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Up & Running

Getting DataCAD up and running is a simple matter with its flexible and interactive setup program. Use the "[System Requirements](#)" and "[Pre-Installation Checklist](#)" in this chapter to make sure you're ready to begin installing the program. The setup instructions that follow take you step-by-step through the installation process.

To get the most from this manual, please refer to "[How to Use This Manual](#)" in this chapter. Please note that the online help included with DataCAD contains all the information included in this manual, along with some updated material that became available after this manual went to press.

DATA CAD LLC offers 30 days of free installation support when you purchase DataCAD. If you have any questions about installing DataCAD, you can contact DATA CAD LLC Technical Support Monday through Friday, 8:00 a.m. – 5:00 p.m. eastern time, except during normally observed holidays, via phone at (860) 677-2829, via fax at (860) 677-2883, or via e-mail at techsupport@datacad.com.

In this chapter:

- ⊕ How to use this manual
- ⊕ System requirements
- ⊕ Pre-installation checklist
- ⊕ Installing DataCAD

How to Use This Manual

This manual begins with two chapters introducing DataCAD to you. "[The Drawing Board](#)" chapter covers such topics as opening and saving drawings, using coordinate entry, and common menus in the program.

Following this introduction, four additional chapters guide you through a basic tutorial, designed to quickly teach you how to use DataCAD's basic functions to produce a plan drawing. All of these exercises focus on a single project.

The remainder of the manual gives detailed explanations as well as step-by-step instructions on how to use each DataCAD function and menu to complete drawing tasks. These instructions may include typing words or characters, using [keyboard shortcuts](#), selecting entities, and choosing menu options.

Words or characters to be typed are printed in bold; type them exactly as they appear. Keys to be pressed are shown just as they appear on your keyboard, with parentheses around the character(s), as shown below:

Press (Enter).

Press (Ctrl) + (M).

Combinations like the second example above should be executed by pressing and holding down the first key, pressing the second key, then releasing both simultaneously.

Some instructions ask you to "click" or "right-click" on a button or icon. To click on a button, move your mouse until your cursor arrow is over the button and then press the left mouse button once. To right-click, press the right mouse button once. Right-clicking generally doesn't require specific placement of the cursor arrow; however, it must be somewhere over the DataCAD window.

Getting Help in DataCAD

DataCAD includes online help, accessible by pressing [Ctrl] + [F1] or using the Help Menu in DataCAD. The online help closely follows this Reference Manual and includes a table of contents, an index, searching capabilities, and an annotation feature to let you make your own notes directly in the online help. The online help may also include information that became available after this manual went to press.

You can also access the online help from outside of DataCAD. Use Windows Explorer to open the DATACAD\HELP folder and then double-click on DCADWIN.HLP.

Installing DataCAD

To install DataCAD on your computer, first review the "[System Requirements](#)" and "[Pre-Installation Checklist](#)" sections below to make sure you have everything you need to install the program. Then follow the instructions in "[Running the Setup Program](#)" on the next page.

Information in this section requires you to be familiar with your computer hardware and your computer's operating system (Microsoft Windows 95, 98, or ME, Windows NT 4.0 or later, or Windows 2000).

To check available free hard disk space prior to installation, double-click on the My Computer icon on your Windows desktop. Right-click on the drive you want to install DataCAD on and select Properties. The hard disk space already in use will be displayed as well as the amount that is still free.

System Requirements

Make sure you have the following minimum requirements to install and run DataCAD for Windows:

- Personal or multimedia computer with a Pentium-based processor (Pentium II recommended)
- Windows 95 or later or Windows NT 4.0 or later
- 64MB of memory for use on Windows 95, Windows 98, or Windows ME (128MB recommended); 128MB for use on Windows NT workstation, Windows 2000, or Windows XP (256MB recommended)
- Minimum of 100MB free hard disk space. 300MB is recommended for a complete installation. If you do not have enough free disk space in your computer when you try to run DataCAD, the message *Not enough room to create swap files* is displayed. Free up disk space and then restart DataCAD.

To check available free hard disk space prior to installation, double-click on the My Computer icon on your Windows desktop. Right-click on the drive you want to install DataCAD on and select Properties. The hard disk space already in use will be displayed as well as the amount that is still free.

- CD-ROM drive
- VGA, or better, resolution graphics card capable of displaying 16-bit color or better for using the Shader
- Microsoft-compatible mouse or digitizer (3-button mouse recommended)

Pre-installation Checklist

- Do you have the minimum system requirements for DataCAD?
- Where do you want to install DataCAD? If you're upgrading your current version of DataCAD, install the new version to the same drive and folder so that the program is properly updated.
- If you're upgrading your current version of DataCAD, have you made a copy of your DataCAD folder to back up all your files before installing the new version? This is simply a precaution against losing any of your drawing files or customized settings.
- Remember to fill out and send in your registration card to qualify for technical support benefits and to receive new product information as it becomes available.

Running the Setup Program

The DataCAD setup program guides you step-by-step through installing DataCAD. Please exit all Windows applications completely before running the DataCAD setup program. If your computer uses Windows NT or 2000, you must have Administrative Privileges to run this setup program. (This means you must be logged on as the Administrator.)

1. Place the DataCAD CD-ROM in your CD-ROM drive; the setup program should start automatically. If it doesn't, click on Run in the Start menu. In the Open input box, type **e:\setup**, where "e" is the letter of your CD-ROM drive. Press **[Enter]** or click on OK. To cancel the installation at any time, press **[Esc]** or click Cancel.
2. The setup program begins. Simply follow the instructions on the screen. To properly update your current version of DataCAD, install this new version to the same folder.
3. Once setup is complete, you can view the ReadMe file, which contains important information about DataCAD that is not documented anywhere else. Click Finish to complete the setup program and open the ReadMe file.

The setup program will ask you to enter your serial number. Your serial number is printed on your registration card and looks similar to DCXW12345; the five characters after the hyphen are always numbers. Be sure not to substitute the letter "O" for a zero when entering your serial number.

The Drawing Board

A user-oriented menu system makes DataCAD easy to learn and convenient to use. Because it was developed by architects for architecture, the program focuses on design, not computer technology. Even draftsmen and designers with no previous computer experience can learn DataCAD quickly.

DataCAD's dual menu system combines familiar Windows menus in the Menu Bar with a comprehensive, nested menu structure in the Menu Window. This unique system, along with toolbars and numerous keyboard shortcuts, allows constant access to the tools you use to draw, instead of covering your drawing area with menus that are only temporarily visible.

From your initial design work, massing studies, and interior perspectives to your final set of construction drawings, DataCAD provides you with the tools you need to produce quality work quickly, accurately, and with greater flexibility than ever before.

In this chapter:

- ⊕ Creating drawings
- ⊕ Opening drawings
- ⊕ Importing and exporting drawings
- ⊕ The DataCAD screen
- ⊕ Common menus found throughout DataCAD
- ⊕ Planning your drawing project
- ⊕ Basic drawing and editing
- ⊕ Viewing your project
- ⊕ Referencing drawings
- ⊕ Naming and saving files

Opening a Drawing

DataCAD has four options for opening a drawing:

- opening an existing DataCAD drawing
- creating a new DataCAD drawing
- creating a new DataCAD drawing using a default drawing
- importing a DWG or DXF drawing file

If DataCAD does not start, review Chapter 1 to make sure you meet all [system requirements](#). You have 30 days of free installation support; see the first page of Chapter 1 for details on how to [contact us](#).

When you open a file, you may see *Attention! The drawing file 'xxx' is currently marked "In Use." Continuing will result in data loss. Are you sure you want to continue?* This happens if someone on your network has the file open or if the file was exited abnormally the last time it was used. If somebody else is currently working on the file, click No to cancel.

If nobody else is working on a file and you get the *Attention . . .* message, what should you do? That depends on what happened to the file in an earlier drawing session. Often, there is an autosave (.ASV) copy of the drawing. For more information about autosave files, see “More About Recovering Lost Work” near the end of this chapter. If you get the *Attention . . .* message but cannot find an autosave (.ASV) copy of the drawing, try opening the drawing and using Save As in the File pull-down menu to give it a new name.

Opening an Existing Drawing

You can open a DataCAD drawing when you first start DataCAD or while you're working on another DataCAD drawing.

➔ To open an existing DataCAD drawing:

1. Start DataCAD. If DataCAD is already running, click Open in the File pull-down menu. A dialog box opens, listing all files in the current folder.
2. Select the type of file you want to open from the Files of Type drop-down box. You can open any DataCAD, DataCAD Plus, backup, or autosave files.
3. Click on the drawing you want to open to highlight it. If the Show Preview option is checked, this drawing will be displayed in the preview window on the right side of the dialog box.
4. Click on Open or press (Enter) on your keyboard. The drawing is opened. If a message displays alerting you to orphaned reference files, see the “Referencing Drawings” section in this chapter for instructions on resolving these orphaned files.

Working with Folders in DataCAD's Save and Open Dialog Boxes

You'll probably want to store your DataCAD drawings for each project in separate folders. If you have two projects, you might create two folders named PROJECT1 and PROJECT2 in DataCAD's \DWG folder.

➔ To create a new folder in DataCAD:

1. Start DataCAD or click on New in the File pull-down menu.
2. Click on the Create New Folder icon in the "Enter name of drawing file to edit" dialog box. The Create New Folder icon is at the top of the dialog box. A new folder is displayed in the dialog box.
3. Type the name of the folder and press **[Enter]**.

A new folder contains nothing when you create it. Only after you save drawings to that folder will the drawing filenames appear in the dialog box.

You'll also want to save files to or open files in a different folder than what's displayed in the dialog box. To do this, you'll first have to change folders.

➔ To change folders in a dialog box, do one of the following:

- Click on the arrow for the "Look in:" drop-down box and select a new folder; or
- Click on the Up One Level icon and select a new folder from the list; or
- Double-click on a folder to open it.

Creating a New Drawing

You must name your new drawing before DataCAD will open it.

➔ To create a new drawing:

1. Start DataCAD; if DataCAD is already running, click on New in the File pull-down menu. A dialog box is displayed with the title "Enter name of drawing file to create."

Shortcut: Press **[Ctrl] + [N]** to start a new drawing.

2. Type a name for your new drawing in the Filename input box. The drawing name can be up to 80 characters long and consist of spaces, letters, numbers, and any other characters except the following: + / \ [] ; : . ,
3. Click on Create or press **[Enter]**. The new drawing is opened.

The minimize button in the upper-right corner of every drawing window is disabled. But you *can* minimize the DataCAD program window.

Creating a New Drawing Using a Default Drawing

A default drawing is used as a template for creating a new drawing. Simply choose the default drawing and then create your new drawing. Or you can use the Default Drawing setting in Program Preferences to automatically use a selected default drawing every time you start a new drawing. All DataCAD drawings have a .DC5 file extension.

➔ To create a new drawing using a default drawing:

1. Start DataCAD; if DataCAD is already running, click on New in the File pull-down menu. A dialog box is displayed, prompting you to *Enter name of drawing file to create*.
2. Click on the Default button. A dialog box is displayed, prompting you to *Choose new default drawing name*. This Default button is not available if the Use Default Drawing option is not checked. This option is checked (on) by default.
3. Click on the default drawing you'd like to use and then click on Open. The dialog box closes. You've just told DataCAD which default drawing to use to create your new file. Now you must create the new drawing file.
4. Type a name for the new drawing you'd like to create in the Filename input box of the "Enter name of drawing file to create" dialog box.
5. Click on Create. Your new drawing is created, with all the settings of the default drawing chosen in step 3.

If no default drawing is specified in the Program Preferences dialog box or through the File/New dialog box, DataCAD uses its internal default settings for the new drawing. To use these settings, either toggle off Use Default Drawing on the Misc. tab in the Program Preferences dialog box or toggle off "Use default drawing" in the Create New File dialog box. Also, make sure the Use Default File option is unchecked in the File/New dialog box.

➔ To automatically use the same default drawing for every new drawing:

1. Open any drawing file. (The Tools pull-down menu is not available unless a drawing is open.)
2. Click on Program Preferences in the Tools pull-down menu.
3. Click on the Misc. tab.
4. Choose a default drawing from the Default Drawing File drop-down box.
3. Click on OK to save your new settings and close the dialog box. The default drawing you chose will now automatically be used each time you start a new drawing, unless you specify a different default drawing using the Default button in the File/New dialog box, as described in the preceding instructions.

More About Default Drawings

Default drawing files are really just regular DataCAD drawing files that are stored in your \DATACAD\DEFAULT folder and used in a different way than your regular drawing files. A default drawing contains predefined information and settings and is used much like a template for creating new drawings. You can create your own default drawings with system settings, layer names and colors, text font and size, dimension settings, display options, scale type, object snap settings, linetype spacing, and more already set; so using a default drawing can save you time in setting up each new drawing.

Setting up a system of default drawing files at different scales and for different types of work can significantly reduce production time for drawings and provide consistency in work done by different designers in your office.

➔ To create a new default drawing:

1. Create a new drawing as described earlier in this chapter.
2. Set whatever options you'd like your new drawings to have, including grid settings and layers. See the "Before You Draw" chapter for more information about many of these settings.
3. Save the file in your \DATACAD\DEFAULT folder. It's ready to use whenever you need to create a new drawing.

Importing a DWG or DXF Drawing File

DataCAD's drawing translator can import DWG and DXF files compatible with AutoCAD 12, 13, and 14 as well as AutoCAD 2000. You can choose an automatic translation or an interactive one, where you can decide how each linetype and font should be translated and displayed in DataCAD.

➔ To import a DWG or DXF file and open it in DataCAD:

1. Create a new DataCAD drawing, as described in "Creating a New Drawing" earlier in this chapter.
2. Click on Import in the File pull-down menu. The Import dialog box is displayed.
3. Look at the options in the Files of Type drop-down box. Select Supported File Formats to display all DWG and DXF files, select DWG to display all DWG files, or select DXF to display all DXF files.
4. Select a file to import from the list or type the name of the file to import in the File name input box. By default, DataCAD looks for DXF and DWG files in your DATACAD\XFER folder. If your DXF or DWG files are in a different folder, change to that folder and then click on the file you want to import.
5. Click Open. The DWG Import Entity Assignments dialog box is displayed if the Import section of the DXF/DWG tab of the Program Preferences dialog shows that "Interactively assign . . ." is toggled on. The options in the DWG Import Entity Assignments dialog box allow you to choose how DataCAD will translate linetypes and fonts.

6. Look at the options in the Line Type Assignments box; this lists each AutoCAD linetype used in the file and suggests a DataCAD equivalent. To change any of these assignments, click on the linetype to change. Immediately below the Assignment list box, your selection is reproduced, with the translation linetype in a drop-down box. Choose a new linetype from the drop-down box; your selection is changed in the linetype list box.
7. Repeat step 6 for the Font Assignments.
8. Click on OK to close the dialog box and begin importing the file. Progress is noted in the Message Area. Or click on Cancel or press **[End]** to cancel the import and close the dialog box.

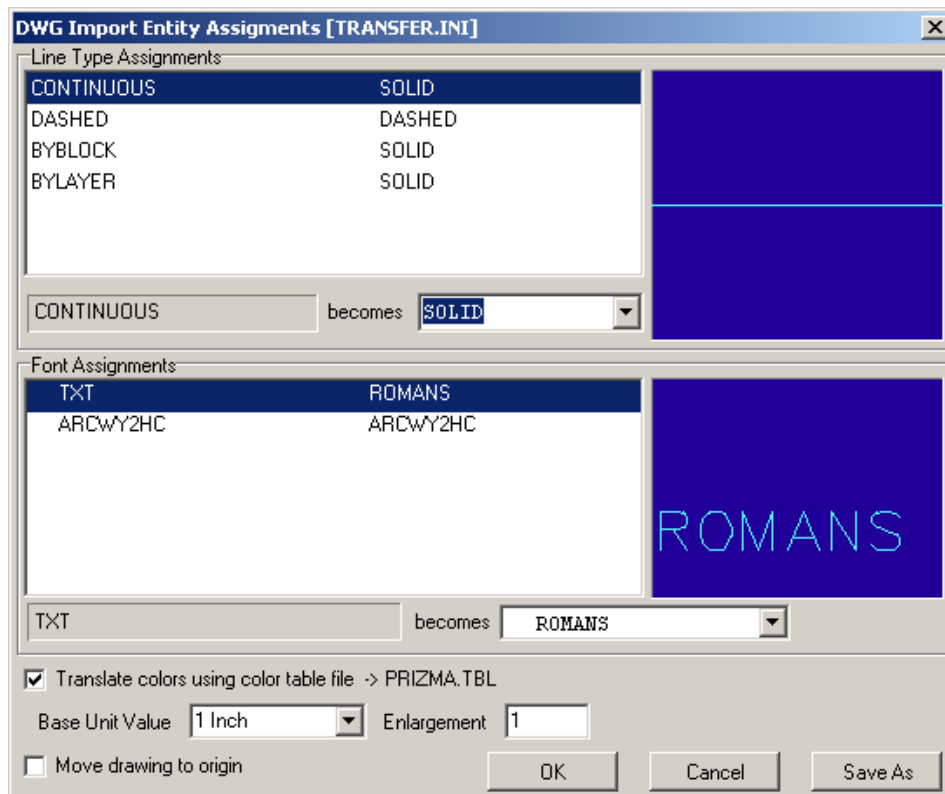


Figure 2.1: The Import Entity Assignments dialog box

More About Importing Files Into DataCAD

The settings and assignments that are displayed in the Entity Assignments dialog box are contained in a file called TRANSFER.INI in your DATACAD\SUP folder. If you change any of the settings in the Entity Assignments dialog box, you have the option to save those settings in a new .INI file to use when translating other files. This new .INI file is saved in the \SUP folder. You can affect how importing files and the Entity Assignments dialog box work through settings on the DXF/DWG tab in the Tools/Program Preferences dialog box.

This could be useful if you normally exchange files with several different offices. If you change the settings and save a separate .INI file for each office, then you can easily switch between .INI files without having to go through entity assignments over and over again.

➔ To save entity assignments:

1. Change settings and assignments in the Entity Assignments dialog box as described in the step-by-step instructions on importing DWG files.
2. Click on Save As. The Save Settings As dialog box is displayed.
3. Type a filename for the new .INI file and click on Save. Or click on Cancel to cancel the save. When you save the .INI file, the setting for the Translator Settings File on the DXF/DWG tab of the Preferences/Options dialog box is changed to that file.
4. Continue with step 8 in the instructions for importing a DWG or DXF file and opening it in DataCAD.

A Brief Tour

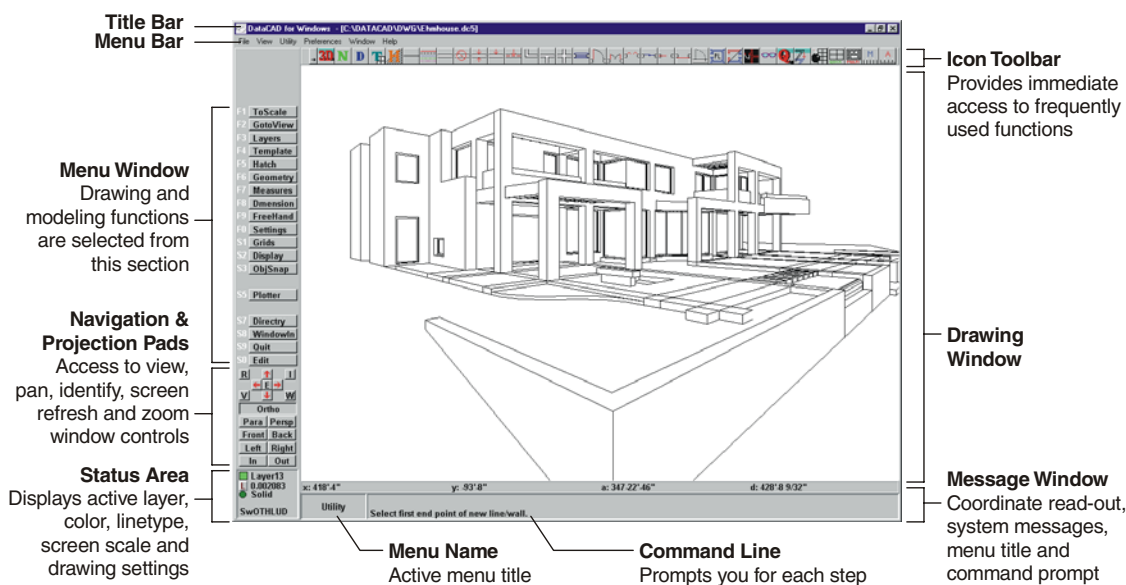


Figure 2.2: The DataCAD screen

The Menu Bar

The Menu Bar is located along the top of the DataCAD screen and contains the File, Edit, View, Insert, Create, Tools, Macros, Window, and Help pull-down menus. These menus give you quick access to standard Windows functions as well as many of DataCAD's most used features.

Notice that some menu options have keyboard shortcuts associated with them. For instance, the Open option in the File menu has **[Ctrl]+[O]**. Instead of using the mouse to choose a menu option, you can simply press the keyboard shortcut to quickly do the same thing.

➔ To choose an option from the Menu Bar:

1. Click on the menu name in the Menu Bar. The menu will drop down, displaying all the options in it.

2. Move your mouse down the menu to the option you want. Notice that each option highlights as your mouse passes over it.
3. Click on the option you want to select.

Toolbars

Toolbars are displayed across the top of the DataCAD window, between the Menu Bar and Drawing Area. It gives you a quick way to access a variety of DataCAD functions, no matter where you are in the program. Icons, representing various DataCAD options, make up the toolbar. If you move your cursor over an icon, a description of that icon, called a *ToolTip*, pops up near the icon; the description is also displayed in the top line of the Message Area. DataCAD's standard toolbars are anchored above the Drawing Area; you cannot move them to a different screen position.

Notice that as you click on different icons, the menu for that function is displayed in the Menu Window. Accessing menus using the toolbar instead of the Menu Window reduces the number of steps to get to a menu and thus saves you time.

For more information on how to turn off display of the toolbar or how to choose which toolbar is displayed, see "Customizing DataCAD" in this chapter. There are even icons in some toolbars that let you switch to a new toolbar.

Because a toolbar can contain more icons than will fit on the screen, left and right arrow icons are available at the far left of the toolbar for easy scrolling through the toolbar icons. The maximum number of toolbar icons is 100.

In addition to the standard DataCAD toolbars, you can also use an undo/redo toolbar and context-sensitive viewing toolbars. You can activate the viewing toolbars by choosing Viewer Toolbar from the View/Toolbars pull-down menu. A different toolbar is displayed depending on the view of your model: you can use the orthographic view toolbar, the parallel view toolbar, or the perspective view toolbar.

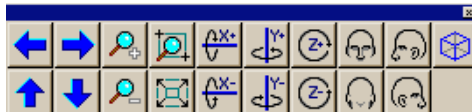


Figure 2.3: Parallel views toolbar



Figure 2.4: Undo/redo toolbar



Figure 2.5: Perspective view toolbar



Figure 2.6: Plan view toolbar

The Menu Window

While DataCAD's Menu Bar has menus that contain shortcuts to many of DataCAD's most frequently used functions as well as basic Window options, the majority of DataCAD's menus appear in the Menu Window. The name of the menu shown in the Menu Window appears at the left end of the Message Area.

There are two main menus in the Menu Window: Edit and Utility. To switch between these two menus, click on the menu option in the Menu Window or simply right-click anywhere in the DataCAD window. Open a drawing in DataCAD and try right-clicking a few times; notice the menu selection changes between the Edit and Utility menus.

Both 2D and 3D menus are displayed in the Menu Window, but these menus are not interchangeable. For example, the Move command in the Edit menu should not be confused with the Move option in the 3D Edit menu.

Menus in the Menu Window are nested. As you can see in Figure 2.2, there are many submenus available from the Edit and Utility menus. Each of these submenus may, in turn, contain additional submenus and menu options. When a submenu is being displayed, right-clicking exits the submenu and displays the previous menu in the Menu Window.

Shortcut: Press **[PgUp]** and **[PgDn]** to scroll forward and backward through menu lists. This only works when you're in a menu with **ScrlFwrd** and **ScrlBack** options. At all other times, these shortcuts are used to change your drawing display scale.

The Menu Window lists the options in the currently selected menu. If a menu contains more items than can be shown in a single list, a scroll forward option (**ScrlFwrd**) is displayed at the bottom of the menu and will be available until you reach the end of the list. Once you scroll forward in a list, a scroll back option (**ScrlBack**) will also be available. As you move through DataCAD's menu structure, the selections in the Menu Window change to reflect the different commands associated with each option.

Some DataCAD menu options are *toggles*, switches that can be turned on or off. When a toggle is on, the button appears depressed and the button name is green. When a toggle is off, the button appears raised and the button name is red.

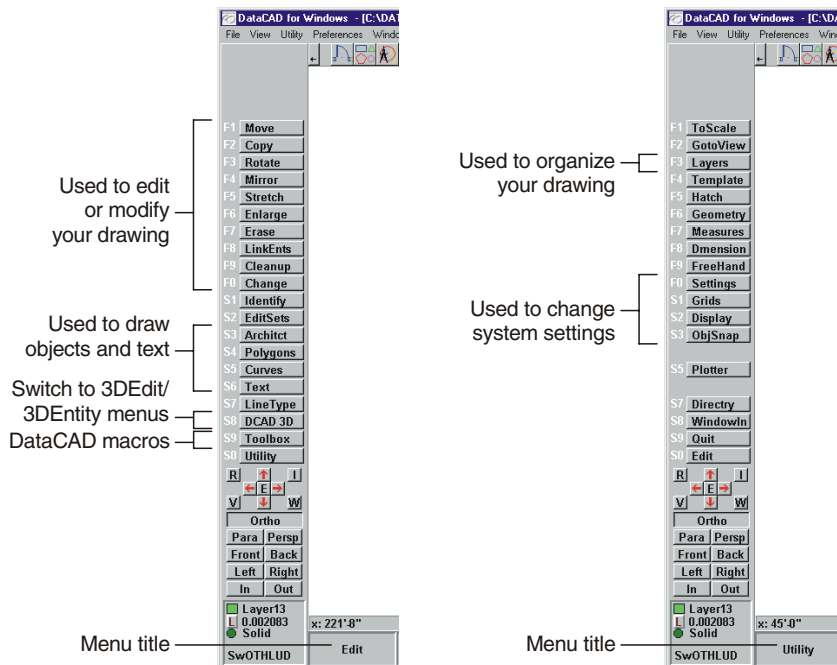


Figure 2.7: The Edit menu (left) and Utility menu

Choosing Menu Options

There are three ways to choose a menu option:

- click on it using your mouse
- press the corresponding function key
- use a keyboard shortcut or command line alias

Many of the most productive DataCAD users use keyboard shortcuts to enter values and choose menu options while using the mouse to draw and enter other drawing information. With one hand on the keyboard and the other on the mouse, you can draw in DataCAD very efficiently and quickly.

➔ To choose a menu option using the mouse:

1. Move the cursor over the option in the Menu Window that you'd like to use. The menu option highlights as the mouse cursor passes over it.
2. Click the left mouse button.
3. Right-click (click the right mouse button) when you are finished using the function; this lets you exit the menu.

Notice that immediately to the left of each menu option in the Menu Window is a label corresponding to the function keys on your keyboard. Function keys are usually located across the top or on the left side of your keyboard and are labeled as "F1," "F2," "F3," etc. To choose a menu option, simply press the corresponding function key. If a menu option's function key label is S1 through S0, press **[Shift]** + **[F1]** through **[Shift]** + **[F10]**.

Keyboard shortcuts let you access a variety of functions quickly just by pressing one or two keys. For example, you can press **[A]** on the keyboard to open the Architect menu *from anywhere in DataCAD* instead of using the mouse to choose the option from the icon toolbar or Menu Window.

In this manual, all keyboard shortcuts are described exactly how they should be executed. If a shortcut requires a lowercase letter, you will be instructed to press only that letter. If a shortcut requires an uppercase letter, you will be instructed to press **[Shift]** and that letter simultaneously. For instance, the snap grid can be toggled on and off by pressing **[X]** (a lowercase "x"); however, to open the Object Snap menu via the keyboard, you must use an uppercase "X", so you must press **[Shift]** + **[X]**.

There are several shortcuts that will be particularly useful to you as you draw:

SHORTCUT FUNCTION

[Home]	Centers a drawing or view on the screen at the cursor's current position. To reposition a drawing, move the cursor to the new center and press [Home] .
[.]	Deletes only the last entity you drew
[<]	Deletes the last group you drew; you can also use Undo in the Edit pull-down menu
[.]	Restores only the last entity you deleted. This shortcut does not undo your previous actions; it just restores the deleted entity.
[>]	Restores the last group you deleted; you can also use Redo in the Edit pull-down menu. This shortcut does not redo your previous actions; it just restores the deleted group.
[Esc]	Redraws the screen to display the most recent version of the drawing

- End** Terminates the screen refresh on the active layer only. To terminate the screen refresh on additional layers, press **End** once for each additional layer. **End** does not affect the drawing file, only that portion of it which is displayed. Save time on large drawings. Once the part of the drawing you want to edit is redrawn, press **End** to cancel the rest of the refresh.
- Delete** Terminates the screen refresh on all layers simultaneously
- PgUp**, **PgDn** Changes the display scale in increments specified in the ToScale menu. To zoom out, press **PgUp**; to zoom in, press **PgDn**.
- P** Restores the previous view and returns to the previous scale in one keystroke. DataCAD remembers up to 25 previous views.

Some keyboard shortcuts can yield unexpected results if you don't clearly understand what the command does. For example, there's a difference between Undo/Redo and Restore commands. The results you get with the keyboard shortcuts (,), (<), (.), and (>) are not the same as what happens when you use Undo or Redo. The keyboard shortcut for Undo (which negates the last command you used) is (Ctrl)+(z); this produces the same results as selecting Undo from the Edit pull-down menu or clicking the Back button on the Undo/Redo Toolbar. The keyboard shortcut for Redo (which reinstates the last command undone by using Undo) is (Ctrl)+(y); this produces the same results as selecting Redo from the Edit pull-down menu or clicking the Forward button on the Undo/Redo Toolbar.

The Appendix has a complete list of all keyboard shortcuts in DataCAD plus instructions on how to program your own shortcuts.

Using the Command Line Alias

The Command Line Alias lets you assign a keyboard shortcut to a single or multi-character shortcut. You can have unlimited multi-keystroke shortcuts.

➔ To use a command line alias:

1. Press **Ctrl** + **Spacebar**. You are prompted to enter the alias you want to use.
2. Choose an alias in either of these two ways:
 - Type the alias you want to use and press **Enter**.
 - Press **?** and then **Enter** to see the list of available aliases. Highlight the alias you want to use and press **Enter** or simply double-click on the alias.

Customizing the Alias Browse Box

The number of lines shown in the list as well as its width can be adjusted in the [Alias] section of the DCADWIN.INI file.

➔ To customize the alias list box:

1. Look in the Windows folder (normally located on your C:\ drive) and locate DCADWIN.INI and open it in a text editor, such as Notepad.
2. Find the Alias section, as shown below:

```
[Alias]
```

```
Popup Width=300  
Popup Lines=4
```

3. Change the width and lines values as necessary and then save the file.

Making an Alias

Aliases are stored in a text file in your SUP folder called DCADWIN.DCA. Aliases use the same shorthand as custom keyboard macros. An item in the Alias file looks like this:

```
[cr]  
comment=draw circle by radius  
alias=^;^S5^S1^
```

➔ To make an alias called “ep” that will take you to the Erase, Partial menu:

1. Press (Ctrl) + (Spacebar).
2. Click on Settings; the Alias Manager appears.
3. Go to the Alias input box; type the shortcut you want to use, in this case **ep**.
4. Go to the Comment box; type a hint to describe the alias: **Erase Partial**
5. Go to the Sequence box; type the required keyboard sequence for the command: **^;^F7^F9^**. Note that the format for a Command Line Alias is the same as a Keyboard Macro. You must separate each function with a carat [^].
6. Click on Add to add the new alias to the list.
7. Click on OK to close the Alias Manager. Your alias is now ready to use.

The Navigation Pad

The Navigation Pad is displayed below the Menu Window and provides an easy way to view your model, recalculate extents, and access the 3DViews, Identify, and WindowIn menus.

Both the Navigation Pad and the Projection Pad appear under the menu buttons if there is enough room. If the menu buttons require too much space for both pads to appear, only the Navigation Pad will be displayed. You can still access projection options from the View pull-down menu in the Menu Bar. The menu buttons may take up so much room that neither pad can be displayed. Or the menu buttons themselves may be cut off at the top of the DataCAD window. To make more room to display the pads, choose a smaller font for the menu buttons using the Interface Settings tab in the Tools/Program Preferences dialog box. You can also choose a higher screen resolution; see Windows Help for more information on this.

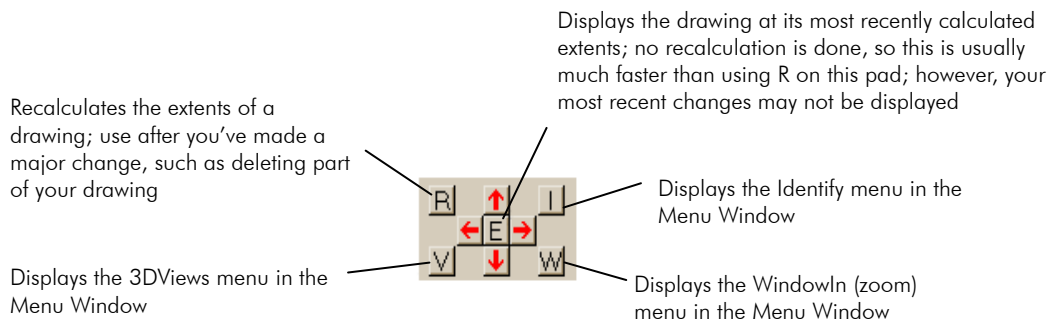


Figure 2.8: Some of the Navigation Pad's buttons can be used to change your view of your drawing.

The Navigation Pad also includes arrow keys to pan, or move the observer around the model on the screen.

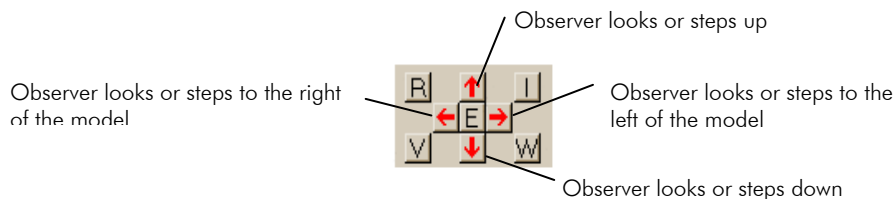


Figure: 2.9: Click the arrow buttons on the Navigation Pad to easily change the view of your model.

You can also use the keyboard arrow keys in the same way you use the Navigation Pad arrows. The pan distance, or how much the display shifts each time you use one of the arrow keys, can be set to any value by using the ScrlDist in the Settings menu. See "Setting the Scroll Distance" in the "Settings" section of the "Before You Draw" chapter.

The Projection Pad

The Projection Pad is a set of nine buttons located directly under the Navigation Pad. These options are also available from the View pull-down menu on the Menu Bar.

Both the Navigation Pad and the Projection Pad appear under the menu buttons if there is enough room. If the menu buttons require too much space for both pads to be displayed, only the Navigation Pad will appear. You can access Projection options from the View pull-down menu in the Menu Bar. The menu buttons may take up so much room that neither pad appears or the menu buttons may be cut off at the top of the DataCAD window. To make more room for the pads, choose a smaller font for the menu buttons using the Interface Settings tab in the Tools/Program Preferences dialog box. You can also choose a higher screen resolution; see Windows Help for more information on this.

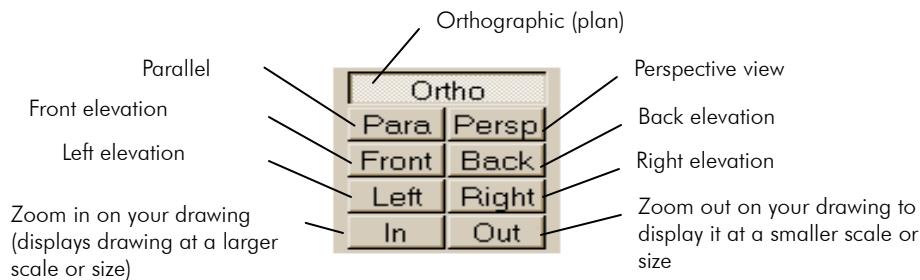


Figure 2.10: The Projection Pad has nine options to change your view of your model.

The Status Area

The Status Area is located in the bottom left corner of the DataCAD window. When you are working on a drawing, this area provides information on the current settings. See “Customizing Display” in the “Before You Draw” chapter for more information on Display menu options.

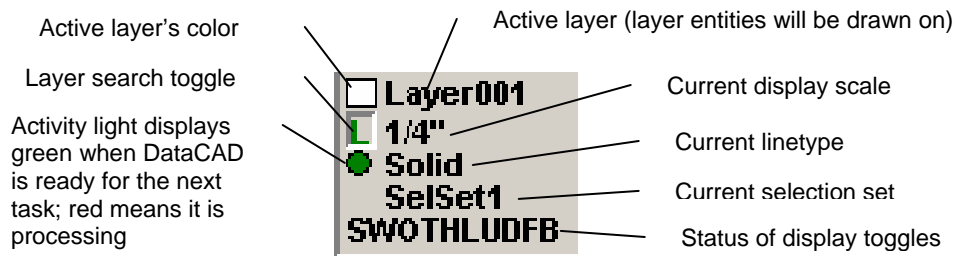


Figure 2.11: The Status Area lets you quickly know information about the active layer and the status of various other settings in DataCAD.

SWOTHLUDFB settings affect how a drawing is plotted as well as how it appears on the screen.

Hotspots

The Status Area gives you one-click access to the Color, Layer, Scale, Linetype, and Selection Set menus. In the Status Area, just left-click on the item (text, color swatch, or square) you want to change. DataCAD instantly takes you to the appropriate menu or dialog box so you can make your new selection. For example, if you click on the active layer’s name, the Layer Manager dialog box appears. Clicking on a hotspot in the Status Area has the following results:

Active layer’s color swatch – the Color Palette appears

Active layer’s name – the Layer Manager appears

Current display scale – the ToScale selection menu appears

Current linetype – the LineType menu appears

Current selection set – the EditSets menu appears

Changing the Active Layer

In a drawing with multiple layers, pressing **[Tab]** changes the active layer by scrolling through all the layers that are on. You'll see the active layer name in the Status Area change each time you press **[Tab]**. Press **[Shift] + [Tab]** to scroll backward through your drawing's layers.

Toggling Layer Search On or Off

You can click on the layer search toggle in the Status Area to quickly toggle layer search on or off. If the toggle button is depressed and the "L" is green, layer search is on; if the toggle button is raised and the "L" is red, layer search is off. We'll discuss DataCAD's layer search option in "Selection Menus" later in this chapter.

Shortcut: Press the apostrophe key (') to toggle the LyrSrch option on or off.

Using SWOTHLUDFB

SWOTHLUDFB in the Status Area lets you know at a glance whether these toggles are on or off. Each letter represents a particular feature in DataCAD. When the letter is in uppercase, the feature it represents is on or active. When the letter is lowercase, the feature is off or inactive. Specific SWOTHLUDFB settings are:

S	Snap grid is on. Your cursor will move along the X and Y axes in increments specified by the SetSnap option in the Utility/Grids/ GridSize menu.
s	Snap grid is off. Your cursor is not restricted to incremental movement; you can draw entities at any point in the Drawing Area.

Shortcut: To toggle snap grid on or off, press **[X]**.

W	Walls mode is on.
w	Walls mode is off.

Shortcut: To toggle wall mode on or off, press **[=]**.

O	Ortho mode is on. Your cursor will move at angle increments specified by the SnapAng option in the Utility/Grids menu.
o	Ortho mode is off. Snap angle settings are not used; you can draw entities at any angle.

Shortcut: To toggle Ortho mode on or off, press **[O]**.

T	Text in your drawing is displayed on the screen as entered.
t	Text in your drawing is represented by appropriately sized boxes.
H	Hatch patterns are displayed.
h	Hatch patterns are not displayed. Drawings refresh faster.
L	Lines are displayed at the weight (thickness) that you drew them.
l	Lines are all displayed with a line weight (thickness) of 1. Drawings refresh faster.
U	Linetypes, or "user-defined lines", are displayed.
u	Linetypes, or "user-defined lines", are all displayed as solid. Drawings refresh faster.
D	Associative dimensions are displayed.
d	Associative dimensions are not displayed. Drawings refresh faster.
F	Fills are displayed
f	Fills are not displayed
B	Bitmaps are displayed

b Bitmaps are not displayed

- ➔ To change the status of SWOTHLUDFB toggles:
- 1. Click on Display in the Utility menu.
 - 2. Click on the appropriate menu option to toggle it on or off.

The Message Area

The Message Area displays information about your drawing. It includes the current coordinate entry mode, the cursor’s position, the menu you are using, program and error messages, the Z-base and Z-height, and program prompts and data entry area.

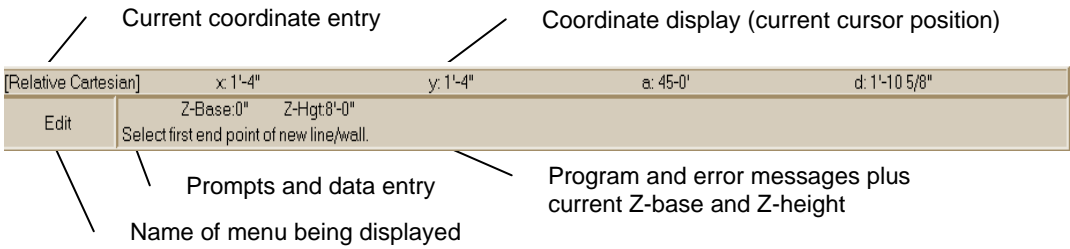


Figure 2.12: The Message Area is located along the bottom of the DataCAD window.

The values of the coordinate display can be shown in architectural, engineering, or metric units. See “Setting Scale and Angle Types” in the “Customizing Settings” section of the “Before You Draw” chapter for more information on how to display values in a different unit of measure.

The Drawing Area

The Drawing Area, the largest in the DataCAD window, displays drawing contents. Lines, arcs, walls, fixtures, and anything else you design are displayed. Although drawings in DataCAD are drawn at full scale in an infinite field, you can display any portion of a drawing at any scale. In other words, you can zoom in and out on your drawing, as needed. To do this, you can use the ToScale and WindowIn options in the Utility menu. See “Viewing Drawings” later in this chapter for more information on zooming in and out on your drawing.

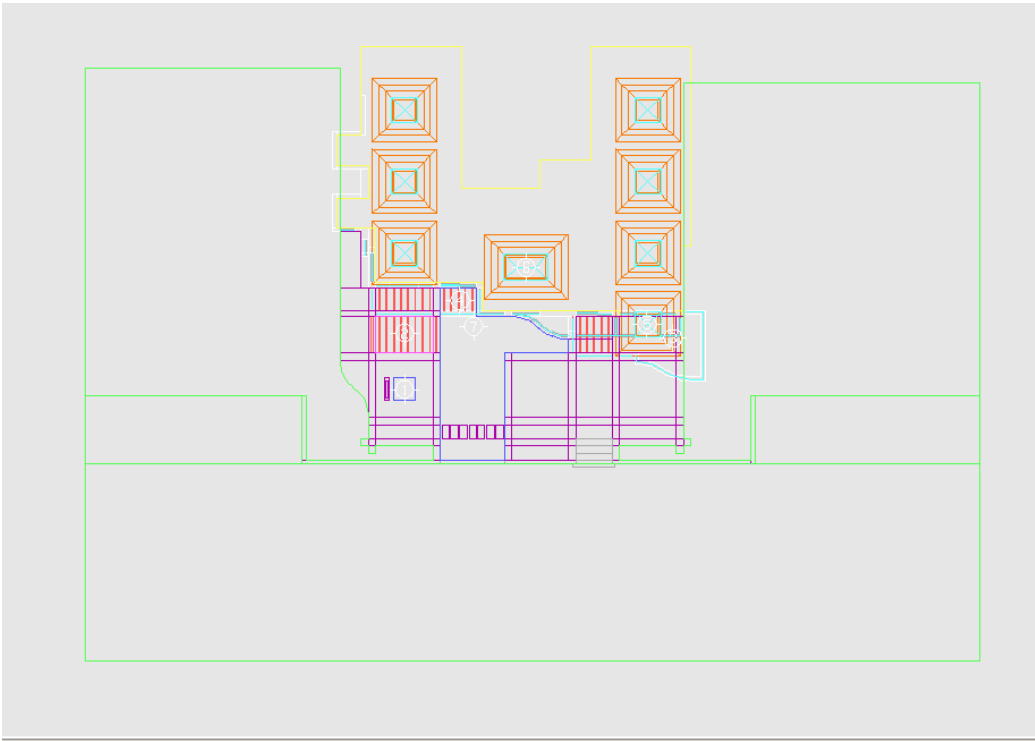


Figure 2.13: The Drawing Area is the largest part of the DataCAD window.

Customizing DataCAD

The Program Preferences dialog box, available from the Tools pull-down menu, lets you customize folders for each DataCAD file type, how the DataCAD screen looks, how the DWG/DXF translator works, and the macros available in the Macros pull-down menu, as well as several miscellaneous settings.

Customizing Pathnames

The Pathnames tab in the Program Preferences dialog box lists the default folders that files are stored in.

To reset all folder names to their defaults, click Defaults in the bottom right corner of the Program Preferences dialog box.

➔ To change a path:

1. Choose Program Preferences from the Tools pull-down menu.
2. Click in the input box of the folder path you'd like to change, and type a new path; or, click on the folder icon at the right of the path to be changed and browse for a new path.
3. Click on OK to save your changes and close the dialog box; click on Defaults to reset all paths to their DataCAD defaults.

If you enter a folder name that doesn't exist, clicking on OK will simply highlight the incorrect folder.

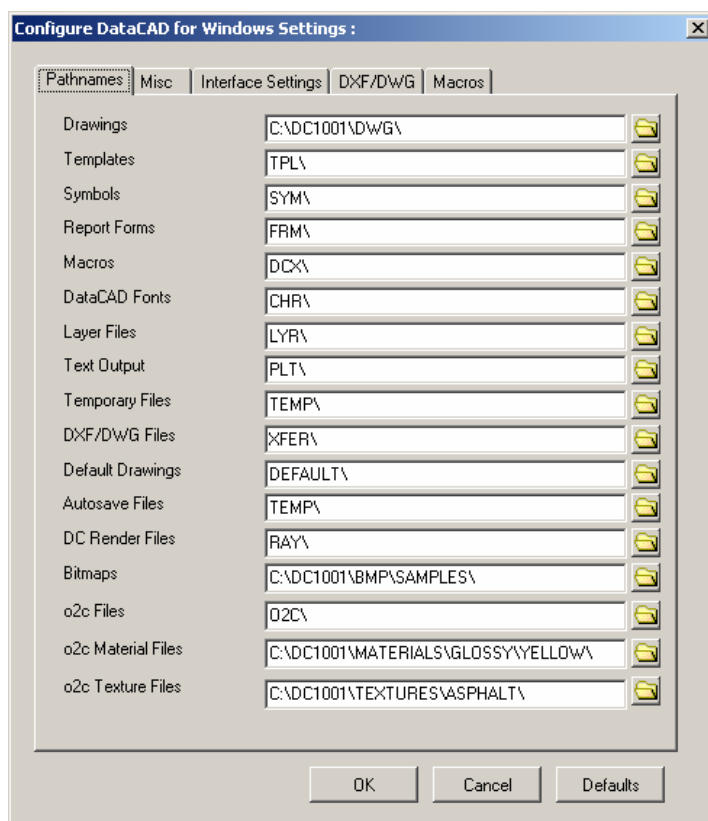


Figure 2.14: The Pathnames tab on the Program Preferences dialog box

DataCAD tries to resolve invalid reference file paths. It does this by checking the current drawing file folder for valid references. If a valid reference file is found in the current drawing file folder, it will be used instead of the file designated in the original path; however, the original path will be retained.

Customizing DataCAD's Interface

The Interface Settings tab on the Program Preferences dialog box contains an interactive copy of the DataCAD window. Using your mouse, move your cursor over the different areas of this image to see Tool Tips. Click on any part of the image to make changes to that part of the interface.

Warning: Make sure the DataCAD window is maximized before you attempt to customize the interface, or erratic behavior may occur when using the program.

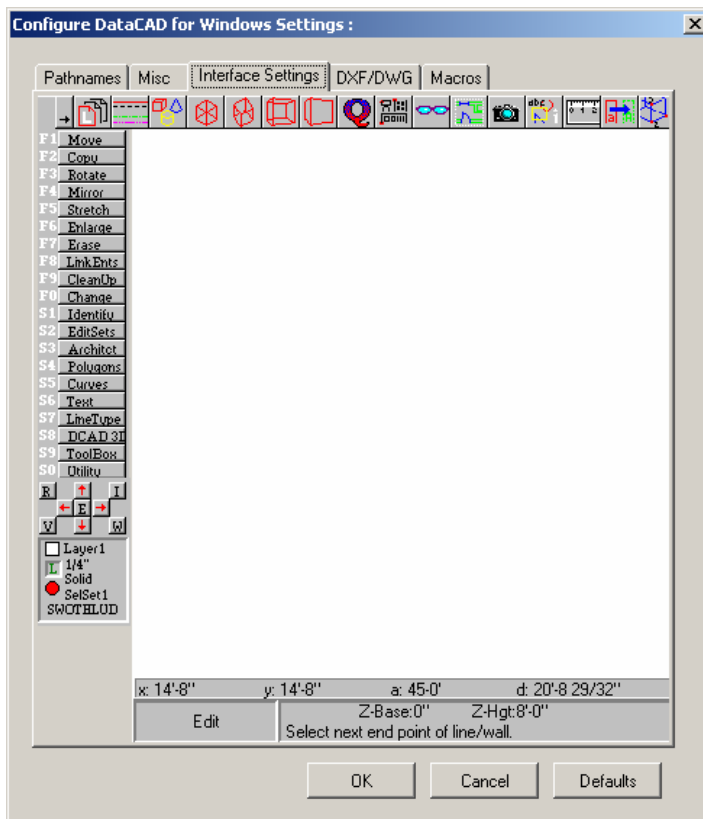


Figure 2.15: The Interface Settings tab on the Program Preferences dialog box

The toolbar is anchored above the Drawing Area; you cannot move it to a different screen position.

➔ To turn the toolbar on or off:

1. Click on Program Preferences in the Tools pull-down menu. The Program Preferences dialog box is displayed.
2. Click on the Interface Settings tab, if it's not already showing.
3. Position your cursor over the toolbar in the interface image or, if the toolbar is already off (not being displayed), above the Drawing Area where the toolbar is normally displayed.
4. Click to toggle the toolbar on or off.
5. Click on OK to close the Program Preferences dialog box. The toolbar display is changed.

You can also turn the toolbar on by checking On/Off in the Default .KEY file section on the Misc tab of the Program Preferences dialog box. Turn the toolbar off by unchecking the On/Off option.

➔ To change the color of the Drawing Area:

1. Click on Program Preferences in the Tools pull-down menu. The Program Preferences dialog box is displayed.
2. Click on the Interface Settings tab, if it's not already showing.

3. Position your cursor anywhere over the Drawing Area in the interface image.
4. Click to open a Color Palette dialog box.

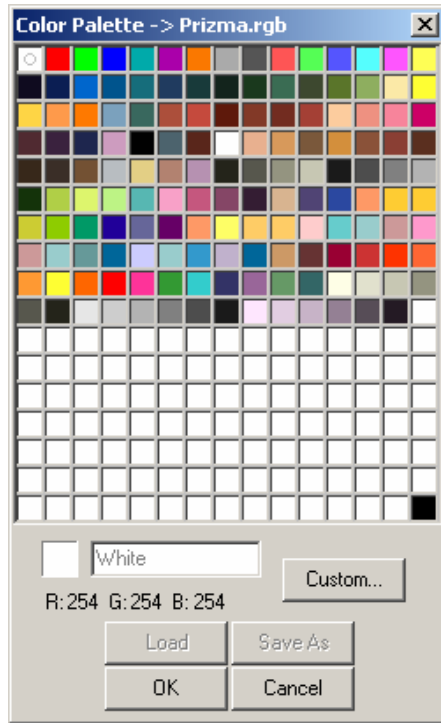


Figure 2.16: The Color Palette dialog box lets you select a new background color for your drawings

5. Choose a new color for the Drawing Area background and click on OK. The Color Palette dialog box closes, and the Drawing Area background color is changed in the Program Preferences Interface Settings display.
 6. Click on OK to close the Program Preferences dialog box. The Drawing Area background color is changed.
- ➔ To move the Menu Window, Navigation Pad, Projection Pad, and Status Area to the opposite side of the DataCAD window:
1. Click on Program Preferences in the Tools pull-down menu. The Program Preferences dialog box is displayed.
 2. Click on the Interface Settings tab, if it's not already showing.
 3. Position your cursor over any part of either pad or the Status Area, or any part of the Menu Window except the menu options.
 4. Click to move these parts of the interface to the opposite side of the DataCAD window.
 5. Click on OK to close the Program Preferences dialog box. The Menu Window is moved to the other side of the window.

- ➔ To change the font used on the menu buttons in the Menu Window or the font used in the Message Area:
1. Click on Program Preferences in the Tools pull-down menu. The Program Preferences dialog box is displayed.
 2. Click on the Interface Settings tab, if it's not already showing.
 3. To change the font in the menu buttons, position your cursor anywhere over the menu buttons in the Menu Window of the interface image. To change the font in the Message Area, position your cursor anywhere over the Message Area of the interface image.
 4. Click to open a Fonts dialog box.
 5. Choose a new font and click on OK. The Fonts dialog box closes.
 6. Click on OK to close the Program Preferences dialog box. The menu button font is changed.

Customizing File Translation

The DXF/DWG tab on the Program Preferences dialog box (Tools pull-down menu) lets you choose an import method, translator default settings, and 3D entity export method. DXF/DWG supports solid fills, TrueType fonts, polylines with voids, and complex hatch patterns (such as StoneCAD).

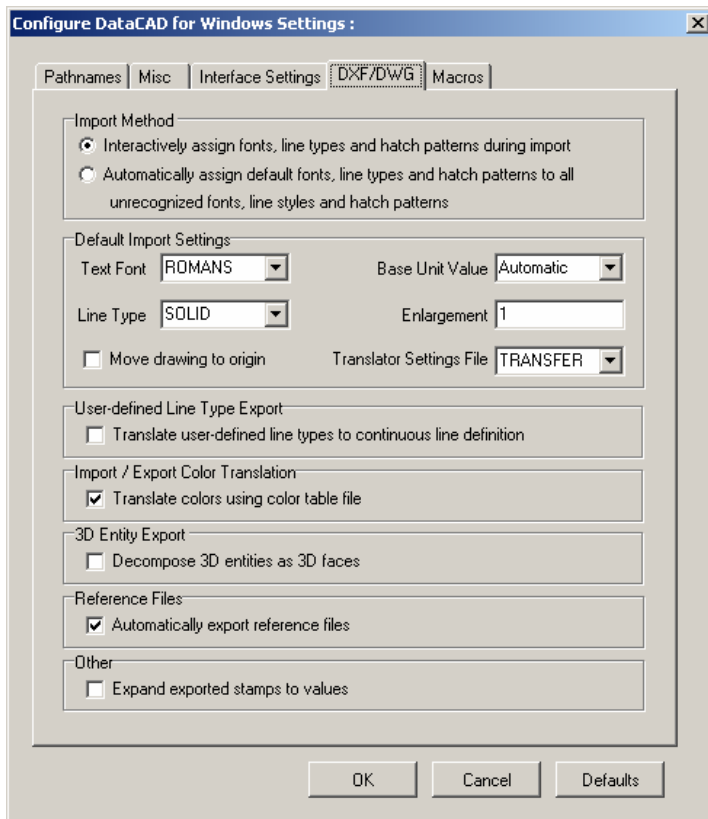


Figure 2.17: The DXF/DWG tab on the Program Preferences dialog box

You can choose between two import methods: the first method allows you to interactively (manually) assign linetypes and fonts, while the second makes these assignments automatically, according to the Translator Default Settings in the Program Preferences dialog box. The interactive option is the default method and uses the DWG Import Entity Assignments dialog box. The DWG Import Entity Assignments dialog box is displayed during the import process only if the interactive option is checked.

There are several Translator Default Settings you can change as well. Set the default font, linetype, base unit, and translator file to use during import. These defaults will be used as assignment suggestions when using the interactive import method; with automatic translation, however, they will be used when DataCAD has no equivalent for an existing entity in the DWG or DXF file. To change any of these settings, click on the arrow to the right of the drop-down box and select another setting. You can also check the “Move drawing to origin” option to move drawing geometry to absolute 0 during import. This option is unchecked (off) by default.

During export, the line definitions and shape codes for DataCAD linetypes are stored in a .SHX file. If you export a DataCAD drawing named MYDRAWING.DC5, the translator will create two files: MYDRAWING.DWG and MYDRAWING.SHX. In order for the linetypes in the DWG file to display properly, the .SHX file must be placed in the \SUPPORT folder in the AutoCAD folder. The AutoCAD user may also place the .SHX file in the same folder as the .DWG file. A .SHX file is only created if line types other than Solid, Dashed, Dotted or Dot-Dash are used.

If you’re not concerned with linetypes displaying exactly as you drew them, check the Export Linetypes toggle in the Tools/Program Preferences dialog box to convert all user-defined linetypes (any linetype except Solid, Dotted, Dashed, and Dot-Dash) to Continuous when exporting a DataCAD drawing to a DWG or DXF file. In this case, a shape file will not be created because none is needed; instead, all user-defined linetypes in MYDRAWING.DC5 will be *permanently* translated to the Continuous linetype. The Export Linetypes toggle is off by default.

Both DataCAD and AutoCAD define colors in terms of their place in each program’s internal color list. For instance, in DataCAD’s code, Color 1 = White, Color 2 = Red, Color 3 = Green, etc. While we see white, red, and green colors on screen, DataCAD recognizes these only as Color 1, Color 2, and Color 3, respectively. AutoCAD works similarly, except AutoCAD displays Color 1, Color 2, and Color 3 as red, yellow, and green, respectively.

So if you don’t have the Import/Export Color Translation toggle checked when you export a DataCAD file, the resulting DWG/DXF file will only associate “Color 1,” “Color 2,” etc., with each entity. When the file is displayed in AutoCAD, the program recognizes “Color 1” and displays the entity as red, instead of white as it was drawn in DataCAD. With the Import/Export Color Translation toggle on, the table file corrects for this, so that white entities in DataCAD are displayed as white when exported to a DWG/DXF file and vice versa.

The Import/Export Color Translation toggle, which is on by default, tells DataCAD to use a table (.TBL) file so that colors translate properly when exported to or imported from a DWG/DXF file. An easy way to create a table file is to simply save DEFAULT.TBL under a new name that matches the custom RGB file. You can select Generate TBL File from the Tools pull-down menu to create a TBL file to match the currently selected RGB file. You must create this file for each custom RGB file. It should be given the same filename as the currently selected RGB file and saved in the DATACAD\SUP folder.

The 3D Entity Export option, located near the bottom of the DXF/DWG tab, is unchecked (off) by default. Any 3D entities in a DataCAD (.DC5) file will be translated as a polyface mesh when exported to a DXF or DWG file; this polyface mesh is a single entity. However, by checking the “Decompose 3D entities as 3D faces” option, each facet of a 3D entity will be turned into an individual polyface; thus, a single sphere in DataCAD can be translated into many separate entities arranged in a sphere shape in a DWG or DXF file.

Checking the Decompose option can make translation faster and the translated entities will look the same after a shade or hidden line removal. However, editing may be more difficult since these translated 3D entities are made up of many smaller and separate entities.

Check the “Automatically export reference files” option in the Reference Files section of the DXF/DWG tab to export any XREFs you’ve used in your drawing at the same time you export the drawing itself. Each reference file is exported as a separate .DWG file. This option is checked by default; if you uncheck it, XREFs will not be exported along with your drawing. If you send your drawing to a co-worker or client, be sure to send all of the exported reference files as well so your drawing will be viewed as you intended.

You can toggle on “Expand exported stamps to values” in the final section of the DXF/DWG tab. When this box is unchecked, the stamp will export in its raw token format (i.e., @PLTTIME). When this box is checked, the stamp will export as it appears in DataCAD (i.e., 12:00 P.M.).

Customizing the Macros Menu

The Macros tab in the Program Preferences dialog box (Tools pull-down menu) lets you choose which macros you want to be displayed in the Macros pull-down menu. To add macros to the menu, click on the macros in the Macros Available list box, then click on the Add button. To delete macros from the menu, click on the macros in the Macros in Menu list box and then click on the Remove button.

The Macros Available list box contains all macros that you have installed (normally found in the DATACAD\DCX folder). It is possible to add macros that are located in other folders: click on the file folder button above the Macros Available list box to browse different folders and select macros to add to the list.

You aren’t uninstalling macros when you remove them from the Macros pull-down menu. Macros removed from the Macros in Menu list simply won’t be displayed as options in the pull-down menu.

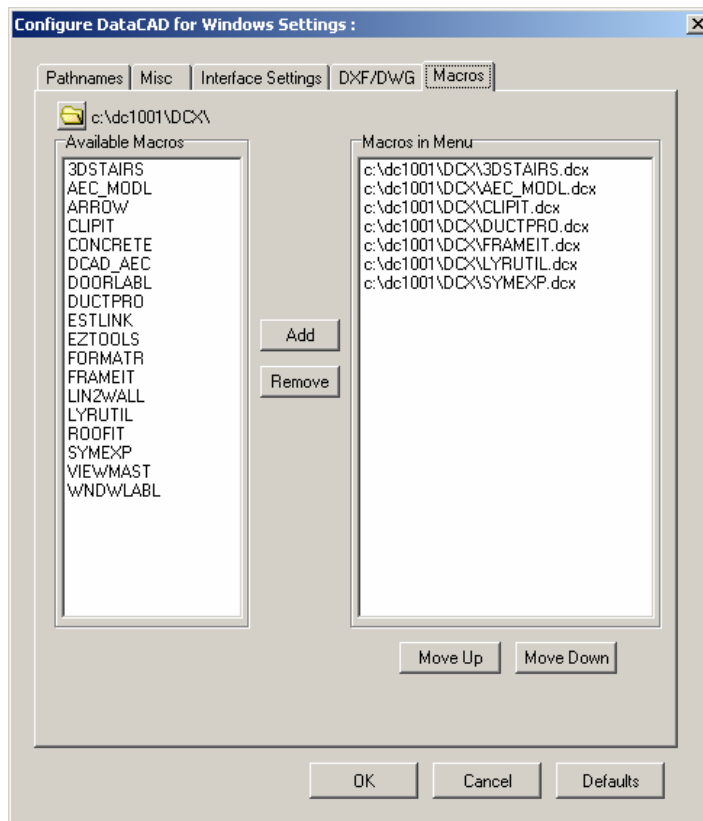


Figure 2.18: The Macros tab in the Program Preferences dialog box

Customizing Miscellaneous Settings

The Misc tab on the Program Preferences dialog box (Tools pull-down menu) has several settings to customize DataCAD's functionality. You can set a default drawing to be used automatically when you create new drawings, select a default toolbar to use, and select a default font for text that you enter into your drawing. Simply click in the appropriate drop-down box and choose a file.

You can activate the Display List to increase display speeds in DataCAD. The *display list* is a sort of internal list that DataCAD keeps of all entities in your drawing. The speed of redrawing the geometry in the Drawing Area when you pan, change views, or press **[Esc]** to update the drawing is increased when this option is checked. Display List is toggled on by default.

Shortcut: Press **[U]** at any time to update the Display List. Note that this is different than pressing **[Esc]** in that **[Esc]** simply redraws the geometry in the current display list while **[U]** updates the display list itself and then redraws all entities.

To find out how many entities and line segments are in the Display List, as well as how much memory is being used to maintain the Display List, choose Status in the Utility / Display / DispList menu.

Although the Max Acceleration toggle next to Display List is unchecked (off) by default, you can check this to increase display speeds even more. To accomplish this, DataCAD takes a few minor shortcuts when redrawing your geometry. For instance, arcs may appear segmented and object snap points may appear enlarged when you zoom in on them. Please note that your geometry has not actually been changed in any way; it is merely a display shortcut. To avoid any display shortcuts, keep Max Acceleration toggled off.

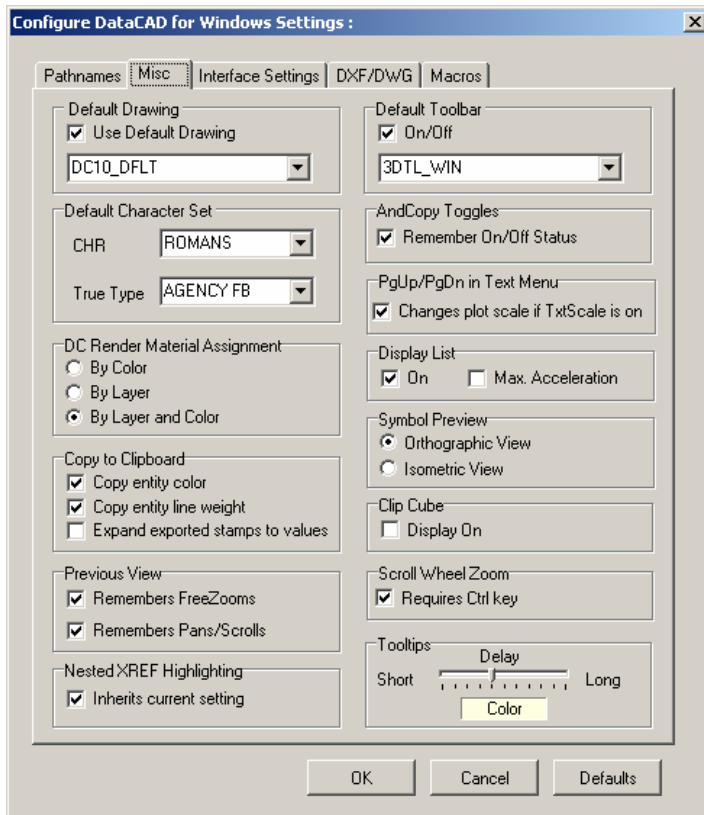


Figure 2.19: The Misc tab in the Program Preferences dialog box

The DC Render Material Assignment section allows you to export .RAY files in which you can assign materials by color, by layer, or by color and layer. Assigning materials by color means you assign one material to all entities of a specified color. Similarly, assigning materials by layer means you assign one material to all entities on a specified layer. And assigning materials by both color and layer means you assign one material to all entities of a specified color on a specified layer. For instance, all red entities on the Walls layer would be assigned one material, all white entities on the Walls layer would be assigned a different material, and all white entities on the Doors layer would be assigned yet a different material.

The Copy to Clipboard section has three options, “Copy entity color,” “Copy entity line weight,” and “Expand exported stamps to values.” The first two options are both checked (on) by default. When you select a DataCAD entity to copy, these options cause the entity’s color and line weight to be copied as well. Then when the entity is pasted into another document, the entity displays as it did in the original drawing.

The “Expand exported stamps to values” option lets you determine the format of stamps when they are either copied to the clipboard or exported via DXF/DWG. When this box is unchecked, the stamp will export in its raw token format (i.e., @PLTTIME). When this box is checked, the stamp will export as it appears in DataCAD (i.e., 12:00 P.M.).

Normally, pressing and changes your drawing scale. If you have PgUp/PgDn in Text Menu checked on the Misc tab and TxtScale is toggled on in the Text menu, pressing and will not only change the drawing scale but also the plotting scale. With TxtScale on in the Text menu, text is always inserted at a size relative to the current plot scale. If the toggle is on in Program Preferences, then you can change the current plot scale by pressing PgUp\PgDn rather than selecting the PltScale button in the Text menu. The PltScale button only appears when the option is OFF in Program Preferences and TxtScale is ON in Text menu.

DataCAD can remember up to 25 previous views (at set scales) of your drawing. If you’d like Free Zooms and pans and scrolls to be included in these previous views, check Remembers FreeZooms and Remembers Pans/Scrolls on the Misc tab.

Shortcut: Press to return to the previous view.

There is also an option to display the boundary of a clip cube. Check Display On in the Clip Cube section of the Misc tab. See the “Viewing Your Model” chapter for more information on using clip cubes.

The AndCopy Toggles option determines whether the status of the AndCopy toggles in such menus as Move, Rotate, Enlarge and Mirror is remembered by DataCAD. By unchecking Remember On/Off Status, the AndCopy option in the menus will always be toggled off when the menu is opened.

DataCAD displays a preview of selected symbols when you insert symbols using the Insert pull-down menu. Using the Symbol Preview setting on the Misc tab, you can choose to display this preview in orthographic or isometric view. You can also change this setting on the Insert Symbol dialog box; check the Isometric View option below the preview window to display symbols in isometric view, or uncheck it to display them in plan view. Symbol previews are displayed in orthographic view by default.

When “Inherits current setting” is unchecked in the Nested XREF Highlighting section, the nested XREFs will retain their highlight settings if you insert them in another drawing file. If the box is checked, the drawing file’s current settings will be used for XREF highlighting.

You can change the color of Tooltips by clicking on the box in that portion of the Misc tab. This brings the Color Palette dialog box to the screen so that you can select the color you prefer. In addition, you can control the delay speed for Tooltips with the slider on the Misc tab.

The Scroll Wheel Zoom section of the Misc tab controls how you zoom in and out using your mouse wheel. As a default, “Requires Ctrl key” is checked. This means that you must press (Ctrl) while you move the mouse wheel to zoom in and out on your drawing. The zoom factor is +/- 25%.

Common Menus in DataCAD

There are several types of menus that appear frequently as submenus throughout DataCAD, including selection, value, and color menus.

Selection Menu

Selection menus appear frequently in submenus throughout DataCAD. *Selection* refers to the way you mark drawing entities for editing purposes. There are three primary ways to select drawing entities: by Entity (one at a time), by Group (when multiple entities have been associated with one another), or by Area (an editing box stretched around entities).

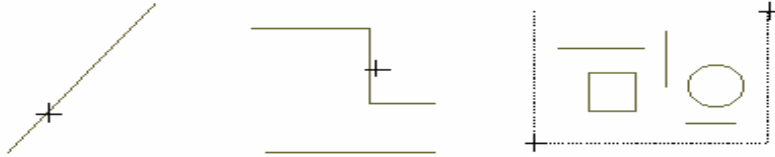


Figure 2.20: From left to right, examples of Entity, Group, and Area selection

Along with Entity, Group, and Area, selection menus include several other methods for selecting parts of the drawing to edit. All of the options are not available in every selection menu; selection menus are tailored for each editing option. Selection menu options can include:

Entity	Select drawing entities one at a time; this is the default selection method
Group	Select entities that are associated as a group
Point	Select entities by a single point; available only when using Stretch
Area	Select geometry wholly contained within an area that you define with a rubberband box
Fence	Select geometry wholly contained within a fence that you define with a temporary polygon boundary
Crossing	Select any entity that crosses an area or fence boundary
SelSet	Select entities that are associated in a selection set
LyrSrch	Editing operations search all layers that are toggled on

More About Selection Menus

An *entity* is a straight or curved geometric element defined by two end points. A line, a circle, an ellipse, and an arc are all entities. A B-spline or Bezier curve or a line of text is also an entity, as is a symbol, associative dimensions, and associative hatching. There are also 3D entities, such as slabs, spheres, and cylinders. Entities can be edited alone or as a group.

A *group* is a set of any number of entities, linked together logically but not necessarily graphically. Entities that do not touch may be part of the same group, and entities that are contiguous may be part of different groups.

Entities are linked as a group in a number of ways. A group of entities drawn in succession without disconnecting the cursor are linked as one group. Similarly a series of lines of text entered at the same time are linked as one group. You can link entities or groups of entities using the LinkEnts option in the Edit menu. Entities are also grouped when they are created at the same time with the Copy option. If the AndCopy option is toggled on when you use Move, then the copies will be a single group. When an existing entity is edited, with the Change or Move options for example, it retains the same group associations.

The grouping together of entities in DataCAD allows you to edit many related items without affecting other entities in the same portion of a drawing. Because groups are also easily redefined, the designer can organize a drawing as a series of elements composed of logically related pieces or entities.

You can also select entities by area or fence. *Area selection* uses a box, which is temporarily drawn around the entities you want to select. All entities and groups enclosed completely within this area box are included in the area selection. Any entities and groups partially enclosed or not enclosed within the area box are not included in the area.

Similar to an area but allowing more flexibility, *fence selection* uses a polygon with up to 36 sides, which is temporarily drawn around the entities you want to select. The fence polygon can be concave or convex and must be closed to complete the selection process.

Crossing is a toggle available during area and fence selection that you can enable after selecting the first point of the rubberband box. Using Crossing, you can select any entity that crosses the rubberband box, along with any entities fully within the box. Crossing works with all editing commands except Stretch.

A *selection set* is a user-defined set of entities and/or groups. Selection sets provide increased efficiency during editing by reducing the time required to select entities or groups, or define areas or fences. A selection set may contain any number of entities or groups, and up to eight different selection sets may be defined. You can only edit active selection sets. Add or remove entities from selection sets with the EditSets option in the Edit menu. See “Working With Selection Sets” in the “Editing Drawings” chapter for more information on using selection sets.

Press **[Alt] + [1]** through **[Alt] + [8]** to change the active selection set when editing. While in an edit command, such as Move or Erase, press **[Alt] + [2]**, for example, to switch to selection set 2.

Layer Search controls whether or not DataCAD scans all layers during selection operations. If you toggle LyrSrch on, you can select entities on all layers that are on (being displayed). When the LyrSrch toggle is off, you can select entities only on the active layer (the layer you’re currently working on). We’ll cover layers in more depth later in this chapter.

Shortcut: Press the apostrophe (single quotation mark) key on your keyboard to toggle layer search on or off.

In addition to the button in the Menu Window, a Layer Search Status Toggle is located in the Status Area of the DataCAD window. If the “L” on the toggle button is green, layer search is on; if it’s red, layer search is off. Click on the toggle button to turn LyrSrch on or off with a single mouse click.

The green and red colors on the layer search toggle come from DataCAD’s default screen color palette. If you change the palette DataCAD uses, different colors may be displayed. See “Customizing Colors” in the “Before You Draw” chapter for more information.

More About 3D Selection Menus

The 3D selection menu options are identical, with a few exceptions for Area and Fence. When using Area and Fence for 3D entities, most entities are selected when all the end points of each displayed line segment and all displayed markers (such as center points for cylinders) lie within area or fence boundaries.

When using 3D Fence selection, however, it's possible to select some 3D entities by enclosing their end points but not enclosing other parts of these entities. If your fence polygon is concave, for instance, you could enclose the end points of a slab and still have the middle part of the slab lie outside the polygon. The entity is still considered in the fence if the end points lie within the fence. But if *any* end point of an entity lies outside the fence, the entity is considered outside the fence and is then not selected.

Selection by Area and Fence is independent of the current Z-base and Z-height settings. You can think of the region enclosed by the area or fence as going back into the screen infinitely. If you can see the region and it falls inside the boundary, the region is selected when you enclose it with an area or fence.

The following list includes the portions of an entity that must lie within the boundaries of the area or fence to be selected:

3D Lines	Entire line
Polygons	End points of all displayed edges, including end points of void edges
Blocks	End points of all edges which directly determine the shape of the block
Slabs	End points of all displayed edges, including end points of void edges
3D Arc	Entire arc
Cylinder	Entire cylinder
Symbol	The symbol reference point
Cone	Entire cone
Truncated Cone	Entire truncated cone
Sphere	Entire sphere
Torus	Entire torus
Mesh Surface	Any control points; the surface and the control grid are ignored
Surface of Rev.	Entire surface of revolution

Value Menus

Some DataCAD functions require you to enter a numerical value. When you choose one of these functions from a menu, a list of values appears in the Menu Window, followed by these options:

Match	Matches an existing value, entity, or angle; click on Match, then click on the entity to match
Add/Subtract	Adds or subtracts one or more values from a given value; click on the option and then the number to add or subtract
Clear	Returns the given value to zero and starts again (Clear does not, however, function like Undo); choosing Clear and then pressing [Enter] will accept a value of 0
To Layer	Puts an entity on more than one layer or on a different layer than you are currently working on; available when you have more than one layer in the drawing

You can select a value from the list, use the options on the previous page, or type any value and press **[Enter]**. The total is displayed in the prompt line in the Message Area but is not entered until you press **[Enter]**.

The decimal separator for all displayed dimensions and numbers is determined by the value of the Windows Decimal Separator. To set this value, click on Control Panel in the Start/Settings menu on your Windows desktop. Double-click on Regional Settings. On the Number tab, set the Decimal symbol value.

DataCAD's Online Calculator

DataCAD's online calculator is available anytime you need to enter a value, such as a distance or angle. Any time DataCAD prompts you to enter a value, you can simply type = and your equation, and DataCAD will automatically calculate the value. The online calculator supports pi; you can type **pi** in any equation.

Distances must be entered in decimal units. The results, however, will be displayed according to the Scale Type setting. For instance, the default setting, Architectural (Imperial) units, will display in feet/inches/fractions of inches. See "Customizing Settings" in the "Before You Draw" chapter for more information on the Scale Type and Angle Type options.

The online calculator assumes angles are entered in radians, but you can enter an angle in degrees simply by typing **d** immediately after the value. Results for angle calculations will be displayed according to the Angle Type setting, with the default being Normal (in degrees/minutes/seconds).

The calculator can do the following kinds of operations:

OPERATION	EXAMPLE	CALCULATED VALUE
Addition	=4.5+5	9.6 (9'-6")
Subtraction	=7-2.5	4.6 (4'-6")
Multiplication	=6*4.5	27.0 (27')
Division	=45/3	15.0 (15')
Parenthetical	=(4+3.5)*(8/2)	30.0 (30')
Square	=SQR(3)	9.0 (9')
Square Root	=SQRT(16)	4.0 (4')
Cosine	=COS(35d)	0.9.27/32 (9 27/32")
Sine	=SIN(35d)	0.6.7/8 (6 7/8")
Tangent	=TAN(35d)	0.8.13/32 (8 13/32")
Arc Tangent	=ATAN(12,16)	0.6435 radians (36.87°)
Natural Logarithm	=LN(9.3)2.23	(Base 2.71828)

Following are two examples to illustrate how to use the online calculator to quickly figure scales and angles.

➔ To define a new plotting scale:

1. Click on EditDefs in the Settings menu.
2. Click on Scales in the EditDefs menu.
3. Click on Change to modify an existing scale.
4. Select the scale to change from the menu.
5. Look at the prompt to "Enter new string for this scale." Enter **1/32** for the menu button name.
6. Respond to the prompt to "Enter new scaling value for this scale" by typing **=(1/32)/12** and pressing **[Enter]** to get the result 0.002604167.
7. Press **[Enter]** to accept this value as a new scale factor.

➔ To calculate an angle:

1. Respond to the prompt to enter an angle by typing **=ATAN(6,12)**, where 6 is the rise and 12 is the run.
2. Press **[Enter]** to calculate an angle of 26.33.54, or 26° 33' 54".

Color Menus

When an option requires you to assign a color, a list of colors is displayed in the Menu Window along with the options Custom, Match, and NoChange.



Figure 2.21: A sample color menu

Custom opens the Color Palette dialog box so you can change a color by adjusting its red, green, and blue (RGB) and hue, saturation, and brightness (HSB) values. To work properly, it requires a 256-color driver.

Warning: When you make a change to a color menu, you must save this new color palette, or the colors will revert back to their original RGB values when you close DataCAD, even if you save the drawing. See “Customizing Colors” in the “Before You Draw” chapter for more information on how to save palettes.

➔ To change the color of one or more entities in your drawing:

1. Click on Custom in any color menu. The Color Palette dialog box appears.



Figure 2.22: The Color Palette dialog box

2. Choose a color swatch, excluding the ones in the top row. (The top row contains the 15 standard colors DataCAD uses.) If you are satisfied with one of the colors in the dialog box, double-click on the swatch to activate that color and dismiss the Color Palette. If you want a color that is not shown on the Color Palette, continue to step 3.
3. Click on Edit. The Color Picker dialog box appears.

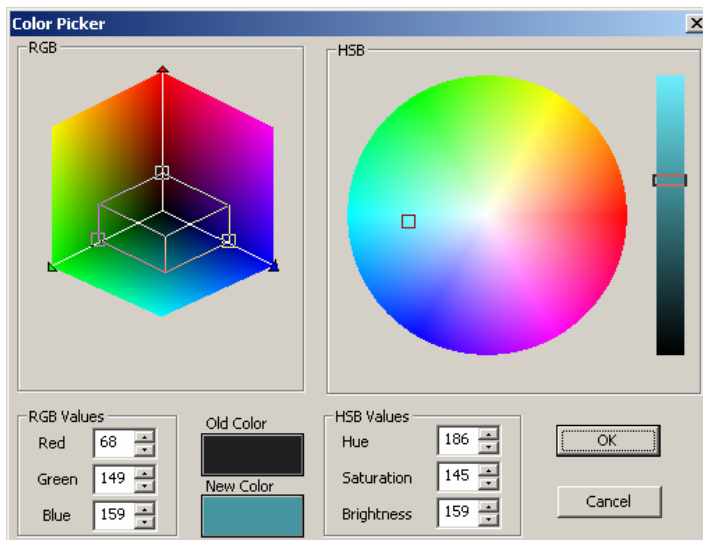


Figure 2.23: Color Picker dialog box

4. Adjust the RGB and HSB values for the current color (shown in the Old Color box) until you see the color you want in the New Color box. Alternatively, you can enter a new value for Red, Green, or Blue as well as for Hue, Saturation, or Brightness to adjust the color. You can click Cancel to close the Color Picker and Color Palette dialog boxes without saving your changes. Click OK to change the color. You return to the drawing where you are prompted to "Select entity to <CHANGE>."
5. Click on the entity whose color you want to change. DataCAD substitutes the new color for the old one.

➔ To change the color of one entity to match that of another entity:

1. Click on Match in the color menu.
2. Select the entity whose color you would like to match.
3. Select the entity whose color you would like to change. You can select multiple entities or groups without repeating steps 1 and 2. Click on NoChange to exit the color menu without making a change.

Planning Your Project

Much of your work will be editing drawings under design development, and DataCAD is a flexible program in terms of allowing you to change what you have drawn. Nevertheless, the more information you have when you first draw something, the less time you'll spend making revisions.

With traditional manual design development, you first do a rough sketch of the project and then gradually refine this idea into a drawing that is dimensionally and materially specific. Computer-aided design (CAD), however, produces a drawing with precise dimensions the first time it is drawn.

This suggests that the ideal process of CAD development might be the opposite of manual design development. With CAD, it's important to know such things as the exterior wall construction *before you start to draw*, so your drawing will be accurate from the beginning. If you finalize your floor plan, however, before you decide on exterior wall construction materials, you'll have to go back and change the thickness of all exterior walls in order to do automatic dimensioning or wall sections and details. This wall thickness change would in turn impact the interior dimensions of rooms and may affect the modular grids of ceiling and floor tile, lighting, etc. Some well-thought out planning before you start to draw can save you valuable time.

Gathering Project Information

Practically speaking, it's impossible to predict the exact dimensions and materials of everything in the building before you finalize the design. A realizable goal, however, is to gather as much information as possible about the construction of the building before you begin the design development or construction documents phases of the project. This keeps time-consuming changes to a minimum and allows the building to be drawn and dimensioned accurately.

You need to have as much information as possible on the following:

Scope of Work	How much do you need to draw? In what detail? What information can be reused for several drawings?
Dimensions	What are the basic dimensions of the structural grid, columns, and load-bearing partitions? What size are the interior partitions? Doors and windows? Sill and jamb conditions? Floor and ceiling heights? Joist sizes?
Materials	What are the walls and foundation made of? What floor and roof systems are you using?
Existing Elements	Which parts of the drawing can be used to create other drawings for this project? Which templates or typical details can be used for this project? Which drawings from the schematic design phase can be reused?

Planning Layers

The concept of layering drawings has been used in designing, drawing, and drafting long before computers. In the past it was referred to as overlay, or pin register, drafting.

With overlay drafting, drawings were developed in layers on trace paper, each aligned with the others. Each sheet contained related information that you could separate and reproduce individually or in any combination. You could also use this layering of drawings during the design process, to overlay and quickly trace, change, and rework drawings. This same procedure is easily accomplished using DataCAD, to form a highly productive drawing process.

The key to efficient CAD is the well-planned use of the almost unlimited number of layers available in each drawing file to isolate related entities, details, and text. By organizing the information displayed on each layer according to type or content, you can perform editing functions on one building component without affecting components on other layers.

There are many layering methods; however, it is important to devise a well thought out strategy based on office standards for the type of work you do. A drawing file with a good layer structure could and probably should contain all the drawings for a single project.

Just as your drawing can contribute to the drawings of others, what others in your office have already drawn can help you. As you plan your drawing layers, make a note of where you can get some of the basic information needed for your particular task. There's no need to redraw all those toilets for the third floor if they've already been drawn for the first floor, or if you already have an appropriate toilet symbol on a plumbing template.

Before you begin drawing, it is a good idea to sit down and make a list of all the layers you intend to create, noting what you expect each layer will contain. Although creating layers is easy with DataCAD, you will find it helpful to have a reference list as you draw to help you prioritize and chart your drawing process. See "Layers" in the "Before You Draw" chapter for more information on how to set up layers in DataCAD.

In DataCAD, you can toggle each layer you create on or off at any time; only layers toggled on will be displayed, and only one of these layers is active. To continue our parallel with overlay drafting, the active layer can be thought of as the top piece of trace paper; it is the layer you're currently drawing on. The remaining layers of trace paper in the overlay stack are all the layers toggled on. Existing layers not currently in the stack are all the layers toggled off (not displayed).

The process of drawing from the most general to the most specific is opposite the process of gathering information to prepare for drawing. You want to understand the building in the greatest detail possible before you draw it. But once you start drawing, you want to document all that information gradually, arriving at the most specific drawing (the detail) only after you have graphically described all the general conditions.

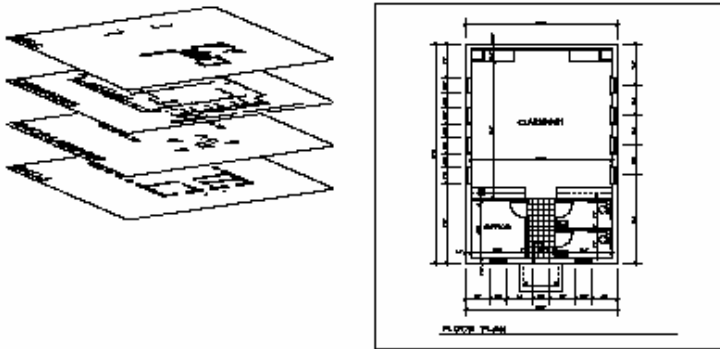


Figure 2.24: The image on the left is how your drawing is organized using layers. On the right is how you see your drawing in DataCAD.

Prioritizing Drawing Tasks

In addition to making a list of all your layers, make a flow chart of your drawing process: what you will draw first, second, third, etc. In order to keep drawing redundancy to a minimum, you will want to draw those parts of your drawing which are the most general first, then proceed to detail your drawing, gradually making it more specific. Along the way, make copies of those parts of your drawing that can be used for other drawings.

Drawing Efficiently

A drawing will, of necessity, contain many entities. The idea behind CAD is to have the computer automatically draw as many of those entities as possible. DataCAD incorporates several functions that make this automation possible; entities may be copied or mirrored, for example, and symbols and linetypes may be added as well.

Any entity, group, or area in your drawing can be copied one or several times automatically or flipped and copied at once to produce a mirror image next to the original image. For example, you only need to draw one side of a corridor of rooms, if you use Mirror to create the opposite side. For more information on using Mirror, see “Mirroring Entities” in the “Other Ways to Draw” chapter.

Any entity, group, or area in your drawing can be stored in a template as a symbol. You can draw a tree once, save it as a symbol in a template, then place trees all over your site plan just by selecting with the mouse all the places in your drawing where you want the trees to appear.

Many material symbols shown in sections and details can be drawn automatically by selecting the proper linetype, or by creating one of your own. DataCAD has linetypes for materials like insulation, brick, plywood, and roof shingles. You could produce, for example, a line of insulation running up a wall just by choosing the Insul linetype and entering the two end points of the line.

DataCAD’s Edit menu provides you with many ways to modify anything you draw without redrawing it, including moving or stretching entities, or changing their attributes.

Use Stretch to select an area or point and move it to a new location, automatically stretching or shrinking everything attached to what you are moving. For example, you can move a window to a new place in the wall, and the wall will be stretched to match the new position of the window. For more information on using Stretch, see “Stretching Entities” in the “Editing Drawings” chapter.

Move is different from Stretch in that only the entity, group, selection set, or area you select is moved to a new location in the drawing; everything connected to those items stays where it is. So if you have two cased openings, one of which contains a door, you can move the door to the other cased opening without moving all the wall and jamb lines attached to the door. For more information on using Move, see “Moving Entities” in the “Editing Drawings” chapter.

Use Change to alter the characteristics or attributes of any entity, group, area, or selection set you’ve created. For example, you can change a wall from double-weight solid lines to single-weight dashed lines, alter its height from 8 to 10 feet, change its color from green to blue, etc. For more information on using Change, see “Changing Entities” in the “Editing Drawings” chapter.

Sample Project

This is an example of how to use DataCAD by outlining a detailed strategy for completing a project; it is not intended to be a tutorial or replace DataCAD training. The drawing files for this sample project are on your DataCAD CD-ROM.

If you do a Typical installation, the files for this sample project will not be installed. To install them after doing a Typical installation, insert your DataCAD CD in your CD-ROM drive, select Custom installation, and then select Sample Drawings.

Scope of Work

Following is a summary of what is known about the building for which you will produce working drawings:

Description	A 12,300 s.f. YMCA facilities building, two stories, flat roof, sloping site, with parking for 34 cars.
Materials	Structure: Clear-span metal frame with steel column grid, 25' x 20', lightweight steel floor and bar joists @ 2'-0" O.C. Exterior Walls: Metal panel over metal framing with batt insulation, CMU block veneer wall with metal framing, batt insulation and GWB interior. Interior Walls: Metal studs with fire-rated GWB.
Dimensions	Plan: 100' x 68'-8" exterior limits of building.
Elevation Hgts	Lower Level Floor @ 83.00' Lower Level Ceiling @ 91.00' Upper Level Floor @ 95.00' Upper Level Ceiling @ 106.75' Top of Roof @ 107.00' Top of Parapet @ 108.75'

Summary of Available Personnel and Drawings Needed

PERSONNEL	DRAWINGS
Job Captain	Site Plan
Draftsman 1	Foundation Plan
Draftsman 2	Lower Level Plan
	Upper Level Plan
	Four Exterior Elevations

North/South Building Section
 Lower Level Electrical Plan
 Upper Level Electrical Plan
 Lower Level Plumbing Plan
 Upper Level Plumbing Plan
 Door, window, and Finish Schedules
 Door and Window Types

Planning the Drawing Time

1. Summarize drawings needed; plan final sheet layout.
2. Draftsman 1 develops base plans while Draftsman 2 begins work on typical details.
3. Draftsman 1 gives Draftsman 2 the plan information. Draftsman 2 begins developing elevations and sections.
4. When plans, elevations, and sections have been developed, develop schedules and details on both workstations using completed drawings as a base for detail drawings.
5. When all drawings have been completed, assemble individual drawings into “sheets” for plotting.

How Draftsman 1 Drew the First Floor Plan

Draftsman 1 first spent some time with the Job Captain, learning what the project involved and the assigned tasks. Then Draftsman 1 sat down and prepared two outlines:

- an outline of the layers required and their names
- a strategy of what to draw first, second, third, etc.

LAYER NAME	DESCRIPTION
A2-BLK	Lower Level Conc. Block
A2-COL	Upper Level Columns
A2-DIM	Upper Level Dimensions
A2-DRS	Upper Level Doors
A2-FND	Upper Level Foundation
A2-HCH	Upper Level Hatch
A2-LAV	Upper Level Lavatories
A2-LBL	Upper Level Labels
A2-NOT	Upper Level Notes
A2-ROM	Upper Level Room Numbers
A2-STR	Upper Level Stair
A2-TIL	Upper Level Tile
A2-TXT	Upper Level Text
A2-WIN	Upper Level Windows
A2-WLS	Upper Level Walls

Draftsman 1’s drawing process outline is as follows:

1. Create drawing file YMC-MSR1.DC5.
2. Create and name drawing layers listed above.
3. Load sheet border from template file.
4. Input all existing site information.

5. Draw Columns, Column Grid, Perimeter, and building elements on appropriate layers.
6. Separate drawing layers needed for first floor by saving them out to the hard disk on separate layer files.
7. Create a detailed elevation drawing called YMC-ELV1.DC5.
8. Create and name the required drawing layers.
9. Load basic first floor plan drawing layers from layer files.
10. Draw interior partitions.
11. Place windows in perimeter wall.
12. Save modified plan layers to a layer file called YMC-PLN1.LYR. This layer will become the exterior wall for the second and third floors, which have identical windows to the first floor.
13. Place doors and windows in perimeter wall and in interior partition walls.
14. Zoom in on lobby area and draw lobby details.
15. Zoom in on restroom area and draw fixture layout and details.
16. Draw reflected ceiling plan.
17. Add dimensions, labels, and notes to drawing.
18. Create a new detail elevation file for the Second Floor Plan called YMC-ELV2.DC5.

Notice that Draftsman 1's outlines do not detail how to best draw each part of the drawing. This strategy simply gives an outline of what to do next while leaving Draftsman 1 free to choose the best drawing method for each part of the drawing. Copy, Mirror, and Stretch will be used to produce, respectively, the column grid, two halves of a symmetrical parking layout, and two handicapped parking spaces that are larger than the other parking spaces.

Basic Drawing and Editing

DataCAD is ready to draw lines immediately upon opening a drawing; notice that in the Create pull-down menu, Line is checked by default. You can even draw with custom linetypes to represent such things as insulation, telephone and electrical lines, and siding. And editing lines, or anything else you draw, is easy with DataCAD's many editing features.

Drawing Using the Mouse

➔ To draw a line by selecting two points:

1. Move the cursor anywhere in the Drawing Area and click. You've just selected the first end point of your first line. In addition to simply clicking to select a point and begin drawing, you can snap to a point on another entity and begin drawing from there. See "Object Snapping" in the "Drawing Tools" chapter for more information.

2. Move the cursor to the point where you want the line to end and click.
The line is drawn. Notice, however, that if you move your cursor, it is still connected to the end of the line. This allows you to continue drawing lines, using the end of the previous line as the beginning of the next line.
3. Repeat step 2 to continue drawing lines until your last end point meets the first end point you selected in step 1.
4. Right-click to disconnect the cursor from the last line.

Drawing Using Coordinate Entry

Coordinate entry lets you draw lines more precisely by entering the exact coordinates of each end point of a line. In DataCAD, you have a choice of five coordinate entry methods, or input modes:

- **Relative polar** requires that you enter a distance and an angle relative to the last point entered.
- **Absolute polar** requires that you enter a distance and an angle from absolute 0.
- **Relative Cartesian** requires that you enter an X and Y distance relative to the last point entered.
- **Absolute Cartesian** requires that you enter an X and Y distance from absolute 0.
- **Distance-Direction** requires that you indicate the direction with the mouse and then enter a distance relative to the last point entered.

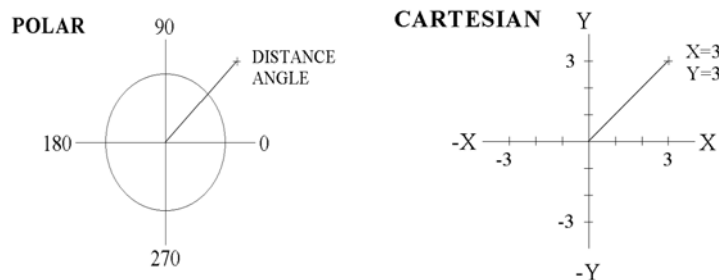


Figure 2.25: Polar and Cartesian coordinate entry methods

Once you select a coordinate entry method, it remains active until you choose a different entry method. The coordinate entry method you're using is always displayed in the Message Area beside the coordinate display. (Coordinates are only displayed when your cursor is in the Drawing Area.)

Four additional coordinate entry methods are available *only* while in the 3DViews/Controls menu: relative and absolute cylindrical (enter distance, angle, and Z coordinate) and relative and absolute spherical (enter distance, plan angle, and rise angle). These methods provide additional variables to the Z component of coordinate entry, which may be helpful in specialized modeling applications like entering a 3D line that represents a solar angle.

➔ To choose a coordinate entry method, either:

- press (Insert) until the name of the method you want appears in the Message Area at the bottom of the DataCAD window, or
- click on Input Mode in the Tools pull-down menu and select a coordinate entry method from the list.

➔ To draw a line using relative or absolute polar coordinate entry:

1. Click in the Drawing Area to select the line's first end point.
2. Press Spacebar.
3. Type the length of the line (the distance from the end point you entered in step 1 to the next end point) and press Enter.
4. Type the angle of the line from the X axis and press Enter. The line is drawn, with your cursor still attached to the end point of the line.
5. Continue drawing lines by repeating steps 2-4. Right-click to disconnect the cursor from the line.

➔ To draw a line using relative or absolute Cartesian coordinate entry:

1. Click in the Drawing Area to select the line's first end point.
2. Press Spacebar.
3. Type the relative or absolute X distance from the first end point and press Enter.
4. Type the relative or absolute Y distance from the first end point and press Enter. The line is drawn, with your cursor still attached to the end point of the line.
5. Continue drawing lines by repeating steps 2-4. Right-click to disconnect the cursor from the line.

➔ To draw a line using direction-distance coordinate entry:

1. Click in the Drawing Area to select the line's first end point.
2. Drag the cursor in the direction you want to draw the line.
3. Press Spacebar.
4. Type the length you want the line to be, and press Enter. The line is drawn, with your cursor still attached to the end point of the line.
5. Continue drawing lines by repeating steps 2-4. Right-click to disconnect the cursor from the line.

More About Coordinate Entry

You can enter distances and angles either by typing a value or by using a value menu. See “Value Menus” earlier in this chapter for more information on how to use them. Depending on the scale type you are using, there may be several different methods you can use to enter coordinates. See “Setting Scale and Angle Types” in the “Before You Draw” chapter for more information. For instance, in the architectural scale type, which is DataCAD’s default, you must enter distances as feet.inches.fractions of inches.

TO ENTER: TYPE:

4'	4
6"	0.6 or .6 or Spacebar 6
3/16"	0.0.3/16 or ..3/16 or ..3.16 or Spacebar Spacebar 3 Spacebar 16
4'-6"	4.6 or 4 Spacebar 6
4'-6 3/16"	4.6.3/16 or 4.6.3.16 or 4 Spacebar 6 Spacebar 3 Spacebar 16

Notice in this example that 3/16" is entered as **0.0.3/16**, indicating 0 feet, 0 inches, and 3/16 of an inch. Notice also that you can leave out the zeros when you enter distances, as long as the decimal points remain as placeholders; so you could type **..3/16**. Further, slashes, decimal points, and spaces are interchangeable, so you could, for instance, replace the slash with another decimal point and type **..3.16**.

Angles are entered in a similar fashion, except the Spacebar method can't be used. For instance, using relative polar coordinate entry, angles are entered as: degrees.minutes.seconds

TO ENTER: TYPE:

38°	38
12'	0.12 or .12
30"	0.0.30 or ..30
38°-12'-30"	38.12.30

Drawing with Linetypes

DataCAD has four standard linetypes and 21 custom linetypes. You cannot redefine the four standard linetypes -- solid, dashed, dotted and dot-dash -- although you can change some of their attributes. You can, however, create your own linetypes, adding up to 175 custom linetypes such as special symbolic lines for site plan utilities, center lines, and highlighted lines for review. See “User-Defined Linetypes” in the Appendix for a list of available custom linetypes and their settings.

➔ To draw a line using a linetype:

1. Go to the Edit menu in the Menu Window; click on LineType. A list of available linetypes is displayed in the Menu Window, along with options to modify a chosen linetype's weight, color, size, factor, or overshoot.
2. Look at a preview of a linetype before you use it by moving the mouse cursor over the linetype option in the Menu Window. Click on ScrlFwrd and ScrlBack to scroll forward and backward through the list of linetypes.
3. Click on a linetype in the Menu Window to use that linetype.

4. Click on the Factor, Spacing, LineWgt, Color, or OverSht options to change the settings for the linetype you chose. See “Linetype Settings” below for details on changing linetype settings.
5. Begin drawing lines as described in “Drawing Lines By Selecting Two Points” or “Drawing Lines Using Coordinate Entry” earlier in this section.

Linetype Settings

All custom linetypes enter building materials at actual size, and they appear at the display scale you specify. You can use the following options to customize a linetype:

Factor	Toggle that lets you quickly switch between using the default spacing factor (Factor toggled off) and the spacing factor set by the Spacing option described below (Factor toggled on).
Spacing	Changes the size of the linetype pattern. Keep in mind that DataCAD defines units in absolute world coordinates. For example, to set dashed lines to repeat every 1" on a 1/4"=1'-0" scale drawing, set the spacing to 4'-0". To set line spacing, click on a value in the value menu or type a spacing value, and press [Enter] .
LineWgt	Sets the thickness of a linetype. The most common line thickness is 1. To set the line weight, click on a line weight in the value menu or type a line weight value, and press [Enter] . For line weights to display, ShowWgt must be toggled on in the Display menu.
Color	Changes the color of a linetype. Any new lines that you enter into the drawing appear in the new color.
OverSht	Displays extensions, or punch, at line intersections, giving drawings an architectural look. DataCAD always dimensions to the actual length of a line, ignoring any displayed overshoot. To set the overshoot, click on OverSht, and then choose or type an overshoot distance and press [Enter] . For line overshoots to display, OverSht must be toggled on in the Display menu. You can change the overshoot distance for existing lines with the OverSht option in the Edit/Change menu.

Basic Editing Tools

DataCAD's Edit pull-down menu provides easy access to most editing functions, including primary editing menus such as Move, Copy, Rotate, Mirror, Stretch, Enlarge, Erase, Change, Partial Erase, Line Cleanup, and Wall Cleanup. Choose any of these editing options to display that menu in the Menu Window. See the “Editing Drawings” chapter for details on these options.

The Edit pull-down menu also includes unlimited, global undo/redo options and cut, copy, and paste. The Undo option reverses the last action, including any drawing or editing, changes made using macros, or the deletion of layers. The Redo option restores the last action. To undo an action, click on Undo in the Edit pull-down menu or press **[Ctrl] + [Z]**; to redo an action, click on Redo in the Edit pull-down menu or press **[Ctrl] + [Y]**. You can also use the Undo/Redo toolbar; click on Undo/Redo Toolbar in the View pull-down menu to display the toolbar.

Cut, Copy, Paste, and Paste Special are four more Edit pull-down menu options you can use to edit drawings or share parts of drawings between documents. Cut removes entities from the drawing to the clipboard, Copy copies entities in the drawing to the clipboard, Paste adds entities previously cut or copied to the active drawing, and Paste Special lets you choose the format (metafile, text, etc.) for entities previously cut or copied and then adds them to the active drawing. To cut or copy DataCAD entities, you must first use the Clipboard Select option in the Edit pull-down menu to select the entities to be cut or copied. You can choose whether to copy just the entity or the entity with its current color and line weight. Check the Copy to Clipboard options on the Misc tab of the Program Preferences dialog box as appropriate.

➔ To move or copy entities from one drawing to another:

1. Click on Clipboard Select in the Edit pull-down menu. A selection menu is displayed in the Menu Window. See “Selection Menus” in this chapter for more information.

Shortcut: Press **[Ctrl] + [E]** to access the Clipboard Select selection menu.

2. Toggle InsPoint on to enter an insertion point for the entities. This point can then be used to paste the entities into another drawing. Toggle InsPoint off to enter entities relative to absolute zero.
3. Choose a selection method by clicking on one of the selection menu options.
4. Select the entities to be moved or copied.
5. Click on Cut in the Edit pull-down menu to remove the entities from the drawing, or click on Copy to simply copy them. If InsPoint was toggled on in step 2, you are prompted to “Select reference point for clipboard cut/copy”. If InsPoint was toggled off, skip to step 7.

Shortcut: Press **[Ctrl] + [X]** to cut entities, or press **[Ctrl] + [C]** to copy entities.

6. Click in the Drawing Area to select a reference point. Your cursor will be attached to this reference point when you paste the entities into another drawing.

Shortcut: Press **[Ctrl] + [V]** to paste entities.

7. Paste them into another DataCAD drawing or into a document in another application:
 - Open the drawing you’d like to add the entities to and click on Paste in the Edit pull-down menu. Toggle ByCenter on to paste the entities by the center of their extents; toggle it off to paste the entities by their insertion point (if InsPoint was toggle on) or at absolute zero.
 - Open the drawing you’d like to add the entities to, click on Paste Special in the Edit pull-down menu, and select a format for the entities.
 - Open the Windows application document that you’d like to add entities to and use that application’s paste function.

You can preserve the exact position of an entity (as measured from absolute zero) when copying it from one drawing to another. For example, if a marker was placed at $x=16'$, $y=16'$ (as measured from absolute zero) in the original file, then pasting it into a new file using AbsZero will place the marker at $x=16'$, $y=16'$ from absolute zero in the new file.

Referencing Drawings

You can reference, or link, an unlimited number of DataCAD drawings to your current drawing. These referenced drawings, also called XREFs, are displayed in your current drawing and will even print as though they were part of your current drawing; however, they are not actually added to your current drawing, allowing you to keep file size as small as possible. Reference file geometry can be moved, copied, mirrored, or enlarged just as any other entities can be, and reference files can be rotated into any plane in space.

➔ To insert a reference file:

1. Click on Reference File in the Insert pull-down menu. A dialog box is displayed, prompting you to "Enter name of reference file to insert".
2. Select a drawing from the list box and click on Open. The dialog box closes and a box representing the extents of the referenced drawing is displayed, attached to your cursor. The InsXREF menu is displayed in the Menu Window.
3. Enlarge the referenced drawing before inserting it by clicking on Enlarge in the InsXREF menu. A value menu is displayed and you are prompted to "Enter XREF enlargement factor."
4. Use the value menu to enter the enlargement factor you want to use and press **Enter**.
5. Decide if you'd like to dynamically rotate the reference file geometry into place when you insert it into your drawing. If you want to do this, toggle DynamRot on. The DynamRot option has no effect if you use AbsZero to place the reference file geometry.
6. Select one of these ways to insert the reference file:
 - To insert the referenced drawing relative to absolute zero, click AbsZero in the Menu Window. The referenced file is inserted into your current drawing.
 - To place the referenced drawing by its center, toggle ByCenter on in the Menu Window. Move the cursor to position the referenced drawing and click to insert it. If you toggled DynamRot on in the previous step, you are prompted to "Position XREF on drawing". Rotate the geometry into place and click to insert it.
 - If the ByCenter toggle is off, your cursor is attached to absolute zero in the referenced file. Move the cursor to position the referenced drawing and click to insert it. If you toggled DynamRot on in the previous step, you are prompted to "Position XREF on drawing". Rotate the geometry into place and click to insert it.

- To use the active clip cube(s) in the referenced file, toggle on UseClip. You are prompted to “Position XREF on drawing.” DataCAD inserts the active clip cube(s) into the current drawing.

You can also use coordinate entry to select the point to position the reference file geometry. See “Using Coordinate Entry” in this chapter for more information.

Self-referencing XREFs are supported. You can place an XREF for drawing A into drawing A. “Circular” XREFs are no problem since endless-loop swap files are not created. Self-referencing XREFs are automatically redefined to reflect the new drawing filename when you use the Save As function.

Referenced files may have unique clipping boundaries. You can have multiple Clip Cube definitions active simultaneously. You can also enter text and dimensions outside of the XREF Clip Cube.

➔ To use XREFs with unique clipping boundaries:

1. Select Reference File Management from the Insert pull-down menu.
2. Select Clip Cube from the extended menu.
3. Choose the XREF for which you want to define a new Clip Cube.

For information about using clip cubes from referenced files, see “Viewing Selected Entities in Your Model” in the “Viewing Your Model” chapter.

Reference File Manager

You can use the Reference File Manager to manage reference files in your current drawing. When you click on Reference File Management in the Insert pull-down menu, you can refresh or reload reference files, resolve orphaned reference files, or open the Reference File Manager.

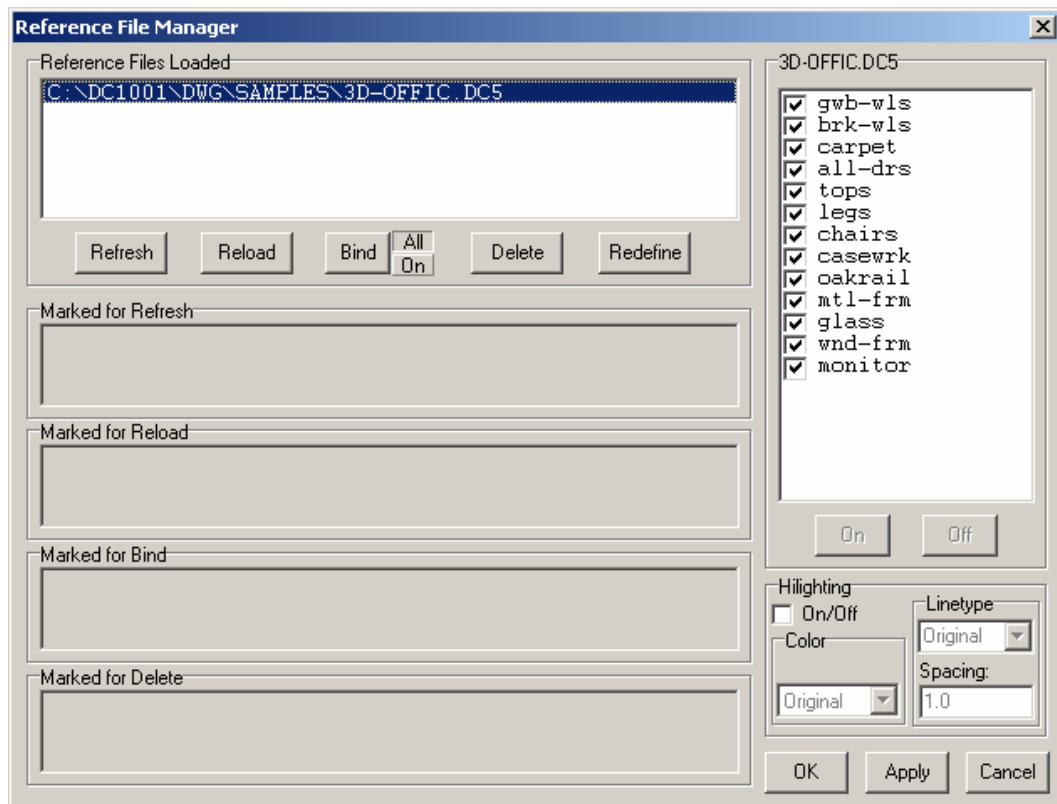


Figure 2.26: The Reference File Manager dialog box

You can also redefine loaded reference file names within the Reference File Manager by clicking on the Redefine button. This allows you to redefine XREF files that were renamed. Redefined XREFs retain the previous XREF's position, scale, and rotation.

Displaying Reference File Layers

➔ To display only selected layers:

1. Click on Reference File Management in the Insert pull-down menu. A submenu is displayed.
2. Click on Manager in the submenu. The Reference File Manager dialog box is displayed. All the reference files in your current drawing are listed in the Reference Files Loaded list box.
3. Click on a reference file in the list to select it. If necessary, you can use the horizontal scroll bar for reference file names that extend beyond the dialog window. All the layers in the reference file are listed in the Layer Control list box, with a checkmark next to the ones that are on (displayed).
4. Click on the checkboxes next to the layers to turn layers in the reference file on or off.
5. Click on Apply to apply the changes you've specified. The checked layers are displayed, and the unchecked layers are not displayed.
6. Click on OK to close the Reference File Manager dialog box when you're finished.

Refreshing and Reloading Reference Files

You can refresh and reload reference files. Refresh will incorporate any changes you've made to the reference file, such as editing the drawing, but will not update any changes to layer status. Reload will update any changes you made to the reference file geometry as well as the reference file's layer status, such as which layers are toggled on and which are off.

➔ To refresh reference files:

1. Click on Reference File Management in the Insert pull-down menu. A submenu is displayed.
2. Click on Manager in the submenu. The Reference File Manager dialog box is displayed. All the reference files in your current drawing are listed in the Reference Files Loaded list box.
3. Click on a reference file in the list to select it. Press **[Ctrl]** or **[Shift]** and click to select multiple files.
4. Click on Refresh. The reference file name is copied into the Marked for Refresh list box.
5. Click on Apply to apply the changes you've specified. The selected reference files are refreshed.
7. Click on OK to close the Reference File Manager dialog box when you're finished.

Double-clicking on a file listed in the Marked for Refresh, Marked for Reload, Marked for Bind, or Marked for Delete list boxes will remove the file from the list and return it to the Reference Files Loaded list box.

➔ To reload reference files:

1. Click on Reference File Management in the Insert pull-down menu. A submenu is displayed.
2. Click on Manager in the submenu. The Reference File Manager dialog box is displayed. All the reference files in your current drawing are listed in the Reference Files Loaded list box.
3. Click on a reference file in the list to select it. Press **[Ctrl]** or **[Shift]** and click to select multiple files.
4. Click on Reload. The reference file name is copied into the Marked for Reload list box.
5. Click on Apply to apply the changes you've specified. The selected reference files are reloaded.
6. Click on OK to close the Reference File Manager dialog box when you're finished.

Double-clicking on a file listed in the Marked for Refresh, Marked for Reload, Marked for Bind, or Marked for Delete list boxes will remove the file from the list and return it to the Reference Files Loaded list box.

Binding Reference Files

XREFs and nested XREFs both bind, based on the On or All toggle in the XREF Manager.

➔ To bind reference files:

1. Click on Reference File Management in the Insert pull-down menu. A submenu is displayed.
2. Click on Manager in the submenu. The Reference File Manager dialog box is displayed. All the reference files in your current drawing are listed in the Reference Files Loaded list box.
3. Click on a reference file in the list to select it. Press **[Ctrl]** or **[Shift]** and click to select multiple files.
4. Look beside the Bind button where there are two small toggles: All and On. Click on All to bind all layers in the reference file to the current drawing, regardless of whether they're on or not. Click on On to bind only the layers in the reference file that are on (displayed). Before you use the On toggle to bind layers of the reference file, use the Layer Control list box to check those layers you want toggled on.
5. Click on Bind. The reference file name is copied into the Marked for Bind list box.

Warning: The drawing file size may increase substantially when you bind referenced files. Binding too many files may cause the drawing file size to exceed the 128MB limit. DataCAD cannot provide a warning when this occurs. Check the Directory (available from the Tools pull-down menu) to monitor the current drawing file size.

6. Click on Apply to apply the changes you've specified. The selected reference files are bound, and their geometry is copied into the current drawing. Once a reference file is bound to the current drawing, it ceases to be a reference file. So, if you choose to bind only a few layers of a reference file, the entire reference file -- even those layers you didn't bind -- is no longer linked to your current drawing.
6. Click on OK to close the Reference File Manager dialog box when you're finished.

Highlighting is lost when an XREF is bound because it is no longer an XREF.

Deleting Reference Files

➔ To delete reference files:

1. Click on Reference File Management in the Insert pull-down menu. A submenu is displayed.
2. Click on Manager in the submenu. The Reference File Manager dialog box is displayed. All the reference files in your current drawing are listed in the Reference Files Loaded list box.
3. Click on a reference file in the list to select it. Press **[Ctrl]** or **[Shift]** and click to select multiple files.

4. Click on Delete. The reference file name is copied into the Marked for Delete list box.
5. Click on Apply to apply the changes you've specified. The selected reference files are deleted.
7. Click on OK to close the Reference File Manager dialog box when you're finished.

Double-clicking on a file listed in the Marked for Refresh, Marked for Reload, Marked for Bind, or Marked for Delete list boxes will remove the file from the list and return it to the Reference Files Loaded list box.

Highlighting Reference File Geometry

You can use color, linetype, or a combination of the two to highlight reference file geometry. Highlighting is lost when an XREF is bound because it is no longer an XREF.

➔ To highlight entities from a reference file:

1. Click on Reference File Management in the Insert pull-down menu. A submenu is displayed.
2. Click on Manager in the submenu. The Reference File Manager dialog box is displayed.
3. Click on the reference file in the Reference Files Loaded list box to select the reference file you want to highlight.
4. Click on the On/Off checkbox in the Highlighting section to turn highlighting on.
5. Choose a highlighting color and linetype/line spacing from the drop-down boxes.
6. Click on Apply to apply the changes you've specified. The highlighting is applied to the geometry for that reference file.
7. Repeat steps 3 – 6 to highlight the geometry from other reference files.
8. Click on OK to close the Reference File Manager dialog box.

Resolving Orphaned Reference Files

Reference files can be orphaned if you move the file to another folder on your computer or if you delete the referenced file. You can resolve the broken link between your current drawing and the referenced file by either deleting the link or by redefining the path to where you moved the reference file. Clicking on the Redefine button allows you to redefine XREF files that became orphaned when they were renamed. Redefined XREFs retain the previous XREF's position, scale, and rotation.

The path between the current drawing and the reference file is limited to 160 characters. This allows you to use the full pathname that includes Drive + Path + Filename.

➔ To resolve orphaned reference files:

1. Click on Reference File Management in the Insert pull-down menu. A submenu is displayed.
2. Click on Orphans in the submenu. The Orphaned Reference Files dialog box is displayed, listing all the orphaned reference files in your current drawing.
3. Click on a reference file in the list to select it.
4. Resolve the orphaned file in one of these two ways:
 - Click on Delete to delete the reference file from your current drawing.
 - Click on Redefine Path to reestablish the link between your current drawing and the reference file. A dialog box is displayed, allowing you to browse for the file; find the referenced file and click on OK. The dialog box closes. Click on Done to close the Orphaned Reference Files dialog box.

Viewing Drawings

Viewing is simply the way you look at your drawing when you work on it. Compare viewing a drawing on the computer with looking at a drawing on your drafting board. As you draft on the board, your eyes are always moving and refocusing, allowing you to concentrate on small details or view the entire drawing sheet quickly and easily. At the computer, you need to consciously decide what you want to focus on and to what degree. To say this in CAD terms, you will zoom in on an area of your drawing when you have to focus on a detail; and when you want to see the entire drawing, you will zoom out to the *extents*, or outermost edges of the drawing.

Go-to-views, hyperview linking, and multi-view windows are three additional viewing features that allow you to jump quickly to preset views of your drawing. See the “Viewing Your Model” chapter.

Zooming In and Out

The WindowIn menu in DataCAD lets you zoom in on an area by displaying the drawing in a larger scale. When you zoom out, you are actually looking at your drawing in a smaller scale, which allows you to see more of your drawing. The scales that the WindowIn menu uses are predefined in the Utility/Settings/EditDefs/Scales menu.

➔ To zoom in on your drawing:

1. Click on WindowIn in the Utility menu, or click on the W in the Projection Pad.
2. Click above and to the left of the area you want to scale.
3. Move the mouse down and to the right to stretch the box around the area you want to zoom in on.
4. Click to complete the box. The drawing rescales automatically.

Shortcuts: Press  to display the WindowIn menu in the Menu Window.

To return to the previous view (essentially undoing the zoom), press **[P]**.
DataCAD remembers up to 25 previous views.

Press **[PgUp]** and **[PgDn]** to quickly zoom in and out on your drawing. To use **PgUp** and **PgDn** on your numeric keypad, make sure **NumLock** is off first.

➔ To zoom to your drawing's extents:

1. Click on **WindowIn** in the **Utility** menu, or click on the **W** in the **Projection Pad**.
2. Click on **Extents** in the **WindowIn** menu. Your drawing is displayed at the largest scale possible while still displaying the drawing in its entirety. If you've made significant changes to your drawing, click on **Recalc** instead of **Extents**. DataCAD will review your drawing and update the extents of the drawing accordingly.

Shortcut: Click on the **E** in the **Projection Pad** to view the drawing at its extents; click on **R** to recalculate the extents.

➔ To zoom in on your drawing and display the area to its extents:

1. Click on **WindowIn** in the **Utility** menu, or click on the **W** in the **Projection Pad**.
2. Toggle **FreeZoom** on or off in the **WindowIn** menu, depending on your preference. When it is toggled on, DataCAD can show a nonstandard display scale. When it is toggled off, DataCAD can only use the predefined display scales from the **Utility\ToScale** menu.
3. Click at one corner of the area you want to zoom in on and move the mouse diagonally to stretch a rubberband box around this area. The area is displayed at its extents.

Shortcut: Press **(Home)** to center your drawing at the current cursor position in the **Drawing Area**.

Stretch a rubberband box around
the area you want to zoom in on

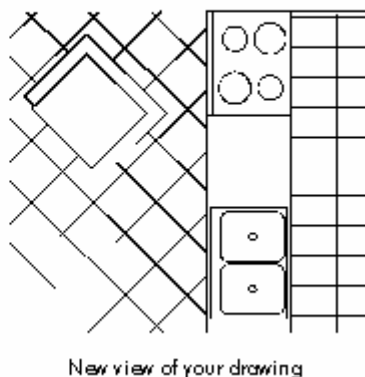
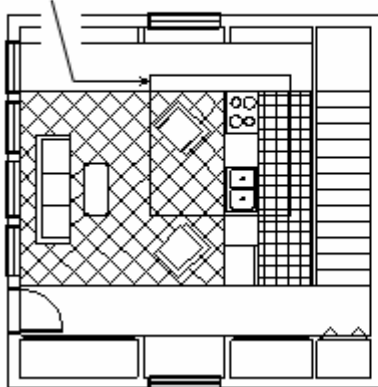


Figure 2.27: Zooming in on your drawing.

Changing Your View Using the Keyboard

In addition to zooming in and out on your drawing, you can use the arrow keys to scroll, or pan, in different directions around the screen. Your view of your drawing changes each time you press an arrow key. Pressing theuparrowkey lets you look up towards the top of your drawing by moving the drawing down. The *scroll distance*, or the distance that the drawing moves each time you press an arrow key, is defined using the ScrlDist option in the Utility/Settings menu. See “Setting the Panning Distance” in the “Before You Draw” chapter for more information on setting the scroll distance.

Saving Drawings

It is a good idea to save your drawing periodically, especially when you walk away from the computer or when you’ve made several changes. There are four ways to save your drawing in DataCAD.

- Press **[Shift] + [F]** (if Caps Lock is on, just press **[F]**). This shortcut is a great way to quickly save your drawing with minimal interruption.
- Click on Save in the File pull-down menu.

Shortcut: Press **[Ctrl]+[S]** to save your drawing.

- Click on Save As in the File pull-down menu to save your changes under a different drawing filename.
- Click on Save All in the File pull-down menu to save changes to all open drawings.

Warning: If you close your drawing without saving it, any changes you’ve made since you last saved it will be lost. If it’s a new drawing that has not yet been saved, all references to the drawing in DataCAD will be erased.

More About Recovering Lost Work

At regular intervals, DataCAD automatically saves the changes you’ve made to your drawing to a temporary file, called an autosave (.ASV) file. The autosave file exists only until you save the drawing. When DataCAD updates your drawing’s autosave file, you’ll see the message “Automatic save in progress” in the user prompt line in the Message Area. See “Automatically Saving Your Drawing” in the “Before You Draw” chapter for information on how to set the automatic save interval.

The autosave file is located in your DATACAD\TEMP\ folder and has the same name as your drawing file but with an .ASV extension; so if your drawing file is named HOUSE.DC5, your autosave file will be HOUSE.ASV.

An autosave file is different from a backup (.BAK) file in that it is a *temporary* file that exists only during your work session in DataCAD. If you save your drawing or close DataCAD, the autosave file is DELETED. It does not overwrite or update your drawing or backup files in any way.

If you experience a power interruption caused by an electrical failure, hardware problem, or other error, any changes you made to your drawing since you last saved it are lost. Fortunately, the temporary autosave file is not disturbed, and you can recover your drawing to the point of the last autosave. There is also a .SWP file that is a copy of the drawing you're working on. If you exit DataCAD abnormally, you can simply rename the .SWP file with a .DC5 extension to recover your work.

Warning: Keep in mind that SWP files can be corrupt or invalid. You should use them only if the ASV or BAK file is either not available or is corrupt as well. If an ASV file exists, it should be your first choice for recovering a file.

It's always a good idea to rename your old drawing file to something like HOUSE.OLD before renaming the autosave file. That way you keep a copy of the original drawing file but have your newly renamed autosave file to work on.

➔ To recover changes made to your drawing since you last saved it:

1. Start DataCAD and open your drawing (.DC5) file. The message "File [drawing name] is already marked in use. Continuing with this file will result in data loss. Are you sure you wish to continue?" is displayed.
2. Click on Yes. The message "An autosave (.ASV) file for this drawing exists. Are you sure you want to continue?" is displayed.
3. Click on Yes to continue.
4. Click on Yes to rename the autosave file.
5. Type a new name for the autosave file and press **Enter**. This name must be different from the name of the original drawing. Consider naming this file with a generation letter or number: DRAWING1.DC5, for instance.
6. Press **Enter** to confirm the new filename. You now have two copies of the drawing; the new file is the most current.
8. Notice that the new filename is displayed in the Enter Drawing File to Edit dialog box. Click on the new file and click on Open. Your drawing opens in the Drawing Area.

A swap (.SWP) file is also created when you're working on a drawing. If you have a power interruption, for example, the swap file remains in the DATACAD\ TEMP folder and can also be renamed to recover your work.

➔ To recover drawing changes without starting DataCAD:

1. Do not start DataCAD again. Instead, in Windows Explorer, change to the folder, \DATACAD\TEMP\.
2. Click on the autosave file to highlight it; you do not need to open it.
3. Click on Rename in the File menu in Windows Explorer.
4. Type a new name for the file, changing the extension to .DC5. So if your autosave file is named HOUSE.ASV, rename it to HOUSE.DC5.

- ➔ To open your drawing without recovering changes in the autosave file:
1. Start DataCAD and open your drawing (.DC5) file. The message “File [drawing name] is already marked in use. Continuing with this file will result in data loss. Are you sure you wish to continue?” is displayed.
 2. Click on Yes. The message “An autosave (.ASV) file for this drawing exists. Are you sure you want to continue?” is displayed.
 3. Click on Yes to continue.
 4. Click on No; you won’t rename the autosave file. The autosave file is deleted and your drawing opens.

Another way DataCAD helps guard against loss of work is by automatically creating a backup (.BAK) file every time you save your drawing. Suppose you’re unable to open your drawing because of a disk failure or file corruption. By replacing your drawing (.DC5) file with the backup file, you can recover your previous drawing version and prevent a loss of your work.

- ➔ To recover your work using a backup file:
1. Start DataCAD.
 2. Go to the Enter Drawing File to Open dialog box. Change the Files of Type drop-down box to Backup Files (*.BAK).
 3. Click on your backup file to highlight it and click on Open. Your drawing is displayed in the Drawing Area, and automatically renamed with a .DC5 extension.
 4. Click on Save in the File pull-down menu. We recommend renaming and keeping your original file instead of overwriting it.

You can run the Layer Utility to clean up the recovered file.

More About Naming Your Files

Naming your drawings is an important step to meaningful and efficient file organization. Ideally, the names of your drawings and the folders you store them in will reflect the contents of the file as well as the project they pertain to.

Identifying your drawings and layers with names like “florplan” or “bricks” or “dmension” becomes confusing and redundant. Which floorplan? What kind of bricks? Dimensions of what?

Despite the fact that everyone would rather read names than numbers, a naming system based on numbers can eliminate confusion between files. How you structure this numbering system is up to you — only you know the specific needs of your office.

In the sample project outlined earlier in this chapter, we used a different naming system. In the filename YMC-MSR1, “YMC” is the job name abbreviation, “MSR” is the drawing type, and “1” is the drawing version number. A similar convention is used for naming layer files in that project.

The system described below is a typical numbering system that has worked well in several architectural offices.

- The first three characters of the filename = the last three digits of the office job number.
- The next two characters of the filename = the letter and number of the type of drawing. (See the following drawing type list.)
- The last two characters of the filename = the chronological revision number of the drawing.

A suggested system for identifying types of architectural drawings follows:

A0 = General (index, symbols, location plan, 3D views)
 A1 = Existing and temporary conditions, site work, demolition
 A2 = Plans, detail plans, roof plan
 A3 = Exterior elevations, detail elevations
 A4 = Building sections, wall sections
 A5 = Interior elevations, toilet elevations
 A6 = Reflected ceiling plan, floor patterns, furnishings
 A7 = Schedules, door and window types
 A8 = Details
 A9 = Alternates

Structural, mechanical, HVAC, and civil drawings can be identified by type in a similar fashion. Under the above system, for example, the drawing of the fourth elevation in job number 85236 would be named 236A304.DC5.

Exporting Files

It may often be necessary to export your DataCAD drawings to other file formats, so they may be used in other applications by you or other offices or consultants you work with. You can export both DWG and DXF files compatible with AutoCAD releases 12, 13, and 14 as well as AutoCAD 2000. You can also export a DC Render (.RAY) file or an o2c object. It's generally a good idea to run your drawing file through Layer Utility before exporting it. For details, see "Layer Utility" in the "Drawing Tools" chapter. See "DXF/DWG Translation" in the "Appendix" for more information about how entities are translated during export.

During export, line definitions and shape codes for DataCAD linetypes are stored in a .SHX file. If you export a DWG file of the DataCAD drawing MYDRAWING.DC5, the translator will create two files: MYDRAWING.DWG and MYDRAWING.SHX. For linetypes in the DWG file to display correctly, the .SHX file must be placed in the \SUPPORT folder in the AutoCAD directory.

DataCAD models can be exported to web-friendly VRML 1.0 (Virtual Reality Markup Language). This means that any of your customers who have a VRML viewer can see your 3D model.

➔ To export your drawing to a DXF or DWG file:

1. Save your drawing before starting the export function. This is especially important if you've made any changes to your drawing.
2. Click on Export in the File pull-down menu. A submenu is displayed.
3. Click on DWG or DXF in the submenu. Two export options are displayed in a submenu: All Layers and On Layers.

4. Click on All Layers to export all the layers in your drawing. Click on On Layers to export only those layers that are toggled on, or currently being displayed. The Export dialog box is displayed.
5. Go to the File name input box and type a name for the file.
6. Choose a file type from the Save As Type drop-down box. If you chose DWG in step 3, the default file type is R14-DWG file (*.dwg); if you chose DXF in step 3, the default file type is R14-DXF file (*.dxf).
7. Click on Save; the export begins. The export's progress is displayed in the Message Area. To cancel the translation, press [End]. If the filename you entered in step 4 already exists, a dialog box will be displayed, giving you the choice to overwrite (replace) the existing file. Also if the DWG/DXF file you're attempting to overwrite is open, a message explaining that the file is currently in use will be displayed. By default, DataCAD saves all exported files to the DATACAD\XFER folder.

Exported .RAY files include eight light sources: one sunlight and seven spotlights.

➔ To export to a DC Render (.RAY) file:

1. Click on Program Preferences in the Tools pull-down menu.
2. Click on the Misc tab in the Program Preferences dialog box.
3. Go to the DC Render Material Assignment section. Select By Color to export a .RAY file in which you can assign a different material to all entities of each color; select By Layer to export a .RAY file in which you can assign a different material to all entities on each layer; or select By Layer and Color to export a .RAY file in which you can assign different materials to all entities of each color on each layer. By Layer and Color is the default selection and is the most comprehensive.
4. Click on OK to save your changes and close the Preferences dialog box.
5. Click on Export in the File pull-down menu. A submenu is displayed.
6. Click on RAY in the submenu. A dialog box is displayed.
7. Type the name of the .RAY file that you'd like to export.
8. Decide what folder you want to use for the .RAY file. DC Render files are saved to your DATACAD\RAY folder by default. If you'd like to save your .RAY file to another folder, use the Save In drop-down box or the Up One Level icon button to change to another folder. To change the default folder that DC Render files are saved to, choose Program Preferences from the Tools pull-down menu and change the pathname for DC Render Files.
9. Click on Save to export your drawing to the named .RAY file.

➔ To export an o2c object:

1. Click on Export in the File pull-down menu. A submenu is displayed.
2. Click on o2c in the submenu. A dialog box is displayed.
3. Type a name for the o2c object that you'd like to export.

4. Decide what folder you want to use for the o2c object. By default, o2c objects are saved to your DATACAD\RAY folder. If you'd like to save your o2c objects to another folder, use the Save In drop-down box or the Up One Level icon button to change to another folder.
5. Click on Save to export your model as the named o2c object.

➔ To export a 3D model to VRML 1.0:

1. Select Export from the File pull-down menu.
2. Click on VRML in the extended menu. A dialog box appears.

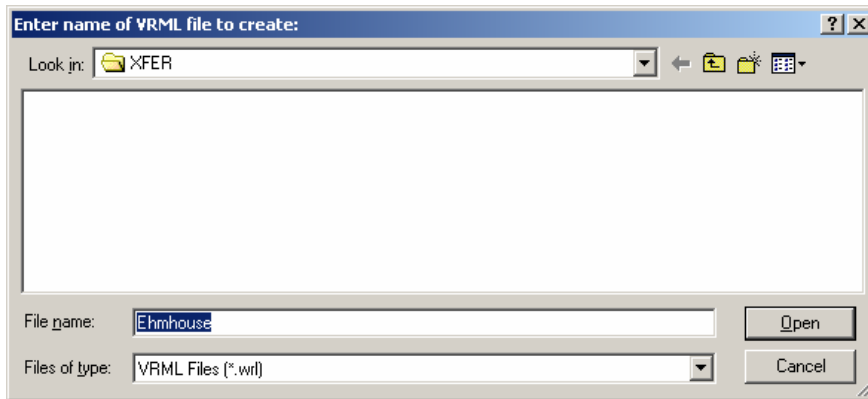


Figure 2.28: Save VRML file format for export

3. Supply a file name for the VRML file that you want to create. Then, click on Open or press (Enter).

More About o2c Objects

You can export your DataCAD model as an o2c object. o2c, an acronym for Objects to See, is a highly compressed 3D format that is optimized for Internet-based communication.

Simply export your DataCAD model as an o2c object and then either distribute it to your client as an e-mail attachment or embed it into one of your web pages.

Using the easy-to-use o2c player, your clients can freely rotate or walk through a fully rendered architectural model. They can view models in wire-frame, hidden-line, flat-shaded, textured, or ray-traced modes. The o2c objects can even contain animated elements that can be played in conjunction with the 3D viewing commands. And because you can generate a high-resolution, photorealistic rendering of the model right in the o2c player, you don't need to transmit excessively large images via email; your clients can simply create their own. While the o2c player provides comprehensive and powerful viewing controls, objects can't be modified.

For more information about o2c, see the chapter on "Viewing Your Model."

Tutorial: Drawing a Schoolhouse

In this chapter, you'll learn how to customize both DataCAD and your drawing before you ever draw a line. Then you'll draw the exterior and interior walls of the schoolhouse and add doors and windows to it.

In this chapter:

- ⊕ Drawing walls
- ⊕ Drawing windows
- ⊕ Drawing doors

Setting Up a New Drawing

DataCAD contains many options for customizing the way your system operates. In this tutorial, you will make only two modifications: adding layers to a new drawing and changing the snap grid setting.

To begin this tutorial, start DataCAD and start a new drawing, as described in “The Drawing Board” chapter.

Setting the Snap Grid for the Tutorial

First, you’ll need to change the snap grid setting.

➔ To set the snap grid:

1. Press **[S]** on your keyboard. Make sure Caps Lock is off. In the Message Area, you are prompted to “Enter X-snap grid.”
2. Type **.1** and press **[Enter]**. You are prompted to “Enter Y-snap grid.”
3. Type **.1** and press **[Enter]**.

You just set the accuracy of the cursor movement to one inch. Move your cursor around the drawing screen and look at the X and Y snap grid values in the Message Area.

Creating New Layers for the Tutorial

When you start a new drawing in DataCAD, it contains only one layer (called Layer1), which is also the active layer. Any settings in the active layer will automatically be used in new layers when you create them. For example, the snap grid setting you entered above will be assigned to each layer you add to your drawing; so you won’t have to set the snap grid in every new layer.

➔ To add layers to this drawing:

1. Right-click to return to the Utility menu from the previous exercise.
2. Click on Layers in the Utility menu.
3. Click on NewLayer in the Layers menu.
4. Type **8** and press **[Enter]**. You’ve just added eight layers to the drawing for a total of nine layers.

Naming Layers for the Tutorial

You’ll be given the layer names for this exercise, but you should develop your own plan for naming layers based on the type of work you do. To verify which menu is displayed in the Menu Window, check the Message Area. See Figure 3.1 to locate the menu name display.

➔ To rename your layers:

1. If you aren’t in the Layers menu, click on Layers in the Utility menu.
2. Click on Name in the Layers menu. A list of all your layers is displayed in the Menu Window.

3. Click on Layer1. You are prompted to “Enter new name.”
4. Type **Walls** and press **[Enter]**. Look at the list of layers in the Menu Window; notice that Layer1 has been changed to Walls.
5. Click on Layer2. You are prompted to “Enter new name.”
6. Type **Doors** and press **[Enter]**. Notice that Layer2 is now named Doors in the layer list in the Menu Window.
7. Rename the rest of your layers as follows: Layer3 to Windows, Layer4 to Plumb, Layer5 to Dim, Layer6 to Hatch, Layer7 to Notes, Layer8 to Misc, and Layer9 to Border.
8. Right-click twice to return to the Utility menu.

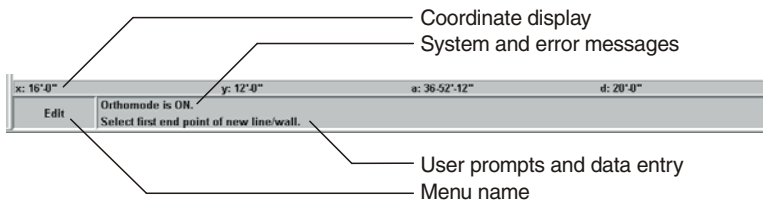


Figure 3.1: The Message Area display

Moving Between Layers in the Tutorial

Now that you’ve created and named your new layers, you can move among them. To draw walls on the Walls layer and doors on the Doors layer, you have to change to the corresponding layer before you begin drawing. In DataCAD terms, before you place a wall on your drawing you will make the Walls layer the active layer.

The Status Area shows the active layer, or the layer that you are drawing on. As you change active layers, the Status Area is updated with the corresponding layer name. You can only draw on the active layer.

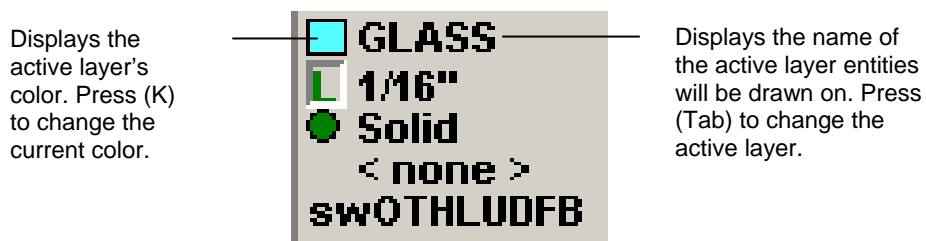


Figure 3.2: Check the Status Area for active layer name and color.

➔ To change the active layer:

1. Press **[Tab]** on your keyboard. Notice that the active layer name in the Status Area changes.
2. Keep pressing **[Tab]**. Eventually you’ll return to the beginning of the layer list. Pressing **[Shift] + [Tab]** cycles backwards through your layers.

Assigning Colors to Layers in the Tutorial

You can assign different colors to layers, so that any entities you add to a layer will be drawn in that layer's color. You'll be able to tell at a glance what layer an entity is on simply by its color. That color is the active color for that layer until you change it.

➔ To assign layer colors:

1. If you're not in the Layers menu, click on Layers in the Utility menu.
2. Press **[Tab]** until the Doors layer is the active layer. Check the Status Area to see which layer is active.
3. Click on Color in the Layers menu.

Shortcut: Press **[K]** to change the color of the active layer. Pressing **[K]** will cycle through the colors; pressing **[Shift] + [K]** cycles backwards through the layers.

4. Click on Green. The color assigned to the Doors layer is now Green, and you can see the layer color box in the Status Area is also displaying green.
5. Press **[Tab]**, so that the Windows layer is active.
6. Click on Color in the Layers menu.
7. Click on Cyan. The color assigned to the Windows layer is now Cyan.
8. Use this same procedure to assign colors to the rest of your layers. Make the Plumb layer Lt. Grey, the Dim layer Lt. Grey, the Hatch layer Brown, the Notes layer Lt. Red, the Misc layer Red, and the Border layer Lt. Green.
9. Right-click to return to the Utility menu.

Drawing Walls, Windows, and Doors

This part of the tutorial introduces you to using DataCAD as an architectural tool. You will learn how to use the wall menu options to begin drawing architectural plans. This tutorial uses Imperial units. See "Customizing Drawing Settings" in the "Before You Draw" chapter for more information on how to change drawing units.

Drawing the Schoolhouse Walls

Each wall that you draw will have two or more lines, rather than just one. In the following exercises, you will draw walls using both the mouse and the keyboard.

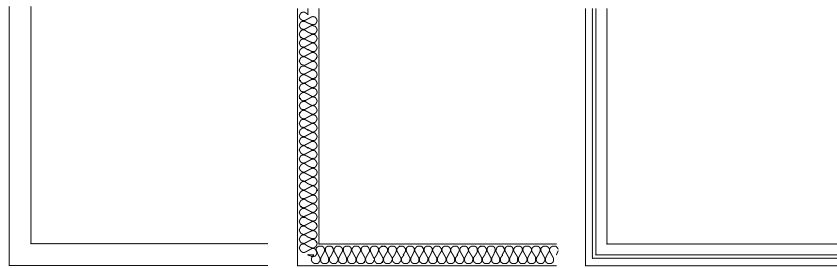


Figure 3.3: Examples, from left to right, of 2-line walls, 3-line walls, and 4-line walls

The Architect menu is used for drawing walls as well as doors and windows.

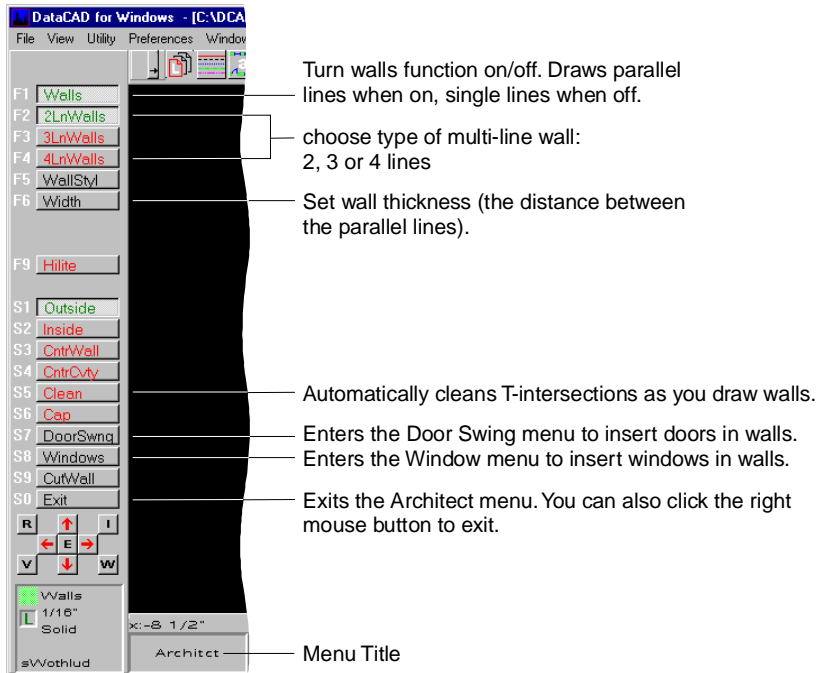


Figure 3.4: The Architect menu is used for drawing walls, windows, and doors.

Drawing the Schoolhouse's Exterior Walls

Before you draw the exterior walls of the schoolhouse, you should make sure your wall settings are correct.

➔ To set up for drawing exterior walls:

1. Continue with the drawing you started in the previous section of this tutorial.
2. In the Edit menu in the Menu Window, click on Architect. The Architect menu is displayed.
3. Click on Walls in the Architect menu to toggle it on; walls will be drawn instead of single lines. Notice that the "w" in SWOTHLUD in the Status Area is now uppercase. This indicates that you will draw walls instead of single lines.
4. Click on 2LnWalls in the Architect menu, if it's not already toggled on.
5. Click on Width in the Architect menu to set the width of the wall. A value menu is displayed in the Menu Window, and you are prompted to "Enter wall width." For more information on using value menus, see "Value Menus" in "The Drawing Board" chapter.
6. Click 1'-0" in the value menu and then right-click; or type 1 and press **Enter**.
7. Click Outside in the Architect menu; any walls you draw will be defined by the outside line of the wall.
8. Right-click to return to the Edit menu.

- Before you begin drawing the exterior walls, look at the Status Area to see that Walls is your active layer. If it's not, press **Tab** until it is.

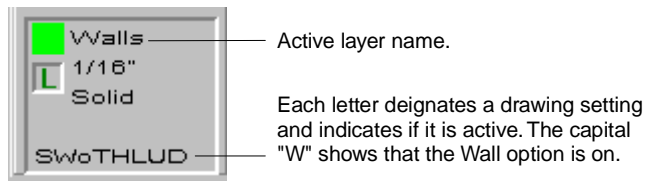


Figure 3.5: You can check both the active layer and the status of the Walls toggle in the Status Area.

Now you are ready to draw the exterior walls.

➔ To draw exterior walls:

- You'll draw the first wall using the mouse. Notice that the prompt in the Message Area reads "Select first end point of new line/wall." Move the cursor to the lower left of the Drawing Area and click to enter the first end point of the new wall. You are prompted to "Select next end point of line/wall."
- Move the cursor to the right until the coordinate readout shows an X measurement of 28'-0" and a Y measurement of 0, and click. This defines the exterior line of the wall. You are prompted to "Select a point to define the inside of the wall."
- Move the cursor to a position anywhere above the displayed line and click. The wall is immediately drawn. You are prompted to "Select next end point of line/wall."

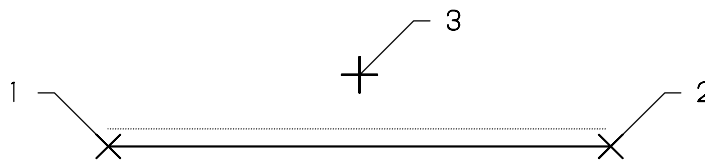


Figure 3.6: Drawing the first wall

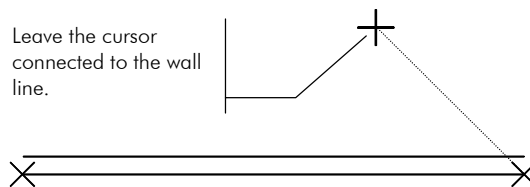


Figure 3.7: The wall is drawn.

- Now you'll draw a second wall using coordinate entry. Press **Insert** until the Message Area reads "Current input mode = relative polar (distance, angle)." You can also use coordinate entry to enter points. See "Drawing Using Coordinate Entry" in "The Drawing Board" chapter to review coordinate entry methods.
- Press **Spacebar** to activate coordinate entry. You are prompted to "Enter relative distance."
- Type **40.6** and press **Enter**. (40.6 means 40 feet 6 inches in architectural units.) You are prompted to "Enter relative angle."

7. Type **90** and press **[Enter]**. The wall is drawn and the corner is automatically cleaned. If the new wall extends beyond your view of the Drawing Area, use the arrow keys to move your drawing into view.

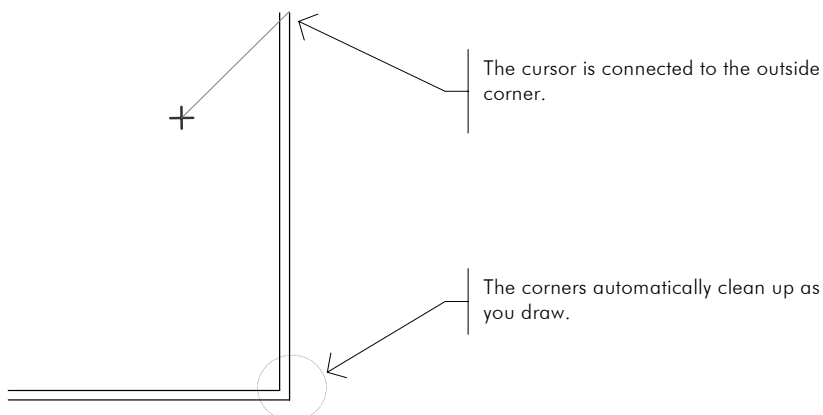


Figure 3.8: The second wall is drawn and the corner is cleaned.

8. Click **W** on the Navigation Pad to go to the WindowIn menu. The WindowIn menu is displayed in the Menu Window.
9. Click on **FreeZoom** in the WindowIn menu to toggle it off.
10. Click on **Extents** in the WindowIn menu to zoom out to the extents of the drawing.
11. Right-click to return to the Edit menu.
12. Press **[Spacebar]**. You are prompted to “Enter relative distance.”
13. Type **28** and press **[Enter]**. You are prompted to “Enter relative angle.”
14. Type **180** and press **[Enter]**.
15. Move the cursor close to the inside line of the corner where you originally began and click. The last corner is cleaned, leaving a clean outline of the schoolhouse exterior walls.

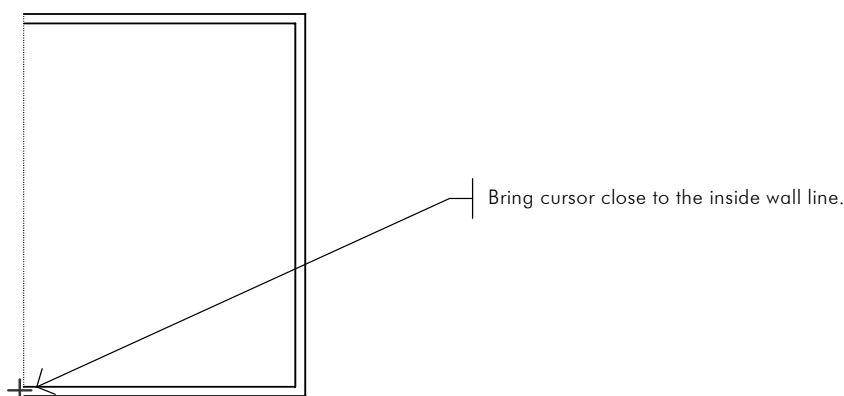


Figure 3.9: Ending your exterior wall

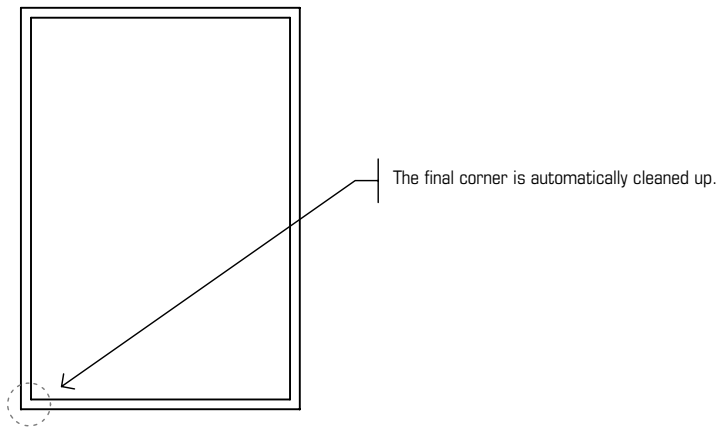


Figure 3.10: The exterior walls are complete.

Drawing the Schoolhouse's Interior Walls

Now you will draw the interior walls. You'll change the wall thickness for the interior walls to four inches.

➔ To set up for drawing the interior walls:

1. In the Edit menu in the Menu Window, click on Architct. The Architct menu is displayed.

Shortcut: Press **[A]** to access the Architct menu at any time.

2. Click on Width in the Architct menu. You are prompted to "Enter wall width."
3. Type **.4** and press **[Enter]**.
4. Click on Inside in the Architct menu to define the wall by the inside line.
5. Click on Clean to toggle on automatic T intersection cleaning. Walls and 2LnWalls should still be toggled on.
6. Right-click once to return to the Edit menu.

➔ To draw interior walls:

1. Click **W** on the Navigation Pad to open the WindowIn menu. You are prompted to "Select first corner of the Zoom window."
2. Position your cursor outside the lower-left corner of the plan and click. You are prompted to "Select second corner of the Zoom window."
3. Move the mouse diagonally, stretching the rubberband box until it encloses the lower half of the plan as in the figure below and click. Your drawing now appears larger.

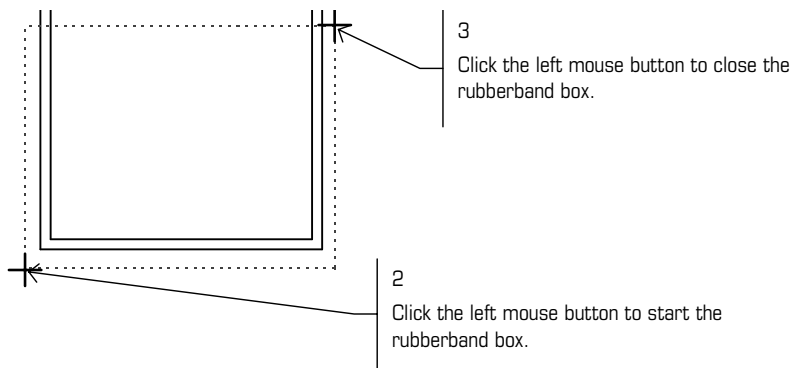


Figure 3.11: Zooming in on the lower half of the plan.

4. Right-click to return to the Edit menu.
5. Press [`] on the keyboard to set X and Y to 0. You are prompted to “Select reference point.” The accent key ([`]) is located in the upper-left corner of your keyboard, just below [Esc] and to the left of [1] on most keyboards.
6. Move the cursor close to the inside left corner of the bottom wall and either click the middle mouse button or press [N] on the keyboard. This sets this corner as the new reference point to work from. Steps 6-9 are the equivalent of placing one end of an architect’s scale at a corner and marking a point at 10’-2”.

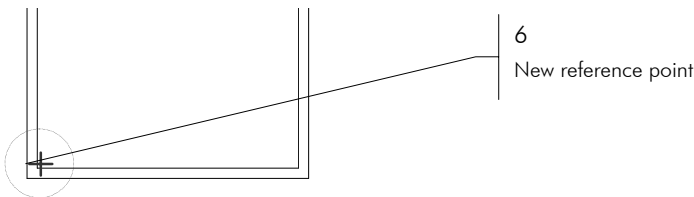


Figure 3.12: Setting a reference point

7. Press [Spacebar]. You are prompted to “Enter relative distance.”
8. Type 10.2 (the inside dimension of the office) and press [Enter]. You are prompted to “Enter relative angle.”
9. Type 0 for the relative angle and press [Enter]. Notice that your cursor is now attached to the wall exactly 10’-2” to the right of the corner you used as your reference point. This will be the first point of the interior wall.

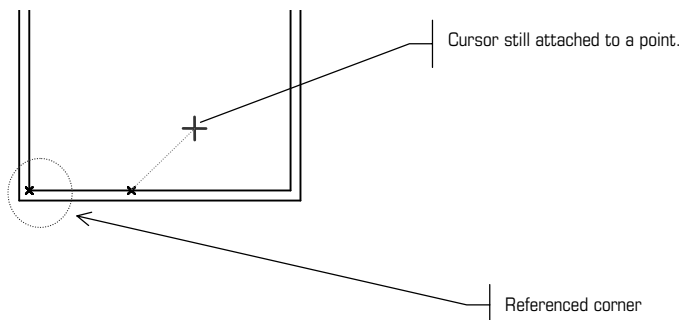


Figure 3.13: Starting your interior wall

10. Press [Spacebar]. You are prompted to “Enter relative distance.”

11. Type **10.8** (the inside dimension of the office) and press **[Enter]**. You are prompted to "Enter relative angle."
12. Type **90** and press **[Enter]**. You are prompted to "Select a point to define the Outside of the wall."
13. Move the cursor to the right of the displayed line and click to indicate the outside of the wall.

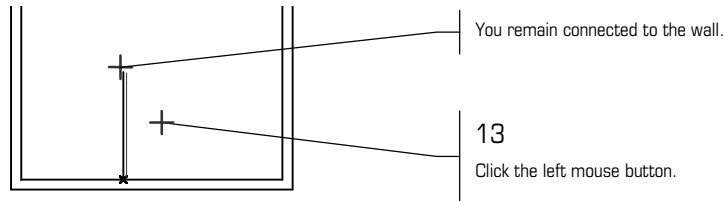


Figure 3.14: Defining the outside of the wall

14. Move the cursor close to the inside line of the left exterior wall and click.

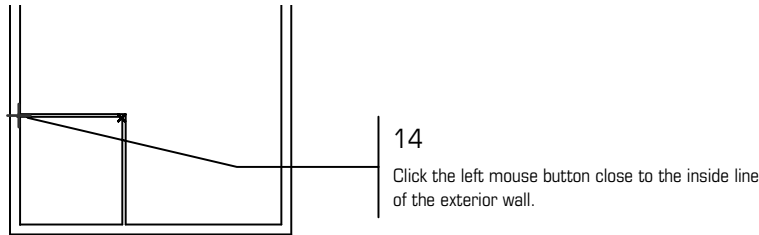


Figure 3.15: Drawing the last office wall

15. Right-click to disconnect from the line. The office area is complete and all intersections are automatically cleaned.

➔ To draw the remaining interior walls:

1. Press **[`]** on the keyboard. You are prompted to "Select reference point."
2. Move the cursor close to the inside right corner of the bottom wall and click the middle mouse button. Remember, if you don't have a three-button mouse you can also press **[N]** on the keyboard.

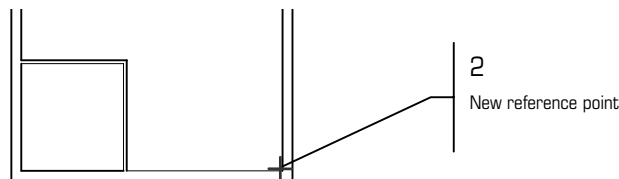


Figure 3.16: Selecting a new reference point

3. Press **[Spacebar]**. You are prompted to "Enter relative distance."
4. Type **10.2** and press **[Enter]**. You are prompted to "Enter relative angle."
5. Type **180** and press **[Enter]**. The cursor connects to a point on the inside line of the exterior wall.
6. Press **[Spacebar]**. You are prompted to "Enter relative distance."
7. Type **10.8** and press **[Enter]**. You are prompted to "Enter relative angle."

8. Type **90** and press **[Enter]**. You are prompted to “Select a point to define the outside of the wall.”
9. Move the cursor to the left of the new wall and click to define the outside of the wall.
10. Move the cursor to the right just before the inside line of the exterior wall and click.
11. Right-click to disconnect from the wall line. The wall is drawn and your intersections are cleaned.

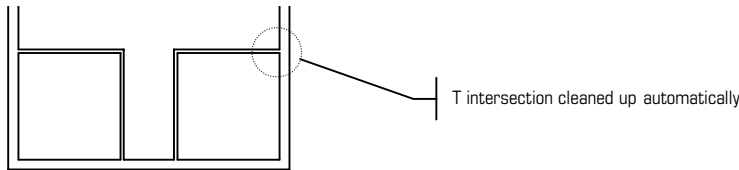


Figure 3.17: The wall is drawn and the intersections are cleaned.

12. Press **[.]**. You are prompted to “Select reference point.”
13. Move the cursor near the inside corner of the wall you just drew and click the middle mouse button.

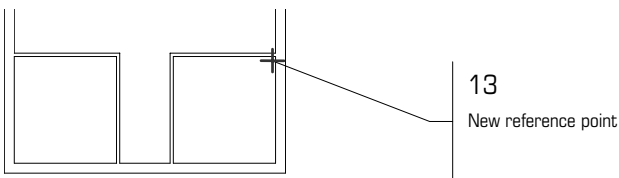


Figure 3.18: Selecting a new reference point

14. Press **[Spacebar]**. You are prompted to “Enter relative distance.”
15. Type **5.2** and press **[Enter]**. You are prompted to “Enter relative angle.”
16. Type **270** and press **[Enter]**. The cursor connects to a point on the inside line of the exterior wall.
17. Move the cursor to the left, close to the inside line of the wall, and click. You are prompted to “Select a point to define the Outside of the wall.” It is important in step 17 to select a point closer to the inside, not the outside, line of the room you just drew so that the correct wall line will be cleaned.
18. Move the cursor below the displayed line and click to define the outside of the wall.

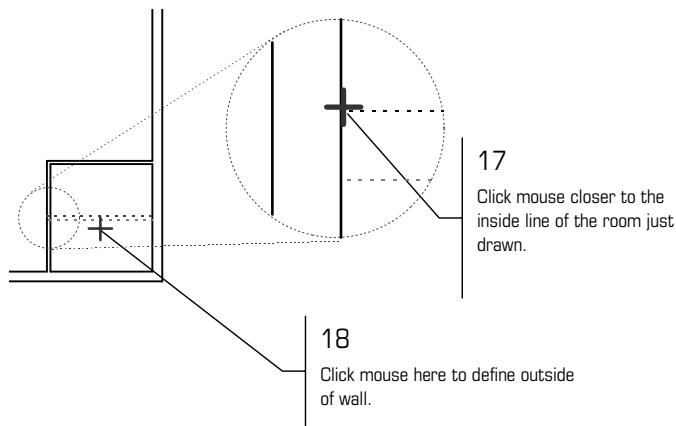


Figure 3.19: Drawing the dividing wall between the bathrooms

19. Right-click to disconnect from the wall.
20. Click on E on the Navigation Pad to view the drawing extents.

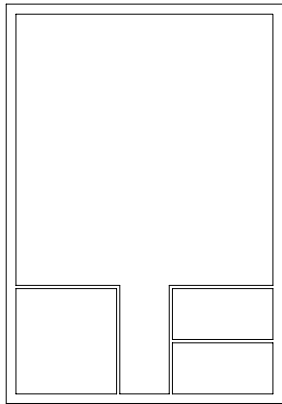


Figure 3.20: The walls of the schoolhouse floor plan

21. Press **Ctrl** + **S** on the keyboard to save the drawing. This is a good habit to get into. You should do it after you enter several items or make a lot of changes to your drawing.

You have now completed all the walls of the schoolhouse plan and saved your drawing. If any of the walls did not clean, and your drawing doesn't look like the figure above, the next section will help you straighten up your plan.

Cleaning the Schoolhouse's Wall Intersections

As you insert walls, DataCAD automatically cleans intersections. If you right-click by accident and disconnect from a wall prematurely, or your intersections clean at the wrong wall line, you will need to clean them.

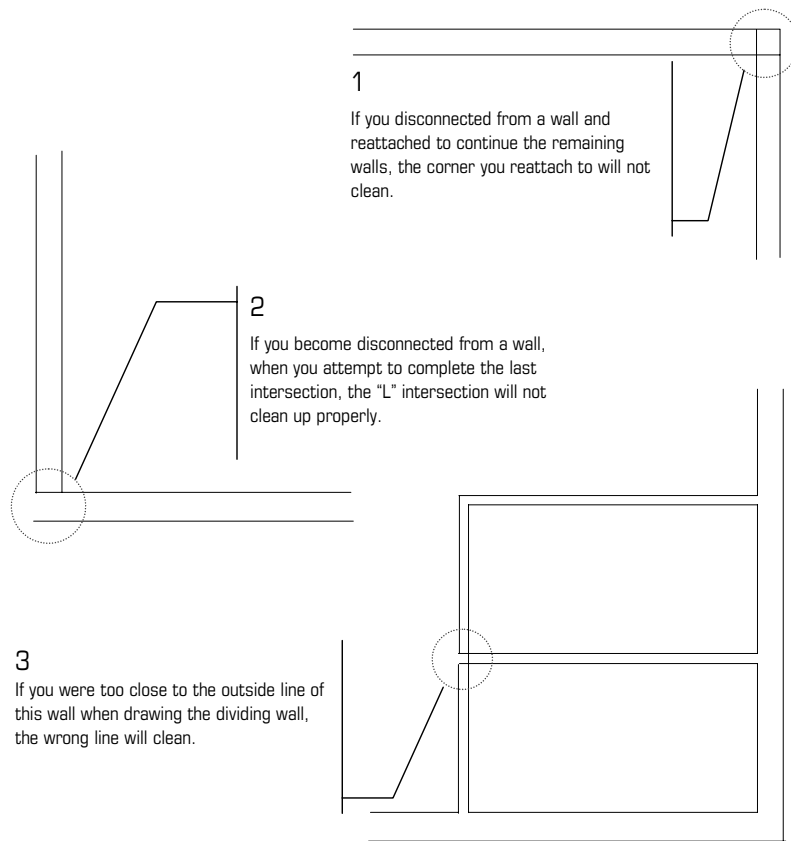


Figure 3.21: Examples of wall intersections that were improperly drawn

If an L intersection doesn't clean properly, despite Clean being toggled on when you drew the walls, it may be because you disconnected your cursor from the end of one wall and then started drawing the next wall. In such a case, you may have to use 2LnTrim in the Cleanup menu so the wall lines intersect properly, and then use Erase to remove any unwanted lines within the intersection.

➔ To fix the error shown in example 1:

1. In the Edit menu in the Menu Window, click on Cleanup.
2. Click on LIntSct in the Cleanup menu. You are prompted to "Select 1st corner around L intersection (wall lines only)."
3. Click above and to the right of the intersection.
4. Diagonally drag the cursor to the lower left of the intersection, stretching the rubberband box, and click.

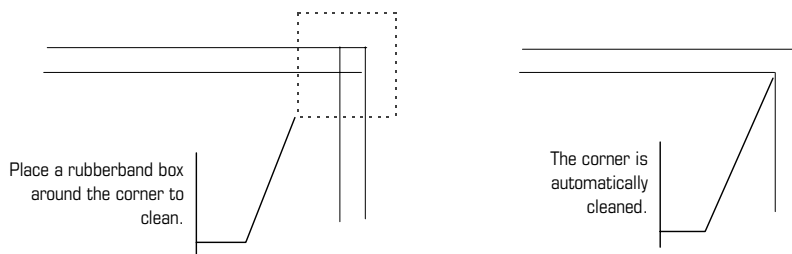


Figure 3.22: Cleaning an L intersection

➔ To fix the error shown in example 2 on the previous page:

1. In the Edit menu in the Menu Window, click on Cleanup.
2. Click on LIntSct in the Cleanup menu. You are prompted to "Select 1st corner around L intersection (wall lines only)."
3. Click above and to the right of the intersection.
4. Diagonally drag the cursor to the lower left of the intersection, stretching the rubberband box around the intersection, and click.

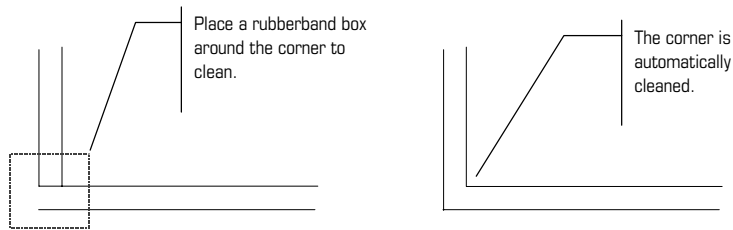


Figure 3.23: Cleaning an L intersection

➔ To fix the error shown in example 3 on the previous page:

1. In the Edit menu in the Menu Window, click on Cleanup.
2. Click on 1LnTrim in the Cleanup menu. You are prompted to "Select first point of line to clip to."
3. Click on Entity in the 1LnTrim menu. You are prompted to "Point to line to trim to."
4. Click on the inside wall line, shown in Figure 3.24. The line you select becomes dashed. You are prompted to "Point to outside."
5. Click to the left of the dashed line. You are prompted to "Select entity to trim."
6. Click on the two wall lines to select them as the lines to trim. The lines are trimmed.

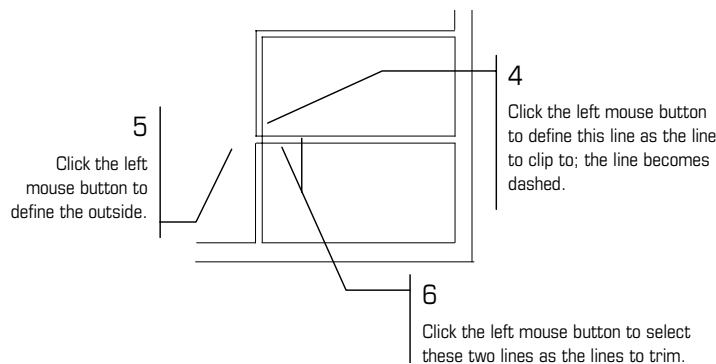


Figure 3.24: Completing a 1-line trim

7. Right-click once to return to the Cleanup menu.

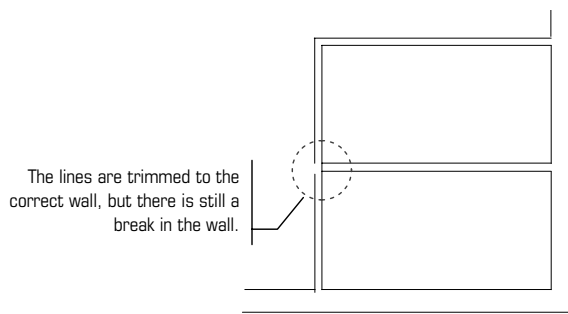


Figure 3.25: Trimming wall lines to the correct wall

8. To fix the break in the outside wall, click on WeldLine in the Cleanup menu. You are prompted to "Select first line to weld."
9. Click on one of the lines of the broken wall to select it; it becomes dashed.
10. Click on the other line of the broken wall to select it. The line mends and becomes a single entity.

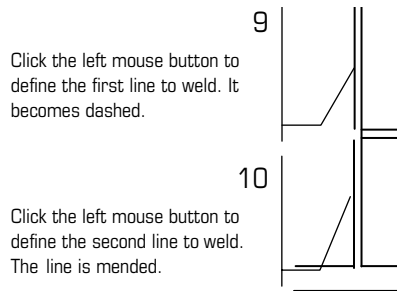


Figure 3.26: Selecting lines to be welded

11. Right-click once to return to the Cleanup menu.

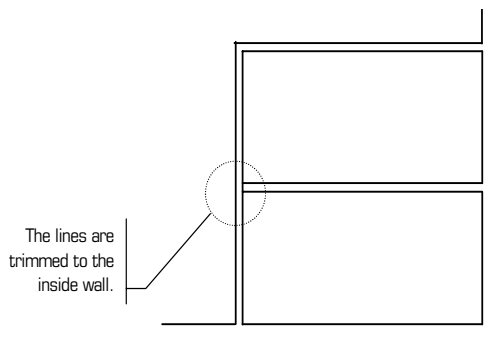


Figure 3.27: The wall line is welded.

12. To clean up the T intersection in the wall, click on TIntsct in the Cleanup menu. You are prompted to "Select 1st corner around "T" intersect (wall line ends only)".
13. Click at the lower left of the intersection you want to clean. You are prompted to "Select 2nd corner around "T" intersect (Wall line ends only)".
14. Move the cursor above and to the right of the intersection to enclose the corner to clean and click. You are prompted to "Point to a wall line to trim to."

15. Click on the inside line to select it as the line to trim to.

16. Right-click twice to return to the Edit menu.

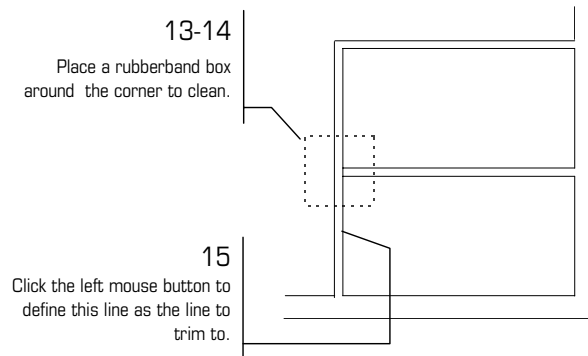


Figure 3.28: Completing a T intersection

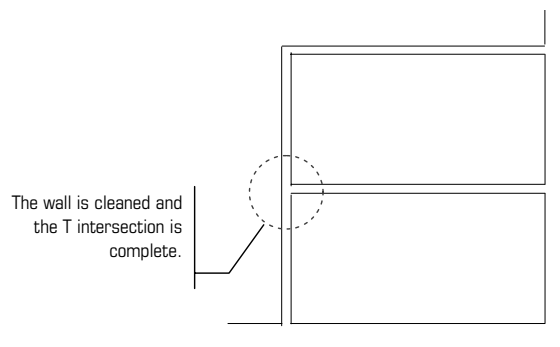


Figure 3.29: The wall intersection is cleaned.

Drawing Countertops in the Schoolhouse

While the 3-line walls function is usually used for drawing walls, we'll use it here to quickly draw parallel lines representing countertops.

➔ To set up to draw countertops:

1. Click on **Architct** in the Edit menu.
2. Click on **3LnWalls** in the **Architct** menu to toggle on the 3-line wall option. This allows you to draw three parallel lines at once. The two outer lines will define the edges of the counter and the centerline will be a dashed line to represent the upper cabinets. Walls should still be toggled on.
3. Click on **CntrLine** in the **Architct** menu to set the attributes of the centerline of the two parallel lines. This will represent the upper cabinets.
4. Click on **LineType** in the **CntrLine** menu. All available linetypes are listed in the Menu Window.
5. Click on **Dashed** in the **LineType** menu to make the centerline dashed.
6. Right-click once to return to the **Architct** menu.
7. Click on **Width** in the **Architct** menu. You are prompted to "Enter wall width."
8. Type **2** and press **[Enter]**. This sets the width of the countertop at two feet.

9. Click on Outside in the Architect menu; the wall will be defined by the outside line.
10. Click on Cap in the Architect menu to automatically cap the end of the parallel lines, or in this case countertops.
11. Right-click to return to the Edit menu.

1.

Now you are ready to draw the countertops.

➔ To draw countertops on the plan:

1. Click W on the Navigation Pad to open the WindowIn menu. You are prompted to “Select first corner of the Zoom window.”
2. Click near the lower-left corner of the bottom wall. You are prompted to “Select second corner of the Zoom window.”
3. Diagonally move the mouse until the rubberband box encloses the lower half of the plan and click. The plan now appears larger.
4. Right-click once to return to the Edit menu.
5. Move the cursor to the corner marked 6 in Figure 3.30 and click the middle mouse button to select it as your first reference point.

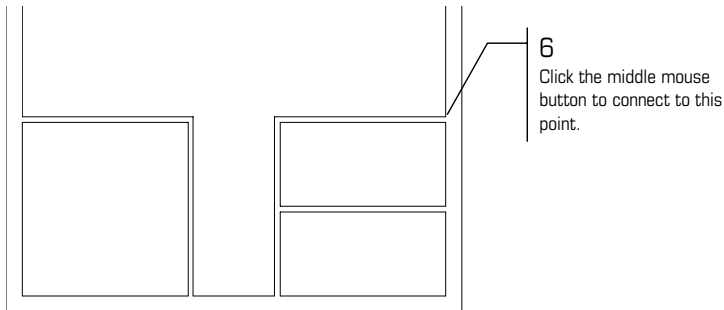


Figure 3.30: Selecting the starting point of the countertop

6. Press **[Spacebar]**. You are prompted to “Enter relative distance”.
7. Type **10.2** and press **[Enter]**. You are prompted to “Enter relative angle”.
8. Type **180** and press **[Enter]**.
9. Move the cursor above the displayed line and click to define the other side of the counter.

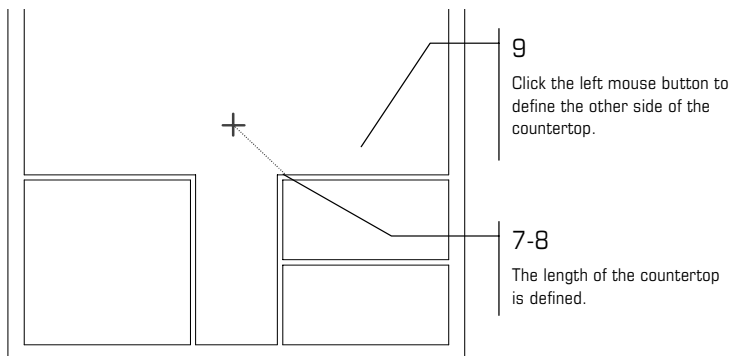


Figure 3.31: Drawing the countertop

10. Right-click to disconnect from the counter. The countertop lines are drawn and the end is capped.

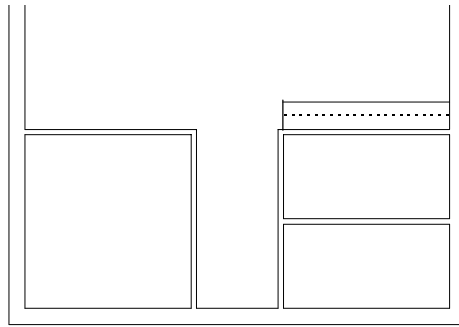


Figure 3.32: The first countertop is drawn.

11. Repeat steps 5 - 10 for the counter on the opposite side. Remember that the relative angle for this counter will be 0.

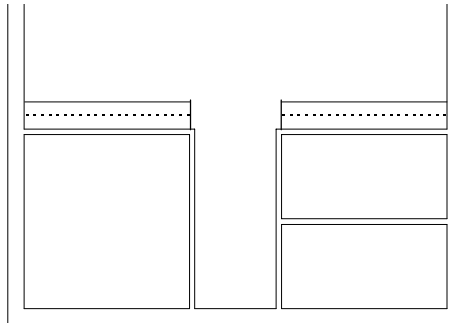


Figure 3.33: The second countertop is drawn.

12. Click E on the Navigation Pad to view the extents of the drawing.
13. Press **[Ctrl] + [S]** to save the drawing.

Drawing the Schoolhouse's Doors and Windows

Now that you have drawn the walls for the schoolhouse plan, you are ready to insert doors and windows.

➔ To set up to draw a door:

1. Before you begin to draw doors, press **[Tab]** until the Doors layer is the active layer in the Status Area.
2. In the Edit menu in the Menu Window, click on Architct.
3. Click on DoorSwng in the Architct menu.

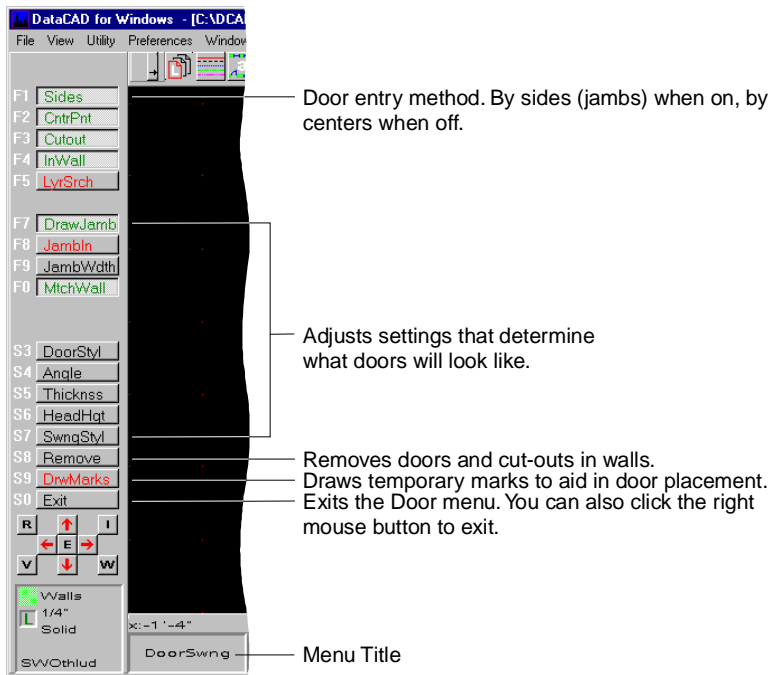


Figure 3.34: The DoorSwng menu

4. Click on DrwMarks in the DoorSwng menu to toggle the Draw Marks option on.
5. Click on LyrSrch to toggle it on. (If LyrSrch is already toggled on, click on it to toggle it off; then toggle it back on again.) You are prompted to “Select layer to search for walls”, and a list of your layers is displayed in the Menu Window.
6. Click on Walls. DataCAD now knows where to find your walls when it needs to cut them for doors and windows.

There are two methods to draw a door: by defining both of its sides and by defining one side and its center. You’ll draw the first door by sides. To position and begin drawing a door, you need to reference and measure from an existing point on the drawing.

➔ To draw a door by sides:

1. Click W on the Navigation Pad to go to the WindowIn menu. You are prompted to “Select first corner of the Zoom window.”
2. Click at the lower-left corner of the bottom wall. You are prompted to “Select second corner of the Zoom window.”
3. Diagonally move the mouse until the rubberband box encloses the lower half of the plan and click. The plan now appears larger.

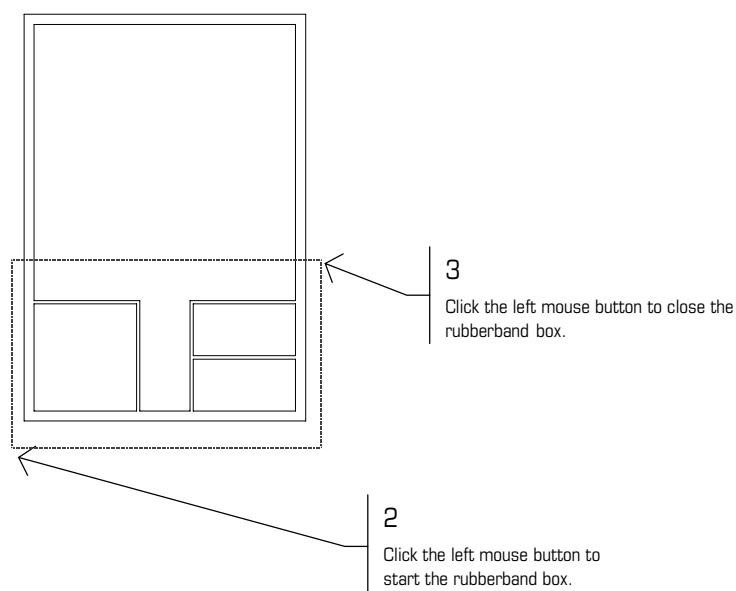


Figure 3.35: Zooming in on the plan

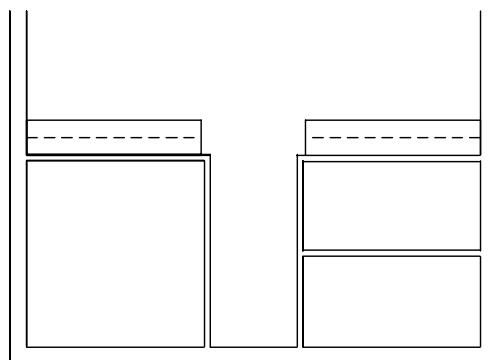


Figure 3.36: The result of zooming in on your drawing

4. Right-click once to return to the DoorSwng menu.
5. Press [`] on the keyboard. You are prompted to “Select reference point”.
6. Click the middle mouse button near the upper-right corner of the first room you drew, as shown in Figure 3.37. Stay close to the inside of the corner. This will be your reference point. You are prompted to “Select hinge side of door”.

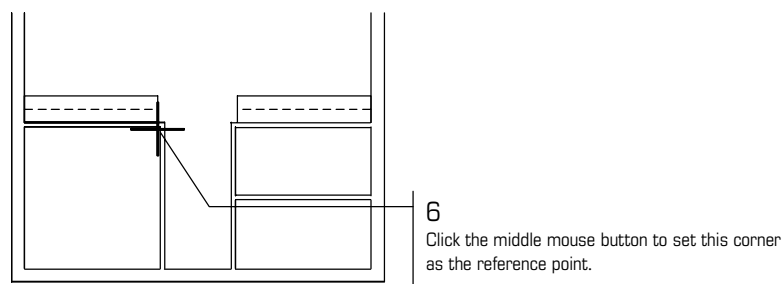


Figure 3.37: Setting a reference point.

7. Press **[Spacebar]**. You are prompted to “Enter relative distance”.
8. Type **.3** and press **[Enter]**. You are prompted to “Enter relative angle”.

9. Type **270** and press **[Enter]**. You are prompted to “Select strike side of door”.
10. Press **[Spacebar]**. You are prompted to “Enter relative distance”.
11. Type **3** and press **[Enter]**. You are prompted to “Enter relative angle”.
12. Type **270** and press **[Enter]**. You are prompted to “Select direction of door swing”.
13. Click anywhere inside the office (on the left side of the wall); this defines the direction of the swing. The wall is cut, and you are prompted to “Select any point on the outside of the wall”.
14. Click to the right of the outside wall to define the non-printing snap point location. The door is drawn.

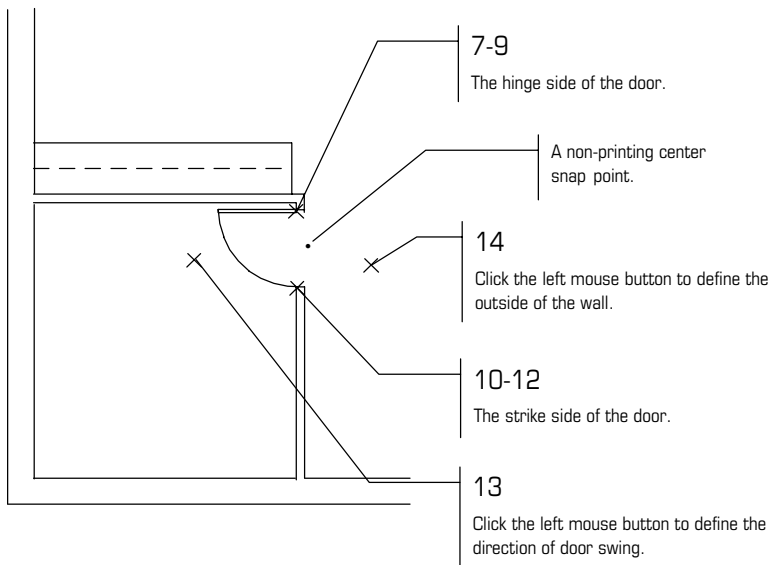


Figure 3.38: Drawing the door

➔ To draw a second door by center:

1. Click on Sides in the DoorSwng menu to toggle off the Sides option. Now you can define doors by entering their center and strike side.
2. Move the mouse to the location shown in Figure 3.39. Click the middle mouse button to snap to the midpoint of the inside front wall. You are prompted to “Select strike side of door”.

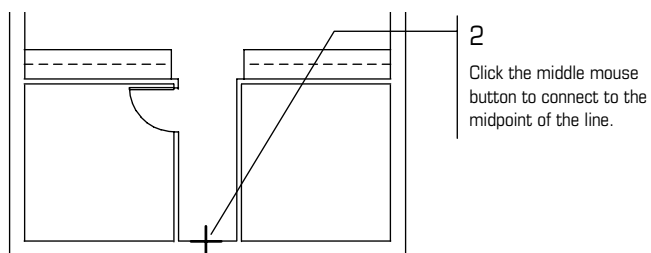


Figure 3.39: Defining the center of the front door

3. Press **[Spacebar]**. You are prompted to “Enter relative distance”.
4. Type **1.6** and press **[Enter]**. You are prompted to “Enter relative angle”.

5. Type **0** and press **[Enter]**. You are prompted to “Select direction of door swing”.
6. Click inside the hall (above the wall) to define the direction of the door swing. The wall is cut. You are prompted to “Select any point on the outside of the wall”.
7. Click anywhere on the outside of the exterior wall where you want to locate the non-printing snap point. The door is drawn.

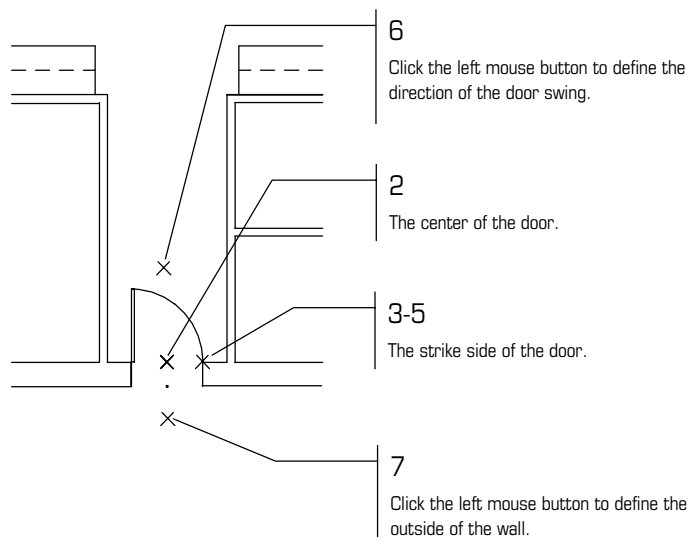


Figure 3.40: The front door is drawn.

8. Click on Sides to toggle on the Sides option.
9. Press **[`]** to set a new reference point.
10. Click the middle mouse button near the upper-left corner of Room A, shown in Figure 3.41. You are prompted to “Select hinge side of door”.

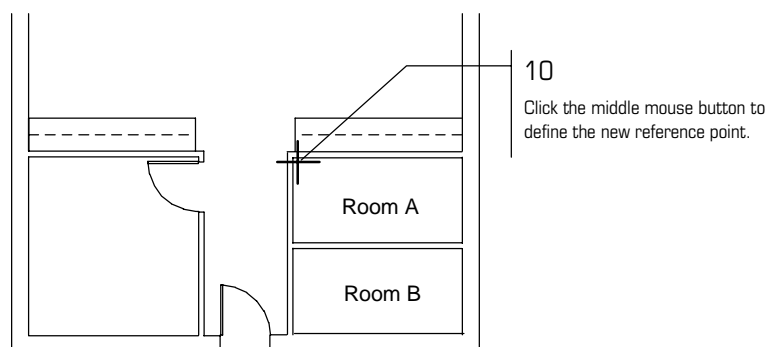


Figure 3.41: Locating the next door

11. Press **[Spacebar]**. You are prompted to “Enter relative distance”.
12. Type **.3** and press **[Enter]**. You are prompted to “Enter relative angle”.
13. Type **270** and press **[Enter]**. You are prompted to “Select strike side of door”.
14. Press **[Spacebar]**. You are prompted to “Enter relative distance”.
15. Type **3** and press **[Enter]**. You are prompted to “Enter relative angle”.

16. Type **270** and press **[Enter]**. You are prompted to “Select direction of door swing”.
17. Click anywhere inside Room A (on the right side of the wall) to define the direction of the swing. The wall is cut. You are prompted to “Select any point on the outside of the wall”.
18. Click inside the hall to define the outside of the wall. This defines the non-printing snap point location. The door is drawn.
19. Now repeat steps 9-18 to create a door for Room B.
20. Right-click twice to return to the Edit menu.

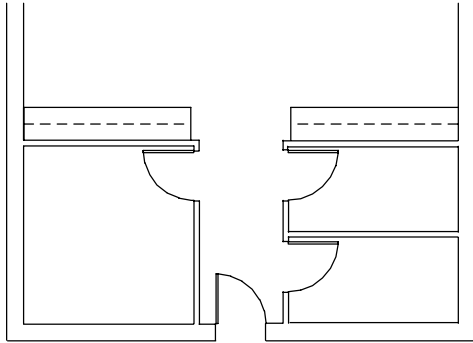


Figure 3.42: The remaining schoolhouse doors are drawn.

21. Click **R** on the Navigation Pad to recalculate the extents of the drawing.
22. Press **[Ctrl] + [S]** on the keyboard to save the drawing.

You have now completed entering all the doors for this exercise and saved them to your drawing file.

➔ To set up to draw windows:

1. Before you begin to draw windows, press **[Tab]** until the Windows layer is the active layer in the Status Area.
2. In the Edit menu in the Menu Window, click on **Architct**.
3. Click on **Windows** in the **Architct** menu. The Windows menu is displayed.

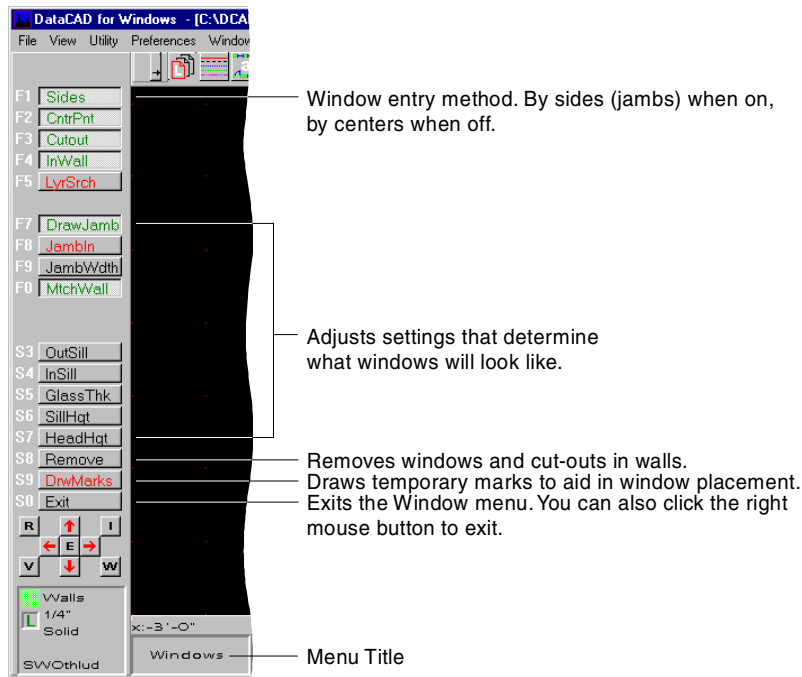


Figure 3.43: The Windows menu

➔ To draw the first window by sides:

1. Click W on the Navigation Pad to open the WindowIn menu. You are prompted to “Select first corner of the Zoom window”.
2. Click at the lower-left corner of the bottom wall. You are prompted to “Select second corner of the Zoom window”.
3. Diagonally move the mouse until the rubberband box encloses the lower half of the plan, and click to zoom in on your plan.

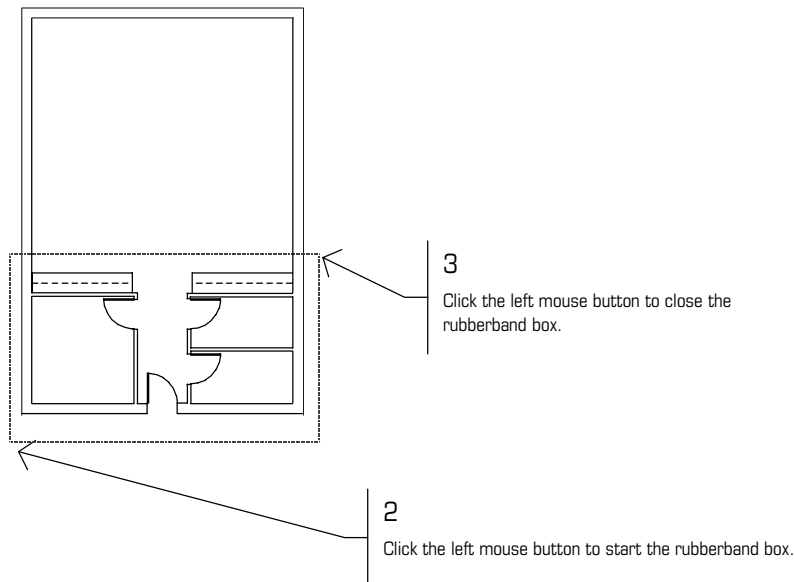


Figure 3.44: Zooming in on the lower half of your drawing

4. Right-click once to return to the Windows menu.

5. Press [```] on the keyboard. You are prompted to “Select reference point”.
6. Click the middle mouse button near the lower left corner of the interior wall shown in Figure 3.45. This will be your reference point. You are prompted to “Select one jamb of window”.

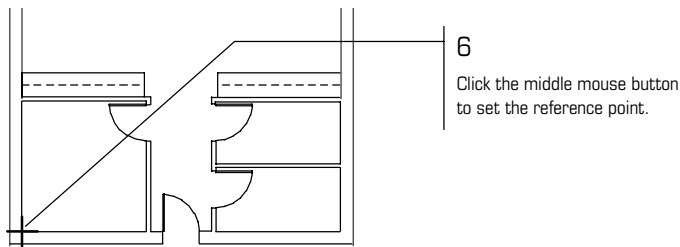


Figure 3.45: Setting a reference point

7. Press [Spacebar]. You are prompted to “Enter relative distance”.
8. Type 3.7 and press [Enter]. You are prompted to “Enter relative angle”.
9. Type 0 and press [Enter]. You are prompted to “Select one jamb of window”.
10. Press [Spacebar]. You are prompted to “Enter relative distance”.
11. Type 3 and press [Enter]. You are prompted to “Enter relative angle”.
12. Type 0 and press [Enter]. You are prompted to “Select any point on the outside of the wall”.
13. Click anywhere on the outside of the exterior wall. The window is drawn.

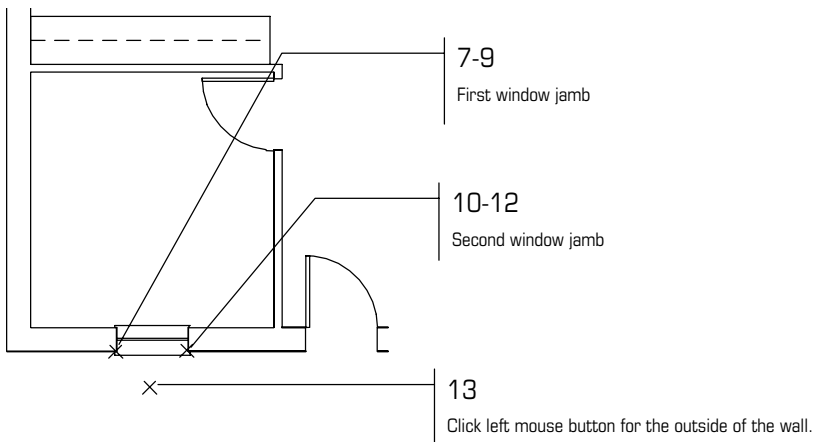


Figure 3.46: Drawing the first window

14. Move the cursor to the bottom right corner of Room B and repeat steps 6 - 14 for the window on the opposite side. Remember that the relative angles will now be 180.
15. Click E on the Navigation Pad to view the extents of the drawing.

➔ To draw multiple windows:

1. Click on Sides in the Windows menu to toggle Sides off; you can now define windows by the center and one side or jamb.

2. Click W on the Navigation Pad to open the WindowIn menu. You are prompted to “Select first corner of the Zoom window”.
3. Click at the upper-right corner of the top wall. You are prompted to “Select second corner of the Zoom window”.
4. Diagonally move the mouse until the rubberband box encloses the upper half of the plan, and click to zoom in on the plan.

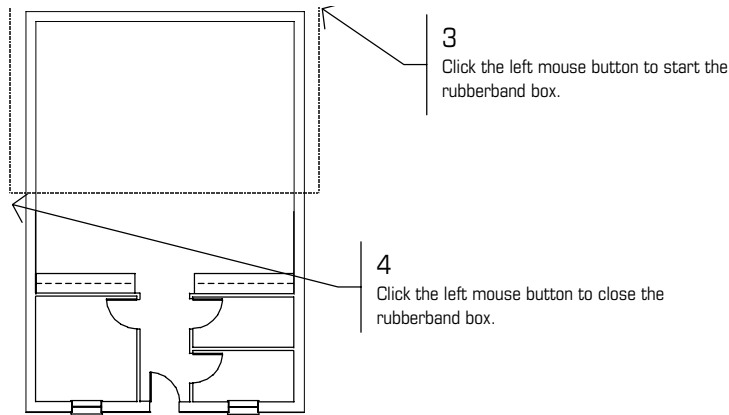


Figure 3.47: Zooming in on the upper half of your drawing

5. Right-click once to return to the Windows menu.
6. Press [`] on the keyboard. You are prompted to “Select reference point”.
7. Move the cursor to the top left corner of the interior wall, as shown in Figure 3.48, and click the middle mouse button to set the reference point. You are prompted to “Select center of window”.

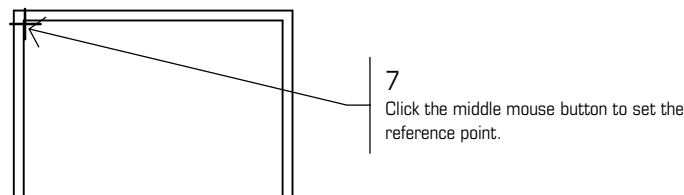


Figure 3.48: Setting a reference point

8. Press Spacebar. You are prompted to “Enter relative distance”.
9. Type 7.8 and press Enter. You are prompted to “Enter relative angle”.
10. Type 270 and press Enter. You are prompted to “Select one jamb of window”.
11. Press Spacebar. You are prompted to “Enter relative distance”.
12. Type 1.6 and press Enter. You are prompted to “Enter relative angle”.
13. Type 270 and press Enter. You are prompted to “Select any point on the outside of the wall”.
2. 14. Click anywhere on the outside of the exterior wall. The wall is cut and the window is drawn.

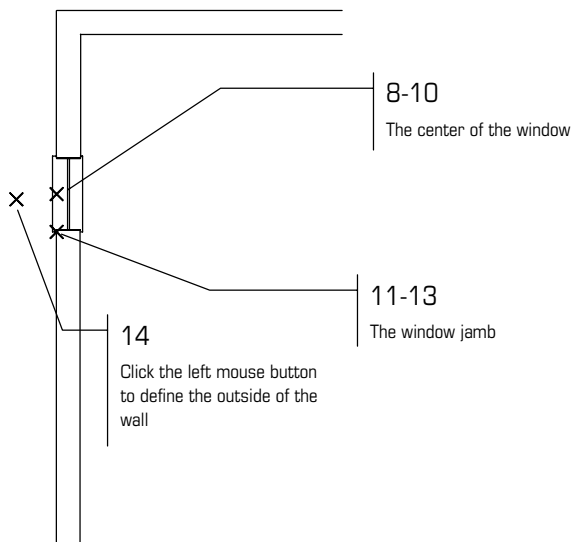


Figure 3.49: The wall is cut and the window is drawn.

15. Press [```] on the keyboard. You are prompted to “Select reference point”.
16. Move the cursor to the center snap point of the window you just drew and click the middle mouse button to set the reference point. You are prompted to “Select center of window”.
17. Press [`Spacebar`]. You are prompted to “Enter relative distance”.
18. Type **5.6** and press [`Enter`]. You are prompted to “Enter relative angle”.
19. Type **270** and press [`Enter`]. You are prompted to “Select one jamb of window”.
20. Press [`Spacebar`]. You are prompted to “Enter relative distance”.
21. Type **1.6** and press [`Enter`]. You are prompted to “Enter relative angle”.
22. Type **270** and press [`Enter`]. You are prompted to “Select any point of the outside of the wall”.
23. Click anywhere on the outside of the exterior wall. The wall is cut and the window is drawn.

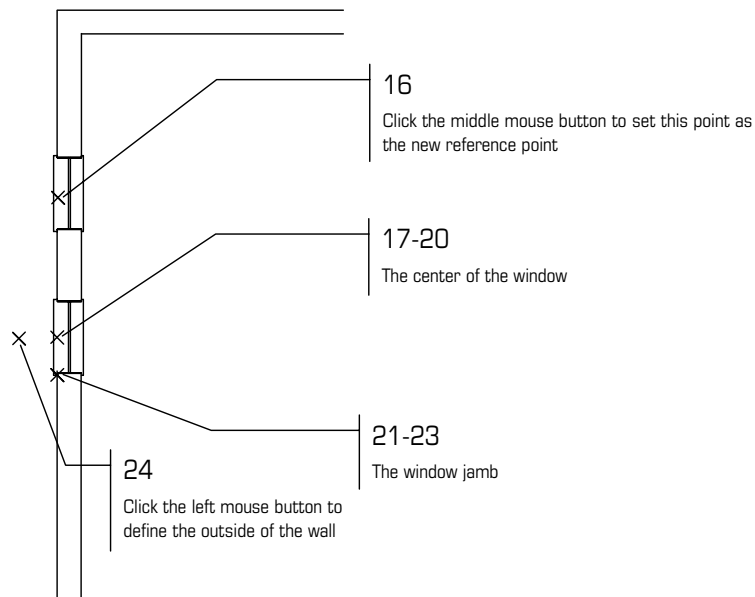


Figure 3.50: The wall is cut and the window is drawn.

24. Repeat steps 15 - 23 for the last window on this wall.
25. Repeat these steps to create the windows on the other wall.
26. Click E on the Navigation Pad to view the extents of the drawing. Your plan should look like Figure 3.51.

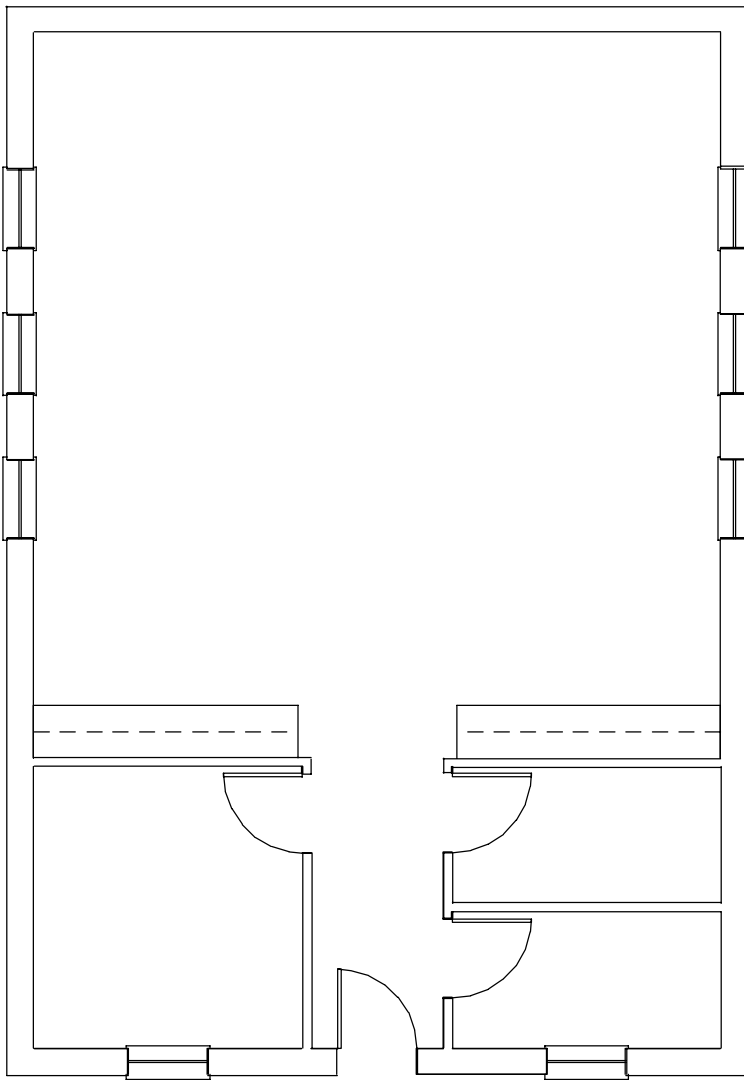


Figure 3.51: The Schoolhouse floor plan.

28. Press **Ctrl** + **S** to save the drawing.

3.

Tutorial: Adding Dimensions, Notes, Symbols, and Hatching

This part of the tutorial covers those functions that transform a drawing into an architectural construction document. You will add text notes and arrows, dimension the drawing, add a tile hatch to the hallway, and insert plumbing symbols in the bathrooms as well as a sink in one of the countertops. Finally, you'll draw a title block and border around the drawing.

In this chapter:

- ⊕ Drawing notes and other text
- ⊕ Dimensioning the drawing
- ⊕ Adding hatch to areas of the drawing
- ⊕ Inserting symbols
- ⊕ Drawing title blocks

Adding Text to the Schoolhouse Drawing

As you enter text, the schoolhouse floor plan will start to take shape. Depending on the size, placement, and font being used, your drawing can develop into something that looks very professional.

➔ To set text attributes:

1. If you are not already working in DataCAD, open the program now. Open the drawing you worked on in the previous chapter.
2. Press **[Tab]** until the Notes layer is the active layer, as displayed in the Status Area.
3. Click on Text in the Edit menu in the Menu Window, or click on Text in the Create pull-down menu. The Text menu is displayed in the Menu Window.

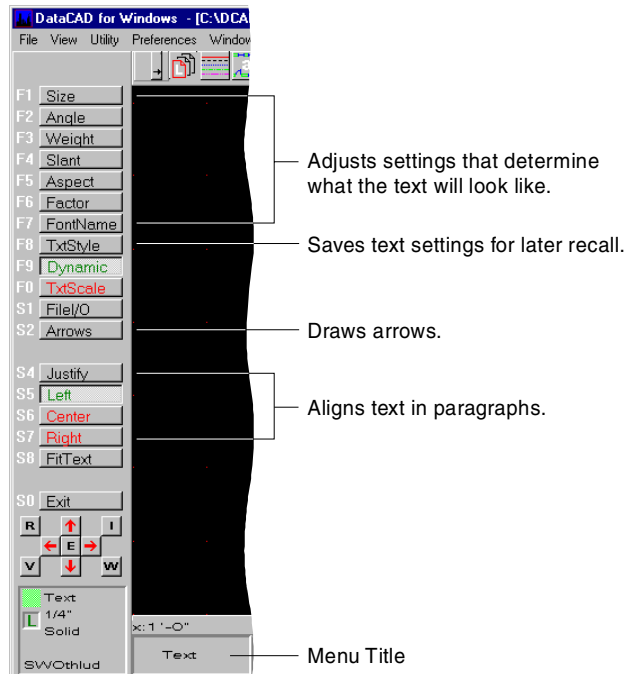


Figure 4.1: The Text menu

4. To set the size of the text, click on Size. A value menu is displayed. The text cursor graphically represents the current size of the text.
5. Click on 1'-0" and right-click, or type 1 and press **[Enter]**.
6. To set the aspect ratio of the text, click on Aspect. A value menu is displayed. An aspect ratio below 1.0 creates wider and fatter characters, while an aspect above 1.0 creates thinner characters.
7. Type .75 and press **[Enter]**.
8. Click on FontName to select a new font. The Set Text Parameters dialog box appears so that you can select CHR or TTF fonts.

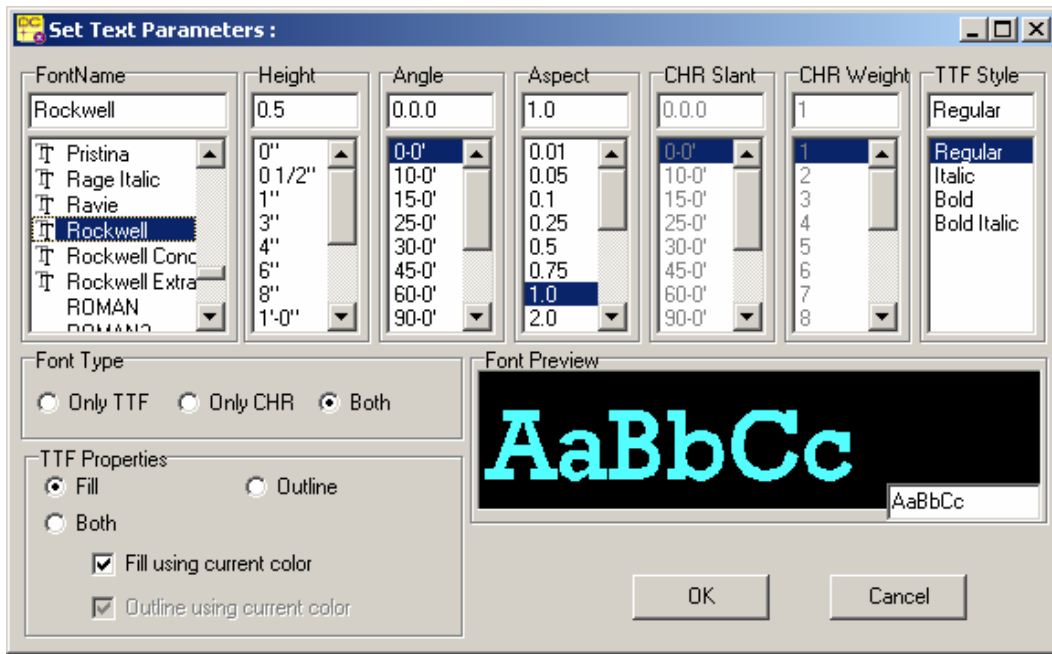


Figure 4.2: The Set Text Parameters dialog box

9. Click on ARCWY2HC.CHR. This font has a nice hand-drawn style.
10. Click on Center to center the text within an area on the drawing.

➔ To insert room names:

1. Move the cursor to the center of the large room. You are prompted to "Position text using cursor, then 'ENTER'".
2. Click to set the position of the text.

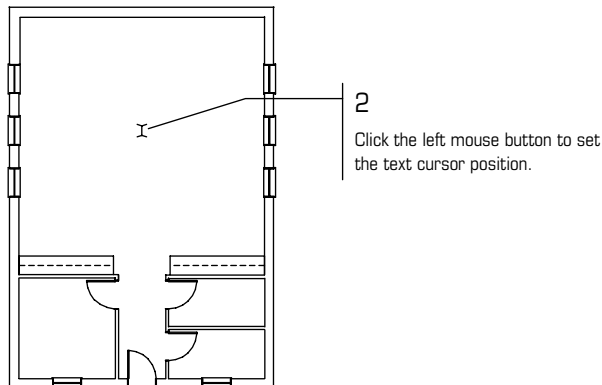


Figure 4.3: Positioning the text cursor

3. Press **[CapsLock]**, if it isn't already on.
4. Type **CLASSROOM**. The text appears in the drawing as you type.
5. Right-click to exit this command and return to the Text menu.
6. Repeat steps 1 - 5 for the office, but type **OFFICE** in the room to the lower left instead of typing **CLASSROOM**.

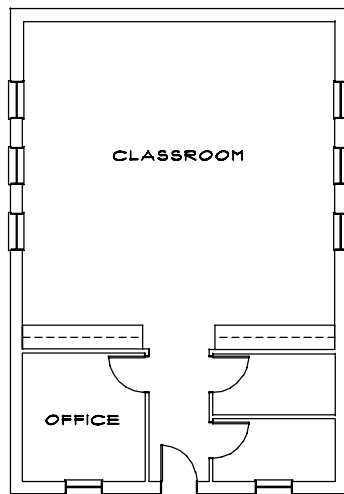


Figure 4.4: Room labels for the classroom and office are added

➔ To insert notes:

1. To set the size of the text, click on Size. A value menu is displayed.
2. Type **.6** and press **Enter**.
3. Click on Aspect. A value menu is displayed.
4. Type **1.25** and press **Enter**. This sets the aspect ratio to create a thinner text.
5. Click on Right to align the text on the right side of the indicated placement point.
6. Click in the center of the room, between the text **CLASSROOM** and the countertops. To zoom in closer to the drawing, press **PageDown**. Press **PageUp** to zoom out.
7. Type the following note, pressing **Enter** after each line:
PLASTIC LAMINATE COUNTERTOP
36" ABOVE FINISHED FLOOR WITH
ADJUSTABLE SHELVES BELOW
8. Right-click to exit this command and return to the Text menu.

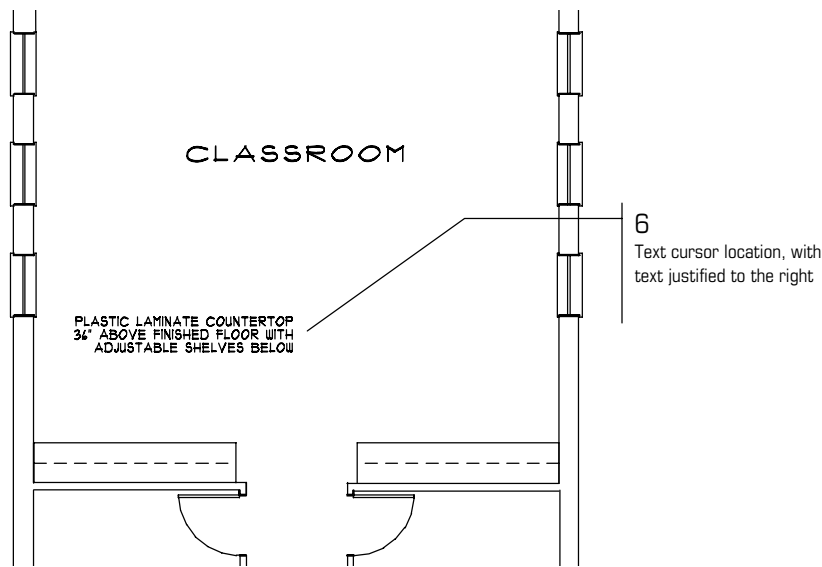


Figure 4.5: The note is placed on the drawing

Drawing Text Arrows in the Schoolhouse Drawing

➔ To draw text arrows:

1. Click on Arrows in the Text menu.
2. Click on Size to change the arrow size. You are prompted to "Enter arrow size".
3. Type .6 and press Enter. You are prompted to "Draw arrow".
4. Click near the right side of the paragraph of text you just entered. This defines the tail end of the arrow.
5. Click to define any bends in the tail of the arrow.
6. Click to define the arrowhead location.

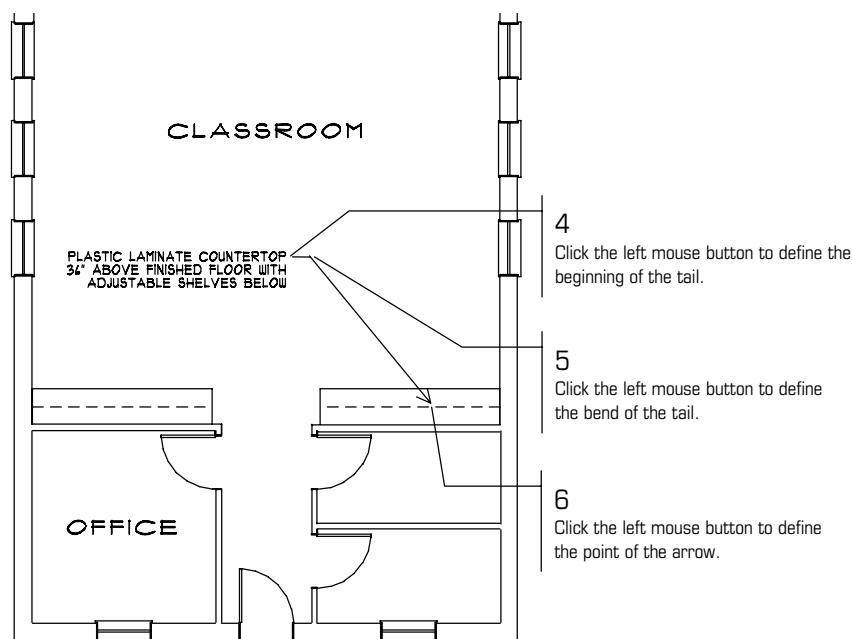


Figure 4.6: Placing an arrow in the drawing

7. Right-click once. The arrow is drawn.
8. Right-click twice to return to the Edit menu.

Editing Text in the Schoolhouse Drawing

➔ To correct existing text:

1. Click on Change in the Edit menu.
2. Click on Text to set the text attribute you want to change.
3. Click on Contents to change or edit the words in existing text.
4. Click on Area to select the text with a window box.
5. Click below and to the left of the text paragraph.
6. Move the mouse diagonally until the rubberband box completely encloses the entire note and click. The first line of text is displayed in the Message Area.

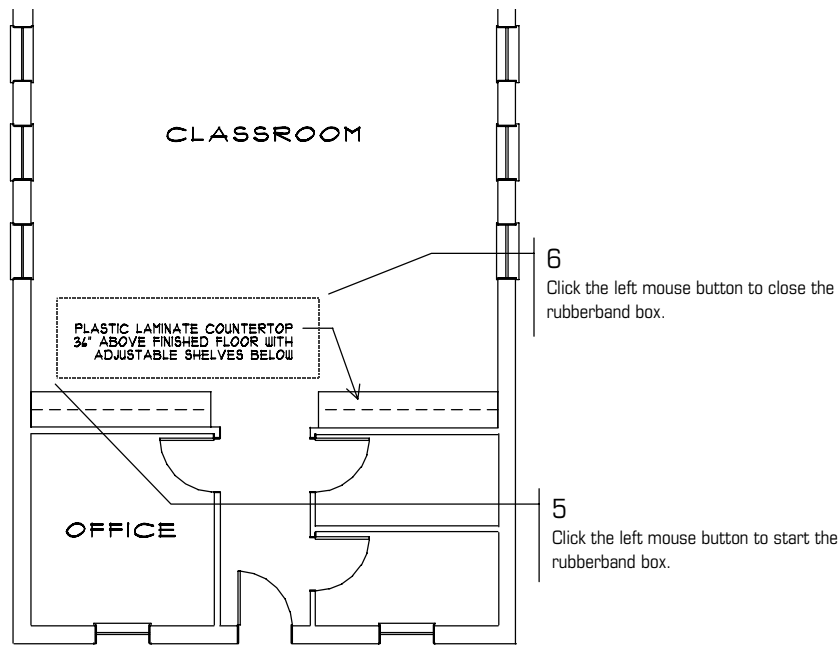


Figure 4.7: Changing text

7. Type over the text to make the following correction, pressing **[Enter]** when you're finished:

CERAMIC TILE COUNTERTOP

The first line of text is corrected and the second line of text is displayed in the prompt.

8. Press **[Enter]** to accept the second line as it is. The third line of text is displayed in the prompt.
9. Press **[Enter]** to accept the third line as it is.
10. Right-click twice to return to the Edit menu.
11. Click on E to view the extents of the drawing.
12. Press **[Ctrl] + [S]** to save your drawing.

Dimensioning the Schoolhouse Drawing

In an architectural office, there are many factors that determine the type of dimension used for a project. In this section of the tutorial, you will insert standard linear dimensions in the schoolhouse floor plan.

➔ To set up to dimension:

1. Press **[Tab]** until the Dim layer is the active layer in the Status Area.
2. Click on Dmension in the Utility menu to display the Dimension menu in the Menu Window.
3. To draw linear dimensions, click on Linear. The Linear menu is displayed.

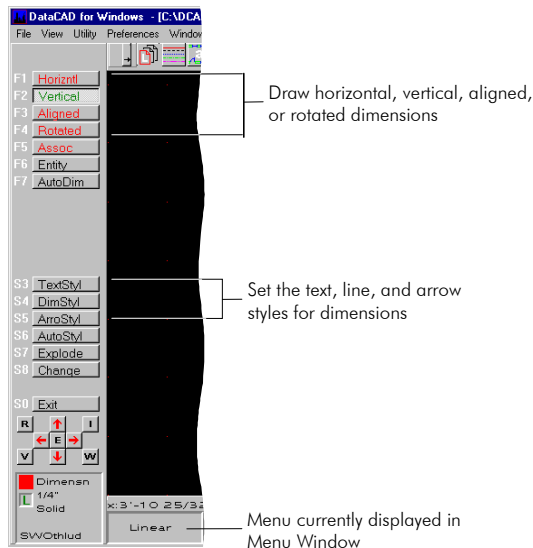


Figure 4.8: Linear dimension menu

4. Click on TextStyl to change the current text style or settings.
5. Click on FontName in the TextStyl menu to change the font.
6. Click on ARCWY2HC.CHR to match the font style of the notes.
7. Right-click to return to the Linear menu.

Drawing a String of Dimensions:

➔ To draw a dimension string:

1. Click on Vertical to draw vertical dimensions. You are prompted to "Select first point of distance to dimension".
2. Move the cursor to the lower left corner of the exterior wall and click the middle mouse button or press **[N]** on the keyboard to object snap to the corner. You are prompted to "Select second point of distance to dimension".
3. Move the cursor to the snap point of the window shown in Figure 4.9, and click the middle mouse button to snap to it. You are prompted with "Dimension which side of line?"

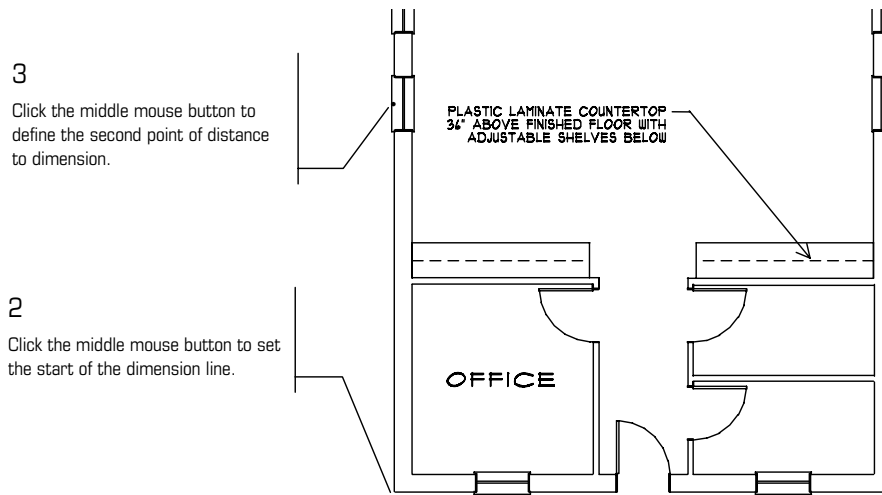


Figure 4.9: Placing the first dimension on the drawing

4. Move the cursor to the outside of the wall and click.
5. Click on StrngLin to continue drawing dimensions in a stringline. You are prompted to "Select next point to dimension to".
6. Move to the snap point of the middle window and click the middle mouse button to snap to the next point to dimension to. You are prompted to "Select next point to dimension to".
7. Move to the snap point of the last window and click the middle mouse button to snap to the next point to dimension to.
8. Click the middle mouse button to select the corner of the wall, which is also the end of the stringline.
9. Right-click to exit the Stringline menu and return to the Linear menu.

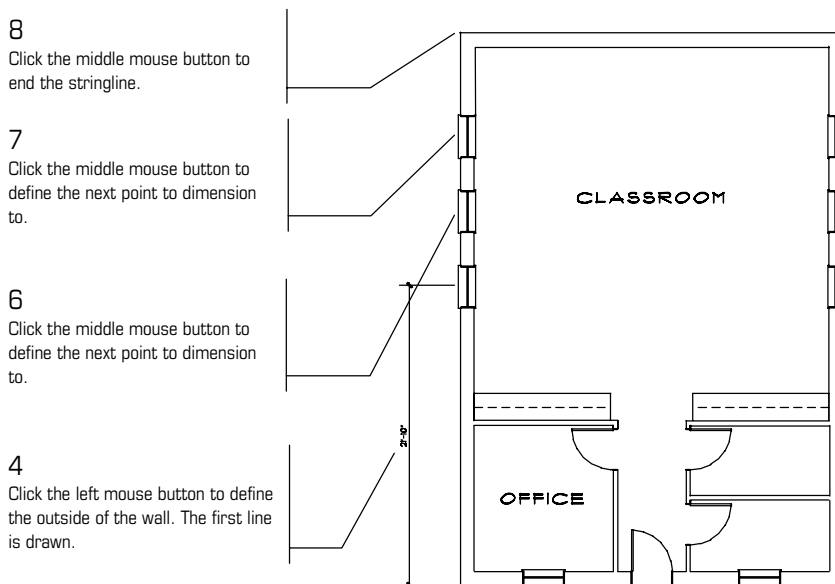


Figure 4.10: Defining a stringline of dimensions

10. Click on Overall in the Linear menu to place an overall dimension of the stringline.

11. Right-click twice to return to the Utility menu.

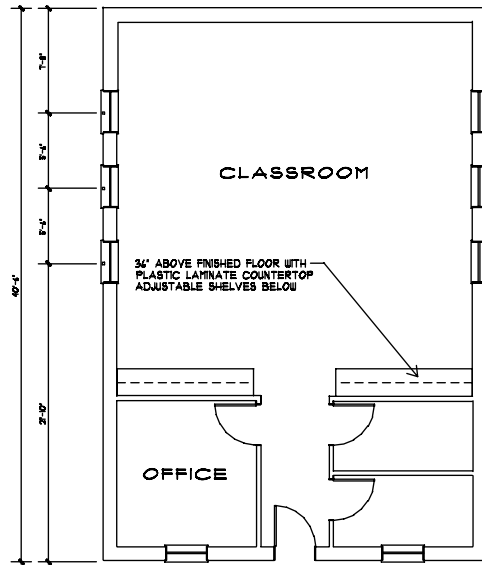


Figure 4.11: The stringline and overall dimensions are drawn

Automatic Dimensioning

You can also use automatic dimensioning; follow the steps below to use this second method of adding dimensions to your drawing.

➔ To use automatic dimensioning:

1. Click on Dmension in the Utility menu. The Dmension menu is displayed in the Menu Window.
2. Click on Linear. The Linear menu is displayed.
3. Click on AutoStyl to change the current automatic dimension settings.
4. Click on OnlyPnts to search and dimension to points only.
5. Click on LyrSrch to enable layer searching. All of the layers will be searched for points to dimension.
6. Click on MissDist to set the distance to look for snap points. 7. Click on 1" and right-click, or type .1 and press **[Enter]**.
8. Right-click to exit the Auto Style menu and return to the Linear menu.
9. Click on Horizntl to draw horizontal dimensions.
10. Click on AutoDim. You are prompted to "Select first endpoint of line to dimension along".
11. Move the cursor to the bottom left corner of the exterior wall and click the middle mouse button or press **[N]** on the keyboard. You are prompted to "Select second endpoint of line to dimension along".
12. Move the cursor to the bottom right corner of the exterior wall and click the middle mouse button. You are prompted with "Dimension which side of line?"

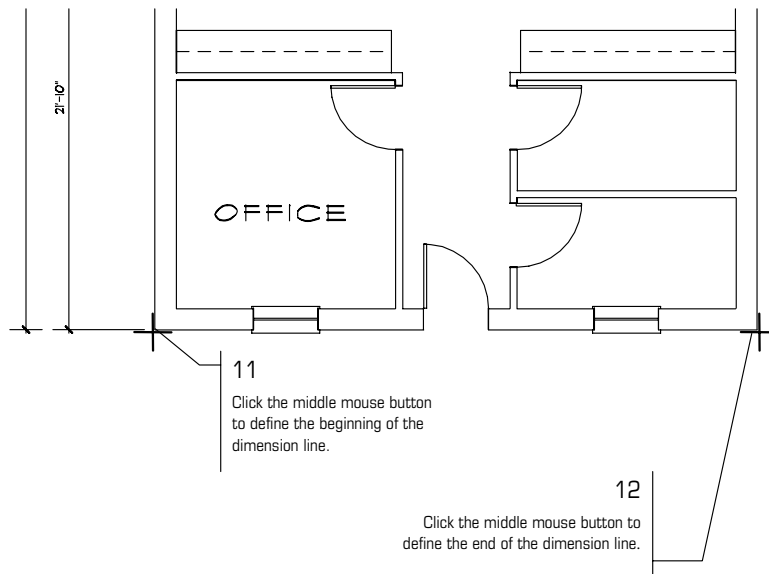


Figure 4.12: Selecting the points of your automatic dimension line

13. Move the cursor below the bottom wall and click. The dimension string is drawn automatically.
14. Right-click to return to the Linear menu.
15. Click on Overall to place an overall dimension. The front wall is dimensioned. (See Figure 4.13)
16. Right-click three times to return to the Edit menu.

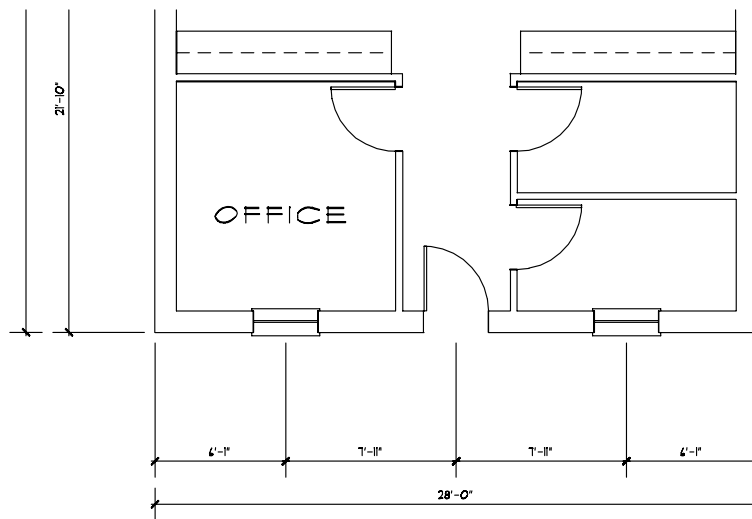


Figure 4.13: The front wall is dimensioned

Stretching Dimensions

➔ To stretch dimensions:

1. Click on Stretch in the Edit menu. You are prompted to “Select the first point of distance to stretch”.

2. Click anywhere on the drawing window to locate the first point of the distance to stretch. You are prompted to "Select the second point of distance to stretch".
3. Press **[Spacebar]**. You are prompted to "Enter relative distance".
4. Type **1.6** and press **[Enter]**. You are prompted to "Enter relative angle".
5. Type **270** and press **[Enter]**. The distance and angle to stretch appear in the Message Area.
6. Click on Area as the selection method. You are prompted to "Select first corner of box to <STRETCH>".
7. Make sure LyrSrch is on.
8. Click outside the upper-left corner of the plan.
9. Diagonally move the mouse until the rubberband box encloses the rear wall and part of the left and right walls. (See Figure 4.14)
10. Click to close the rubberband box. The walls are shortened by 1'-6" and the dimensions are automatically adjusted.

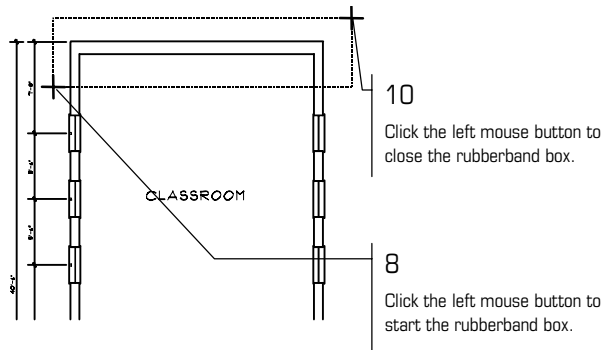


Figure 4.14: Stretching the rear wall

11. Click on Invert to stretch the same distance in the opposite direction. The new angle is displayed in the Message Area. You are prompted to "Select first corner of box to <STRETCH>". You can use Invert for most editing commands. It uses the inverse of the defined angle.
12. Repeat steps 8 - 10 to return the wall to its previous shape.

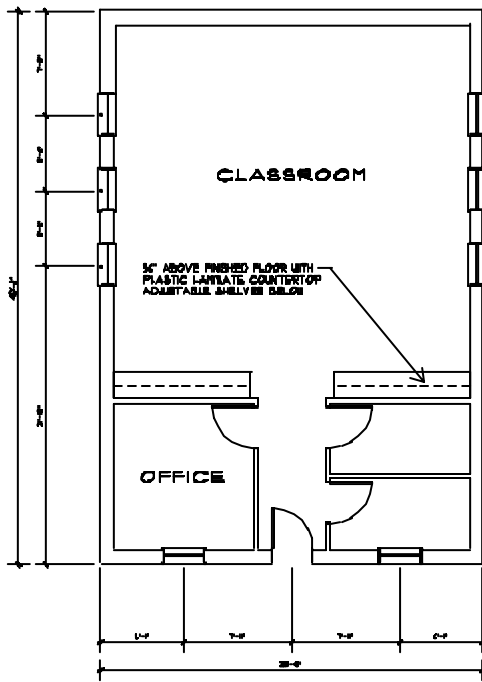


Figure 4.15: The plan is dimensioned

13. Right-click to return to the Edit menu.
14. Press **Ctrl** + **S** to save the drawing.

You have dimensioned the schoolhouse plan. Floor plans for construction of a real building would, of course, require more dimension lines as well as lines from many different angles. You can use DataCAD to create them all.

Hatching the Schoolhouse Drawing

Hatching is adding lines to your drawing that are in patterns, such as square boxes that represent floor tile, for example. As you will soon find out, there are many hatch patterns: to choose from.

➔ To set up to draw a tile hatch pattern:

1. Press **Tab** until the Hatch layer is the active layer in the Status Area.
2. Click on Hatch in the Utility menu.

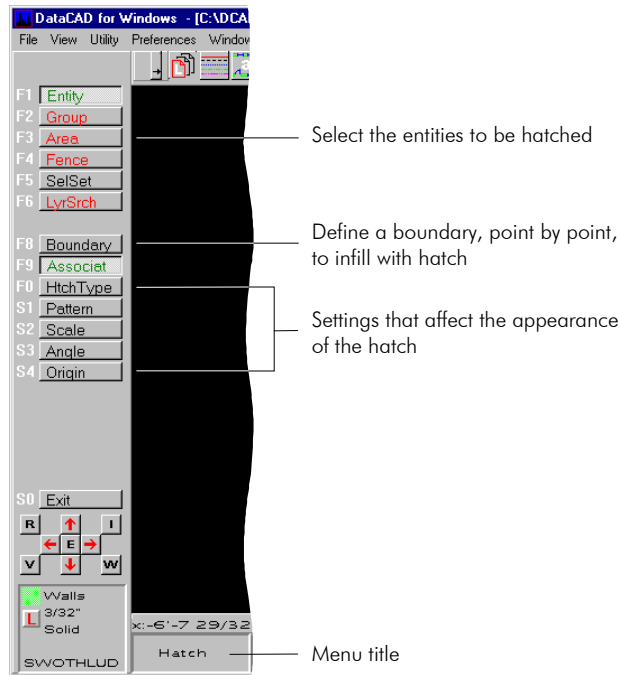


Figure 4.16: The Hatch menu

3. Click on **Associat** to toggle associative hatching on. This creates the entire hatch as a single entity. See “Drawing Associative Hatch” in the “Hatch” chapter for details.
4. Click on **Pattern** to select a hatch pattern. You are prompted to “Enter hatch pattern name”. A list of hatch patterns is displayed in the Menu Window, and a preview window is displayed in the Drawing Area.
5. Move the cursor over the buttons in the Menu Window. Notice that examples of each pattern are displayed in the preview window as your cursor passes over the pattern name.
6. Click on **ScrlFwr** three times to scroll to the end of the list.
7. Click on **12x12til** to select a 12”x12” tile pattern.
8. Click on **Scale** to set the scale of the pattern. You are prompted to “Enter hatch scale (1 to 9999)”.
9. Type **1** and press **[Enter]**.
10. Click on **Angle** to set the angle of the pattern. You are prompted to “Enter hatch angle”.
11. Type **0** and press **[Enter]**.

➔ To draw a tile hatch pattern:

1. Click on **W** from the Navigation Pad to go to the WindowIn menu. You are prompted to “Select first corner of the Zoom window”.
2. Click below and to the left of the exterior wall. You are prompted to “Select second corner of the Zoom window”.

3. Diagonally move the mouse until the rubberband box encloses the lower half of the plan and click. The plan now is displayed larger.
4. Right-click to return to the Hatch menu.
5. Click on Origin to start the hatch pattern at a defined point. You are prompted to "Select hatch pattern origin".
6. Move the cursor to the lower-left corner of the hall and click the middle mouse button.
7. Click on Boundary to define a boundary for the hatch. You are prompted to "Select first point on boundary to hatch to".
8. You will use object snapping to select points for the hatch boundary. To select the first point of the boundary, move the cursor to the lower-left corner in the hall and click the middle mouse button. You are prompted to "Select next point on boundary to hatch to". For more information on object snapping, see "Object Snapping" in the "Drawing Tools" chapter.

Shortcut: Press **[N]** to object snap to a point.

9. Move the cursor upward, and click the middle mouse button to object snap to the corner marked A with. (See Figure 4.17)
10. Move the cursor close to the corner marked B and click the middle mouse button to object snap to that point.
11. Move the cursor close to the corner marked C and object snap to that point.
12. Right-click to close the hatch boundary.

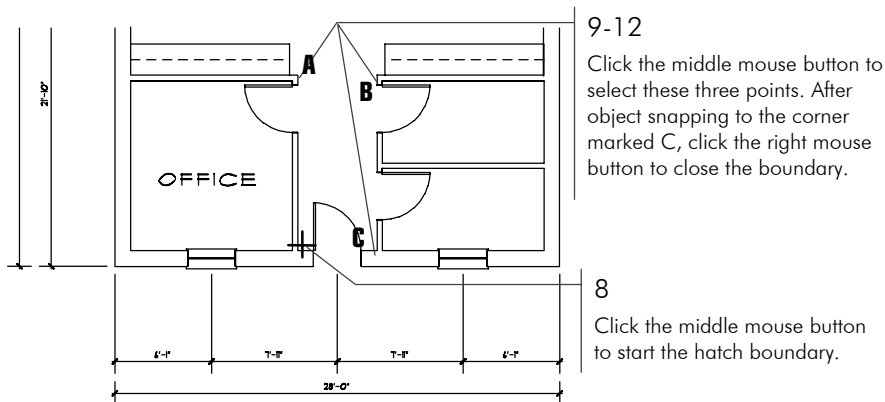


Figure 4.17: Selecting a hatch boundary

13. Click on Begin to draw the hatch pattern. The pattern will fill the boundary you defined in the previous steps.
14. Click on R to recalculate the extents of the drawing.
15. Right-click to return to the Utility menu.
16. Press **[Ctrl] + [S]** to save the drawing.

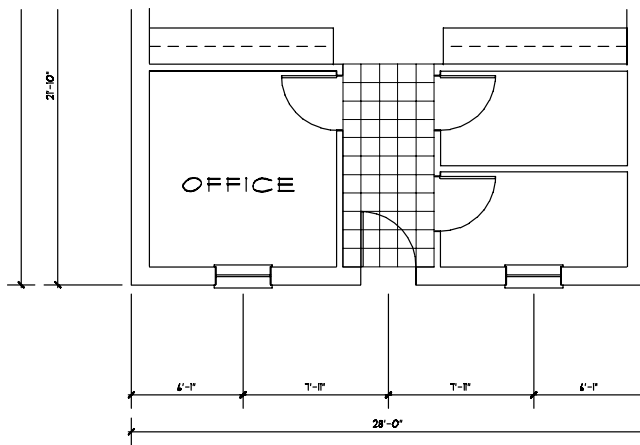


Figure 4.18: The tile hatch is drawn

You have now created a hatch pattern and saved your updated drawing file.

Adding Symbols to the Schoolhouse

Symbols are one of the most powerful features of DataCAD. You can build symbol libraries from the drawings you create and reuse them in other drawings. The symbols can be sections, details, notes, furniture, fixtures, or any part of a drawing that will be drawn repetitively.

You usually store symbols in templates, and each template usually contains symbols that are in some way related to one another. For example, if you have common bathroom configurations, each saved as a symbol, they can all be located in the same template.

Opening a Template Folder

➔ To open a template folder:

1. Click on W in the Navigation Pad to open the WindowIn menu. You are prompted to "Select first corner of the Zoom window".
2. Click at the lower-left corner of the exterior wall. You are prompted to "Select second corner of the Zoom window".
3. Move the mouse diagonally until the rubberband box encloses the lower half of the plan and click. The plan is now displayed larger.
4. Right-click to return to the Utility menu.
5. Click on Template in the Utility menu. A dialog box is displayed, listing the template folders in your DATACAD\TPL folder.
6. Click on the Plumb folder to list the plumbing fixture templates.
7. Click on BP2COM1.TPL. The template is displayed in a grid on the right side of the screen. Each box contains a symbol.
8. Move the cursor over the template window; as you move the cursor over a symbol, a description of the symbol is displayed in the Message Area.

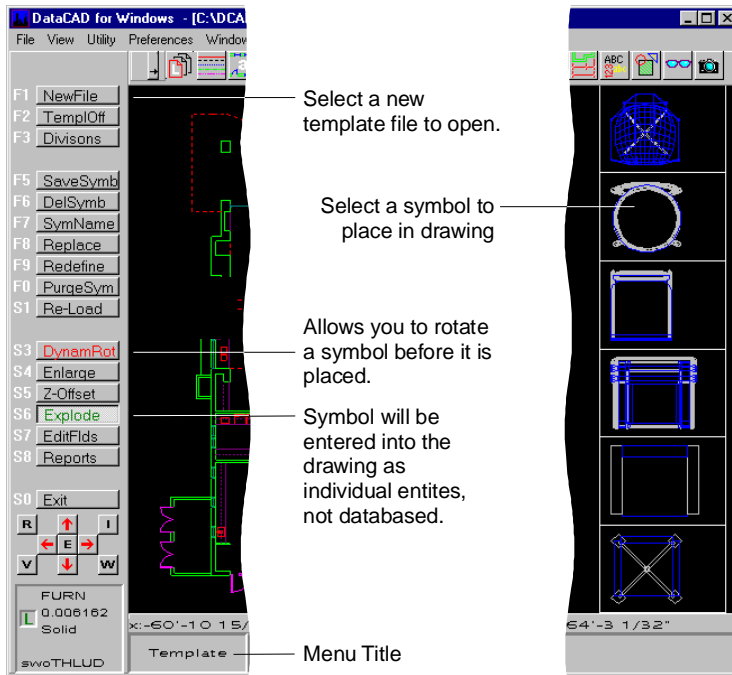


Figure 4.19: The Template menu and an open template

Selecting and Placing Symbols

➔ To select and place symbols:

1. Press **[Tab]** until Plumb is the active layer in the Status Area.
2. Click on DynamRot to toggle dynamic rotation on. This allows you to preview a symbol's position before you place it in the drawing.
3. Move the cursor over the toilet with grab bars (HC Toilet in the left column of the template) and click to select it.
4. Move the cursor into the Drawing Area. A boxed outline representing the toilet and grab bars is now attached to your cursor. The insertion point is at the upper-left corner of the symbol, the point the cursor is connected to. Remember, you can use the arrows on the keyboard to pan around your drawing.
5. Position the cursor near the upper-right corner of the room that is in the lower-right corner of the plan, and click the middle mouse button. This is the insertion point and the center of rotation for the symbol. If you move your cursor slightly up or down, you will see the boxed outline rotate around the insertion point.
6. Position the cursor near the lower-right corner of the bathroom and click the middle mouse button. The toilet with grab bars is drawn. If you place the toilet incorrectly, click on Undo in the Edit pull-down menu to delete it and then place the toilet again.

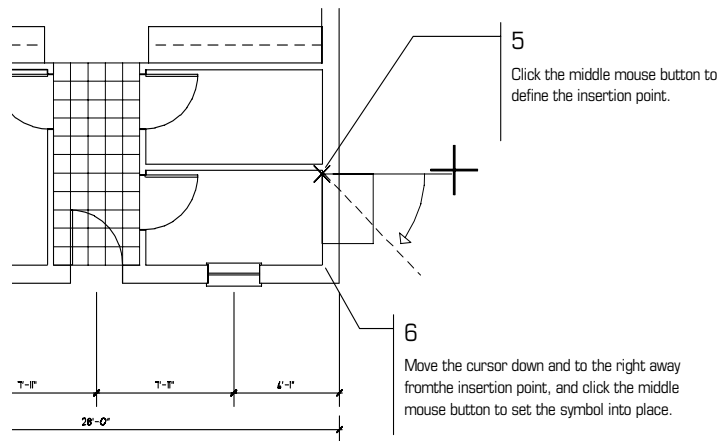


Figure 4.20: Placing the toilet on your plan

7. Right-click to cancel the toilet symbol.

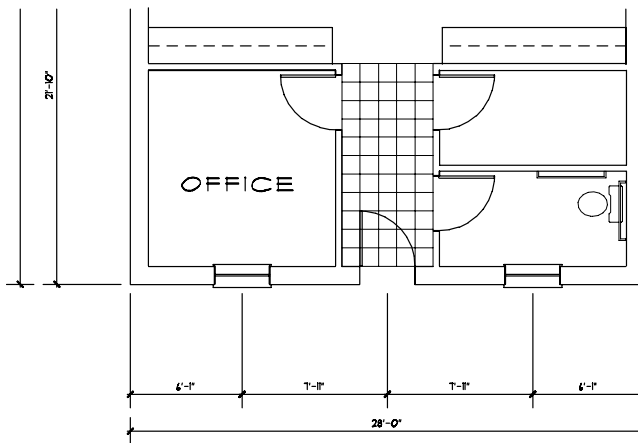


Figure 4.21: The toilet and grab bars are placed on the drawing

8. Move the cursor over the WALL-HUNG SINK (top left symbol in the template) and click on it to select it.
9. Move the cursor into the Drawing Area. A boxed outline representing the sink is now attached to your cursor. The insertion point for this symbol is in the back center of the sink.
10. Position the cursor at the center point, between the window and hallway wall, as shown in Figure 4.22. Click the middle mouse button.

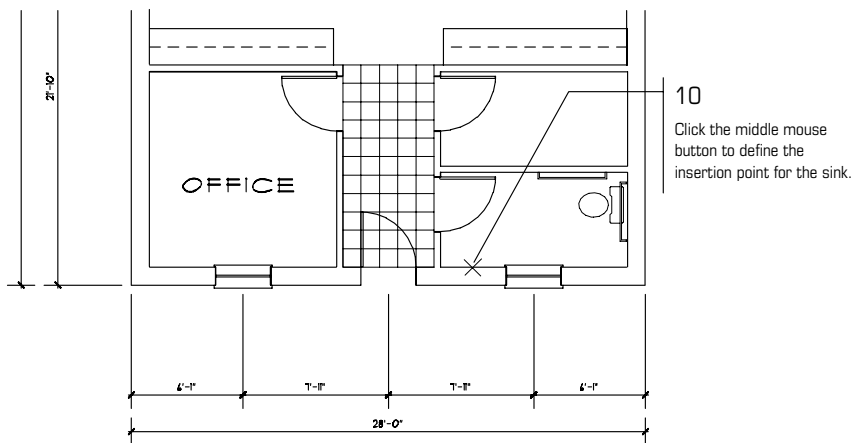


Figure 4.22: Placing the sink in your drawing

11. Rotate the symbol by dragging the cursor away from the insertion point and clicking when it is in place. The sink is drawn. If you place the sink incorrectly, click on Undo in the Edit pull-down menu to delete it and then place the sink again.

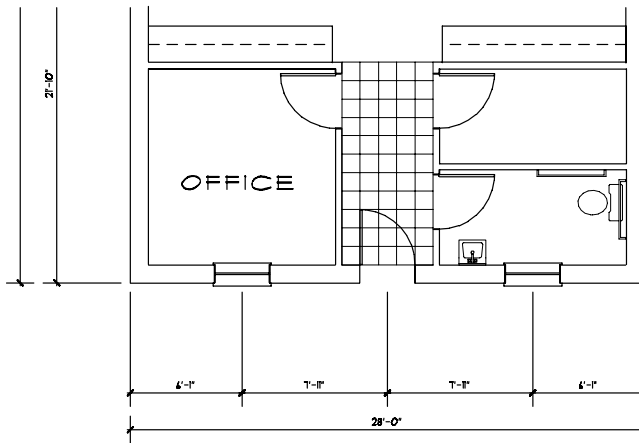


Figure 4.23: The sink is drawn

12. Right-click to cancel the sink symbol.
13. Press **Ctrl** + **S** to save the drawing.

Selecting Another Template File

➔ To select another template file:

1. Click on NewFile in the Template menu to switch to a new template file.
2. Click on the KITCHEN folder to view a list of Kitchen templates.
3. Click on KT2EQIP1 from the dialog box. The grid is updated with the new template.
4. Select the bar sink (upper left in the template) and place it as shown in the figure below. Refer to the previous section if you need help placing this sink.

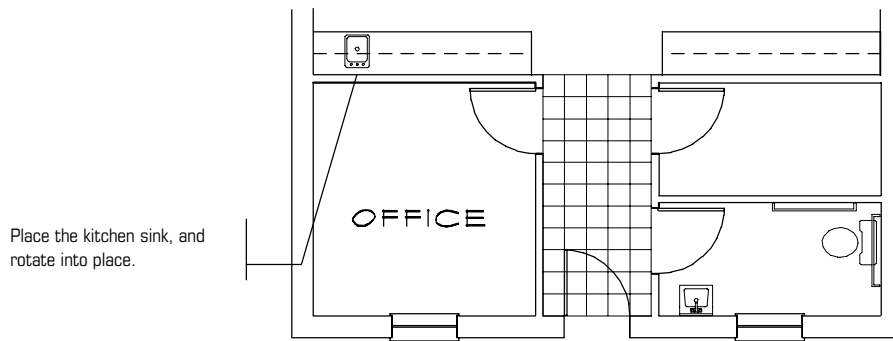


Figure 4.24: The kitchen sink is drawn

5. Click on TempLOff to close the Template window and return to the Utility menu.
6. Right-click to go to the Edit menu.

Copying Symbols

➔ To copy symbols:

1. Click on Copy in the Edit menu in the Menu Window. You are prompted to “Select first point of the distance to copy”.
2. Move the cursor to the upper-right corner of the bathroom you just placed fixtures in, and click the middle mouse button to define the first point to copy from. You are prompted to “Select second point of distance to copy”.
3. Move the cursor to the upper-right corner of the room directly above the bathroom and click the middle mouse button. The distance and angle you just defined are displayed in the Message Area.

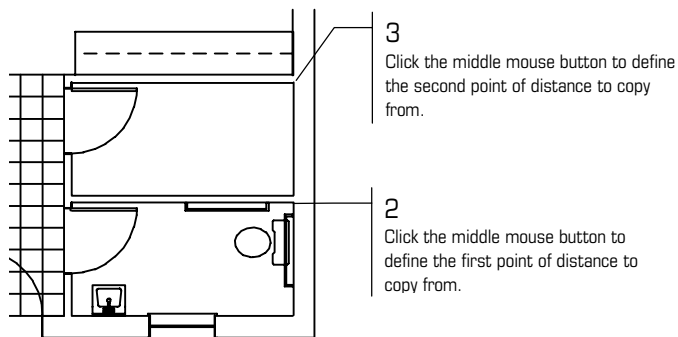


Figure 4.25: Defining a distance to copy

4. Click on Area if it's not already active. You are prompted to “Select first corner of area to <COPY>”.
5. Click on LyrSrch to toggle layer searching off and search only the active layer for items to copy.
6. Click at the lower left of the bathroom sink. You are prompted to “Select second corner of area to <COPY>”.

7. Diagonally move the mouse to the upper right of the toilet and grab bars, until the area box encloses all bathroom fixtures and their insertion points.
8. Click to close the box. The sink, toilet, and grab bars are copied into the other bathroom.

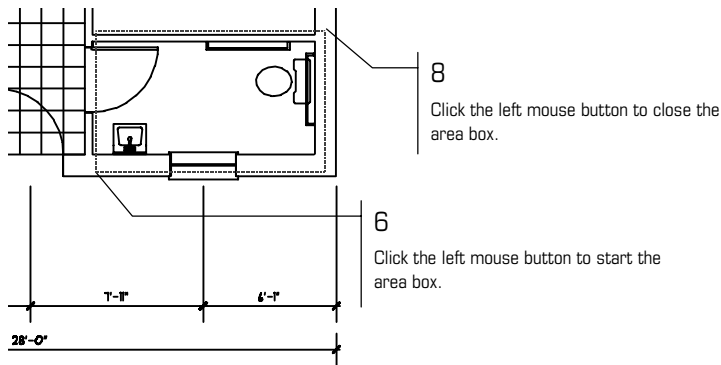


Figure 4.26: Selecting the bathroom fixtures to copy

9. Click on R to recalculate the extents of the drawing.

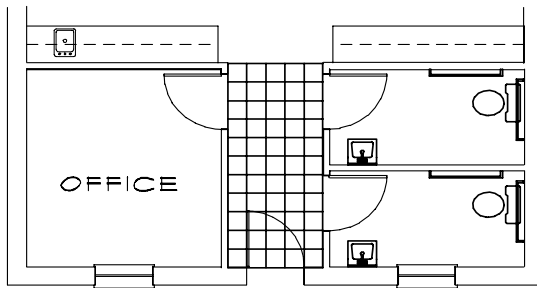


Figure 4.27: The bathroom fixtures are copied to the other bathroom

10. Right-click to return to the Edit menu.
11. Press **Ctrl** + **S** to save the drawing.

Drawing a Title Block and Border

To complete the schoolhouse plan, you will create a simple border that will plot with the floor plan. The paper size will be 8 1/2" x 11".

➔ To draw a title block and border:

1. Press **Tab** until Border is the active layer in the Status Area.
2. Press **=** to toggle the Walls option off. Note the *w* in SwOTHLUD in the Status Area is now lowercase.
3. Click on Polygons in the Edit menu in the Menu Window.
4. Click on RectAngl to draw a rectangle.
5. Move the cursor to the lower-left corner of the drawing where you want the border to start and click. If necessary, press **PageUp** once to zoom out and give yourself more space to work around your drawing.

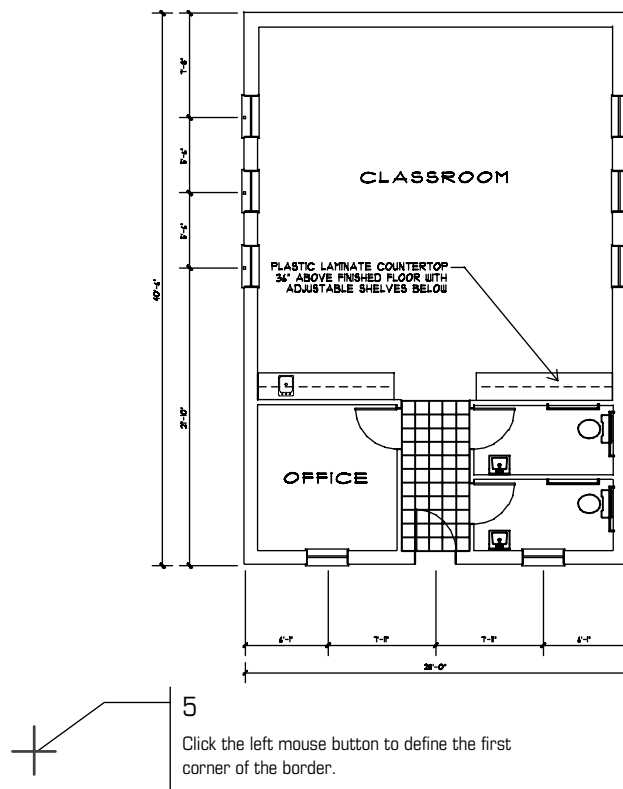


Figure 4.28: Creating a border

6. Press **[Insert]** until the prompt in the Message Area reads “Current input mode = relative Cartesian (x, y)”.
7. Press **[Spacebar]**. You are prompted to “Enter relative X-distance”.
8. Type **80** and press **[Enter]**. You are prompted to “Enter relative Y-distance”.
9. Type **60** and press **[Enter]**. The border is drawn.
10. Right-click to return to the Edit menu.
11. Click on **E** to view the extents of the drawing.

Moving the Border

➔ To move the border:

1. Click on **Move** in the Edit Menu.
2. Click on **Drag**.
3. Click on **Group**. You are prompted to “Select group to <MOVE>”.
4. Move the cursor to the center of the rectangle you just created and click the middle mouse button. The rectangle is displayed in dashed lines to indicate its selection. You are prompted to “Select the point to drag from”.
5. With the cursor still at the center of the rectangle, click the middle mouse button. Notice that you can now move the border by moving the mouse.

6. Press **[O]** to turn off ortho mode. This allows free movement when you place the border. Note the “o” in SwoTHLUD in the Status Area is now lowercase.
7. Using **[PageUp]** or **[PageDown]** for a better view if you need to, move the border over the floor plan so the plan is displayed centered within it.
8. Click to set the border into place.
9. Right-click twice to return to the Edit menu.
10. Click on R to recalculate the extents of the drawing.

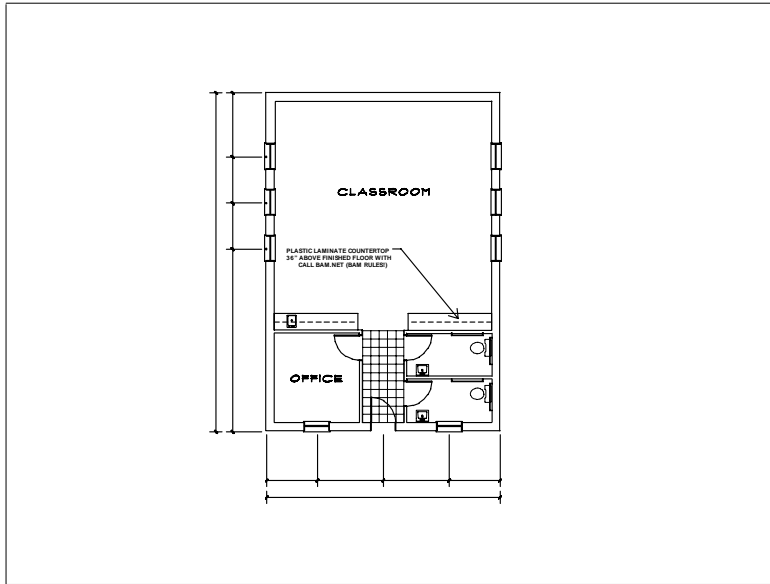


Figure 4.29: The border is placed

Drawing a Title Block

➔ To draw a title block:

1. Click on Copy in the Edit menu. You are prompted to “Select first point of the distance to copy”.
2. Move the cursor to the lower right corner of the border and click the middle mouse button. You are prompted to “Select second point of the distance to copy”.
3. Press **[Spacebar]**. You are prompted to “Enter relative X-distance”.
4. Type **-5** and press **[Enter]**. You are prompted to “Enter relative Y-distance”.
5. Type **0** and press **[Enter]**. The distance and angle to copy are displayed in the Message Area.
6. Click on Entity in the Menu Window. You are prompted to “Select entity to <COPY>”.
7. Select the right vertical line of the border by clicking on it. The line is copied.

8. Right-click to return to the Edit menu.
9. Press ['] to set a new reference point.
10. Move the cursor to the lower right outside corner of the border and click the middle mouse button to set a new reference point.

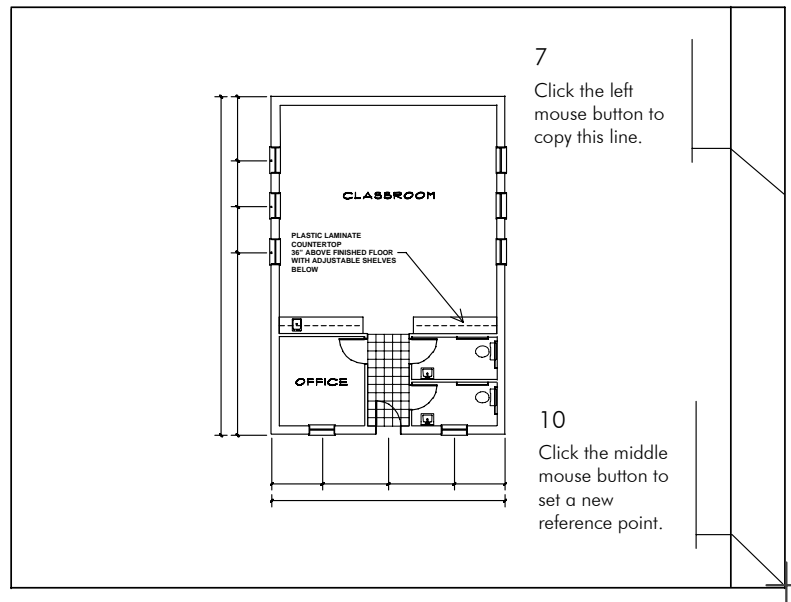


Figure 4.30: Completing the title block

11. Press [Spacebar]. You are prompted to "Enter relative X-distance".
12. Type 0 and press [Enter]. You are prompted to "Enter relative Y-distance".
13. Type 5 and press [Enter]. You connect to a point on the right line of the border.
14. Press [Spacebar]. You are prompted to "Enter relative X-distance".
15. Type -5 and press [Enter]. You are prompted to "Enter relative Y-distance".
16. Type 0 and press [Enter]. A line is drawn.
17. Right-click to disconnect from the line and return to the Edit menu.
18. Click on Move in the Edit menu.
19. Click on Drag.
20. Click on AndCopy. You are prompted to "Select entity to <MOVE>".
21. Select the line you just drew by clicking on it. The line becomes dashed. You are prompted to "Select the point to drag from".
22. Select an end point of the dashed line with the middle mouse button.
23. Press [O] to toggle ortho mode on.
24. Move the cursor upwards and click to place the line in a few different locations to section off the title block, as in Figure 4.31.
25. Right-click three times to return to the Edit menu. The title block is drawn.

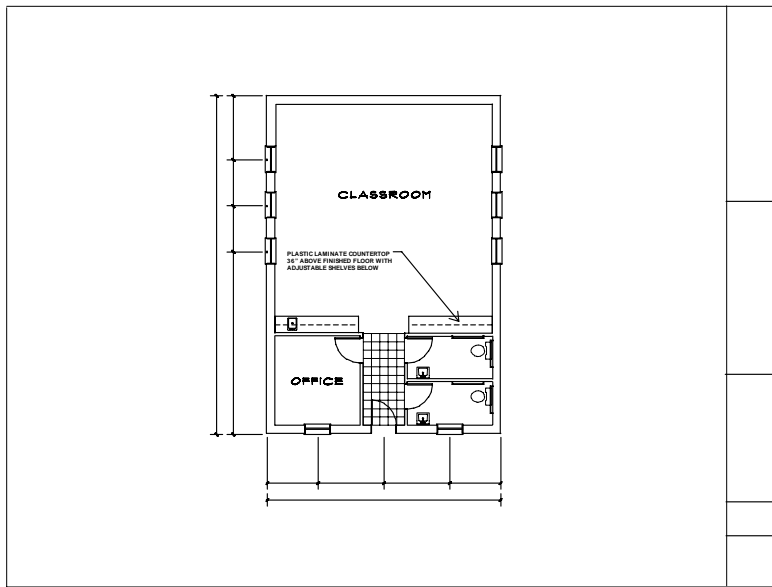


Figure 4.31: The drawing with title block

Tutorial: Printing a Drawing

DataCAD has many options available for plotting or printing your drawing. This chapter will focus on printing your plan on an 8 1/2" x 11" sheet of paper.

In this chapter:

- ⊕ Printing the schoolhouse drawing
- ⊕ Setting printing options

Printing the Schoolhouse Drawing

If you are not in DataCAD, start DataCAD now. Choose the drawing you worked on in the previous tutorials. Before plotting the schoolhouse floor plan, make sure:

- a printer or plotter is either connected to your computer or available via a network connection
- the printer or plotter is turned on and is online or ready to receive data
- you have the corresponding Windows driver for your printer installed on your computer (see your printer's manual for more information about installing a printer driver)

➔ To plot the schoolhouse:

1. Choose Print from the File pull-down menu in the Menu Bar, or choose Plotter from the Utility menu in the Menu Window. The following message is displayed: *A printer has not yet been selected for this drawing. Please click OK to open the Print Setup dialog box and assign a printer to this drawing file.* Since this is the first time you've printed this drawing, DataCAD requires you to choose a printer, paper size, and orientation before you assign pens, complete a print layout, and plot your drawing. To accept the default settings, click OK and skip to step 5; otherwise, continue with step 2 below.
2. In the Printer/Plotter Settings dialog box, choose a printer from the Printer Name drop-down box.
3. Choose a paper size. The sizes available are read from your printer driver. If a size you want is not displayed in the drop-down box, your printer may not support it. Choose the closest appropriate size or use the User-Defined option if available, or use a different printer.
4. Click OK to close the Settings dialog box. The Plotter menu is displayed in the Menu Window. You can exit DataCAD and open the printer driver to see what paper sizes are supported. Many times, the default settings of the driver do not include all supported sizes. You must check them to activate them.

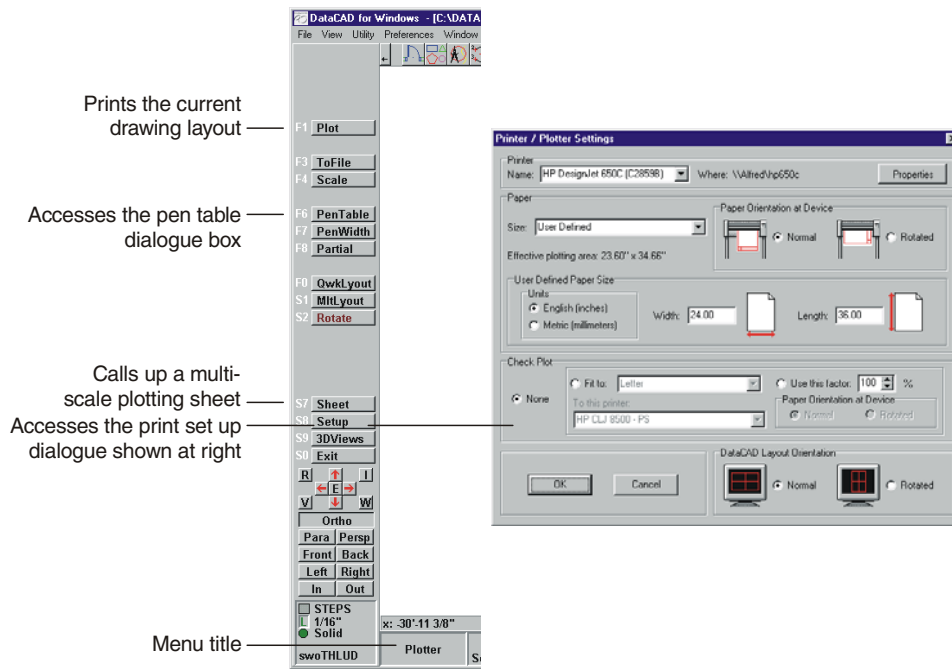


Figure 5.1: The Plotter menu and Printer/Plotter Settings dialog box

5. Choose Scale to set the plotting scale. The current plotting scale is displayed in the Message Window.
6. Choose 1/8" if it's not already the current plotting scale.
7. Choose QwkLyout from the Plotter menu. A dashed box appears representing the 8 1/2" x 11" sheet. If you move the cursor over the Drawing Window, you will find a movable copy of this box. You are prompted: *Locate sheet of paper using cursor, "ENTER" to end.*
8. With the box centered over the drawing, click the middle mouse button. This object snaps to the center of the title block (the centered point of the rectangle) and centers the title block. You can also move the box anywhere you want and click with the left mouse button.
9. Choose Plot to send the drawing directly to the plotter or printer. The schoolhouse floor plan is plotted.

Printing Options

There are many printing options that help you get the plotted output you want. In the next set of exercises, you will use a few of the images that were created in this book to build a plot that contains multiple drawings. Each drawing added to the plot can be set to a different scale.

Assigning Pens

Use Pen Assignments in the Pen Table dialog box to set a hierarchy of line weights, which help add line definition to your drawings. For example, if your walls were a heavier line weight than your doors and windows, the door and window openings would appear clearer. This difference in line weight helps define the content of your drawing.

➔ To set the pens for printing the schoolhouse drawing:

1. Choose Plotter.
2. Choose PenTable to open the Pen Table dialog box. In the Pen Assignments section, you will assign pen numbers to match the colors used in the drawing. You'll notice that the first color in the Pen Assignments list box is highlighted.

The Pen Table dialog box also includes a Pen Settings section that allows you to choose the pen width, color, and color density for each pen you assigned in the Pen Assignments portion of the dialog box. See "Using Pen Tables" in the "Printing Your Drawing" chapter for detailed information on using the Pen Table.
3. Select White in the Pen Assignments box, if it's not already highlighted. Notice that just below this list box is a small box displaying the highlighted color, followed by an equal sign and a number in a small input box. This number is the pen that will be used to plot all white lines in your drawing.
4. Type 1 in the input box. The color white is now set for pen number 1.
5. Select Red in the Pen Assignments box or press to highlight Red.
6. Type 3 in the input box. The color red is now set for pen number 3.
7. Set the remainder of the colors: for Green use pen 1, for Cyan use 3, for Magenta use 3, for Brown use 4, for Lt. Grey use 4, for Lt. Red use 2, for Lt. Green use 3, and for Yellow use 2.
8. Click OK to save the pen assignments and close the Pen Table dialog box.
9. Right-click to return to the Utility menu.

Using Multi-scale Plotting

The GotoViews menu will help you automate the viewing process which, in turn, is used with multi-scale plotting to place differently scaled details on the same sheet. You can choose GotoView in the Utility menu, in the 3DViews menu, or from the View pull-down menu to assign an unlimited number of views. Each view can have its own set of assigned layers. For example, in this schoolhouse tutorial, there are 14 different layers. If you want to view only the Walls, Doors, and Windows layers, you can create a view using GotoView with only these three layers visible. When you choose that view, all layers except Walls, Doors and Windows will automatically be toggled off. Choose another predefined view to turn on a different set of layers. Using GotoView, you can quickly change your view and continue working.

The GotoView option is a helpful tool to use when you have a drawing that contains floor plans, elevations, sections, and 3D models. In the following steps, you will establish two views, which help complete the remainder of this schoolhouse project.

➔ To create views:

1. Choose Orthographic from the View pull-down menu for plan view.
2. Recalculate the extents of the drawing.

3. Choose GotoView from the View pull-down menu.
4. Choose Add View. You are prompted: *Enter name of new view?*
5. Name the view **3DPlan**.
6. Toggle the Doors, Windows, Plumb, Dim, Hatch, and Notes layers on.
7. Toggle 3DWalls, 3DWindow, 3DDoors, 3DRoof, and 3DMisc layers off.
8. Recalculate the extents of the drawing.
9. Choose GotoView from the View pull-down menu.
10. Choose Add View. You are prompted: *Enter name of new view?*
11. Name the view **2DPlan**.

When you need to switch between 3D layers and 2D layers, use the GotoView menu. In the next section, you will use the views you just set.

➔ To use multi-scale plotting:

1. Press **[L]** to go to the Layers menu.
2. Make Border the only active layer using ActvOnly from the Layer menu.
3. Recalculate the extents of the drawing by clicking on R on the Navigation Pad or by choosing WindowIn Recalc from the View pull-down menu in the Menu Bar.
4. Right-click to return to the Utility menu.
5. Choose Plotter.
6. Choose MltLayout.
7. Choose Layout. The drawing appears as a box attached to the cursor; the dashed box represents the 8 1/2" x 11" sheet. You are prompted: *Locate detail on sheet of paper, "ENTER" to end.*
8. Center the drawing in the dashed box and click to set the title block in place. You are prompted: *Enter name of new detail: detail 1.*
9. Type **Titleblk** and press **[Enter]**.
10. Right-click twice to return to the Utility menu.
11. Choose GotoView.
12. Choose 2DPlan. Right-click once.
13. Recalculate the extents of the drawing by clicking on R on the Navigation Pad or by choosing WindowIn Recalc from the View pull-down menu in the Menu Bar.
14. Choose Plotter from the Utility menu.
15. Choose MltLayout.
16. Choose Layout. The drawing appears as a box attached to the cursor; the dashed box represents the 8 1/2" x 11" sheet. You are prompted: *Locate detail on sheet of paper, "ENTER" to end.*

17. Center the drawing over the dashed box and click the left mouse button to position the plan. You are prompted: *Enter name of new detail: detail 2.*
18. Type **Plan** and press **[Enter]**.
19. Recalculate the extents of the drawing by clicking on R on the Navigation Pad or by choosing WindowIn Recalc from the View pull-down menu in the Menu Bar.
20. Make FrntElev the only active layer. The elevation that was created earlier is now on the screen.
21. Choose Plotter from the Utility menu.
22. Choose Scale to change the scale factor.
23. Choose 1/16".
24. Choose MltLayout.
25. Choose Layout. The drawing appears as a box attached to the cursor; the dashed box represents the 8 1/2" x 11" sheet. You are prompted: *Locate detail on sheet of paper, "ENTER" to end.*
26. Move the drawing into the upper-right quarter of the dashed box. Click to set the elevation in place. You are prompted: *Enter name of new detail: detail 3.*
27. Type **FrntElev** and press **[Enter]**.
28. Recalculate the extents of the drawing by clicking on R on the Navigation Pad or by choosing WindowIn Recalc from the View pull-down menu in the Menu Bar.
29. Make Hide1 the only active layer.
30. Choose Plotter from the Utility menu.
31. Choose Scale to change the scale factor. You can press **[PgUp]** and **[PgDn]** to change the scale of any detail on-the-fly before you place it on the sheet.
32. Choose 1:20.
33. Choose MltLayout.
34. Choose Layout. The drawing appears as a box attached to the cursor and the dashed box that appears representing the 8 1/2" x 11" sheet. You are prompted: *Locate detail on sheet of paper, "ENTER" to end.*
35. Move the drawing into the lower-right quarter of the dashed box. Click to set the perspective in place.
36. Add a new detail named Perspect.
37. Return to the Plotter main menu.
38. Choose Plot to send the drawing directly to the plotter.

Tutorial: Modeling

This chapter guides you through some of the more commonly used macros to help you complete the schoolhouse. In order to keep this exercise simple, and to help you understand a little about 3D spatial relationships, you will work in orthographic (plan) view and then use the 3DViews menu to view your work.

In this chapter:

- ⊕ Adding 3D windows and doors
- ⊕ Adding a roof
- ⊕ Viewing the model in 3D

Modeling the Schoolhouse

You will create five new layers for the 3D items that you are going to add to the drawing. This enables your drawing file to contain both your plan drawings, used for construction documentation, and 3D drawings, used to present your designs.

➔ To add layers to your drawing:

1. If you are not in DataCAD, start DataCAD now and choose the drawing file that you worked on in the previous chapters.
2. Add five new layers to your drawing file.
3. Name the new layers: 3DWalls, 3DDoors, 3DWindow, 3DRoof, and 3DMisc.
4. Go to the Layers menu.
5. Turn off Doors, Windows, Plumb, Dim, Hatch, Notes, Misc, and Border.
6. Press **[Esc]** on the keyboard to refresh the drawing display.
7. Window in on the lower half of the plan.

Adding Windows

You will work with the walls, doors, and windows that you have already placed in your drawing. Do not erase them; they are an essential part of your drawing file.

➔ To add windows:

1. Make the 3DWindow layer the active layer.
2. Choose Toolbox from the Edit menu.
3. Choose AEC_MODL. You are prompted: *Select building component to model.*
4. Choose Windows to go to the WindElev menu.
5. Choose In Plan to draw the windows in plan view.
6. Choose UnitType. You are prompted: *Select type of window unit.*
7. Choose DblHung to draw double-hung windows.
8. Return to the WindPlan menu.
9. Choose Glass to go to the Glass menu.
10. Choose PaneHorz to define the number of glass panes (divisions) across the window. You are prompted: *Number of panes in the horizontal direction.*
11. Choose 3.
12. Choose PaneVert to define the number of glass panes top to bottom in the window. You are prompted: *Number of panes in the vertical direction.*
13. Choose 2.

14. Return to the WindPlan menu. You are prompted: *Enter a point on first inside corner of window.*
15. Click the middle mouse button on an inside corner of the front window opening in the office. You are prompted: *Enter a point on second inside corner of window.*
16. Click the middle mouse button the other inside corner of the window opening. You are prompted: *Enter a point on outside surface of wall.*
17. Click the middle mouse button on the outside of the wall. The window is created.

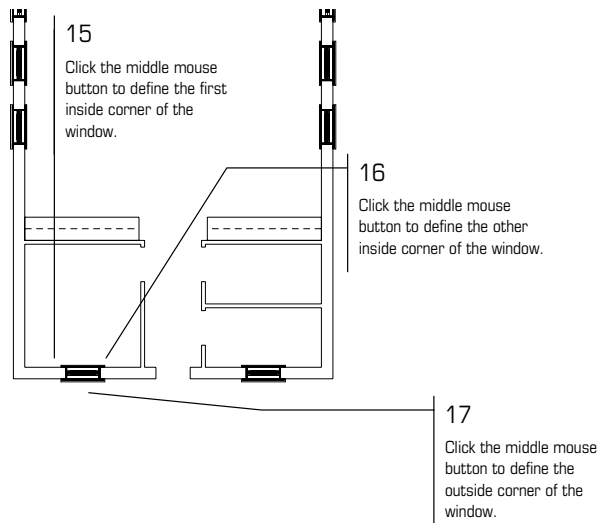


Figure 6.1: Placing windows

18. Repeat steps 15 - 17 to create the remaining windows. Remember that you can use the arrow keys on the keyboard to move around the drawing.

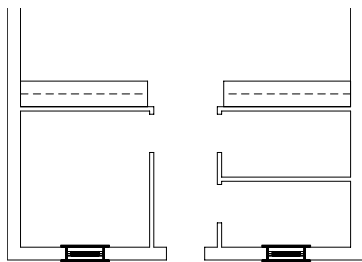


Figure 6.2: The 3D windows are drawn.

19. When you are finished placing windows, return to the AEC_MODL menu.

Adding 3D Doors

➔ To add doors:

1. Make 3DDoors the active layer.
2. Choose Doors from the AEC_MODL menu to go to the DoorPlan menu. You are prompted: *Enter a point on first inside corner of door.*

3. Move the cursor to an inside corner of the door opening in the office and click the middle mouse button. You are prompted: *Enter a point on second inside corner of door.*
4. Move the cursor to the other inside corner of the door and click the middle mouse button. You are prompted: *Enter a point on outside surface of wall.*
5. Move the cursor to the outside corner of the wall and click the middle mouse button. The first door is created.
6. Choose Door. You are prompted: *Select door data to change.*
7. Choose HingLft to change the hinge side of the front door to the left side.
8. Return to the DoorPlan menu.
9. Repeat steps 3 - 5 for the remaining doors.

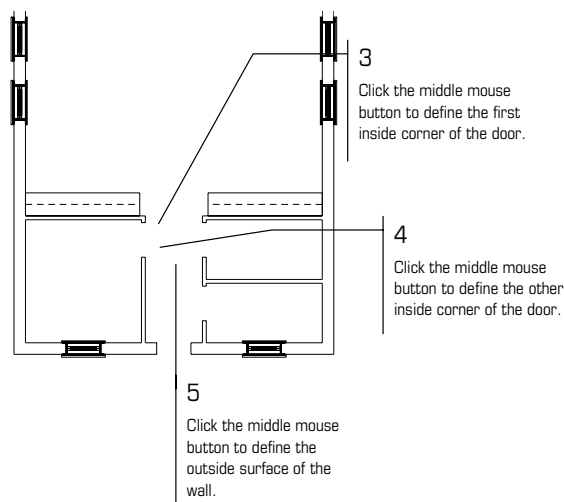


Figure 6.3: Placing the first door

10. Return to the Edit menu.
11. Recalculate the extents of the drawing.

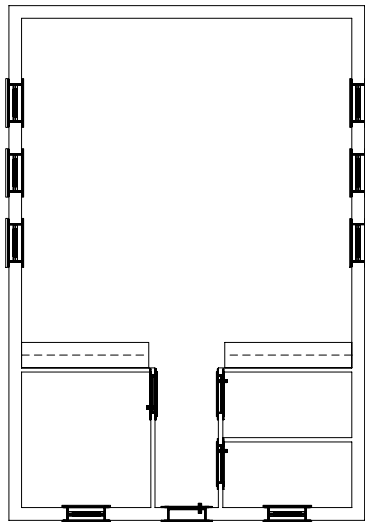


Figure 6.4: The doors are drawn.

There are many other options you can use when placing doors and windows.

Viewing the 3D Model

Choose Isometric from the View menu on the Menu Bar. You will learn more viewing techniques later in this chapter.

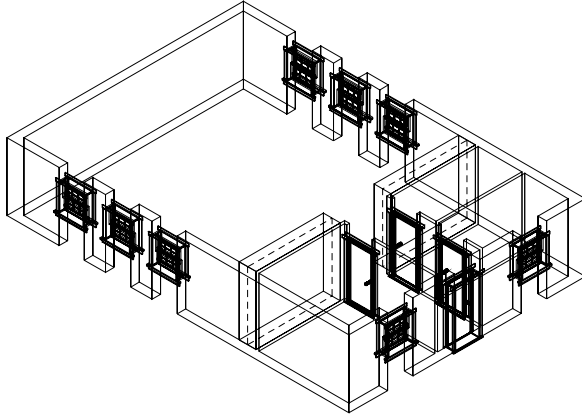


Figure 6.5: An isometric view of the schoolhouse walls, windows, and doors

Adding a 3D Roof

As mentioned throughout this book, layers allow you more control over your drawing. Later in this section of the Tutorial you will use these different layers to switch between viewing construction plans and presentation graphics.

➔ To add a roof:

1. Change to the 3DRoof layer.
2. Change the color of the 3DRoof layer to Magenta.
3. Choose Toolbox from the Edit menu.
4. Choose ROOFIT from the dialog box to go to the RoofIt menu.
5. Choose Gable to create a gable roof.
6. Choose Settings.
7. Set the wall thickness to 1 foot.
8. Return to the RoofIt menu. You are prompted: *Pick the 1st point of roof boundary.* It is important to select the points of the roof boundary in a clockwise direction to ensure that the roof will be placed correctly.
9. Select the lower-left corner of the exterior wall with the middle mouse button. You are prompted: *Enter second point of roof length.*
10. Select the upper-left corner of the exterior wall with the middle mouse button. You are prompted: *Enter second point of roof width.*
11. Select the upper-right corner of the exterior wall with the middle mouse button. The roof appears on top of the walls.
12. Return to the Edit menu.

13. Choose Isometric from the View menu on the Menu Bar to view the drawing in isometric.

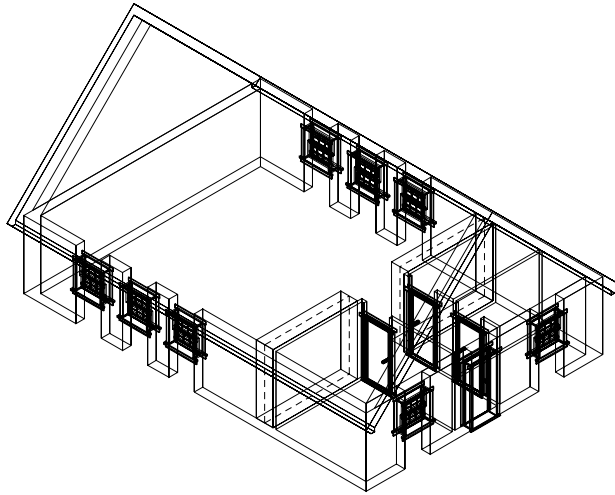


Figure 6.6: An isometric view of the schoolhouse with the new roof

Finishing the Schoolhouse Drawing

Notice that in the isometric view, portions of the wall above and below the windows are missing. You will fix them in the next exercise, but these wall segments are going to be put on a different layer than the existing Walls layer. Later in this chapter, you will see the reason for having two wall layers.

➔ To finish the drawing:

1. Choose Orthographic from the View menu on the Menu Bar to view the schoolhouse in plan view.
2. Choose Layers from the Utility menu in the Menu Window.
3. Make 3DWalls the active layer, with only walls turned on. All other layers should be off.
4. Change the 3DWalls layer color to yellow.
5. Toggle the Walls option on.
6. Toggle the 2-line wall option on.
7. Set a new wall width of 1 foot.
8. Toggle Outside on.
9. Toggle Cap off, if it is on.
10. Press **[Z]** on the keyboard. You are prompted: *Enter new Z-base.*
11. Type **0** and press **[Enter]**. You are prompted: *Enter new Z-height.*
12. Type **3.4** and press **[Enter]**. You have just established a new 3D setting for all new items you draw. All lines will now be drawn with a Z-base of 0 and a Z-height of 3'-4".
13. Window in on the lower half of the plan.

14. Return to the Edit menu. You are prompted: *Select first end point of new line/wall.*
15. Move the cursor to an outside corner of the window opening in the office and click the middle mouse button. You are prompted: *Select next end point of line/wall.*
16. Move the cursor to the other outside corner of the window and click the middle mouse button. You are prompted: *Select a point to define the Inside of the wall.*
17. Move the cursor anywhere on the inside of the plan and click. The cutout is filled.
18. Right-click to disconnect from the wall you just drew.
19. Repeat steps 15 - 18 for the remaining exterior window openings. Be sure to disconnect from each wall segment you draw, before drawing the next. Remember to use the arrow keys on the keyboard to move around the drawing.
20. Press **[Z]** on the keyboard. You are prompted: *Enter new Z-base.*
21. Type **6.8** and press **[Enter]**. You are prompted: *Enter new Z-height.*
22. Type **8** and press **[Enter]**. You have just established a new 3D setting for all new items you draw. All lines will now be drawn with a Z-base of 6.8 and a Z-height of 8'-0".

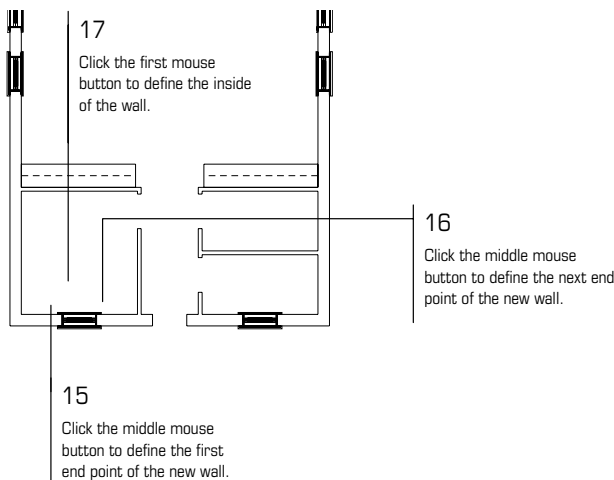


Figure 6.7: Placing headers and sills

23. Repeat steps 15 - 18 to fill in over top of the sills to create window headers. You are prompted: *Select first end point of new line/wall.*
24. Click the middle mouse button at the outside corner of the front door opening. You are prompted: *Select next end point of line/wall.*
25. Move the cursor to the other outside corner of the door and click the middle mouse button. You are prompted: *Select a point to define the Inside of the wall.*
26. Move the cursor anywhere on the interior of the plan and click. The door header is drawn.

27. Press ☐ on the keyboard. The Message Window reads Walls will NOT be drawn.
28. Press ☐ again. You are prompted: Enter new wall width.
29. Type .4 and press . You are now set to draw the headers above the interior doors.
30. Move the cursor to an outside corner of the interior door opening in the office.
31. Repeat steps 24 - 26 to complete this header and the remaining interior headers.
32. Choose Isometric from the View menu on the Menu Bar to view the isometric view.
33. Toggle the 3DDoors, 3DWindow, and 3DRoof layers on.
34. Return to the Utility menu.
35. Recalculate the extents of the drawing.

Viewing Your Drawing in 3D

Perhaps one of the most exciting features of DataCAD is that everything you draw has 3D parameters. The V button on the Navigation Pad opens the 3DViews menu.

➔ To change your view:

1. Choose V from the Navigation Pad or choose DCAD 3D/3DViews from the Edit menu to go to the 3DViews menu.
2. Choose Isometric. If your drawing is not centered choose E from the Navigation Pad. The lines that you drew earlier now appear in 3D.
3. Right-click twice to return to the Edit menu.
4. Choose Orthographic from the View pull-down menu in the Menu Bar to view your drawing in plan view.
5. Choose Isometric to view the isometric view.
6. Choose Orthographic to view your drawing again in plan view. Switching between 2D and 3D helps you better understand what you are drawing.
7. Press + on the keyboard to save the drawing.

Advanced 3D Viewing

Throughout this tutorial you moved around the drawing using a few of DataCAD's different viewing methods. One of the most exciting features of DataCAD is viewing your work in 3D. This section steps you through a few viewing exercises of 2D (orthographic) and 3D.

➔ To use advanced viewing techniques:

1. Choose GotoView from the Utility menu.

2. Choose 3DPlan.
3. Choose V from the Navigation Pad to go to the 3DViews menu.

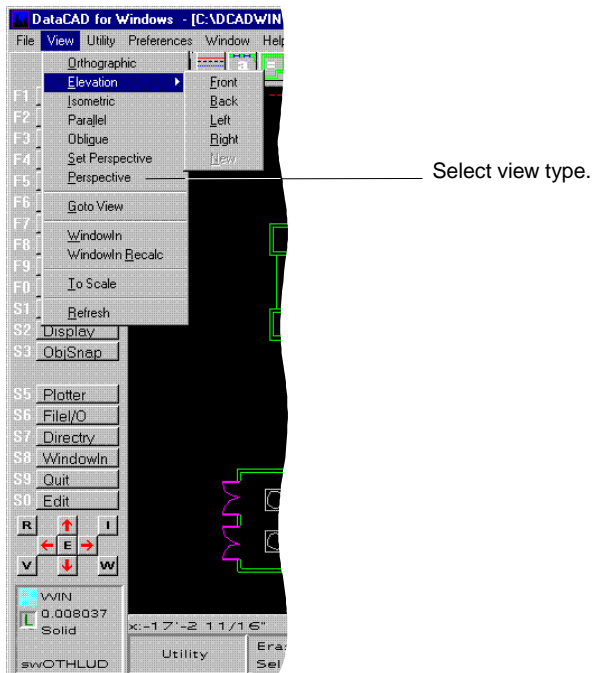


Figure 6.8: The View pull-down menu

Each time you start a new drawing, the Parallel and Prspect options in the 3DViews menu do not do anything. They become active after you set the viewing parameters for each button. When you selected Isometric when working with hatch earlier in the tutorial, you could choose Ortho and Parallel and view 3D views. If you chose Prspect, nothing happened.

Perspective Viewing

➔ To create a perspective view:

1. Press **[PageUp]** twice to zoom out.
2. Choose SetPersp from the 3DViews menu. You are prompted: *Select position of eye point for perspective view.*
3. Click at the lower left corner of the screen. You are prompted: *Select position of center of perspective view.*
4. Press **[O]** to turn ortho mode off and have more control over cursor movement.
5. Click at the upper-right corner of the screen. A perspective view of the schoolhouse appears.

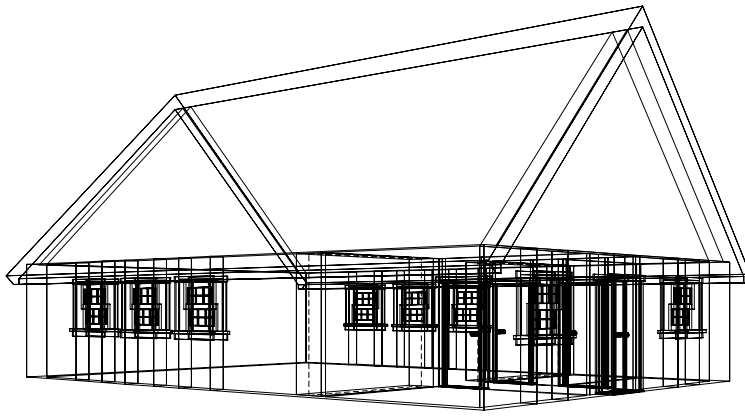


Figure 6.9: A perspective view of the completed schoolhouse drawing

6. Choose WalkThru from the 3DViews menu. This displays a menu used for moving around the 3D model. You can also use the Viewer Toolbar in the View pull-down menu to display a toolbar with options for walking through perspective views.
7. Choose WalkBack. You just stepped away from the model.
8. Experiment with a few of the other options in this menu.
9. Exit the 3D menu and choose Orthographic from the View pull-down menu on the Menu Bar to view the drawing in plan.
10. Choose Perspective from the View pull-down menu on the Menu Bar to view the perspective again.
11. Choose Isometric from the View pull-down menu on the Menu Bar to view the isometric.
12. Choose Orthographic on the Menu Bar to view the plan view again.

Once you establish the viewing options, all of these options display a different image of the drawing. This is an easy way to change the view of a drawing during any stage of a project.

Hidden Line Removal

As you become more experienced with DataCAD, you will be adding more complex 3D elements to the drawing. The more you add to a drawing, the harder it can be to understand a 3D view. The Hide option in the 3DEdit menu is used to hide the lines in your model that would not be visible if it were an actual building. This 3D view can be saved as a layer and edited.

➔ To hide lines in your model:

1. Choose DCAD 3D from the Edit menu to enter DataCAD's 3D menus: 3DEdit and 3DEntity.
2. Choose Perspective from the View pull-down menu on the Menu Bar to display the perspective of the schoolhouse.
3. Choose Hide.
4. Choose Begin. The hidden line removal generates a new image without most of the unwanted lines.

5. At the completion of the hidden line removal, choose NewLyr. You are prompted: *Enter name of new layer to be added.*
6. Type **Hide1** and press **[Enter]**. You are prompted: *Do you want the new layer ON or OFF?*
7. Choose Off. You will toggle this layer on later in the chapter. The hidden line removal is complete and you have added a new layer to the drawing.
8. Choose Orthographic to view the schoolhouse in plan view.
9. Go to Layers.
10. Choose ActvOnly.
11. Choose Hide1. You'll notice that there are still a few lines visible that should not be there, such as the vertical lines above and below windows and doors. You can erase these lines from this view without adversely affecting your drawing.
12. Choose Erase from the Edit menu.
13. Choose Entity as the selection method for erasing lines.
14. Using your mouse, select any lines on the perspective that should not be there.
15. Return to the Utility menu. If you choose Perspective from the View pull-down menu on the Menu Bar, you will see that the image is flat. This is because the hidden line removal created a flat line drawing from the perspective. This drawing has no connection to the original 3D model.

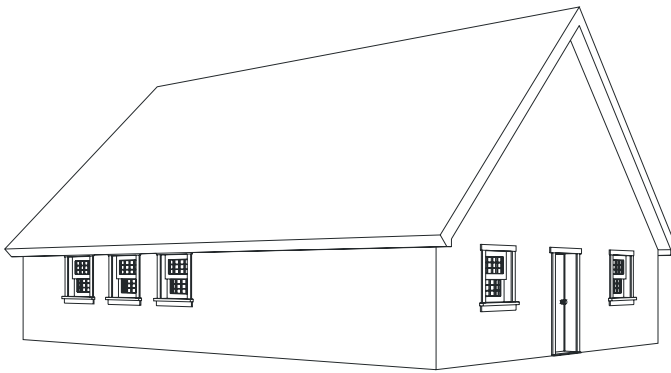


Figure 6.10: The perspective view after a hidden line removal

16. Choose GotoView.
17. Choose 3DPlan.
18. Choose Orthographic to view the schoolhouse in plan view.

See “Hidden Line Removal” later in this chapter for more information on the Hide option.

Creating Elevations

This section shows you how to create elevations from the work you have already completed. You will use the same procedure discussed in the Hide section for creating your elevations.

➔ To create elevations:

1. Choose Elevation from the View pull-down menu on the Menu Bar, and then select Front to display the front elevation of the schoolhouse.
2. Recalculate the extents of the drawing.

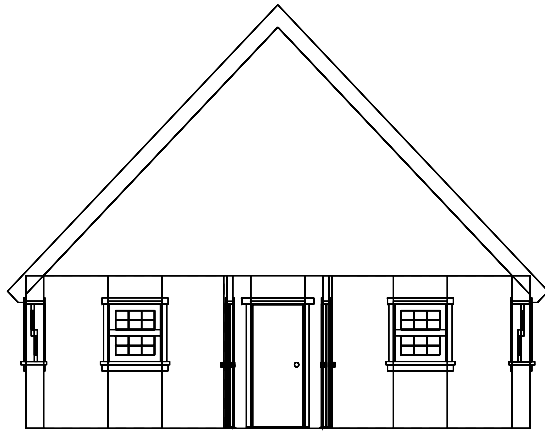


Figure 6.11: The front elevation of the schoolhouse

3. Choose DCAD 3D from the Edit menu to go to the 3DEdit menu.
4. Choose Hide.
5. Choose Begin. The hidden line removal process generates a new image with everything behind the front wall not shown.



Figure 6.12: The elevation after a hidden line removal

6. At the completion of the hidden line removal, choose NewLyr. You are prompted: Enter name of new layer to be added.
7. Type **FrntElev** and press **[Enter]**. You are prompted: Do you want the new layer ON or OFF?
8. Choose Off. You will turn this layer on later in the tutorial. The hidden line removal is complete and a new layer has been added to the drawing.
9. Choose Orthographic from the View pull-down menu on the Menu Bar for the plan view.
10. Choose DCAD 2D to return to the Edit menu.
11. Choose Layers from the Utility menu.

12. Choose ActvOnly. You are prompted: Select layer to be the only layer on.
13. Choose Scr1FwrD to view the end of the layer list.
14. Choose Elevation/Front from the View pull-down menu on the Menu Bar to display the schoolhouse front elevation.
15. Return to the Edit menu.
16. Recalculate the extents of the drawing.
17. Choose Erase from the Edit menu.
18. Choose Entity as the selection method for erasing lines.
19. Select any lines on the front elevation that should not be there.
20. Return to the Utility menu.

Experiment with a few templates, hatch patterns, etc.

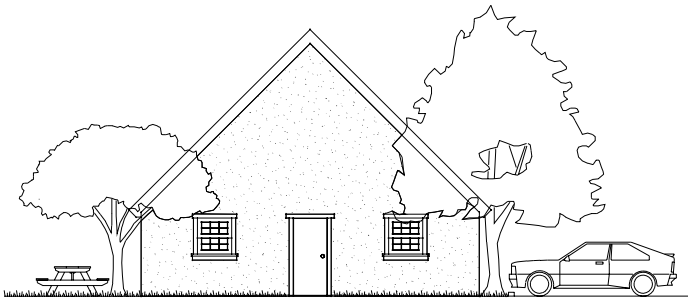


Figure 6.13: The schoolhouse elevation with templates and hatch

This completes the DataCAD Tutorial. For additional instruction, please contact DATA CAD LLC for information on training materials and support options, or visit us online at www.datacad.com.

Before You Draw

There are many settings that you can use to customize how you'll draw in DataCAD. You can keep track of time spent on a drawing and enter other administrative information directly into the drawing. The Settings menu has a variety of options to customize your work session, including setting scale and angle types, creating drawing passwords, setting cursor accuracy, adjusting the time delay for creating autosave files, and editing the definitions of DataCAD's scale, angle, and distance lists.

You can also set the size and angle of both the snap grid and the two display grids and choose whether the display grids are visible. And, perhaps most importantly, you can create and name new layers for your drawing, choose which of these will be displayed, assign different colors for each, and even save them to a separate file.

All of these settings can be used with default drawings to increase efficiency and save time. First, create a new drawing file; then change the settings, grids, and layers as you'd like them to be set in all your new DataCAD drawings; and finally, save the file in your default drawing folder. (See "The Drawing Board" chapter for more information about using default drawings.) Then in the Tools/ Program Preferences dialog box, set the Default Drawing File option to your new default drawing. All of your customized settings will automatically be set in every new drawing you create --- simply start drawing!

In this chapter:

- ⊕ Setting the snap grid
- ⊕ Setting the display grids
- ⊕ Creating and naming new layers
- ⊕ Moving between layers
- ⊕ Setting the active layer
- ⊕ Assigning colors to layers
- ⊕ Entering project costing information
- ⊕ Selecting the scale type and angle type
- ⊕ Creating passwords

Managing Your Project

The Project Directory provides the basis for a comprehensive management system based on actual computer usage and application. You can use Directory to review and enter project information directly into the drawing file. You can also save the project information and time data to a text file, and even purge unused symbol information from your drawing. Simply click on Directory in the Utility menu in the Menu Window or click on Directory in the Tools pull-down menu in the Menu Bar to display the Directory dialog box. Even if you delete a symbol from your drawing, the symbol's name will still be listed in the Directory. To purge the names of all unused symbols from your drawing, click on Purge Symbols in the Directory dialog box.

In the Billing Rate, Department Number, Phase Number, and Service Number text input boxes, you can enter a number from 0 to 32,000. In the Project text input box, you can enter up to nine alpha-numeric characters; in the Employee text input box, you can enter up to seven alpha-numeric characters.

Directory of current drawing

Project: Department Number:
 Employee: Phase Number:
 Billing Rate: Service Number:

Show information for:
☐ Project ☒ Layers ☐ Symbols ☐ SysPaths

Total : 47:12:00 Current drawing size = 488 K
 Since last posting : 47:12:00 Free disk space :
 This session : 7:49:00 Drawing files = 422 MB
 Virtual files = 422 MB

Name of the layer	ON/OFF	Color	Num. Entities
GLASS	ON	Lt Cyan	133
METAL	ON	Lt Grey	417
STUCCO	ON	White	677
COPPER	ON	Brown	256
WOOD	ON	Lt Red	29
ROOF	ON	Yellow	1

Figure 7.1: The Directory dialog box

Setting Up Your Drawing Grids

There are three separate grids in DataCAD: a snap grid and two display grids. The snap grid controls cursor movement while drawing. The two display grids, used primarily for visual reference while drawing, are shown as a series of dots and help you create drawings quickly and accurately. Each layer in DataCAD has its own snap and display grids as well as grid origin and sizes. This lets you move quickly between layers without having to change grid sizes or toggle grids on or off.

Using the Snap Grid

Although the snap grid doesn't display in the Drawing Area, it helps you draw more precisely. The snap grid causes the cursor to jump, or snap, from one point of the snap grid to the next as you move the mouse across the Drawing Area. See "Changing the Grid Size" and "Setting the Snap Grid Angle" later in this chapter for more information about the snap grid.

Look at the X and Y coordinate display in the top line of the Message Area to see the exact coordinates of the cursor as you move it over the Drawing Area. With the snap grid toggled on, notice the coordinate display reflects the cursor's "jumps" from one invisible snap point to the next across the snap grid. When you toggle the snap grid off, the cursor advances freely across the Drawing Area, without snapping to grid points.

➔ To toggle the snap grid on or off:

1. Go to the Utility menu in the Menu Window and click on Grids. The Grids menu is displayed.
2. Click on SnapGrid.

Shortcut: Press [X] to toggle the snap grid on or off. Note that the "S" in SWOTHLUDFB in the Status Area is displayed in lowercase when snap grid is toggled off and uppercase when it's toggle on.

Using the Display Grids

There are two display grids in DataCAD. One is a series of white dots in the Drawing Area, while the second display grid is a series of small yellow crosses. These grids are visual aids only and can't be printed. The display grid colors are white and yellow by default. However, you can change these colors; see "Changing Display Grid Color" later in this chapter.

➔ To toggle display grid 1 on or off:

1. Go to the Utility menu in the Menu Window and click on Grids. The Grids menu is displayed.
2. Click on DspGrid1.

Shortcut: Press [I] to toggle DspGrid1 on and off.

➔ To toggle display grid 2 on or off:

1. Go to the Utility menu in the Menu Window and click on Grids. The Grids menu is displayed.
2. Click on DspGrid2.

Shortcut: Press [J] to toggle DspGrid2 on and off.

Changing Grid Size

The snap grid's default setting is 4", display grid 1's is 4'-0", and display grid 2's is 16'-0". You can change any or all of these grid sizes for *each* layer in your drawing. To change a grid size for another layer, simply change the active layer before following the step-by-step instructions in the next paragraph.

➔ To set the snap grid's size for the active layer:

1. Go to the Utility menu in the Menu Window and click on Grids. The Grids menu is displayed.
2. Click on GridSize in the Grids menu. The GridSize menu is displayed.
3. Click on SetSnap in the GridSize menu. A value menu is displayed.
3. Use the value menu to select a new value or click Custom; type a new snap grid value and press **Enter**.

Shortcut: Press **S** to set the snap grid size.

➔ To set display grid 1's size for the active layer:

1. Go to the Utility menu in the Menu Window and click on Grids. The Grids menu is displayed.
2. Click on GridSize in the Grids menu. The GridSize menu is displayed.
3. Click on SetGrid1 in the GridSize menu. A value menu is displayed.
4. Use the value menu to select a new value, type a new display grid value and press **Enter**. For more information on using value menus, see "Value Menus" in "The Drawing Board" chapter.

➔ To set the display grid 2's size for the active layer:

1. Go to the Utility menu in the Menu Window and click on Grids. The Grids menu is displayed.
2. Click on GridSize in the Grids menu. The GridSize menu is displayed.
3. Click on SetGrid2 in the GridSize menu. A value menu is displayed.
4. Use the value menu to select a new value or click Custom; type a new display grid value and press **Enter**.

You can use the Custom option to enter a new value for any of these grid settings. For more information on how to use the Custom option, see "Value Menus" in "The Drawing Board" chapter.

Changing Display Grid Color

You can change the color of both of the display grids; however, this setting is global, meaning the colors will change on every layer in your drawing. You can't set separate grid colors on each layer.

➔ To change the color of display grid 1:

1. Go to the Utility menu in the Menu Window and click on Grids. The Grids menu is displayed.
2. Click on GridColr in the Grids menu. The GridColr menu is displayed.
3. Click on SetGrid1 in the GridColr menu. A color menu is displayed.
4. Choose a new color from the color menu in the Menu Window. For more information on using color menus, see "Color Menus" in "The Drawing Board" chapter.

➔ To change the color of display grid 2:

1. Go to the Utility menu in the Menu Window and click on Grids. The Grids menu is displayed.
2. Click on GridColr in the Grids menu. The GridColr menu is displayed.
3. Click on SetGrid2 in the GridColr menu. A color menu is displayed.
4. Choose a new color from the color menu in the Menu Window.

Setting the Snap Angle

The *snap angle* is the angle increment that you can draw lines at when ortho mode is on. When you're drawing a line or wall, for instance, once you click to place the first end point, you can move the cursor around that point. Notice though that your cursor jumps, or snaps, in increments around that first end point. These increments are set with the snap angle. With the snap angle set to 4, your cursor moves in 90° increments. If you change the setting to 36, the cursor snaps to 10° increments. If you toggle ortho mode off, your cursor will move freely, and you'll be able to draw at *any* angle.

Shortcut: Press ☐ to toggle ortho mode on or off. The "O" in SWOTHLUDFB in the Status Area will display as lowercase when ortho mode is off and uppercase when ortho mode is toggled on.

➔ To set the snap angle:

1. Go to the Utility menu in the Menu Window and click on Grids. The Grids menu is displayed.
2. Click on SnapAng in the Grids menu. A value menu is displayed in the Menu Window.
3. Use the value menu to set a new snap angle or type a new value, and press . The maximum setting for the snap angle is 36.

Setting the Grid Angle

Rotating the snap grid and both display grids can make it easier to draw in some situations. The Angle option will rotate all three grids. You can't rotate each grid separately.

➔ To rotate all three grids:

1. Go to the Utility menu in the Menu Window and click on Grids. The Grids menu is displayed.
2. Click on Angle in the Grids menu. A value menu is displayed.
3. Choose or type a new angle value, from 0° to 90°, and press .

➔ To rotate the grids to the same angle as an entity in your drawing:

1. Go to the Utility menu in the Menu Window and click on Grids. The Grids menu is displayed.
2. Click on Angle in the Grids menu. A value menu is displayed.

3. Click on Match: in the Angle menu.
4. Click on an entity to match its angle; the angle of this entity is displayed in the Message Area. Check the Message Area as you choose different options; it will display the angle associated with each of the options in step 5.
5. To use this angle, right-click to exit the menu. To use the inverse of this angle, click on Invert in the Menu Window. To use the complementary angle of the original angle, click on Complmnt. To use the supplementary angle of the original angle, click on Suplemnt. Click on Original to go back to the original angle.
6. Right-click to return to the value menu, and press **[Enter]** to accept the angle value. The grid rotates to the Angle setting.

Setting the Grid Origin

You can change the origin for the snap grid and both display grids on each layer or for all layers simultaneously. The grid origin is *not* the drawing origin, which is a fixed point. The grid origin is changed on *all* layers, whether they're toggled on (and currently being displayed) or not.

➔ To change the grid origin on all layers in your drawing:

1. Go to the Utility menu in the Menu Window and click on Grids. The Grids menu is displayed.
2. Click on GridOrg in the Grids menu. The GridOrg menu is displayed.
3. Look at the GridOrg menu to make sure SetAll is toggled on.
4. Click anywhere in the Drawing Area or use coordinate entry to set the new grid origin. The display grids realign themselves based on your new origin. For more information on using coordinate entry, see "Using Coordinate Entry" in "The Drawing Board" chapter.

To display the grids at their new origin, you may have to press **[Esc]** to refresh the Drawing Area. You can press **[Esc]** at any time while you work in DataCAD to refresh your drawing. Use object snapping to more precisely choose the point in your drawing for the new grid origin. See "Object Snapping" in the "Drawing Tools" chapter.

➔ To change the grid origin on the active layer only:

1. Go to the Utility menu in the Menu Window and click on Grids. The Grids menu is displayed.
2. Click on GridOrg in the Grids menu. The GridOrg menu is displayed.
3. Look at the GridOrg menu to make sure SetAll is toggled off.
4. Click anywhere in the Drawing Area or use coordinate entry to set the new grid origin. The display grids realign themselves based on your new origin.
5. If you'd like to change the grid origins on other layers in your drawing, press **[Tab]** to change the active layer and repeat steps 2 through 4.

Working With Layers

You can organize your drawing into groups, or layers, of related information to help increase your editing speed and productivity. By organizing the information on each layer according to type or content, you can display only the information that you need for the current drawing task. See “Planning Layers” and “Sample Project” in “The Drawing Board” chapter for an introduction to planning your layering system.

Creating Layers

When you create a new drawing, only one layer exists, named Layer001. But you can create up to 999 layers in every drawing, either one at a time or in groups.

Save time when you create new drawings by using a default drawing that contains all the layer names and settings you’d normally use. Your new drawings will automatically have all the layers in your default drawing and you can start drawing right away. See “Creating a New Drawing Using a Default Drawing” in “The Drawing Board” chapter.

➔ To create layers in your drawing:

1. Go to the Utility menu in the Menu Window and click on Layers. The Layers menu is displayed.
2. Click on NewLayer in the Layers menu. A value menu is displayed in the Menu Window.
3. Use the value menu to enter the number of new layers you’d like to create, or type a value, and press **[Enter]**. The number of layers created is displayed in the Message Area, along with the total number of layers now in your drawing.

More About Creating Layers

The active layer in your drawing acts as a seed layer for any new layers you create. When you create a new layer, it takes on the settings of the active layer. To save time and avoid redefining settings for each new layer, change the active layer to the layer in your drawing whose settings most closely match those you want your new layer to have.

Each layer has several independent settings that are customizable, including grid size, grid angle, whether the snap grid is toggled on or off, whether ortho mode is toggled on or off, and layer color. See “Setting Up Your Drawing Grids” earlier in this chapter for more information on grid settings.

A *seed layer* then is a layer that is used as a basis for subsequent layers. For example, you might have a basic perimeter plan of a multi-story building you’re drawing. This plan might contain the perimeter and bearing walls, the columns, the column centerlines, and the common core elements, each on a different layer. These layers could be used as seed layers for the partitioning layout on the third floor, which in turn could be used as the seed layer for the reflected ceiling plan or HVAC layout for the third floor. Any layers you want to use as seed layers should be kept as generic and uncluttered as possible.

Setting the Active Layer

The *active layer* is the layer on which you're currently drawing. To quickly check which layer is your active layer, check the Status Area.

➔ To change the active layer:

1. Go to the Utility menu in the Menu Window and click on Layers. The Layers menu is displayed.
2. Click on SetActiv in the Layers menu. A list of layers is displayed in the Menu Window.
3. Click on a layer in the list to make it your active layer, or type the layer name, and press **[Enter]**. The active layer changes, and the active layer name is updated in the Status Area.

Shortcut: You can also use the keyboard to change the active layer. Press **[Tab]** to scroll through all the layers in your drawing that are toggled on. To scroll backward through the layers, press **[Shift] + [Tab]**. Notice the active layer name in the Status Area changes as you scroll through the layers.

➔ To change the active layer and simultaneously toggle all other layers off:

1. Go to the Utility menu in the Menu Window and click on Layers. The Layers menu is displayed.
2. Click on ActvOnly in the Layers menu. A list of layers is displayed in the Menu Window.
3. Click on a layer in the list to make it your active layer, or type the layer name, and press **[Enter]**. The active layer name is updated in the Status Area.

Displaying Layers

One of the most useful things about using layers in your drawings is that you can view only those layers that you need to at any given time. For instance, you don't need to see the site plan while you're working on a reflected ceiling plan. In DataCAD, you simply toggle layers on to display them or off to hide them.

You can toggle certain layers on to view them in the Drawing Area. You can then use go-to-views to save that view of your drawing. See the "Viewing Your Model" chapter.

➔ To toggle a layer on or off:

1. Go to the Utility menu in the Menu Window and click on Layers. The Layers menu is displayed. The Menu Window displays up to 15 layer names at one time; when there are more than 15 layers in your drawing, options to scroll forward (ScrlFwr) and scroll backward (ScrlBack) through the list are available.
2. Click on On/Off in the Layers menu. A list of all the layers in your drawing is displayed in the Menu Window.

3. Click on the layer you want to toggle on or off. You can't toggle all layers off in DataCAD. If the active layer is the only one toggled on, you can't toggle it off until you make a different layer active. If there's at least one other layer toggled on besides the active layer, you can toggle the active layer off; the next layer in the list automatically becomes the active layer.

➔ To toggle on all the layers in your drawing simultaneously:

1. Go to the Utility menu in the Menu Window and click on Layers.
2. Click on AllOn in the Layers menu. In the Message Area, you are prompted with "Are you sure you want to turn on all layers?".
3. Click on Yes. All layers are toggled on.

More About Displaying Layers

Refresh speed should also be considered when you plan and create the layers in your drawing. While your computer's speed determines the time it takes to refresh the display of your drawing, you can dramatically increase refresh speed using some of the Layers options.

Shortcut: There are many reasons why you may want to refresh the display of your drawing during your work session. Press **[Esc]** at any time to refresh your drawing.

As you toggle each layer on, DataCAD can automatically update, or refresh, your drawing in the Drawing Area. To do this, toggle the layer refresh option (LyrRfsh) on in the Layers menu. To quickly toggle several layers on without waiting for your drawing to refresh after each layer, make sure LyrRfsh is toggled off before you begin toggling layers on. After you've finished toggling layers on, press **[Esc]** to refresh the display of your drawing.

Each time you zoom in or out on your drawing or change the view – two things you'll be doing frequently – DataCAD must refresh the display of your drawing to reflect the change. Since only those layers that are actually displayed are refreshed, toggling on only those layers that are necessary for your current drawing task will speed up refresh rates. This is one of the many reasons why planning and maintaining a good layer structure is important.

Additionally, controlling the number of entities on each layer will increase refresh rates. A drawing containing many layers with a few entities on each will refresh faster than a drawing containing a few layers with many entities on each. In other words, you can edit ten layers containing 100 lines faster than one layer containing 1,000 lines.

Locking Layers

You can lock a layer to prevent others from editing the entities on that layer. In the list of layers in the Menu Window, a line appears through the layer name of a locked layer.

You can't modify the layer in any way when it's locked. If you try to edit a locked layer, a message saying that no entities are selected is displayed in the Message Area. The layer lock status is saved with the layer when SaveLyr, LoadLyr, and the Layer Utility macro (located in the Toolbox in the Edit menu) are used.

➔ To lock a layer:

1. Go to the Utility menu in the Menu Window and click on Layers. The Layers menu is displayed.
2. Click on Lock in the Layers menu. A list of all the layers in your drawing is displayed in the Menu Window.
3. Click on the layer you want to lock in the list, or type the layer name, and press **[Enter]**. A line through the layer name appears, and if you make that layer the active layer, a small "X" is displayed over the layer color box in the Status area. The layer can't be edited until it's unlocked.

➔ To unlock a layer:

1. Go to the Utility menu in the Menu Window and click on Layers. The Layers menu is displayed.
2. Click on Lock in the Layers menu. A list of all the layers in your drawing is displayed in the Menu Window.
3. Click on the layer you want to unlock in the list, or type the layer name, and press **[Enter]**. The line through the layer name disappears; the layer can now be edited.

Renaming Layers

When you create new layers, DataCAD assigns generic names to the layers in numerical order, such as Layer001, Layer002, etc. You can either rename the layers when you create them or rename them at some other time.

➔ To rename an existing layer:

1. In the Utility menu in the Menu Window, click on Layers. The Layers menu is displayed.
2. Click on Name in the Layers menu. A list of layers in your drawing is displayed in the Menu Window.
3. Click on the layer you want to rename.
4. Type a new layer name, up to 88 characters, and press **[Enter]**. You cannot use spaces or the following characters in layer names: , . * + / \ [] ; : .

Changing Layer Color

You can assign a color to a layer, so that all new entities you create on that layer appear in that color. Assigning a different color to each layer gives you an easy way to tell at a glance what layer an entity is on. If you assign yellow to your Walls layer, for instance, all entities drawn on the Walls layer will display in yellow. By default, white is the color assigned to all layers.

When you change a layer's color, entities already on that layer will not change to the new color. To change these entities' colors, you must use the Change menu. For more information, see "Changing Entity Attributes" in the "Editing Drawings" chapter.

➔ To change the color assigned to a layer:

1. Go to the Utility menu in the Menu Window and click on Layers. The Layers menu is displayed.
2. Click on Color in the Layers menu. A color menu is displayed in the Menu Window. See “Color Menus” in “The Drawing Board” chapter for more information on how to use them.
3. Click on a color in the list, or use the color menu options Adjust, Custom, or Match, to assign a new color to the layer.

Shortcut: To change the color of the active layer, press [K] to scroll through available colors; press [Shift] + [K] to scroll backwards through the color menu.

Viewing a Layer

There may be times when you’re working with your layers that you need to temporarily view a specific layer. Perhaps you need to quickly verify the contents of a layer before you toggle it on.

➔ To temporarily view a layer:

1. Go to the Utility menu in the Menu Window and click on Layers. The Layers menu is displayed.
2. Click on ViewLyr in the Layers menu. See “ViewLyr Menu Options” below for more information on these settings.
3. Make sure Extents, LyrRfsh, and Border are toggled on in the ViewLyr menu.
4. Click on Select in the ViewLyr menu. A list of all the layers in your drawing is displayed in the Menu Window.
5. Click on the layer you’d like to view, or type the layer name, and press [Enter]. The contents of that layer are displayed in the Drawing Area.
6. Press (Esc) when you’re finished viewing layers. This will refresh the Drawing Area and return to the original view of your drawing, before you entered the ViewLyr menu.

ViewLyr Menu Options

Select	Lists the layers in your drawing; click on the layer name in the list or type the layer name, and press [Enter]
Extents	Toggle on to display the layer at its extents; toggle off to display the layer at the current scale
LyrRfsh	Toggle on to refresh the Drawing Area as you select each layer
Border	Toggle on to use an invisible border to display each layer in
ImgSize	Sets the percentage of the Drawing Area to use to display layers; only available when border is toggled on

Deleting Layers and Their Contents

When a layer is no longer useful to your project, you can erase only the contents of the layer or delete the actual layer as well as its contents.

➔ To erase the contents of a layer:

1. Go to the Utility menu in the Menu Window and click on Layers. The Layers menu is displayed.
2. Click on EraseLyr in the Layers menu. A list of all the layers in your drawing is displayed in the Menu Window.
3. Click on the layer that contains the entities you want to erase, or type the layer name. The prompt “Are you sure you want to erase: <layer name>” is displayed in the Message Area.
4. Click on Yes to erase the layer’s contents. The layer itself is not deleted and isn’t changed in any other way. Click on No to exit the EraseLyr menu without erasing the layer’s contents.

➔ To delete a layer and its contents from the drawing:

1. Go to the Utility menu in the Menu Window and click on Layers.

Warning: As a precaution, use the SaveLyr option to save, or backup, a layer before deleting it. If you find you still need the layer later, you can then use LoadLyr to copy it back into your drawing.

2. Click on DelLayer in the Layers menu. A list of all the layers in your drawing is displayed in the Menu Window.
3. Click on the layer that you want to delete, or type the layer name. The prompt “Are you sure you want to delete layer <layer name>” is displayed in the Message Area.
4. Click on Yes to delete the layer. Click on No to exit the DelLayer menu without deleting a layer. You cannot delete the active layer.

Selecting Layers Without Using Layer Names

You may have noticed the Match option in several of the Layers submenus. Match is used to select a layer by selecting an entity on that layer instead of having to find the correct layer name in a list. You can set the active layer, move entities from one layer to another, or point DataCAD to a specific layer, all without having to remember layer names.

You can use Match with any of the following Layers menu options: On/Off, SetActiv, ActvOnly, Lock, Name, EraseLyr, DelLayer, SaveLyr, ViewLyr/Select. It is also available when using ToLayer in the Move or Copy menus or when using LyrSrch.

➔ To set the active layer using Match:

1. Go to the Utility menu in the Menu Window and click on Layers. The Layers menu is displayed.
2. Click on SetActiv in the Layers menu. The SetActiv menu is displayed.
3. Click Match in the SetActiv menu.
4. Click on an entity in the Drawing Area to select the layer it is on. The active layer setting is changed to that layer.

➔ To move entities to another layer using Match:

1. Go to the Edit menu and click on Move.
2. Click on ToLayer in the Move menu. You are prompted to “Select layer to move to” in the Message Area. The ToLayer option is only available if your drawing has more than one layer. See “Creating Layers” earlier in this chapter for more information on creating additional layers.
3. Click Match in the ToLayer menu. You are prompted to “Select entity to match” in the Message Area.
4. Click on an entity in the Drawing Area to select the layer to move the entity *to*. You are prompted to “Select entity to move” in the Message Area.
5. Click on the entity you want to move. The entity is moved to the layer you selected in step 4.

➔ To point DataCAD to a specific layer using Match:

1. Indicate the wall layer. When drawing doors or windows on a different layer from walls, you must tell DataCAD which layer your walls are on so they can be cut properly. Make the Doors layer your active layer, and click on Architct in the Edit menu. See “Setting the Active Layer” earlier in this chapter for more information.
2. Click on DoorSwng in the Architct menu.
3. Toggle LyrSrch on in the DoorSwng menu. If it’s already toggled on, toggle it off and then back on again to display the list of layers in your drawing.
4. Respond to the prompt to “Select layer to search for walls” in the Message Area. Click on Match at the bottom of the list of layers.
5. Respond to the prompt to “Select entity on layer to match” in the Message Area. Click on a wall in your drawing to select the layer that your walls are drawn on.
6. Draw your door. The wall will be cut properly on the Walls layer. This same procedure can be used with the Windows or CutWall options. For details on how to draw doors and windows, see the “Drawing Walls, Windows, and Doors” chapter.

Searching for Layers

You may have noticed the Filter option in several of the Layers submenus. Filter is used to search for layers with similar names. For example, if you’ve put your plumbing and electrical drawings on separate layers, you might have layers named 1FLPLUMB, 2FLPLUMB, 3FLPLUMB, 1FLELECT, 2FLELECT, and 3FLELECT. To search for a plumbing layer, you’d use Filter and enter **???plumb**. Or you could search for first floor layers by entering **1fl*** or **1fl????**.

Use an asterisk (*) in place of any number of characters and a question mark (?) in place of each character you're unsure of. The filter disregards any characters following an asterisk. Therefore, if you enter ***plumb** to search for plumbing layers, DataCAD recognizes only * as the search parameter and all of your layers will be displayed.

➔ To search for layers:

1. Go to the Utility menu in the Menu Window and click on Layers. The Layers menu is displayed.
2. Click on the Layers option you want to use, On/Off, for example.
3. Click on Filter in the Layers option submenu to toggle Filter on.
4. Type the search parameter, including any wildcard character(s), for the layers you want to find. Your search results are displayed in a list in the Menu Window.
5. Choose the layers you want to use.

More About Searching for Layers

DataCAD groups layers with affixes (prefixes or suffixes) similar to what you entered. If your search doesn't find the layer you want, you can scroll through these groups to find the appropriate layer using the FiltBack and FiltFrwd options that are displayed at the bottom of the Menu Window, below your search results.

DataCAD remembers the last search you did using Filter. When you next use Filter, the search parameter is displayed in the Message Area and your search results are displayed in the Menu Window.

Saving and Loading Layers

You may occasionally need to copy a layer from one drawing to another, you may simply want to save a copy of a layer before you delete it in case you find you need it again, or perhaps you'd like to save a hidden line removal of your model to a layer file to avoid increasing your drawing file size. The SaveLyr and LoadLyr options in the Layers menu can help you do all of these things. SaveLyr saves a layer to a layer (.LYR) file; LoadLyr pastes a layer file into a drawing.

➔ To save a layer:

1. Go to the Utility menu in the Menu Window and click on Layers. The Layers menu is displayed.
2. Click on SaveLyr in the Layers menu. A list of your layers is displayed in the Menu Window.
3. Click on the layer you want to save. A dialog box is displayed.

4. Type the name of layer file and click on Save or press **[Enter]**. DataCAD saves the layer with a .LYR extension. If you already have a layer file by that name, DataCAD prompts you to verify that you want to overwrite the existing file, or in other words replace the old version of the file with this new one. Press **[Enter]** to overwrite the file. You are returned to the Layers menu. By default, files are saved in your DATACAD\LYR folder. You can change the folder that layer files are saved to using the Pathnames tab in the Tools/ Program Preferences dialog box.

➔ To load a layer into a drawing:

1. Go to your drawing and change the active layer to the layer that you want to copy the layer file onto.
2. Go to the Utility menu in the Menu Window and click on Layers. The Layers menu is displayed.
3. Click on LoadLyr in the Layers menu. You are prompted with “Delete active layer?” in the Message Area.
4. Click on Yes in the Menu Window to erase anything on the active layer. Click on No to add the entities on the layer you will load to the entities already on your active layer. A dialog box is displayed, listing all your layer files.

Warning: If you choose Yes to delete the active layer, you cannot restore those entities.

5. Click on the layer file you want to load or type the filename in the Filename input box.
6. Click on Open or press **[Enter]**. The layer file is copied to the active layer in your drawing.

More About Naming a Layer File

Just as you planned how to name the layers in your drawing, you should also plan a logical, useful naming scheme for your layer files. To keep track of where the layer came from, the layer filename should be as similar as possible to the name of the drawing file from which it originated. One way to name layer files is to use the drawing filename followed by a descriptive name for the layer. For example, if you saved the layer that contained a hidden line removal from the drawing file HOUSE.DC5, you might name the layer file, HOUSE_HL.LYR.

Using the Layer Manager

The Layer Manager gives you control over each layer’s number, status (on, off, locked, and unlocked), color, and name. It displays the number of entities on each layer in the Count column.

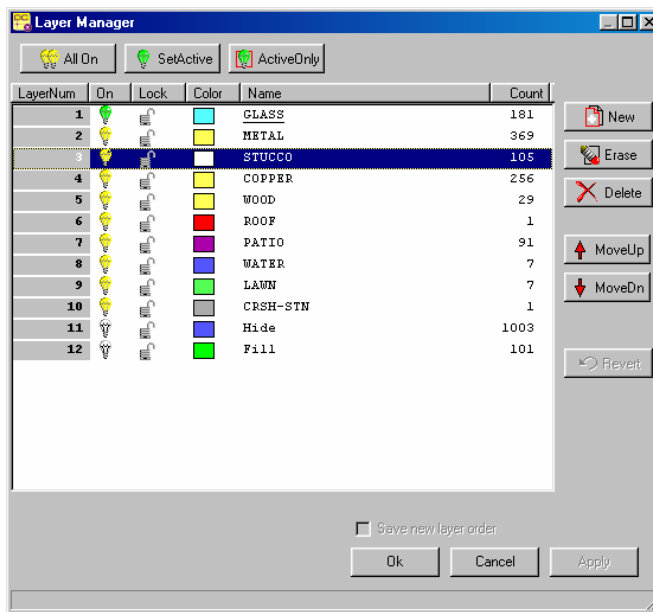


Figure 7.2: The Layer Manager dialog box

Accessing Layer Manager

➔ To open the Layer Manager, use one of these options:

- Select Layers from the Utility menu and click on Manage in the Layer menu. The Layer Manager dialog box appears.
- Or select Layer Manager from the Tools pull-down menu. The Layer Manager dialog box appears.
- Or use the keyboard shortcut (L) for Layer and (S8) for Manage. The Layer Manager dialog box appears.
- Or use the keyboard shortcut (Ctrl) + (L). The Layer Manager dialog box appears.
- Or click on the active layer's name in the status window.

Initially, the only active buttons are AllOn, New, Ok, and Cancel. When you highlight the layer you want to work with, other buttons will become active.

Changing a Layer's Number

➔ To change a layer's number:

1. Click on the number of the layer you want to change. The layer you selected is highlighted in the Layer Manager dialog box. Notice that in addition to AllOn and New, the SetActive, ActiveOnly, Erase, Delete, MoveUp, and MoveDn buttons are active.

Hint: To select several successive layers (for example, layers 3 through 6) click on the first one, press down and hold (Shift), and click on the last layer. Release the (Shift) key when you have highlighted the successive layers you want to work with in the Layer Manager dialog box.

To select several layers that are not in consecutive order (for example, layers 3, 5, and 7), you select the first one, press down and hold (Ctrl), and click on the other layers. Release the (Ctrl) key when you have highlighted the nonconsecutive layers you want to work with in the Layer Manager dialog box.

2. Click on the MoveUp or MoveDn button until you are satisfied with the layer's new position. Alternatively, you can use the mouse to reposition the layer with drag and drop. Just click on the number, drag your mouse, and release the left mouse button when you reach the desired new number to dynamically reposition the layer.

Hint: When you make changes to the position of the layers by using the MoveUp or MoveDn buttons or the drag-and-drop method, DataCAD automatically inserts a checkmark in the Save new layer order box. If this box does not contain a checkmark, your changes will *not* be saved and the layers will return to their original positions when you leave the Layer Manager dialog.

3. Click on the Apply or Ok button when you finish rearranging the layers. This saves your layers in the new arrangement and deactivates the Revert button. When you click the Ok button, the Layer Manager dialog box disappears and you return to the drawing window.

➔ To return the layers you moved to their original positions, use one of these options:

- Click on the Revert button if this option is available. If you selected Apply, the Revert button will not be active. If you selected Ok, the Layer Manager dialog box will not be displayed on your screen.
- Or click on the Undo arrow once if the Layer Manager dialog box is not on your screen to return to the positions that were last active.
- Or press (Ctrl)+(Z) for Undo if the Layer Manager dialog box is not on your screen.

Sorting the Layers

Use Layer Manager to sort your drawing's layers by clicking on the column titles. Clicking the same column title twice will reverse the sorting order. For example, you may want to sort the layers alphabetically A-Z by the information in the Name column. If you click the Name column a second time, the layers will appear alphabetically Z-A.

Hint: Sorting your layers by clicking on the column titles does not automatically place a checkmark in the Save new layer order box. If you want to save the layers in the newly sorted order, make sure you click in the Save new layer order box so that a checkmark appears *before* you exit from the Layer Manager dialog box.

If you just want to quickly check the on/off or lock/unlock status of the layers, you may not want to store the layers in that order. In these cases, you probably wouldn't want to put a checkmark in the Save new layer order box. Leaving the box unchecked means that DataCAD will return the layers to the order they were in when you last opened the Layer Manager dialog box.

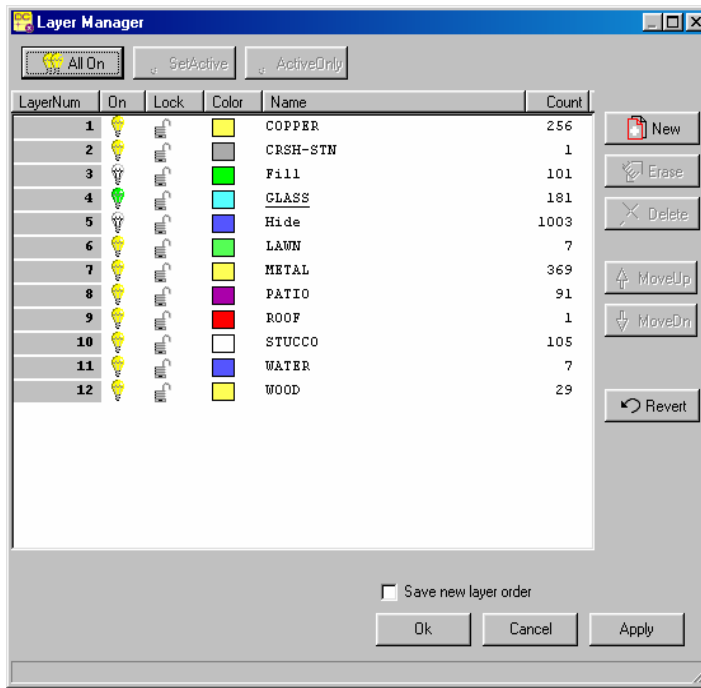


Figure 7.3: Layers sorted alphabetically

This is what happens when you use the column titles to sort your layers:

- **LayerNum:** Layers are arranged in numerical order. Clicking LayerNum again reverses the layers.
- **On:** Layers that are On will appear first. Clicking On again makes the Off layers appear first.
- **Lock:** Layers that are locked will appear first. Clicking Lock again makes the unlocked layers appear first.
- **Color:** Layers will be sorted from white through the darkest shades. Clicking Color again will sort layers from darkest shades to white.
- **Name:** Layers will be sorted alphabetically from A to Z. Clicking Name again will sort layers from Z to A.
- **Count:** Layers will be sorted numerically from the smallest to the largest number. Clicking Count again will sort layers from the largest to the smallest numbers.

Hint: Count indicates the number of entities contained on a particular layer. You can sort by count to quickly find the layers that do not contain any drawing entities. This makes it easy for you to delete empty layers from your file.

Changing Layer Settings

Each layer has different settings that you can change. These settings are on/off, lock/unlock, color, and name.

On/Off

A solid yellow light bulb in the On column indicates that a layer is on; a clear bulb means that a layer is off. The green bulb designates the active layer. Only one bulb in the layer list can be green (meaning that layer is on and active); however, any number of bulbs may be yellow (on) or clear (off).

➔ To change the on/off setting for one layer:

1. Click on the light bulb of the layer you want to change. If the bulb was originally yellow, it changes to clear and vice versa.
2. Click on Apply if you want to make other changes in the Layer Manager or click Ok to return to the drawing window, where your change is in effect.

➔ To change the on/off setting for several layers:

1. Highlight the layers you want to change by using (Shift) for successive or (Ctrl) for nonconsecutive layers.
2. Click on the light bulb for one of the highlighted layers. All the bulbs in the highlighted layers are changed to the same setting.

Hint: This is true even if some of the bulbs in the layers you selected were on and others were off. For example, if you highlighted layers 3 (on), 5 (off), and 6 (on) before you clicked on the bulb in layer 3, DataCAD would switch all three bulbs off.

3. Click on Apply if you want to make other changes in the Layer Manager or click Ok to return to the drawing window, where all your changes are in effect.

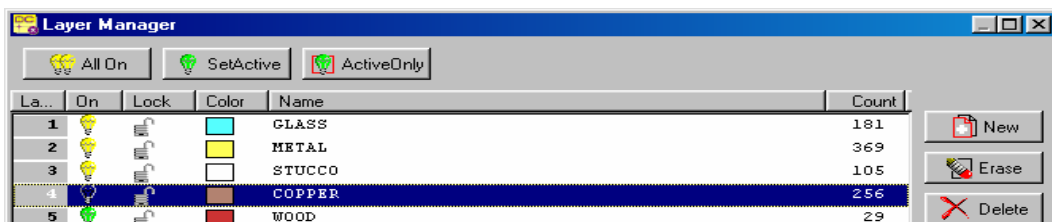


Figure 7.4: Changing the On/Off status of a layer

➔ To turn all the layers on:

1. Click the AllOn button.
2. Click on Apply if you want to make other changes in the Layer Manager or click Ok to return to the drawing window, where all the layers are on.

➔ To designate the active layer:

1. Select the layer you want to change to active status.
2. Click on the SetActive button. The bulb for the highlighted layer changes to green and its name is underlined. The bulb for the layer that was previously active changes to yellow (on) and its name is no longer underlined.

3. Click on **Apply** if you want to make other changes in the Layer Manager or click **Ok** to return to the drawing window. Notice that the layer you selected is active in the status box in your drawing window.
- ➔ To display only the layer you designate as active:
1. Highlight the layer you want to designate as active.
 2. Click the **ActiveOnly** button. The highlighted layer's bulb is green (indicating it is on and active); all other bulbs are clear (off). The layer's name is also underlined.
 3. Click on **Apply** if you want to make other changes in the Layer Manager or click **Ok** to return to the drawing window. Notice that the active layer is the only one that appears in the drawing window.

Lock/Unlock

You can edit unlocked layers, which are designated by an open padlock. Locked layers (red, closed padlock) are protected and cannot be edited.

- ➔ To lock or unlock one layer:
1. Click on the padlock you want to change. DataCAD changes that layer's lock status accordingly.
 2. Make changes to other layers in the Layer Manager if necessary. Remember that if you locked a layer, DataCAD will not allow you to make any changes to it.
 3. Click **Ok** to return to the drawing window.
- ➔ To lock or unlock several layers simultaneously:
1. Highlight successive layers by using (Shift) or several nonconsecutive layers by using (Ctrl) to select the padlocks you want to change.
 2. Click on a padlock in one of the selected layers. If the padlock you clicked was originally unlocked, DataCAD will lock it and all the others in the layers you selected in step 1.
 3. Click **Ok** to return to the drawing window.

Color

In the Layer Manager, the color of each layer appears in a square swatch. Changing the color that applies to entities on a layer is easy. Keep in mind that the color change will apply only to the entities you create on that layer; the new color doesn't affect the entities that were on the layer.

- ➔ To change the color of a layer:
1. Click on the square swatch in the Layer Manager for the layer you want to change. The Color Palette dialog box appears.



Figure 7.5: The Color Palette dialog box

2. Select a new color from the sample swatches in the palette.
3. Click OK. Notice that the new color you selected appears on the swatch in the Layer Manager.

If you do not like any of the available colors, you can easily mix your own.

Caution: Make sure that the color you want to edit is unique among the layers in your drawing. If several layers use the same color, change the selected layer's color as shown in the previous steps so that it is different from the others. If the color of the layer you want to change is not unique, all other layers sharing that color will also be changed in the editing process.

If you want to mix your own color for a layer, refer to the section on "Customizing Colors" at the end of this chapter.

Name

DataCAD allows you to use up to 88 characters (including spaces) for meaningful layer names. You can change a layer's name in several ways with the Layer Manager. When you click on a layer's name once, that layer is highlighted or selected; however, you cannot edit the name.

➔ To change a layer's name use one of these methods:

- Double-click on the layer name you want to change, type the new name (up to 88 characters long), and press (Enter). The new name appears on the list in the Layer Manager.

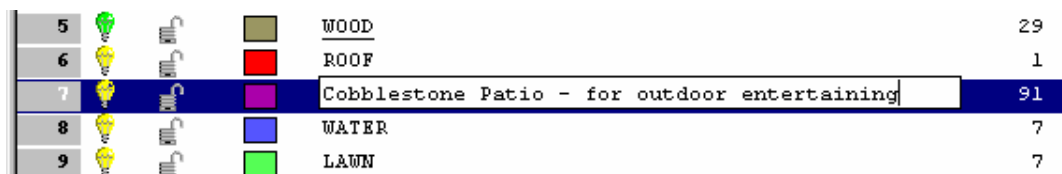


Figure 7.6: Double-clicking on the layer name lets you type a new name

- Press (Enter) when a layer is highlighted, type the new name, and press (Enter).

- Press (F2) when a layer is highlighted, type the new name, and press (Enter).

Using Other Layer Functions

With the Layer Manager, you can add new layers, erase all the entities on a layer, delete a layer, or revert to previous layer settings. To access these basic functions, you use the buttons on the right side of the layer list.



Figure 7.7: Buttons to the right of the layer list show options for changing settings

Add New Layer

The Layer Manager makes it easy to add new layers to your drawing. While you add the layer, you also set its name, on/off and lock/unlock status, and color. Initially, the new layer will not contain any drawing entities; those accumulate in the Count column as you use the layer.

➔ To add a new layer to your drawing file:

1. Highlight the layer that should appear before or above the new one you are creating. DataCAD adds the new layer after the highlighted one. For example, if you want the new Brick layer to come before Stucco, you would highlight Metal.

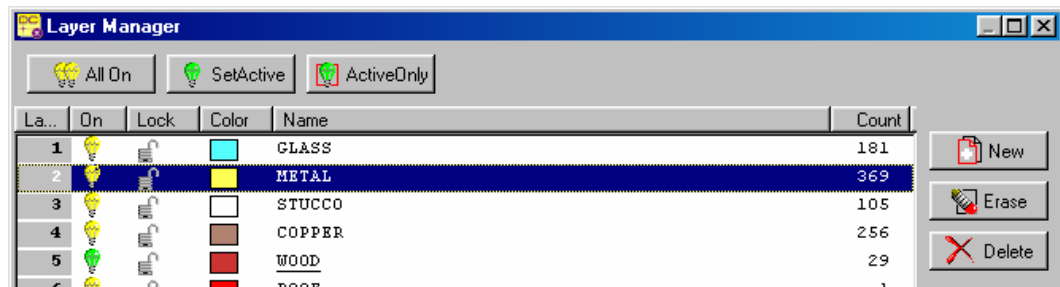


Figure 7.8: Highlight the layer above the position where you want the new layer to appear.

2. Click on the New button to the right of the layer list. The Create New Layer(s) dialog box appears and you are prompted to “Enter the number of new layers to create.”

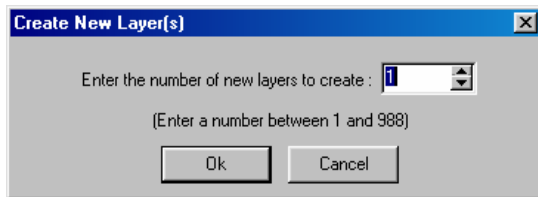


Figure 7.9: The Create New Layer(s) dialog box

3. Input the number of layers you want to add to your drawing and press (Enter) or click Ok. The new layer appears above the one you initially highlighted.

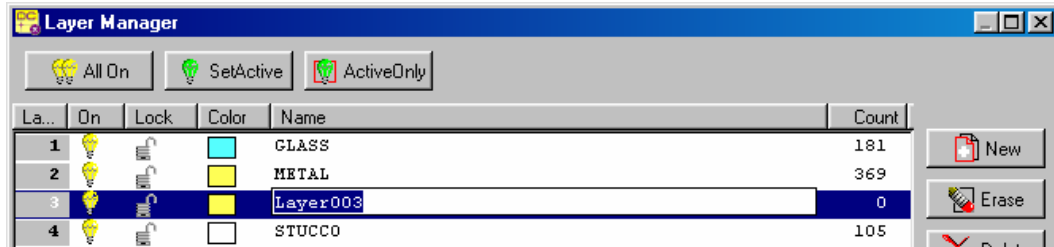


Figure 7.10: The new layer appears below the layer you highlighted in step 1.

4. Type a name for the new layer and press (Enter). Notice that Count contains a zero because there are no entities in this layer yet.

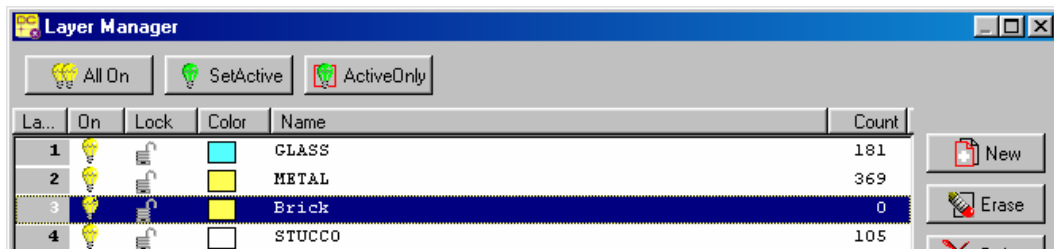


Figure 7.11: Type the new layer's name.

5. Continue adding new layers by repeating steps 1 through 4.
6. Click in the box at the bottom of the Layer Manager to put a checkmark, indicating that you want DataCAD to save the new layer order. When you are finished adding new layers to your drawing file, click on Apply (if you want to continue working in the Layer Manager) or Ok (if you do want to leave the Layer Manager and return to your drawing window now).



Figure 7.12: Check the "Save new layer order" box.

Erase Entities from a Layer

You can quickly eliminate all the entities on a layer. This can help you save time when you want to make major changes on a particular layer of your drawing.

➔ To erase entities from a layer:

1. Highlight the layer that contains the entities you want to eliminate.
2. Click on the Erase button to the right of the layer list. A Warning box appears.

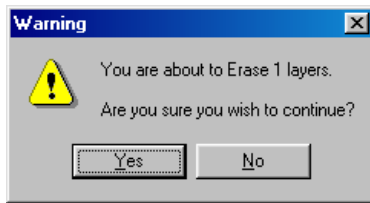


Figure 7.13: Warning box

3. Click on the Yes button or press (Enter). DataCAD puts an eraser in the Count column to signify that you are eliminating all of the entities on that particular layer.

Hint: The entities will be erased from your drawing when you click Apply or Ok (to exit the Layer Manager dialog box), but DataCAD gives you a chance to change your mind and keep those entities. To change your mind, simply select either the Revert or Cancel button.



Figure 7.14: An eraser appears in the Count column to show that entities will be erased from that particular layer.

4. Click on Apply if you want to have the entities erased while you continue working in the Layer Manager. Click Ok if you want to have the entities erased and return to your drawing window. DataCAD erases the entities on the layer.



Figure 7.15: Entities were eliminated from the layer.

Delete a Layer

You can eliminate an entire layer from your drawing. It doesn't matter whether the layer contains hundreds of entities or none.

➔ To delete a layer:

1. Highlight the layer you want to eliminate from your drawing file.
2. Click on the Delete button to the right of the layer list. A warning box appears, giving you a chance to change your mind.

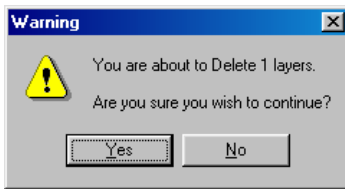


Figure 7.16: Warning box

3. Select Yes or press (Enter) to indicate that you want to continue. The layer disappears from the list.

Oh No! What if you accidentally deleted the wrong layer? Don't panic because DataCAD gives you two more chances to change your mind. You can click Revert to restore the layer immediately; the other alternatives are to select Cancel or use the Undo function.

Changing an Incorrect Layer Setting

There are three ways for you to change settings if you made a mistake in the Layer Manager dialog box. Depending on when you notice the mistake, you can select Revert or Cancel or use the Undo button if you don't catch the error until you leave the Layer Manager dialog box.

Revert to Previous Settings in the Layer Manager

If you accidentally do something to a layer (such as eliminate all its entities, turn a layer off, or eliminate the layer entirely), you can go back to the previous settings with Revert.

Revert is available *after* you do something on the Layer Manager's list and only applies to what you just did. For example, if you change a layer's status to Off and then give the layer a new name, Revert will give you a chance to return to the older name but will not turn the status to On. If Revert is unavailable (the button is gray), it's too late to change your mind about a previous decision.

Cancel

If Revert is not available (the button is gray) or you notice a previous error (such as eliminating all the entities in the wrong layer) you made during the current Layer Manager session, you can use the Cancel button. Selecting Cancel closes the Layer Manager and puts you back in the drawing window, ignores all the changes you made since you entered the dialog box, and returns all layer settings to those that existed previously in your drawing file.

Cancel affects *all* the settings you changed in the Layer Manager dialog box since you opened it. You can't selectively eliminate some changes and retain others. If selecting Cancel eliminated settings that you wanted to keep, you must reopen the Layer Manager dialog box, make the changes again, and save them when you exit the dialog by choosing the Ok button.

Undo

Use the Undo arrow on your button bar or select Undo Layer Manager from the Edit pull-down menu if the Layer Manager dialog box is not active on your screen. This eliminates all the things you changed the last time you used the Layer Manager dialog box. If you restored your old layer settings with Undo but want to keep them after all, just click on the Redo button or select Redo Layer Manager from the Edit pull-down menu.

Exiting from the Layer Manager

The buttons on the bottom of the Layer Manager dialog box give you several exit options:

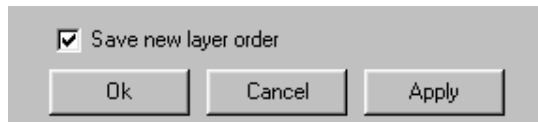


Figure 7.17: Exit options

- **Apply.** When you click on this button, the changes you make in the layer list take effect in your drawing while the Layer Manager dialog box remains open.
- **Save new layer order.** This check box is available if any of your changes involved the original layer order or if you added or deleted layers. By default, when this box is initially active, it contains a checkmark. Remember to uncheck this box if you want to go back to the layer order that was in effect *before* you opened the Layer Manager.
- **Cancel.** This disregards everything you did since opening the Layer Manager dialog box. Clicking this button returns you to the drawing window and retains your layer settings to those that were in effect *before* you opened the Layer Manager.
- **Ok.** Clicking this button returns you to your drawing screen and makes all the layer setting changes take effect in your drawing file.

Customizing Drawing Settings

Passwords, scale and angle types, and a variety of display options can be set and modified at any time during your work session.

Setting a Password

A password can provide a level of security to individual drawing files. DataCAD's password option requires that the password be entered correctly to open the drawing.

Warning: There is no way to circumvent this setting if you forget your password. Use a password you can easily remember or store a reminder of it in a safe place.

➔ To set a password for your drawing:

1. Go to the Utility menu in the Menu Window and click on Settings. The Settings menu is displayed.

2. Click on Password in the Settings menu. A small dialog box is displayed.
3. Enter a password, up to ten characters long, and press **Enter** or click OK. DataCAD prompts you to verify the password.
5. Type the same password again, and press **Enter** or click OK. The password is set. Make sure you save your drawing before closing it so that the new password is saved with the drawing file.
6. Repeat these steps if you need to change the password.

Setting Scale and Angle Types

You can change the scale and angle types your drawing uses. When you change the scale type, all associative dimensions in your drawing will be updated.

➔ To change the scale type:

1. Go to the Utility menu in the Menu Window and click on Settings.
2. Click on ScaleTyp in the Settings menu.
3. Click on the scale type you want: architectural, engineering, decimal, metric, inch fractions, decimal inches, meters, centimeters, millimeters, DIN standard, or AS1100 standard.
4. Choose how dimensions are displayed, using the Units, DoFloat, SigDigit, and StakFrac options. Toggle these on or off, as you like. See “ScaleTyp Menu Options” later in this section for details on these settings.
5. Right-click to exit the ScaleTyp menu when you’re finished.

➔ To change the angle type in your drawing:

1. Go to the Utility menu in the Menu Window and click on Settings.
2. Click on AngleTyp in the Settings menu.
3. Click on the angle type you want: normal, bearing, decimal degrees, radians, or gradients. See “AngleTyp Menu Options” later in this section for details on these settings.
4. Choose how angles are used. You can do this by using the Compass, ClkWise, and StartAng options.
5. Right-click to exit the AngleTyp menu when you’re finished.

4.

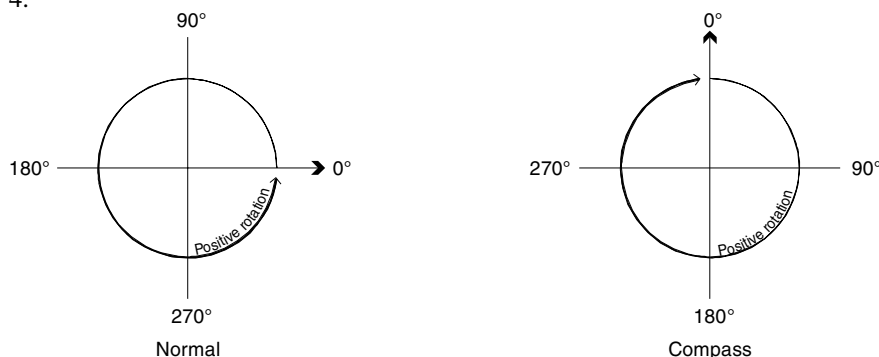


Figure 7.18: The Normal angle type as is (left) and with Compass applied (right).

ScaleTyp Menu Options

Arch	Uses and displays dimensions and coordinates in the architectural scale, or Imperial units (feet, inches, and fractions of inches); this is DataCAD's default scale type
Engr	Uses and displays dimensions and coordinates in the engineering scale (feet and decimal inches)
Decimal	Uses and displays dimensions and coordinates in decimal feet
Inch/Frc	Uses and displays dimensions and coordinates in inches and fractions of inches
Inch/Dec	Uses and displays dimensions and coordinates in decimal inches
Metric	Uses and displays dimensions and coordinates in meters, centimeters, and millimeters
Meters	Uses and displays dimensions and coordinates in decimal meters
CentiMtr	Uses and displays dimensions and coordinates in decimal centimeters
MilliMtr	Uses and displays dimensions and coordinates in decimal millimeters
DIN	Uses and displays dimensions and coordinates in the form <i>meters.centimeters</i> with superscripted millimeters
AS1100	Uses and displays dimensions and coordinates in millimeters with a space separating thousands, hundred thousands, millions and so on; input is done in meters
Units	Toggle on to use and display dimensions and coordinates with unit abbreviations
DoFloat,	Toggle on so that distance readouts use a floating decimal point; only available with the Decimal, Inch/Dec, Meters, CentiMtr, or MilliMtr scale types. When DoFloat is toggled off, the number of significant digits to display is displayed in the Message Area and the SigDigit option becomes available.
SigDigit,	Sets the number of decimal places, or significant digits (0 to 3), to display; only available with the Decimal, Inch/Dec, Meters, CentiMtr, or MilliMtr scale types, when DoFloat is toggled off
StakFrac	Toggle on so that fractions of an inch are displayed "stacked" (with the numerator over the denominator) rather than inline; available only with the Arch scale type. Some third-party fonts have improper character definitions, which may cause the fraction to overwrite the non-fractional portion of the dimension when using StakFrac. Should this occur, try using a different font.

AngleTyp Menu Options

Normal	Toggle on to set zero degrees equal to the positive X axis, a horizontal line to the right
Bearing	Toggle on to enter angles using bearings
DecDeg	Toggle on to enter angles in decimal degrees
Radians	Toggle on to enter angles in radians
Grads	Toggle on to enter angles in gradients
Compass	Sets zero degrees equal to the positive Y axis, a vertical line pointing north; this is not a toggle, but simply clicking on it activates it
ClkWise	Toggle on to increase the angle in a clockwise direction; toggle off to increase the angle in a counterclockwise direction. When you click on Compass, ClkWise is automatically toggled on.
StartAng	Sets zero degrees equal to two specified points

Editing Scale, Angle, and Distance Lists

With some functions, DataCAD will list scale, angle, or distance values for you to choose from. You can modify these scale, angle, and distance lists associated with your drawing. You can list the current values, add new values, change existing values, or delete values you don't use.

➔ To list scale, angle, or distance definitions:

1. Go to the Utility menu in the Menu Window and click on Settings. The Settings menu is displayed.
2. Click on EditDefs in the Settings menu. The EditDefs menu is displayed.
3. Click on the definition list you'd like to edit: Scales, Angles, or Distnces.
4. Click on List. The definitions are displayed in the Menu Window.

5. Right-click anywhere on the screen or press any key on your keyboard to exit the list.

➔ To add a value to a definition list:

1. Go to the Utility menu in the Menu Window and click on Settings. The Settings menu is displayed.
2. Click on EditDefs in the Settings menu. The EditDefs menu is displayed.
3. Click on the definition list you'd like to edit: Scales, Angles, or Distnces. You are limited to 18 definitions in a scale, distance or angle list. If you already have 18 in the list you want to add to, you must either delete or change an existing one.
4. Click on Add. If you're adding a new angle or distance, you are prompted to enter the new definition in the Message Area. If you're adding a new scale you are first prompted to "Enter a new string for this scale". Enter the text that will appear in the list and press **Enter** before continuing with the next step.
5. Type a new definition and press **Enter**. The new value is added to the definition list. You can save your customized scale, angle, or distance lists to a separate file (with a .SCL, .ANG. or .DIS file extension, respectively) and load them into DataCAD at any time for use in your drawings.

➔ To delete a value from a definition list:

1. Go to the Utility menu in the Menu Window and click on Settings. The Settings menu is displayed.
2. Click on EditDefs in the Settings menu. The EditDefs menu is displayed.
3. Click on the definition list you'd like to edit: Scales, Angles, or Distnces.
4. Click on Delete. You are prompted to select the definition to delete. User prompts are displayed in the Message Area.
5. Click on the definition in the list that you want to delete. The definition is deleted.

➔ To change a value in a definition list:

1. Go to the Utility menu in the Menu Window and click on Settings. The Settings menu is displayed.
2. Click on EditDefs in the Settings menu. The EditDefs menu is displayed.
3. Click on the list to be edited: Scales, Angles, or Distnces.
4. Click on Change. You are prompted to select the definition that you want to change.
5. Click on the definition you want to change. If you're changing an angle or distance, you are prompted to enter the new definition in the Message Area. If you're changing a scale you are prompted to "Enter a new string for this scale"; enter the text that will appear in the list and press **Enter** before continuing with the next step.
6. Enter a new definition to replace the existing one and press **Enter**.

➔ To save your customized definition list:

1. Go to the Utility menu in the Menu Window and click on Settings. The Settings menu is displayed.
2. Click on EditDefs in the Settings menu. The EditDefs menu is displayed.
3. Click on the definition list you'd like to edit: Scales, Angles, or Distnces.
4. Click on SaveFile. A dialog box is displayed.
5. Type a name for the customized list file, and click on Save or press **Enter**. The customized list is saved.

➔ To load a customized definition list for use in your drawing:

1. Go to the Utility menu in the Menu Window and click on Settings. The Settings menu is displayed.
2. Click on EditDefs in the Settings menu. The EditDefs menu is displayed.
3. Click on the definition list you'd like to edit: Scales, Angles, or Distnces.
4. Click on LoadFile. A dialog box is displayed.
5. Click on the name of the customized list file that you want to load, and click on Open or press **Enter**. The customized list is loaded into DataCAD.

More About Architectural Scales

To calculate additional scales, you can use DataCAD's online calculator. For example, to calculate the scale value for 1/4"=1'-0", type **=(1/4)/12** when you're prompted for a new scaling value. Or type **=.25/12** to get the same result: 0.02083333". For more information on the calculator, see "DataCAD's Online Calculator" in "The Drawing Board" chapter.

An architectural scale list example follows:

SCALE	DECIMAL FORM
1/32"=1'-0"	.00260416
1/16"=1'-0"	.00520833
3/32"=1'-0"	.00781250
1/8"=1'-0"	.01041666
3/16"=1'-0"	.01562500
1/4"=1'-0"	.02083333
3/8"=1'-0"	.03125000
1/2"=1'-0"	.04166666
3/4"=1'-0"	.06250000
1"=1'-0"	.08333333
1 1/2"=1'-0"	.12500000
2"=1'-0"	.16666666
3"=1'-0"	.25000000
6"=1'-0"	.50000000
12"=1'-0"	1.00000000
24"=1'-0"	2.00000000

More About Engineering Scales

To calculate additional scales, you can use DataCAD's online calculator. For example, to calculate the scale value for 1:20, type $=(1/20)/12$ and press when you're prompted for a new scaling value. The calculated value is .00416666. An engineering scale list example follows:

SCALE	DECIMAL FORM
1:10	.00833333
1:20	.00416666
1:33	.00250000
1:40	.00208333
1:50	.00166666
1:80	.00104166
1:100	.00083333
1:200	.00041666
1:400	.00020833
1:600	.00013888
1:1000	.00008333

More About Metric Scales

To calculate additional scales, simply type the ratio in decimal form. To use DataCAD's online calculator for this, type $=1/20$ and press to get the new scale value 0.05000000. For more information on the calculator, see "DataCAD's Online Calculator" in "The Drawing Board" chapter.

A metric scale list example follows:

SCALE	DECIMAL FORM
1:1	1.00000000
1:2	0.50000000
1:5	0.20000000
1:10	0.10000000
1:20	0.05000000
1:50	0.02000000
1:100	0.01000000
1:200	0.00500000
1:500	0.00200000
1:1000	0.00100000
1:2000	0.00050000
1:5000	0.00020000
1:10000	0.00010000

Setting Cursor Accuracy

Your cursor's miss distance is the length, in pixels, extending from the center of your cursor's crosshairs outward. The smaller the miss distance, the closer you'll have to place your cursor to an entity to select it, for example. This distance affects object snapping and other editing commands where you use your cursor to select a point or entity. The default miss distance is 10.

➔ To set your cursor's miss distance:

1. Go to the Utility menu in the Menu Window and click on Settings. The Settings menu is displayed.
2. Click on MissDist in the Settings menu. A value menu is displayed in the Menu Window. For more information on using value menus, see "Value Menus" in "The Drawing Board" chapter.

3. Choose a new distance value from the list or type a new value, and press **Enter**.

Setting Display Options

There are several display options that you can customize in DataCAD, including grid display, reference mark display, cursor size, and information displayed in the Message Area.

When you zoom out from your drawing, the grid spacing becomes smaller. The grid can become so small that it's difficult to work with and even distracting. You can use the SmalGrid option in the Settings menu to turn off the display grids when it reaches a specific pixel spacing. A good setting is usually between 15 and 20 pixels.

Shortcut: Press **PgDn** and **PgUp** to zoom in and out on your drawing,

➔ To set the smallest grid size at which the display grid will still display:

1. Go to the Utility menu in the Menu Window and click on Settings. The Settings menu is displayed.
2. Click on SmalGrid in the Settings menu. A value menu is displayed in the Menu Window. For more information on using value menus, see "Value Menus" in "The Drawing Board" chapter.
3. Choose a new value from the list or type a new value for the smallest grid size that will display, and press **Enter**.

Another display option is the Draw Marks toggle in the Settings menu. Toggle DrwMarks on to have DataCAD automatically draw temporary visual references for every point and entity you draw. These small Xs do not plot, and you can't snap to them. When you refresh the Drawing Area, the marks disappear.

Shortcut: Press **Esc** to refresh the display of your drawing in the Drawing Area.

You can also toggle the BigCursr option on in the Settings menu to display a large cursor that spans the Drawing Area; toggle it off to display a smaller cursor. When you toggle BigCursr off, you can specify the size of the cursor.

Shortcut: Press **+** to toggle between large and small cursors.

➔ To set the size of a small cursor:

1. Go to the Utility menu in the Menu Window and click on Settings. The Settings menu is displayed.
2. Click on BigCursr to toggle it on. If it's already toggled on, skip to step 3.
3. Click on BigCursr to toggle it off. A value menu is displayed in the Menu Window. For more information on value menus, see "Value Menus" in "The Drawing Board" chapter.
4. Use the value menu or type a new value for the cursor size, in pixels, and press **Enter**.

Two additional options in the Settings menu, NegDist and ShowZ, allow you to customize the Message Area display. To allow the display of negative coordinates in the Message Area, toggle NegDist on. Toggle ShowZ on to continuously display the current Z-base and Z-height settings in the Message Area.

Setting the Panning Distance

You can adjust the distance the screen scrolls when you use the arrow keys to pan around your drawing. For example, a setting of 50% scrolls 50% of the Drawing Area width each time you press an arrow key.

Shortcut: Press the arrow keys to pan around your drawing.

➔ To adjust the scroll distance:

1. Go to the Utility menu in the Menu Window and click on Settings. The Settings menu is displayed.
2. Click on ScrlDist in the Settings menu. A value menu is displayed in the Menu Window. For more information on value menus, see “Value Menus” in “The Drawing Board” chapter.
3. Choose a new scroll distance from the list or type a new value, and press .

Automatically Saving Your Drawing

DataCAD’s automatic save feature creates an autosave (.ASV) file that you can use to recover your drawing file should DataCAD unexpectedly close before you’re able to save the changes you’ve made to your drawing. The SaveDlay option in the Settings menu determines how often DataCAD automatically saves your entire drawing, updating the autosave (.ASV) file. When an autosave is in progress, the activity light in the Status Area next to SWOTHLUDFB turns red briefly, and other operations are momentarily interrupted. If there are a lot of entities on a layer, you may want to adjust this save delay to a higher setting to avoid frequent interruptions until you’re finished working on that layer.

The autosave function does not permanently save your drawing, and so your drawing (.DC5) file is not updated. See “More About Recovering Lost Work” in “The Drawing Board” chapter for information on how to use autosave files.

➔ To adjust the time interval between automatic saves:

1. Go to the Utility menu in the Menu Window and click on Settings. The Settings menu is displayed.
2. Click on SaveDlay in the Settings menu. A value menu is displayed in the Menu Window. To turn off the automatic save function, set SaveDlay to zero. This, however, is not recommended, as it leaves you no way to recover work done since you last saved your drawing (.DC5) file.
3. Use the value menu to choose a value for the time interval or type a new time value, and press .

Synchronizing the Distance Display

In DataCAD, you can rotate the grids along with your cursor to make it easier to draw at an angle. The DistSync option in the Settings menu is toggled on by default so that the coordinate display in the Message Area matches the *rotated* cursor position. If you are working with a rotated cursor but want to view the distance relative to standard X and Y coordinates, toggle DistSync off.

Customizing Display

You can use the Display menu options to toggle the text, dimensions, hatching, or certain drawing attributes on or off. Because the text, dimensions, and hatching that you add to your drawing are made up of many entities, you may want to toggle one or all of these options off to speed up the refresh time while you draw. Toggle these options on again before you plot the drawing.

➔ To customize the display of your drawing:

- 1. Go to the Utility menu in the Menu Window and click on Display. The Display menu is displayed.
- 3. Display all the text, associative dimensions, and hatch patterns that you’ve added to your drawing, or you can choose not to display them while you work to decrease drawing refresh times. Toggle ShowTxt on to display text; toggle it off to display text as boxes. Toggle ShowDim on to display associative dimensions; toggle it off to hide associative dimensions. Toggle ShowHtch on to display hatch patterns; toggle it off to hide hatch patterns.

Some DataCAD menu options are *toggles*, switches that can be turned on or off. When a toggle is on, the button appears depressed and the button name is green. When a toggle is off, the button appears raised and the button name is red.

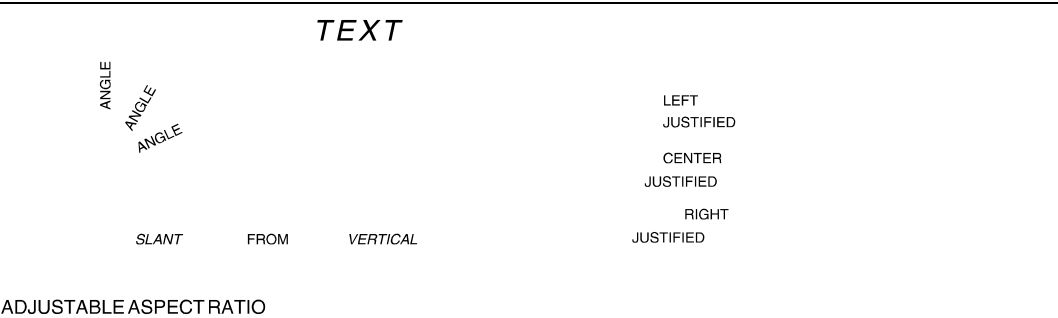


Figure 7.19: ShowTxt toggle on

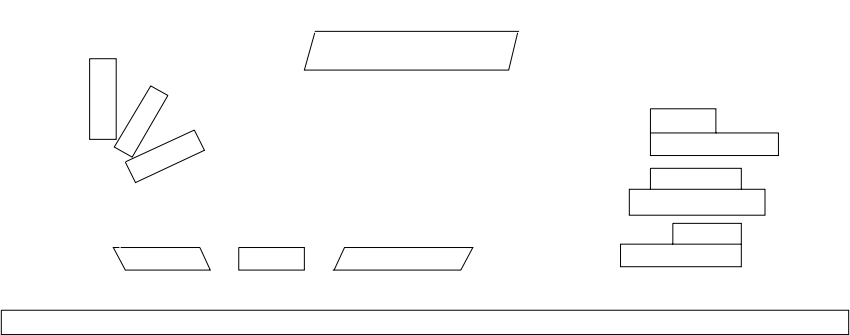
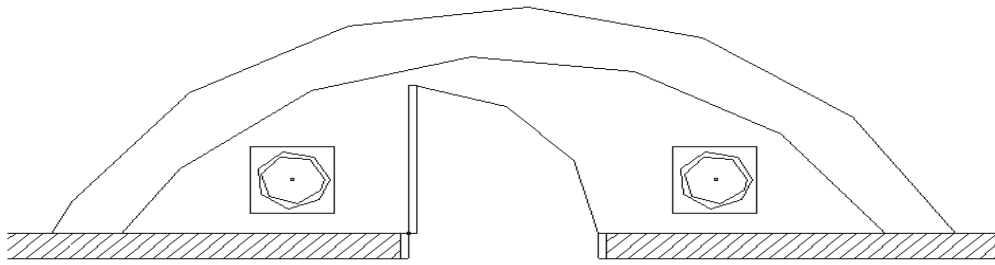


Figure 7.20: ShowTxt toggle off

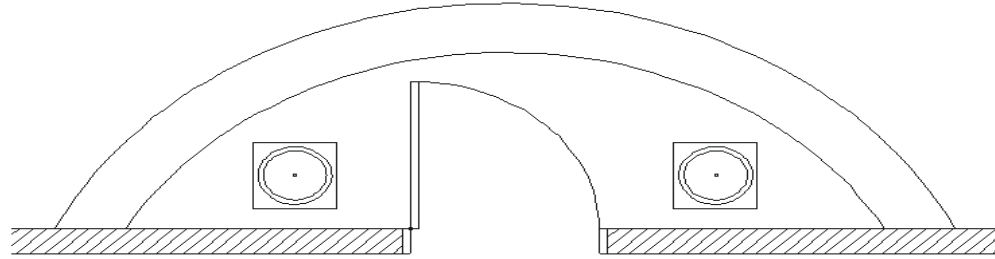
3. Look at the toggles that change how the lines in your drawing are displayed. Toggle ShowWgt on to display line weights you've applied to your drawing; toggle it off to display lines with a weight greater than 1 as single solid lines. You can also toggle on UserLine to display any user-defined linetypes you've used in your drawing. If UserLine is toggle off, user-defined lines, such as insulation, hedge, and shingles, appear as single solid lines. When UserLine is toggled off causing user-defined lines to display as solid, these lines are *not* changed in any way; they are simply represented on screen by a simple solid line. Toggle OverSht on to display line overshoots.
4. Change display settings for symbols as well. Toggle ShowIns on to display symbol insertion points; toggle it off to hide symbol insertion points. You can also click on SmallSym to set the smallest displayable symbol size. Any symbol smaller than the SmallSym setting will be displayed as a box. A setting of 0 displays all symbols, regardless of size. See "Value Menus" in "The Drawing Board" chapter for more information on using value menus.
5. Change settings for displaying text by clicking on SmallTxt or BoxColor. Similarly to SmallSym in step 4, SmallTxt sets the smallest text size displayable on screen; any text smaller than the SmallTxt setting is displayed as a box. A setting of 0 displays all text, regardless of size. BoxColor sets the color of the boxes that replace small text or that replace all text if ShowTxt is toggled off. See "Color Menus" in "The Drawing Board" chapter for more information on using color menus. You can select a global text box color or click on NoChange to display the text boxes in the same color as the text they replace.
6. Display control points of associative dimensions by toggling DimPoint on.
7. Display the center points of 2D curves by toggling CurveCtr on.
8. Display visible attributes added with a macro by toggling ShowAttr on.
9. Click on LyrOrder to assign the order in which to refresh drawing layers. Toggle First on to refresh the active layer first. Toggle Last on to refresh the active layer last. Toggle Inorder on to refresh the layers in the order you created them. Only one of these three options can be toggled on at any one time.

If Display List is on, DispList is also available in the Utility / Display menu. Click DispList; then click Status to see how many entities and line segments are in the Display List, as well as how much memory is used to maintain the list. For more information about Display List, see "Customizing Miscellaneous Settings" in "The Drawing Board" chapter.

10. Look at the ArcFactr option; it changes how arcs and curves are displayed. The higher the setting, the smoother curves are displayed. Use a lower setting while drawing to increase refresh speeds; then increase the setting when drawing is complete. (See Figure 7.21.)
11. Right-click to exit when you've finished changing Display menu settings.



Display with arc factor set to a value of 0.5



Display with arc factor set to a value of 5.0

Figure 7.21: Arc factor settings

Customizing Colors

You can use any customized palette by simply loading it in the Color Palette dialog box.

➔ To use a customized palette:

1. Click on Palettes in the Tools pull-down menu. The Color Palette dialog box is displayed.

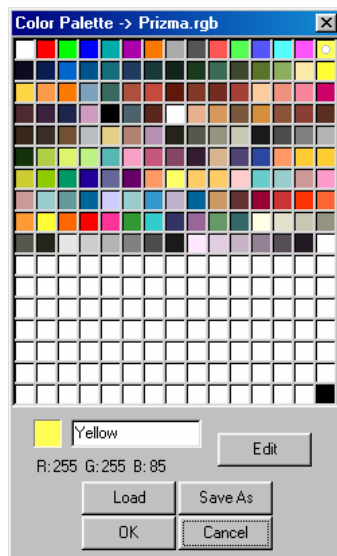


Figure 7.22: The Color Palette dialog box

2. Click on Load in the Color Palette dialog box. A dialog box is displayed, prompting you to “Enter name of the RGB palette to load”.

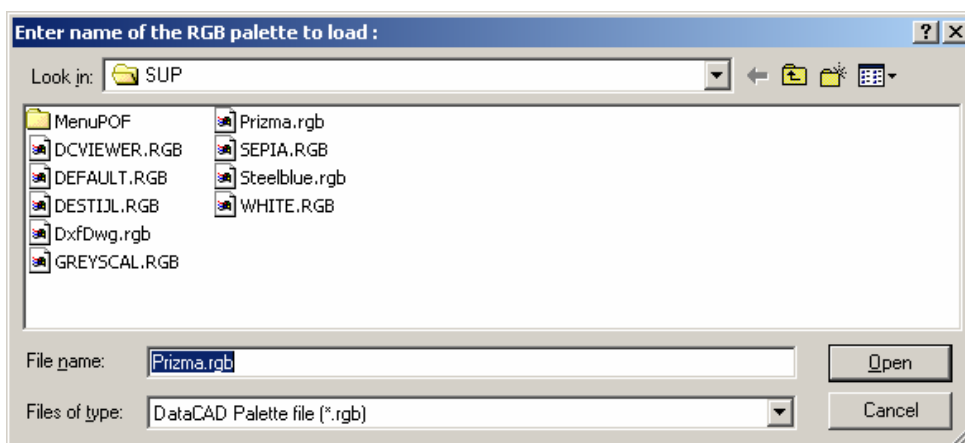


Figure 7.23: RGB palette name

3. Click on the name of the .RGB file you want to use, and click on Open. DataCAD will automatically use your new palette in subsequent drawing sessions.

➔ To mix your own custom color:

1. Click on Palettes in the Tools pull-down menu to display the Color Palette dialog box.
2. Click on the swatch you want to customize in the Color Palette dialog box.
3. Click on Edit to open the Color Picker dialog box.
4. Use your mouse in the RGB (red, green, blue) and/or HSB (hue, saturation, brightness) window to find the color you prefer. Since the original and new colors appear at the bottom of the Color Picker dialog box, it is easy to compare the two.

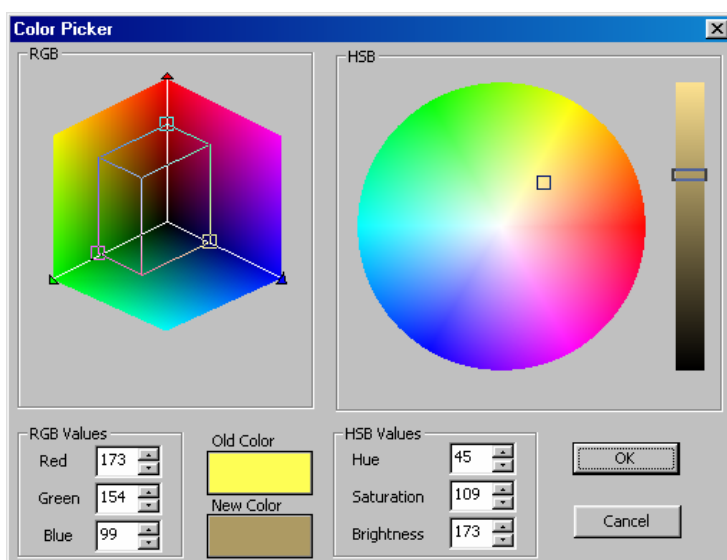


Figure 7.24: The Color Picker dialog box

5. Select OK when you are satisfied with the new color. The Color Palette reappears with your new color in the swatch.

6. Give the custom color a special name in the input box beside the color swatch.



Figure 7.25: Supply a name for the custom color

8. Click OK to exit the Color Palette dialog box.

Drawing Walls, Windows, & Doors

In this chapter:

The Architct menu in the Edit menu combines the three most basic drawing elements under a single menu. Architct includes options for drawing 2-line, 3-line, and 4-line walls as well as windows and six different door styles, all with a variety of customizable options.

With a single choice from the Architct menu, you can begin drawing walls whose intersections can be cleaned automatically. Windows, doors, and other wall openings can then be inserted with just a few clicks and, if necessary, removed just as easily.

- ⊕ Drawing 2-line, 3-line, and 4-line walls
- ⊕ Drawing and removing doors
- ⊕ Drawing and removing windows
- ⊕ Drawing and removing openings in walls

Drawing Walls

Walls can be drawn by simply toggling on the Walls option in the Architct menu. DataCAD automatically cleans wall intersections, including the last intersection where the first and last walls drawn meet.

You can draw 2-line walls using the 2LnWalls option, 3-line walls using the 3LnWalls option, or 4-line walls using the 4LnWalls option. Two line walls draw an inside and an outside line, 3-line walls add a centerline, and 4-line walls add two cavity walls. The 2LnWalls, 3LnWalls, and 4LnWalls toggles are mutually exclusive; when one is toggled on, the others are automatically toggled off.

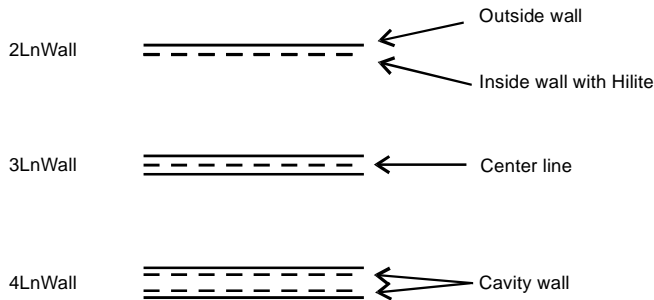


Figure 8.1: DataCAD's wall types

You can also use the Walls option in the DCAD_AEC macro to draw walls. This Walls option works the same way that Walls in the Architct menu does, although it does have fewer settings to customize wall display.

➔ To draw walls:

1. Make sure Ortho is toggled on. Walls can only be drawn in orthographic, or plan, view; if you're not already in plan view, click on Ortho in the Projection Pad or click Orthographic in the View pull-down menu.
2. Go to the Edit menu in the Menu Window and click on Architct.

Shortcut: Press ☐ to quickly toggle Walls on or off from anywhere in DataCAD.

3. Click on Walls in the Architct menu to toggle it on; Wall is checked in the Create pull-down menu, and you can now draw walls instead of lines.
4. Click on 2LnWalls to draw 2-line walls, click on 3LnWalls to draw 3-line walls, or click on 4LnWalls to draw 4-line walls.
5. Set the wall attributes using the Hilite option along with Width for 2-line walls, CntrLine for 3-line walls, and Exterior, Interior, and Cavity for 4-line walls. See "More About Setting Wall Attributes" later in this chapter.

6. Select two points to draw a wall. When you do this, the line connecting those two points becomes the outside wall, inside wall, center of the wall, or center of the wall cavity, depending on whether Outside, Inside, CntrWall, or CntrCvty is toggled on, respectively. Click on Outside to define walls by the outside of the wall, click on Inside to define them by the inside of the wall, click on CntrWall to define them by the center of the wall, or click on CntrCvty to define them by the center of the wall cavity. When drawing 2-line walls or 3-line walls, CntrWall and CntrCvty work the same way. You can choose either option to create walls based on the wall center.
7. Click on Clean to automatically clean wall intersections as you draw your walls; click on Cap to cap wall ends as you draw your walls. You are prompted to "Select first end point of new line/wall". For more information on using Clean and Cap, see "More About Finishing Walls" later in this chapter.
8. Click anywhere in the Drawing Area or use coordinate entry to start drawing your walls. You are prompted to "Select next end point of new line/wall".
9. Move the cursor using your mouse and click again or use coordinate entry to specify the end of the first wall.
10. Click anywhere on the inside or outside of the wall, depending on what you did in step 6. If you chose Outside, CntrWall, or CntrCvty in step 6, click anywhere on the inside of the wall; if you chose Inside in step 6, click anywhere on the outside the wall.
11. Notice that your cursor is still connected to the end of your first wall. You can continue selecting wall end points, as you did in step 9; all intersections will be automatically cleaned.
12. Finish drawing walls by right-clicking to disconnect your cursor from the wall, or clicking near the first endpoint of your first wall to close your wall plan. Your cursor is automatically disconnected.

Sometimes, you may need to fix wall intersections that didn't clean properly when they were drawn. Use the Cleanup menu's TIntsct, LIntsct, and XIntsct options to clean wall intersections. Use the Cleanup menu's WeldLine and WeldWall options to mend openings in walls. See "Cleaning Wall Intersections" and "Welding Lines and Walls" in the "Editing Drawings" chapter.

More About Setting the Width of Walls

When toggling on the Walls option in the Architct menu, you are prompted for a new wall width. A *wall width* is the distance between the inside and outside wall. You can also change wall width anytime while drawing 2-line or 3-line walls.

➔ To change the width of walls to be drawn:

1. Go to the Edit menu in the Menu Window and click on Architct.
2. Click on Width in the Architct menu. A value menu is displayed in the Menu Window. The Width option is not available when 4LnWalls is toggled on

3. Use the value menu to enter a new width, or type a new width and press **[Enter]**. DataCAD sets the new wall thickness and subsequent walls will be drawn with that thickness. For more information on using value menus, see “Value Menus” in “The Drawing Board” chapter.

More About Setting Wall Attributes

You can assign different attributes to outside or inside walls, including colors, linetypes, and line weights by using the Hilite option in the Architct menu. Other attribute options apply only to 3-line or 4-line walls.

- ➔ To draw walls with either the outside or inside wall in a different color, linetype, or line weight:
 1. Go to the Edit menu in the Menu Window and click on Architct.
 2. Click on Hilite in the Architct menu. The Hilite menu is displayed in the Menu Window. You can only set Hilite options for either the inside wall or the outside wall; for example, you can't assign a color to the outside wall and a linetype to the inside wall.
 3. Click on Outside in the Hilite menu to highlight the outside of your walls. Click on Inside to highlight the inside of your walls.
 4. Click on Color to choose a highlight color, click on LineWgt to choose a line weight to highlight your walls with, or click on LineType to choose a linetype as a highlight.
 5. Right-click to return to the Architct menu.
 6. Begin drawing your walls. Notice that the color, linetype, or line weight attributes you set are applied as walls are drawn.
- ➔ To set attributes for the center line when drawing 3-line walls:
 1. Go to the Edit menu in the Menu Window and click on Architct.
 2. Click on CntrLine in the Architct menu. The CntrLine menu is displayed in the Menu Window. The CntrLine option is only displayed when 3LnWalls is toggled on.
 3. Click on Color to choose a highlight color, click on LineWgt to choose a line weight to highlight your walls with, click on LineType to choose a linetype as a highlight, or click on Spacing to set the spacing for the center line. The Spacing setting lengthens both the lines and the spaces in a linetype, so that the aspect ratio of the linetype is preserved.
- ➔ To set the distance from the exterior wall to the exterior cavity wall when drawing 4-line walls:
 1. Go to the Edit menu in the Menu Window and click on Architct.
 2. Click on Exterior in the Architct menu. A list of width values is displayed in the Menu Window.
 3. Use the value menu, or type a new value and press **[Enter]** to set the exterior wall thickness. For more information about how to use value menus, see “Value Menus” in “The Drawing Board” chapter.

- ➔ To set the distance from the interior wall to the interior cavity wall when drawing 4-line walls:
1. Click on **Architct** in the Edit menu.
 2. Click on **Interior** in the **Architct** menu. A list of width values is displayed in the Menu Window.
 3. Use the value menu, or type a new value and press **Enter** to set the interior wall thickness.
- ➔ To set the attributes for the cavity wall when drawing 4-line walls:
1. Click on **Architct** in the Edit menu.
 2. Click on **Cavity** in the **Architct** menu. The **Cavity** menu is displayed in the Menu Window.
 3. Click on **Width** to set the width from the exterior cavity wall to the interior cavity wall; click on **Color** to choose a color for cavity walls; click on **LineWgt** to set the line weight of cavity walls; click on **LineType** to set the linetype for cavity walls; click on **Spacing** to set the spacing for cavity walls.

More About Finishing Walls

As you saw in the tutorial in this manual, DataCAD always cleans the intersections of walls as you draw them, as long as you don't disconnect your cursor from the last wall you drew. For T intersections, you can toggle on **Clean** in the **Architct** menu to automatically trim the interior wall lines as you draw the walls so that your walls look continuous, without internal lines at corners.

Sometimes you may need to fix wall intersections that didn't clean properly when they were drawn. Use the **Cleanup** menu's **TIntsct**, **LIntsct**, and **XIntsct** options to clean wall intersections. Use the **Cleanup** menu's **WeldLine** and **WeldWall** options to mend openings in walls.

Shortcut: Press **↵** to toggle **Clean** on or off.

The **Cap** option in the **Architct** menu can be toggled on to insert a wall cap or closure on the end of a wall. When you draw walls, wall ends are automatically capped instead of remaining open.

The **Clean** and **Cap** toggles are mutually exclusive; when one is toggled on, the other is automatically toggled off.

Using Wall Styles

You can save wall styles for 2-line, 3-line, or 4-line walls with the wall attributes you use most often, and then use them in any drawing file. The following attributes are saved in a wall style:

- line color, linetype, line spacing, and line weight of wall lines. The **Spacing** setting lengthens both the lines and the spaces in a linetype, so that the aspect ratio of the linetype is preserved.
- whether you want to highlight the inside or outside line of your walls using color, line weight, and linetype.

- wall widths and center line and cavity wall attributes.

➔ To save a new wall style:

1. Go to the Edit menu in the Menu Window and click on Architct.
2. Click on WallStyl in the Architct menu. The Wall Style Manager dialog box is displayed.
3. Click on SaveAs in the Wall Styles Available section at the bottom of the dialog box. Another small dialog box is displayed, prompting you to enter a new wall style name.
4. Type a name for your new wall style, and press **[Enter]** or click on OK. The wall style is saved and now appears in the drop-down list in the bottom left corner of the dialog box.
5. Click on OK to close the dialog box.

➔ To save changes to a wall style:

1. Go to the Edit menu in the Menu Window and click on Architct.
2. Click on WallStyl in the Architct menu. The Wall Style Manager dialog box is displayed.
3. Using the drop-down list in the Wall Styles Available section at the bottom of the dialog box, select the wall style you want to save changes to.
4. Change wall settings as necessary.
5. Click on Save in the Wall Styles Available section at the bottom of the dialog box. The wall style is saved with any changes you made to wall settings.

➔ To use a wall style:

1. Go to the Edit menu in the Menu Window and click on Architct.
2. Click on WallStyl in the Architct menu. The Wall Style Manager dialog box is displayed.
3. Use the drop-down list in the Wall Styles Available section at the bottom of the dialog box to select the wall style you want to use. Wall settings in the dialog box are changed to those of the wall style you selected.
4. Click on OK to close the dialog box.

➔ To delete a wall style:

1. Go to the Edit menu in the Menu Window and click on Architct.
2. Click on WallStyl in the Architct menu. The Wall Style Manager dialog box is displayed.
3. Use the drop-down list in the Wall Styles Available section at the bottom of the dialog box to select the wall style you want to delete.
4. Click on Delete. The selected wall style is deleted.
5. Click on OK to close the dialog box.

➔ To display the settings for a wall style:

1. Go to the Edit menu in the Menu Window and click on Architct.
2. Click on WallStyl in the Architct menu. The Wall Style Manager dialog box is displayed.
3. Use the drop-down list in the Wall Styles Available section at the bottom of the dialog box to select the wall style whose settings you want to view. Wall settings in the dialog box are changed to those of the wall style you selected.
4. Click on OK to use those wall style settings and close the dialog box, or click on Cancel to close the dialog box without changing wall settings.

Cutting Walls

You can use CutWall in the Architct menu to remove sections of walls.

➔ To cut openings in walls using the mouse:

1. Go to the Edit menu in the Menu Window and click on Architct.
2. Click on CutWall in the Architct menu. The CutWall menu is displayed.
3. Determine which layer your walls are on. If your walls are on a layer other than the active layer, click on LyrSrch to toggle it on. If LyrSrch is already on, toggle it off and then back on again to choose your walls layer. The layers in your drawing are listed in the Menu Window.
4. Click on the layer your walls are drawn on.
5. Select one side of the opening by clicking on a point on the wall or using coordinate entry to specify the exact coordinates. To choose a coordinate entry method, use the Input Mode option in the Utility pull-down menu.
6. To select the other side of the opening, move the mouse along the wall and click on any point or use coordinate entry. The wall is cut and the wall ends are capped automatically.

If the wall opening is a result of adding a door or window when Cutout was toggled off, you must use Erase in the Edit menu to erase the framed opening. You will then need to use WeldWall in the Edit/Cleanup menu to weld the wall.

➔ To remove an opening from your drawing:

1. Go to the Edit menu in the Menu Window and click on Architct.
2. Click on CutWall in the Architct menu. The CutWall menu is displayed.
3. Determine which layer contains your walls. If your walls are on a layer other than the active layer, click on LyrSrch to toggle it on. The layers in your drawing are listed in the Menu Window.
4. Click on Remove in the CutWall menu.
5. Click to one side of the opening and move your mouse diagonally to stretch the rubberband box around it, completely enclosing all elements of the opening. You can stretch the box around multiple openings to remove them all at once.

6. Click to complete the box. DataCAD removes the opening and welds the wall. If any remnant of the opening remains, click **[Esc]** to refresh the drawing.

Drawing Doors

Not only can you draw walls using the **Architct** menu, you can also use it to draw doors, complete with door swings, in a few quick steps. DataCAD places a jamb into the wall, erases the wall between the jambs, draws the door at the specified thickness, and draws the door swing at the specified direction and angle.

You can draw doors by selecting the sides of the door or by selecting the center and strike side of the door. When toggled on, the **Sides** option lets you draw doors by indicating the door sides (doorjambs). When **Sides** is toggled off, doors are drawn by indicating their center and strike side.

If **LyrSrch** is not toggled on or you have not selected the layer your walls are on, you may see the error message “No walls found to cut. Please check search layer”. Toggle **LyrSrch** on (if it’s already on, toggle it off and then on again) to display the layers in your drawing. Click on the layer your walls are drawn on in the list in the **Menu Window**.

➔ To draw a door using the mouse:

1. Go to the **Edit** menu in the **Menu Window** and click on **Architct**.
2. Click on **DoorSwng** in the **Architct** menu. The **DoorSwng** menu is displayed. You can also access the **DoorSwng** menu in the **Edit/ Architct** menu using the **Doors** option in the **DCAD_AEC** macro.
3. Set the door jamb, angle, style, thickness, head height, and swing style as necessary. See the following pages for more information on setting these door options.
4. Look at the **Sides** toggle switch. If **Sides** is toggled on, you are prompted to “Select hinge side of door.” If **Sides** is toggled off, you are prompted to “Select center of door.” To enter the point you’re prompted for, click on a wall or use coordinate entry. You are prompted to “Select strike side of door.”
5. Click on the wall or use coordinate entry to enter the strike side of the door. You are prompted to “Select direction of door swing.” See “More About Coordinate Entry” in “The Drawing Board” chapter.
6. Click either inside or outside the wall, to indicate which direction the door should swing. You are prompted to “Select any point on the outside of the wall.”
7. Click anywhere outside the wall. The door and doorjamb are drawn.

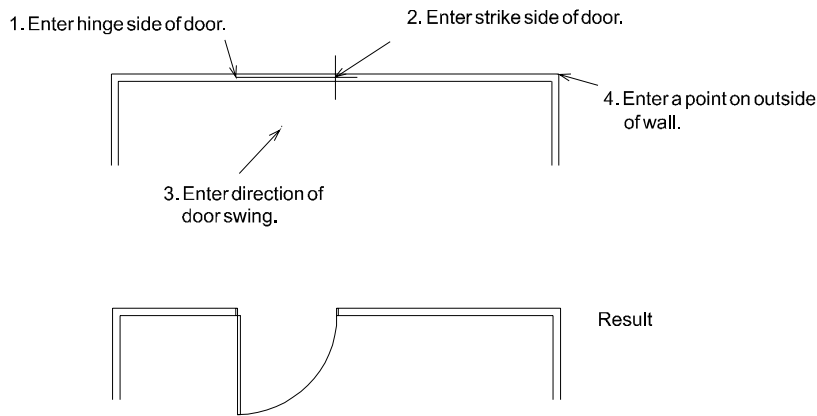


Figure 8.2: Drawing a door

More About Door Styles

The DoorStyl menu allows you to choose from six door styles: single, double, bi-fold, sliding, pocket, and overhead. See Figure 8.3 for examples of door styles. Single creates single-leaf, hinged doors of any size. This is the default setting for the DoorStyl menu. Click on Double to draw two opposing doors hung on a single frame. Bi-fold creates simplified folding doors, while Sliding creates 2-panel sliding doors, and Pocket creates pocket doors. Click on Overhead to create overhead garage doors. A dotted line representing the overhead door in a raised position will be drawn, with the dimensions equaling the width and head height of the door.

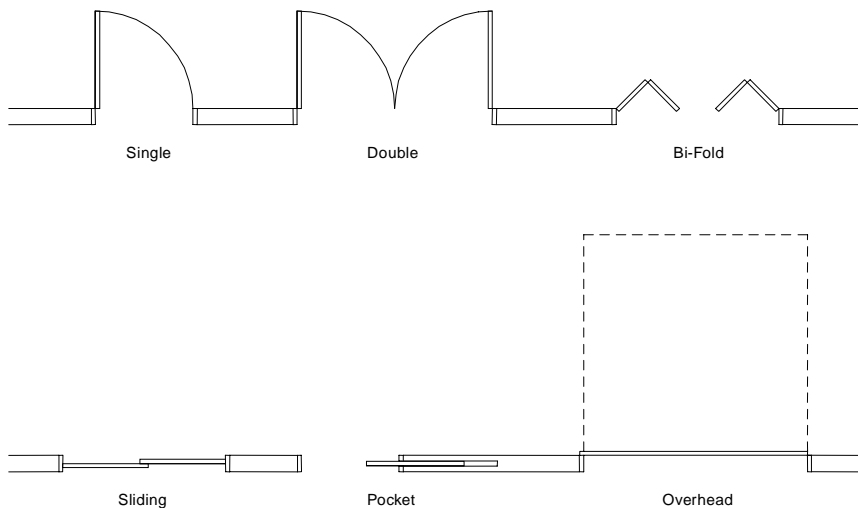


Figure 8.3: Door styles available in the DoorStyl menu

More About Door Jambs

Some of the DoorSwng menu options affect how the door is drawn in the wall. Click on DrawJamb, InWall, or Cutout to toggle the option on. The DrawJamb option draws door jambs for each wall opening, while InWall precisely centers door openings between the two wall faces, and Cutout removes the wall segment between the two jambs. An added benefit to drawing doors with Cutout toggled on is that the walls will automatically weld themselves if you use Remove to erase the door. You may need to use LyrSrch with Cutout if you're drawing doors on a different layer than your walls are on; click on LyrSrch and then click on the layer your walls are on.

JambIn is a toggle that determines whether the door will be drawn by rough opening or finished opening. *Rough opening* means that the door jamb is drawn inside the opening specified; *finished opening* means the door jamb will be drawn outside the opening specified. By default, JambIn is toggled off, meaning the door will be drawn by finished opening. When JambIn is toggled on, DataCAD draws the door by rough opening.

To explain further, when using Sides to draw a door with JambIn off, the two points you select mark the inside of the finished opening; therefore, the jamb will be drawn outside the two points specified. If you toggle JambIn on and draw another door, the two points you select mark the rough opening in the wall, and the jamb is drawn inside the two points.

You can also change the jamb width for your doors.

➔ To set a new jamb width:

1. Go to the Edit menu in the Menu Window and click on Architct.
2. Click on DoorSwng in the Architct menu. The DoorSwng menu is displayed.
3. Click on JambWdth in the DoorSwng menu.
4. Choose or type a jamb width value and press .

MtchWall in the DoorSwng menu is toggled on by default, causing jamb depth to automatically match the wall width. When MtchWall is toggled off, the JambDpth option becomes available; you can use JambDpth to set the jamb depth independently of the wall width. JambDpth is only available from the menu when MtchWall is toggled off.

More About Door Angles

Click on Angle in the DoorSwng menu to determine the door opening. An angle set to 0-0' will result in the doors being drawn closed, while a 90-0' angle will result in the doors being drawn fully open. For bi-fold doors, a 90° angle creates a right angle at the intersection of each door. When an angle between 0-0' and 90-0' is entered, DataCAD will draw single and double doors open to varying degrees.

DataCAD will draw sliding and pocket doors open to varying degrees when an angle between 10-0' and 90-0' is chosen. When any angle 90-0' or greater is chosen, sliding and pocket doors will be drawn fully open. When fully open, sliding doors will be drawn to one side and at the first point chosen for the door opening, making the opening at the second point. When doors are drawn closed, the door overlap is equal to double the door thickness. The Angle setting does not affect Overhead doors; they are always drawn the same, regardless of angle.

➔ To set the angle of the door opening:

1. Go to the Edit menu in the Menu Window and click on Architct.
2. Click on DoorSwng in the Architct menu. The DoorSwng menu is displayed.
3. Click on Angle in the DoorSwng menu.
4. Use the value menu to enter a new angle or type an angle value and press **[Enter]**. An angle of 0-0' draws doors closed. An angle of 90-0' draws most doors fully open. See "Value Menus" in "The Drawing Board" chapter for more information on how to use them.

More About Door Swing

The SwngStyl option in the DoorStyl menu lets you choose the type and color of the door swing. You can choose to draw either a line or arc swing style or to have no door swing drawn at all. The Line option draws a straight line from the corner of the doorjamb to the corner of the open door to represent the door swing. The Arc option draws a curve from the corner of the jamb to the corner of the door.

With both Line and Arc, you can toggle MtchDoor on in the SwngStyl menu to match the door swing color to the door color. This option is toggled on by default. When MtchDoor is toggled off, the SwngColr option is available in the menu. Click on SwngColr to choose a color for the door swing, independent from the door color.

More About Other Door Settings

Regardless of which way you draw a door, you can use CntrPnt to add a point in the center of the door. Then you can use the automatic dimensioning feature to find and dimension to this center point. See "Drawing Linear Dimensions" in the "Dimensions" chapter.

The DrwMarks option in the DoorSwng menu is another helpful, visual drawing aid. With DrwMarks toggled on, DataCAD draws small crosses on the screen where you've selected points in the Drawing Area. These marks are a temporary visual reference only; they disappear when you press **[Esc]** to refresh the screen. They don't plot, and you can't object snap to them.

This DrwMarks option is independent of DrwMarks in the Utility/Settings menu and affects only doors and windows. See "Setting Display Options" in the "Before You Draw" chapter for more information on the Settings menu's DrwMarks option.

The DoorSwng menu also has options to set the door thickness and head height.

➔ To set the door thickness:

1. Go to the Edit menu in the Menu Window and click on Architct.
2. Click on DoorSwng in the Architct menu. The DoorSwng menu is displayed.
3. Click on Thicknss in the DoorSwng menu. A value menu is displayed in the Menu Window.
4. Use the value menu to enter a door thickness, or type a thickness value and press **[Enter]**. See “Value Menus” in “The Drawing Board” chapter for more information on how to use them.

➔ To set the door head height:

1. Go to the Edit menu in the Menu Window and click on Architct.
2. Click on DoorSwng in the Architct menu. The DoorSwng menu is displayed.
3. Click on HeadHgt in the DoorSwng menu. A value menu is displayed in the Menu Window.
4. Use the value menu to enter a door thickness, or type a height value and press **[Enter]**.

Removing a Door

You can quickly remove any door from your drawing and automatically weld the wall with the Remove option in the DoorSwng menu. Keep in mind, however, that Remove only erases the door if Cutout was toggled on in the DoorSwng menu when the door was drawn. If you didn't use Cutout when you drew the door, use Erase to erase the door as a group. You will then need to use WeldWall in the Edit/Cleanup menu to weld the wall. See “Erasing Entities” and “Welding Lines or Walls” in the “Editing Drawings” chapter.

➔ To remove a door from your drawing:

1. Go to the Edit menu in the Menu Window and click on Architct.
2. Click on DoorSwng in the Architct menu. The DoorSwng menu is displayed.
3. Click on Remove in the DoorSwng menu. You are prompted to “Select first point of box around door or window to remove.”
4. Click above and to the left of the door to be removed.
5. Move your mouse diagonally, completely enclosing all elements of the door, and click. DataCAD removes the door. You can stretch the box around multiple doors to remove them all at once.

Drawing Windows

The Windows option draws windows with cased openings in walls, either by selecting the sides of the window or by selecting the center and strike side of the window. You can customize the detail of the windows by changing basic window components, including jamb size, sill projection, glass thickness, sill height, and head height.

➔ To draw a window:

1. Go to the Edit menu in the Menu Window and click on Architct.
2. Click on Windows in the Architct menu. The Windows menu is displayed in the Menu Window.
3. Set the window jamb, exterior and interior sill projections, thickness of glass, and sill height and head height as necessary. See the following pages for more information on setting window options.
4. Toggle Sides on to draw windows by selecting the window sides (jamb); toggle Sides off to draw windows by selecting their center point and one side. If Sides is toggled on, you are prompted to “Select one jamb of window.” If Sides is toggled off, you are prompted to “Select center of window.”
5. Click on a wall to enter one side of the window jamb or the center of the window. You are prompted to “Select second jamb of window” or “Select one jamb of window”, depending on the status of the Sides toggle.

If LyrSrch is not toggled on or you have not selected the layer your walls are on, you may encounter the error message “No walls found to cut. Please check search layer”. Toggle LyrSrch on (if it’s already on, toggle it off and then on again) to display the layers in your drawing. Choose the layer your walls are drawn on from the list in the Menu Window.

6. Click on the wall to enter the side of the window jamb. You are prompted to “Select any point on the outside of the wall.”
7. Click outside the wall. The window is drawn.

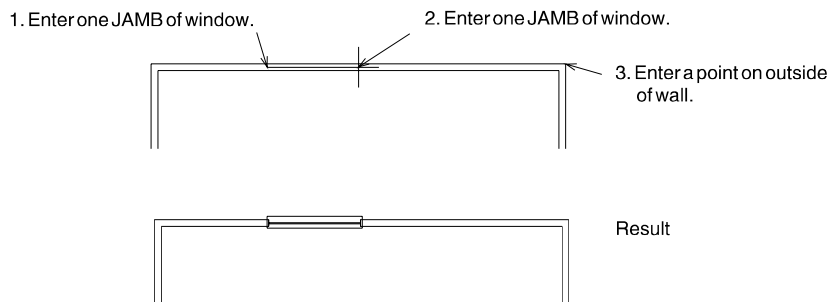


Figure 8.4: Drawing a window with Sides toggled on

More About Window Jambs

Some of the Windows menu's options affect how the window is drawn in the wall. The DrawJamb option draws window jambs for each wall opening, while InWall precisely centers window openings between the two wall faces, and Cutout removes the wall segment between the two jambs. Click on DrawJamb, InWall, or Cutout to toggle the option on. An added benefit to drawing windows with Cutout toggled on is that the walls will automatically weld themselves if you use Remove to erase the window. You may need to use LyrSrch with this if you're drawing windows on a different layer than your walls are on; click on LyrSrch and then click on the layer your walls are on.

JambIn is a toggle that determines whether the window will be drawn by rough opening or finished opening. *Rough opening* means that the window jamb is drawn inside the opening specified; *finished opening* refers to the window jamb being drawn outside the opening specified. By default, JambIn is toggled off, meaning the window will be drawn by finished opening. When JambIn is toggled on, DataCAD draws the window by rough opening.

To explain further, when using Sides to draw a window with JambIn off, the two points you select mark the inside of the finished opening, and the jamb will be drawn outside the two points specified. If you toggle JambIn on and draw another window, the two points you select mark the rough opening in the wall; the jamb is drawn inside the two points.

MtchWall is toggled on by default, causing the jamb depth of the window to automatically match the wall width. When this option is toggled off, you can use JambDpth to set the jamb depth independently of the wall width. JambDpth is only available from the menu when MtchWall is toggled off.

You can also change the jamb width for your windows.

➔ To set a new jamb width:

1. Go to the Edit menu in the Menu Window and click on Architct.
2. Click on Windows in the Architct menu. The Windows menu is displayed in the Menu Window.
3. Click on JambWdth in the Windows menu.
4. Use the value menu to enter a jamb width, or type a jamb width value and press .

More About Sill and Glass Settings

Other Windows options you'll want to set include distances for the exterior and interior sill projections, the thickness of the glass, and the sill height and head height.

➔ To change the distance of the exterior sill projection:

1. Go to the Edit menu in the Menu Window and click on Architct.
2. Click on Windows in the Architct menu. The Windows menu is displayed in the Menu Window.

3. Click on OutSill in the Windows menu. A value menu is displayed in the Menu Window.
 4. Use the value menu to enter a new exterior sill projection, or type a sill projection value and press **[Enter]**.
- ➔ To change the distance of the interior sill projection:
1. Go to the Edit menu in the Menu Window and click on Architct.
 2. Click on Windows in the Architct menu. The Windows menu is displayed in the Menu Window.
 3. Click on InSill in the Windows menu. A value menu is displayed in the Menu Window.
 4. Use the value menu to enter a new interior sill projection, or type a sill projection value and press **[Enter]**.
- ➔ To change the thickness of the glass:
1. Go to the Edit menu in the Menu Window and click on Architct.
 2. Click on Windows in the Architct menu. The Windows menu is displayed in the Menu Window.
 3. Click on GlassThk in the Windows menu. A value menu is displayed in the Menu Window.
 4. Use the value menu to enter a new thickness, or type a new value and press **[Enter]**. If you enter a thickness of 0", DataCAD draws a single sheet of glass.
- ➔ To enter a new window sill height relative to the current Z-base elevation:
1. Go to the Edit menu in the Menu Window and click on Architct.
 2. Click on Windows in the Architct menu. The Windows menu is displayed in the Menu Window.
 3. Click on SillHgt in the Windows menu. A value menu is displayed in the Menu Window.
 4. Use the value menu to enter a new exterior sill projection, or type a new height value and press **[Enter]**.
- ➔ To enter a new window head height relative to the current Z-base elevation:
1. Go to the Edit menu in the Menu Window and click on Architct.
 2. Click on Windows in the Architct menu. The Windows menu is displayed in the Menu Window.
 3. Click on HeadHgt in the Windows menu. A value menu is displayed in the Menu Window.
 4. Use the value menu to enter a new exterior sill projection, or type a new height value and press **[Enter]**.

More About Other Window Settings

Regardless of which way you draw a window, you can use CntrPnt to add a point in the center of the window. You can use DataCAD's automatic dimensioning feature to find and dimension to this center point.

The DrwMarks option in the Windows menu is another helpful, visual drawing aid. With DrwMarks toggled on, DataCAD draws small crosses on the screen when you select points in the Drawing Area. This DrwMarks option is independent of DrwMarks in the Utility/Settings menu and affects only doors and windows. These marks are a temporary visual reference only; they disappear when you press **[Esc]** to refresh the screen. They don't plot, and you can't object snap to them.

Removing a Window

You can quickly remove any window from your drawing and automatically weld the wall with the Remove option in the Windows menu. Keep in mind, however, that Remove only erases the window if Cutout was toggled on in the Windows menu when the window was drawn. If you didn't use Cutout when you drew the window, use Erase to erase the window as a group. You will then need to use WeldWall in the Edit/Cleanup menu to weld the wall.

➔ To remove a window from your drawing:

1. Go to the Edit menu in the Menu Window and click on Architct.
2. Click on Windows in the Architct menu. The Windows menu is displayed in the Menu Window.
3. Click on Remove in the Windows menu. You are prompted to "Select first point of box around door or window to remove."
4. Click above and to the left of the window to be removed.
5. Move your mouse diagonally, completely enclosing all elements of the window, and click. DataCAD removes the window. You can stretch the box around multiple windows to remove them all at once.

Drawing Linear Bays of Windows

With the Windows menu in the DCAD_AEC macro, you can draw 2D linear bays of windows. Click OneWindw in the Windows menu in the DCAD_AEC macro to access the Window menu in the Edit/Architct menu.

➔ To draw a linear bay of windows:

1. Click DCAD_AEC in the Macros pull-down menu. If DCAD_AEC is not listed in your Macros menu, click Configure in the Macros menu and add DCAD_AEC to the Macros in Menu list.
2. Click Windows in the DCAD_AEC menu. The Windows menu is displayed. To model 3D windows, use the Windows menu in the AEC_Modl macro.
3. Click HeadHght in the Windows menu. A value menu is displayed, and you are prompted to "Enter new window header height" in the Message Area.

4. Type a new header height or use the value menu to enter a new height.
5. Press **[Enter]** or right-click to accept the new value and exit the menu. The Windows menu is displayed again.
6. Click SillHght in the Windows menu. A value menu is displayed, and you are prompted to “Enter new window sill height” in the Message Area.
7. Type a new sill height or use the value menu to enter a new height.
8. Press **[Enter]** or right-click to accept the new value and exit the menu. The Windows menu is displayed again.
9. Click LinrWndw in the Windows menu to display the linear windows menu.
10. Decide if you want to draw a set quantity of equally spaced bays, as many bays of a set size that can be fit into a space, or a set number and size of bays. In the Linear menu, click Eq Bays to draw equally spaced bays, click FitBays to fit bays into a space, or click SetBays to specify the number and size of the bays. When using the SetBays option, the opening length is determined by the number and size of the bays, as you specify.
11. Set bay and window options as necessary. See “Linear Window Bay Options” on the following pages for more information. You are prompted to “Enter starting point of window wall”.
12. Use the mouse or coordinate entry to select a starting point for your bay of windows. You are prompted to “Enter ending point of window wall”. For more information on using coordinate entry, see “Drawing Using Coordinate Entry” in “The Drawing Board” chapter.
13. Use the mouse or coordinate entry to select an ending point for your bay of windows. You are prompted to “Enter a point on the outside of the wall”.
14. Click anywhere outside the wall. The bays of windows are drawn.

Linear Window Bay Options

Bay Qty	Set the number of window bays you want drawn; not available when using FitBays because bay quantity is automatically calculated
Bay Size	Set the width of each window bay, which will be multiplied to create the window wall; not available when using Eq Bays because individual bay size is automatically calculated
WallWdth	Set the width of the wall, which in turn determines the depth of the sills; not available when Cutout is toggled on because the wall width, and therefore the sill depth, is calculated automatically
Cutout	Toggle on to automatically calculate the wall width and cut an opening; of the three options Cutout, CntrLin, and InnrEdg, only one can be toggled on at any given time. When you use Cutout, the window wall opening will be cut from Z-base to Z-height (from the bottom of the wall to the top), just as it is with the Cutout option in the Architct menu.
CntrLin	Toggle on to center the depth of the wall along the starting and ending points you enter; of the three options Cutout, CntrLin, and InnrEdg, only one can be toggled on at any given time
InnrEdg	Toggle on to draw the inside face of the wall along the starting and ending points you enter; of the three options Cutout, CntrLin, and InnrEdg, only one can be toggled on at any given time

By Ends	Toggle on to draw a window wall by selecting the starting and ending points of the wall; of the two options By Ends and By Cent, only one can be toggled on at any given time
By Cent	Toggle on to draw a window wall by selecting the starting point and the center point of the wall; of the two options By Ends and By Cent, only one can be toggled on at any given time
Mullions	<p>The width, depth, and offset for window mullions are set using value menus. For more information on value menus, see “Value Menus” in “The Drawing Board” chapter. Customize the dimensions and position of window mullions; following are the Mullions menu options:</p> <p>DoMulln Models mullions; toggle this option off to draw windows without mullions</p> <p>Width Sets the width of mullions; the <i>width</i> is the mullion dimension along the face of the wall</p> <p>Depth Sets the depth of mullions; the <i>depth</i> is the mullion dimension through the wall</p> <p>Offset Sets the distance from the inner edge of the wall to the inner edge of the mullion</p> <p>DoMarks Toggle on to display a snapping point to the center of each mullion</p> <p>No Ends Toggle on to draw windows without mullions at either end of the window wall; of the three options No Ends, HalfEnd, and FullEnd, only one can be toggled on at any given time</p> <p>HalfEnd Toggle on to draw mullions of half-width on either end of the window wall; of the three options No Ends, HalfEnd, and FullEnd, only one can be toggled on at any given time</p> <p>FullEnd Toggle on to draw mullions of full-width on either end of the window wall; of the three options No Ends, HalfEnd, and FullEnd, only one can be toggled on at any given time</p> <p>Color Sets the color of mullion lines. For more information on using color menus, see “Color Menus” in “The Drawing Board” chapter.</p> <p>Linetype Sets the linetype of mullion lines</p> <p>LineWdth Sets the line width of mullion lines; make mullion lines wider by increasing this value</p> <p>LinSpncg Sets the line spacing of window mullion lines. The LinSpncg setting lengthens both the lines and the spaces in a linetype, so that the aspect ratio of the linetype is preserved.</p>
Glass	<p>Customize the dimensions and position of window glass; following are the Glass menu options:</p> <p>DoGlass Models glass; toggle this option off to draw windows without glass</p> <p>Thicknss Sets the thickness, or distance between the panes, of the glass; this setting has no effect when 1-Line is toggled on. The thickness and offset for window glass are set using value menus. For more information on value menus, see “Value Menus” in “The Drawing Board” chapter.</p> <p>Offset Sets the distance between the inner edge of the wall and the inside face of the glass pane</p> <p>1-Line Toggle on to model glass as a single line between mullions; of the three options 1-Line, 2-Line, and 3-Line, only one can be toggled on at a time</p> <p>2-Line Toggle on to model glass as a double line between mullions; of the three options 1-Line, 2-Line, and 3-Line, only one can be toggled on at a time</p> <p>3-Line Toggle on to model glass as a triple line between mullions; of the three options 1-Line, 2-Line, and 3-Line, only one can be toggled on at a time</p> <p>Color Sets the color of glass lines. For more information on using color menus, see “Color Menus” in “The Drawing Board” chapter.</p> <p>Linetype Sets the linetype of glass lines</p> <p>LineWdth Sets the line width of glass lines; make glass lines wider by increasing this value</p> <p>LinSpncg Sets the line spacing of window glass lines. The LinSpncg setting lengthens both the lines and the spaces in a linetype, so that the aspect ratio of the linetype is preserved.</p>

Sills	Customize the dimensions and position of inner and outer window sills. The inside and outside sills, header depth, and sill depth for window sills are set using value menus. For more information on value menus, see “Value Menus” in “The Drawing Board” chapter. Following are the Sills menu options:
DoSill	Models sills and headers; toggle this option off to draw windows without sills and headers
Insill	Sets the distance from the inner edge of the window wall to the edge of the inner sill
Outsill	Sets the distance from the outer edge of the window wall to the edge of the outer sill
HeadDpth	Sets the depth of the window header
SillDpth	Sets the depth of the window sill
Color	Sets the color of sill lines. For more information on using color menus, see “Color Menus” in “The Drawing Board” chapter.
Linetype	Sets the linetype of sill lines
LineWdth	Sets the line width of sill lines; make sill lines wider by increasing this value
LinSpnng	Sets the line spacing of window sill lines

Drawing Curved Bays of Windows

With the Windows menu in the DCAD_AEC macro, you can draw 2D curved bays of windows. Click OneWindw in the Windows menu in the DCAD_AEC macro to access the Window menu in the Edit/ Architct menu.

➔ To draw a curved bay of windows:

1. Click DCAD_AEC in the Macros pull-down menu. If DCAD_AEC is not listed in your Macros menu, click Configure in the Macros menu and add DCAD_AEC to the Macros in Menu list.
2. Click Windows in the DCAD_AEC menu. The Windows menu is displayed. To model 3D windows, use the Windows menu in the AEC_Modl macro
3. Click HeadHght in the Windows menu. A value menu is displayed, and you are prompted to “Enter new window header height” in the Message Area.
4. Type a new header height or use the value menu to enter a new height.
5. Press **[Enter]** or right-click to accept the new value and exit the menu. The Windows menu is displayed again.
6. Click SillHght in the Windows menu. A value menu is displayed, and you are prompted to “Enter new window sill height” in the Message Area.
7. Type a new sill height or use the value menu to enter a new height.
8. Press **[Enter]** or right-click to accept the new value and exit the menu. The Windows menu is displayed again.
9. Click CircWndw in the Windows menu to display the Circular windows menu.

10. Decide if you want to draw a set quantity of equally spaced bays, as many bays of a set size that can be fit into a space, or a set number and size of bays. You can draw a set quantity of equally spaced bays, as many bays of a set size that can be fit into a space, or a set number and size of bays. In the Circular menu, click Eq Bays to draw equally spaced bays, click FitBays to fit bays into a space, or click SetBays to specify the number and size of the bays. When using the SetBays option, the opening length is determined by the number and size of the bays that you specify.
11. Set bay and window options as necessary. See “Curved Window Bay Options” below for more information. You are prompted to “Enter center point of window wall arc”.
12. Use the mouse or coordinate entry to select a center point for the arc of your bay of windows. You are prompted to “Enter starting point of window wall arc”.
13. Use the mouse or coordinate entry to select a starting point for the arc of your bay of windows. You are prompted to “Enter ending point of window wall”. For more information on using coordinate entry, see “Drawing Using Coordinate Entry” in “The Drawing Board” chapter.
14. Use the mouse or coordinate entry to select an ending point for the arc of your bay of windows. The bay of windows is drawn.

Curved Window Bay Options

Bay Qty	Set the number of window bays you want drawn; not available when using FitBays because bay quantity is automatically calculated
BayAngle	Set the separation angle between individual window bays, which will be multiplied to create the total curvature of the window wall; not available when using Eq Bays because individual bay angle is automatically calculated
WallWdth	Set the width of the wall, which in turn determines the depth of the sills
CntrLin	Toggle on to center the depth of the wall along the starting and ending points you enter; of the two options CntrLin and InnrEdg, only one can be toggled on at a time
InnrEdg	Toggle on to draw the inside face of the window wall along the starting and ending points you enter; of the two options CntrLin and InnrEdg, only one can be toggled on at a time
Clckwse	Toggle on to model windows in a clockwise direction; toggle off to model windows in a counterclockwise direction
Averagd	Available only when FitBays is toggled on; toggle on to slightly increase or decrease the separation angle of each bay to fit the number of bays you specified precisely into the defined opening, leaving no extra space at one end of the window wall; if Averagd is toggled off, any leftover space at either end of the curved window wall is filled with bays of a smaller width. Increasing or decreasing the separation angle of each bay will increase or decrease, respectively, the width of each bay.
Mullions	Customize the dimensions and position of window mullions. The width, depth, and offset for window mullions are set using value menus. For more information on value menus, see “Value Menus” in “The Drawing Board” chapter. Following are the Mullions menu options:
DoMulln	Models mullions; toggle this option off to draw windows without mullions
Width	Sets the width of mullions; the <i>width</i> is the mullion dimension along the face of the wall
Depth	Sets the depth of mullions; the <i>depth</i> is the mullion dimension through the wall
Offset	Sets the distance from the inner edge of the wall to the inner edge of the mullion
DoMarks	Toggle on to display a snapping point to the center of each mullion

No Ends	Toggle on to draw windows without mullions at either end of the window wall; of the three options No Ends, HalfEnd, and FullEnd, only one can be toggled on at any given time
HalfEnd	Toggle on to draw mullions of half-width on either end of the window wall; of the three options No Ends, HalfEnd, and FullEnd, only one can be toggled on at any given time
FullEnd	Toggle on to draw mullions of full-width on either end of the window wall; of the three options No Ends, HalfEnd, and FullEnd, only one can be toggled on at any given time
Color	Sets the color of mullion lines. For more information on using color menus, see “Color Menus” in “The Drawing Board” chapter.
Linetype	Sets the linetype of mullion lines
LineWdth	Sets the line width of mullion lines; make mullion lines wider by increasing this value
LinSpncg	Sets the line spacing of window mullion lines. The LinSpncg setting lengthens both the lines and the spaces in a linetype, so that the aspect ratio of the linetype is preserved.
Glass	Customize the dimensions and position of window glass; following are the Glass menu options:
DoGlass	Models glass; toggle this option off to draw windows without glass
Thicknss	Sets the thickness, or distance between the panes, of the glass; this setting has no effect when 1-Line is toggled on. The thickness and offset for window glass are set using value menus. For more information on value menus, see “Value Menus” in “The Drawing Board” chapter.
Offset	Sets the distance between the inner edge of the wall and the inside face of the glass
1-Line	Toggle on to model glass as a single line between mullions; of the three options 1-Line, 2-Line, and 3-Line, only one can be toggled on at a time
2-Line	Toggle on to model glass as a double line between mullions; of the three options 1-Line, 2-Line, and 3-Line, only one can be toggled on at a time
3-Line	Toggle on to model glass as a triple line between mullions; of the three options 1-Line, 2-Line, and 3-Line, only one can be toggled on at a time
Color	Sets the color of glass lines. For more information on using color menus, see “Color Menus” in “The Drawing Board” chapter.
Linetype	Sets the linetype of glass lines
LineWdth	Sets the line width of glass lines; make glass lines wider by increasing this value
LinSpncg	Sets the line spacing of window glass lines
Sills	Customize the dimensions and position of inner and outer window sills. The inside and outside sills, header depth, and sill depth for window sills are set using value menus. For more information on value menus, see “Value Menus” in “The Drawing Board” chapter. Following are the Sills menu options:
DoSill	Models sills and headers; toggle this option off to draw windows without sills and headers
Insill	Sets the distance from the inner edge of the window wall to the edge of the inner sill
Outsill	Sets the distance from the outer edge of the window wall to the edge of the outer sill
HeadDpth	Sets the depth of the window header
SillDpth	Sets the depth of the window sill
Color	Sets the color of sill lines. For more information on using color menus, see “Color Menus” in “The Drawing Board” chapter.
Linetype	Sets the linetype of sill lines
LineWdth	Sets the line width of sill lines; make sill lines wider by increasing this value
LinSpncg	Sets the line spacing of window sill lines

Drawing Other Geometry

DataCAD offers several additional drawing tools outside of the Architct menu. The Polygons and Curves menus in the Edit menu allow you to draw a number of geometric shapes, from simple rectangles to Bezier curves. The DCAD_AEC macro has options to automate drawing stairs, elevators and columns as well as a Polyline option to draw complex figures or boundaries for elements such as hatch patterns.

In this chapter:

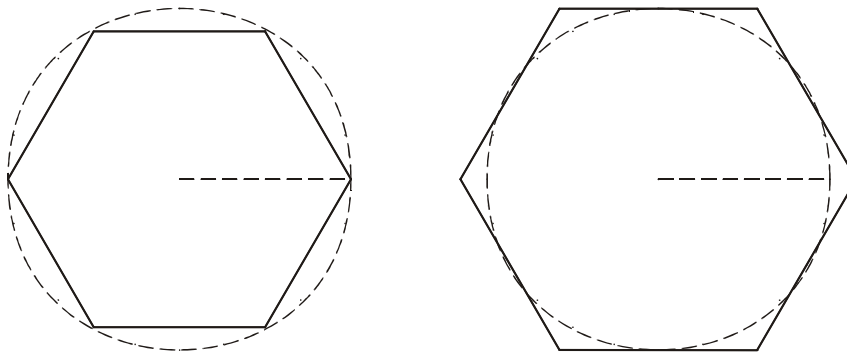
- ⊕ Drawing polygons
- ⊕ Drawing curves
- ⊕ Drawing polylines
- ⊕ Drawing stairs
- ⊕ Drawing columns
- ⊕ Drawing elevators

Drawing Polygons

With the Polygon menu, you can draw rectangles or polygons with up to 36 sides. DataCAD uses a temporary circle to define a polygon. The relationship between this circle and the resulting polygon is dependent on the Inscribe option (see Figure 9.1). The points you select to draw the polygon will determine the orientation of the polygon in your drawing.

➔ To draw a polygon:

1. Go to the Edit menu in the Menu Window and click on Polygons. The Polygons menu is displayed.
2. Look at the toggle switches. The Dynamic, CntrPnt, Vertex, Diameter, and Inscribe options are toggled on by default. Change these settings as necessary. See “Polygon Menu Options” below for more information.
3. Click on No.Sides in the Polygons menu. A value menu is displayed, and you are prompted to “Enter number of sides”.
4. Use the value menu or type a number, and press **[Enter]**. You are prompted to “Select first point on polygon”.
5. Click in the Drawing Area or use coordinate entry to select the first point for the polygon, as prompted. See “Drawing Using Coordinate Entry” in “The Drawing Board” chapter to review coordinate entry methods.
6. Click to select the second point on the outside of the temporary circle used to define the polygon. With Diameter toggled off, the distance between these two points defines the radius of the temporary circle; with Diameter toggled on, the distance between the points is the diameter of the circle. The polygon is drawn.



Polygon drawn with Inscribe on

Polygon drawn with Inscribe off

Figure 9.1: Polygons are actually defined by two points on a circle. These circles are not displayed.

Polygon Menu Options

When you create polygons with both Vertex and Diameter toggled on, the two points you specify are vertices opposite each other on the polygon. With Vertex on and Diameter off, you specify the center point and one vertex. With Vertex off and Diameter on, you specify the mid-point of two sides of the polygon that are opposite each other.

Dynamic	Toggle on to see the size and shape of the polygon you're drawing before you choose the second point; Dynamic is toggled on by default
CntrPnt	Toggle on to display a snapping point, marking the center of the polygon; CntrPnt is toggled on by default
Vertex	Toggle on to draw polygons by defining two vertices of the polygon; toggle off to define polygons by the mid-points of opposite sides of the polygon; Vertex is toggled on by default
Diameter	Toggle on to draw polygons by defining the diameter of the polygon (two points opposite each other); toggle off to define the radius of the polygon (the center point and one side or vertex); Diameter is toggled on by default
Inscribe	Toggle on to draw polygons as though they were inscribed within a circle; toggle off to draw polygons as though a circle was inscribed within them (see Figure 9.1); Inscribe is toggled on by default. See Figure 9.1 for more information about using Inscribe.

Drawing Rectangles

To draw a rectangle, you must select two diagonally opposite corner points. The X and Y coordinates of these diagonal corners is displayed in the Message Area as you draw the rectangle.

➔ To draw a rectangle:

1. Click on Polygons in the Edit menu.
2. Click on RectAngl in the Polygons menu.
3. Toggle CntrPnt on to automatically add a snap point marking the center of the rectangle. If Walls are toggled on and the Outside or Inside option in the Architct menu is toggled on as well, you will be prompted to indicate the inside or outside of the wall, respectively, after you enter the second point for your rectangle.
4. Click in the Drawing Area or use coordinate entry to select the first point of the rectangle. For more information on using coordinate entry, see "Drawing Using Coordinate Entry" in "The Drawing Board" chapter.
5. Move the cursor diagonally and click or use coordinate entry to enter the second point of the rectangle. The rectangle is drawn.

Drawing Curves

You can draw circles, arcs, ellipses, and other curvilinear entities using a wide variety of methods.

Drawing 2-Point Arcs

Two-point arcs are drawn by indicating the radius of the arc and then defining the length of the arc.

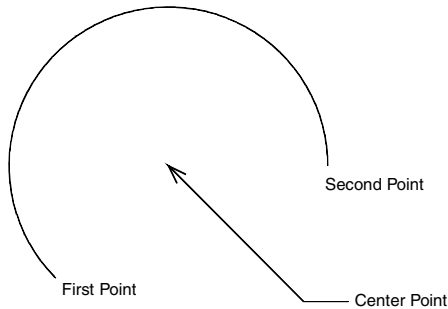


Figure 9.2: 2-point arc

➔ To draw a 2-point arc:

1. Click on Curves in the Edit menu in the Menu Window. The Curves menu is displayed in the Menu Window.
2. Click on 2PtArc in the Curves menu. You can also click on 2D Curves in the Create pull-down menu and then click on 2-Point Arc in the submenu. The 2PtArc menu is displayed.
3. Toggle Dynamic on in the 2PtArc menu to preview the arc before you enter the final point. With Dynamic toggled on, you can move your mouse around the center point to preview the arc at various angles before selecting the second point. You are prompted to “Select center of arc”.
4. Select the center point of the arc by clicking in the Drawing Area or use coordinate entry. For more information on coordinate entry, see “Drawing Using Coordinate Entry” in “The Drawing Board” chapter. You are prompted to “Select start point of arc”.
5. Select the first point on the arc by clicking in the Drawing Area or use coordinate entry. The ClkWise option is now available in the menu. You are prompted to “Select end point of arc”.
6. Toggle ClkWise on to draw the arc clockwise from the first point; toggle it off to draw the arc counterclockwise.
7. Select the second point on the arc by clicking in the Drawing Area or use coordinate entry. This point is used only to determine the angle of the arc, so it doesn’t have to be exactly on the arc. The arc is drawn.

Drawing 3-Point Arcs

Three-point arcs are drawn by selecting the end points of the arc as well as one other point on the arc.

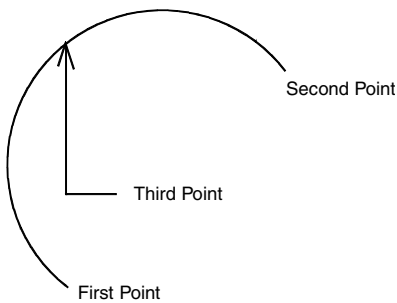


Figure 9.3: 3-point arc

➔ To draw a 3-point arc:

1. Click on Curves in the Edit menu in the Menu Window. The Curves menu is displayed in the Menu Window.
2. Click on 3PtArc in the Curves menu. You can also click on 2D Curves in the Create pull-down menu and then click on 3-Point Arc in the submenu. The 3PtArc menu is displayed.
3. Toggle Dynamic on in the 3PtArc menu to preview the arc before you enter the final point. With Dynamic toggled on, you can move your mouse to preview the arc at various angles before selecting the third point. You are prompted to “Select first end point of arc”.
4. Select the first end point of the arc by clicking in the Drawing Area or using coordinate entry. You are prompted to “Select second end point of arc”. For more information on coordinate entry, see “Drawing Using Coordinate Entry” in “The Drawing Board” chapter.
5. Select the second end point of the arc by clicking in the Drawing Area or using coordinate entry. Keep in mind when selecting the second end point that the arc will be drawn in a counterclockwise direction. You are prompted to “Select any point on the arc”.
6. Select the third point on the arc by clicking in the Drawing Area or using coordinate entry. The arc is drawn, along with its center point.

Drawing Arcs By Center Angle

You can draw arcs based on a center angle that you define.

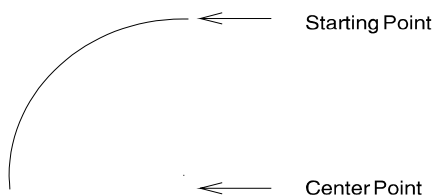


Figure 9.4: An arc defined by selecting an end point, the center point, and an angle

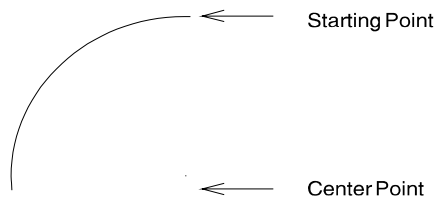
➔ To draw an arc by defining the center angle:

1. Click on Curves in the Edit menu in the Menu Window. The Curves menu is displayed in the Menu Window.

2. Click on **CentAngl** in the Curves menu. You can also click on **2D Curves** in the Create pull-down menu and then click on **Center Angle Arc** in the submenu. The **CentAngl** menu is displayed, and you are prompted to “Select starting point of arc”.
3. Select the first end point of the arc by clicking in the Drawing Area or using coordinate entry. For more information on coordinate entry, see “Drawing Using Coordinate Entry” in “The Drawing Board” chapter. Keep in mind when selecting this starting point that the arc will be drawn in a counterclockwise direction. You are prompted to “Select center of arc”.
4. Select the center point of the arc by click in the Drawing Area or using coordinate entry. You are prompted to “Enter included angle”, and a value menu listing angle values is displayed in the Menu Window.
5. Type an angle or use the value menu to enter an angle, and press **[Enter]**. For more information on using value menus, see “Value Menus” in “The Drawing Board” chapter. The arc is drawn.

Drawing Arcs By Arc Length

You can draw arcs by defining their length.



Arc Length = 10'-0"

Figure 9.5: An arc defined by the starting point, center point, and length of the arc

➔ To draw an arc based on the arc's length:

1. Click on **Curves** in the Edit menu in the Menu Window. The **Curves** menu is displayed in the Menu Window.
2. Click on **CentArc** in the Curves menu. You can also click on **2D Curves** in the Create pull-down menu and then click on **Center Arc** in the submenu. The **CentArc** menu is displayed, and you are prompted to “Select starting point of arc”.
3. Select the first end point of the arc by clicking in the Drawing Area or using coordinate entry. For more information on coordinate entry, see “Drawing Using Coordinate Entry” in “The Drawing Board” chapter. Keep in mind when selecting this starting point that the arc will be drawn in a counterclockwise direction. You are prompted to “Select center of arc”.
4. Select the center point of the arc by clicking in the Drawing Area or using coordinate entry. You are prompted to “Enter arc length”; a value menu listing length values is displayed in the Menu Window.

5. 5. Type a length or use the value menu to enter a length, and press **[Enter]**. For more information on using value menus, see “Value Menus” in “The Drawing Board” chapter. The arc is drawn.

Drawing Arcs By Chord Length

You can draw arcs by defining the length of a chord, measuring from the starting point of the arc to an end point.

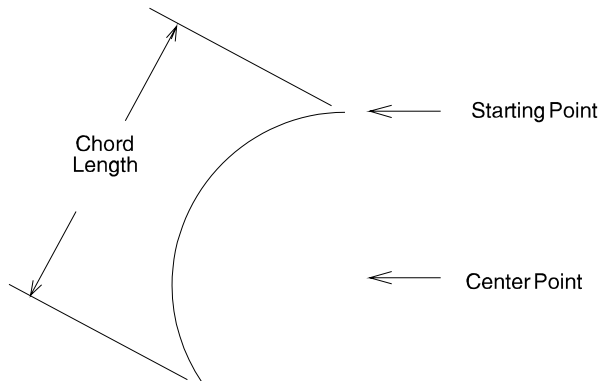


Figure 9.6: An arc defined by selecting the starting point of the arc, its center point, and the length of a chord from end point to end point.

➔ To draw an arc based on chord length:

1. Click on Curves in the Edit menu in the Menu Window. The Curves menu is displayed in the Menu Window.
2. Click on CentChrd in the Curves menu. You can also click on 2D Curves in the Create pull-down menu and then click on Center Chord Arc in the submenu. The CentChrd menu is displayed, and you are prompted to “Select starting point of arc”.
3. Select the first end point of the arc by click in the Drawing Area or using coordinate entry. For more information on coordinate entry, see “Drawing Using Coordinate Entry” in “The Drawing Board” chapter. Keep in mind when selecting this starting point that the arc will be drawn in a counterclockwise direction. You are prompted to “Select center of arc”.
4. Select the center point of the arc by clicking in the Drawing Area or using coordinate entry. You are prompted to “Enter chord length”; a value menu listing length values is displayed in the Menu Window. For more information on using value menus, see “Value Menus” in “The Drawing Board” chapter.
5. Type a length or use the value menu to enter a length, and press **[Enter]**. The arc is drawn.

Drawing Arcs By Radius

You can draw arcs by selecting two endpoints and then entering the length of the arc’s radius.

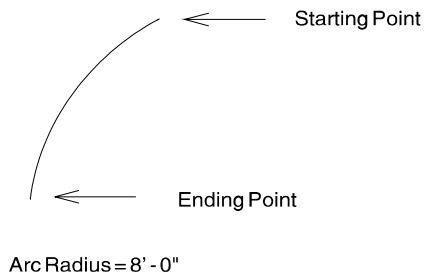


Figure 9.7: An arc defined by selecting a starting point and an ending point for the arc and then setting an arc radius

➔ To draw an arc based on a specified radius:

1. Click on Curves in the Edit menu in the Menu Window. The Curves menu is displayed in the Menu Window.
2. Click on EndPtRad in the Curves menu. You can also click on 2D Curves in the Create pull-down menu and then click on End Point Radius Arc in the submenu. The EndPtRad menu is displayed, and you are prompted to "Select starting point of arc".

To properly define an arc, you must make the radius value larger than half the distance between the starting and ending points. If the message "These points do not define an arc" is displayed, increase the radius length.

3. Select the first end point of the arc by clicking in the Drawing Area or using coordinate entry. Keep in mind when selecting this starting point that the arc will be drawn in a counterclockwise direction. You are prompted to "Select ending point of arc".
4. Select the end point of the arc by clicking in the Drawing Area or using coordinate entry. You are prompted to "Enter radius of arc"; a value menu listing radius length values is displayed in the Menu Window.
5. Type a radius length or use the value menu to enter a radius, and press **[Enter]**. The arc is drawn.

Drawing Arcs By Included Angle

You can draw arcs by selecting the end points of the arc and then entering an arc angle.

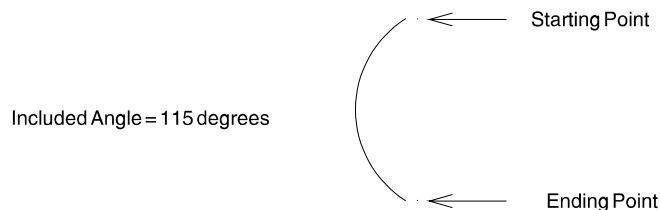


Figure 9.8: An arc defined by selecting a starting point and an ending point and then entering the included angle for the arc.

➔ To draw an arc based on its included angle:

1. Click on Curves in the Edit menu in the Menu Window. The Curves menu is displayed in the Menu Window.
2. Click on EndPtAng in the Curves menu. You can also click on 2D Curves in the Create pull-down menu and then click on End Point Angle Arc in the submenu. The EndPtAng menu is displayed, and you are prompted to “Select starting point of arc”.
3. Select the first end point of the arc by clicking in the Drawing Area or using coordinate entry. For more information on coordinate entry, see “Drawing Using Coordinate Entry” in “The Drawing Board” chapter. Keep in mind when selecting this starting point that the arc will be drawn in a counterclockwise direction. You are prompted to “Select ending point of arc”.
4. Select the end point of the arc by clicking in the Drawing Area or using coordinate entry. You are prompted to “Enter included angle”; a value menu listing angle values is displayed in the Menu Window.
5. Type an angle or use the value menu to enter an angle, and press **[Enter]**. For more information on using value menus, see “Value Menus” in “The Drawing Board” chapter. The arc is drawn.

Drawing Arcs Using a Tangent Line

The StartDir option in the Curves menu uses a temporary tangent line to define the angle of the arc.

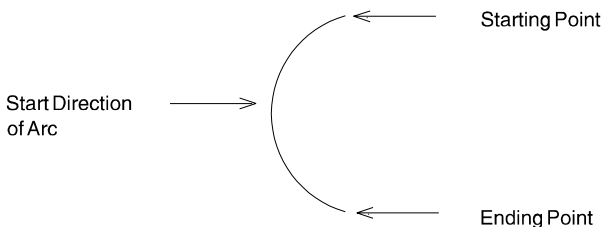


Figure 9.9: An arc defined by selecting two end points and then setting the arc angle by placing a temporary tangent line

➔ To draw an arc using a temporary tangent line:

1. Click on Curves in the Edit menu in the Menu Window. The Curves menu is displayed in the Menu Window.
2. Click on StartDir in the Curves menu. You can also click on 2D Curves in the Create pull-down menu and then click on Start Direction of Arc in the submenu. The StartDir menu is displayed, and you are prompted to “Select starting point of arc”.
3. Select the first end point of the arc by clicking in the Drawing Area or using coordinate entry. For more information on coordinate entry, see “Drawing Using Coordinate Entry” in “The Drawing Board” chapter. Keep in mind when selecting this starting point that the arc will be drawn in a counterclockwise direction. You are prompted to “Select ending point of arc”.

4. Select the end point of the arc by clicking in the Drawing Area or using coordinate entry. You are prompted to “Select starting direction of arc”. Notice that a line is now connected to the starting point you entered, and the line moves as you move your mouse.
5. Move your cursor until the line is positioned as a tangent line to the arc you want to draw.
6. Click to set the tangent line. The arc is drawn. If Ortho mode is toggled on, the tangent line will “jump” in increments equal to the snap angle setting. For more information on about Ortho mode, see “Using SWOTHLUDFB” in “The Drawing Board” chapter.

Drawing Tangent Arcs

You can draw arcs that are tangent to an existing line or arc by simply selecting the entity to be tangent and then selecting an end point.

➔ To draw arcs tangent to another entity:

1. Click on Curves in the Edit menu in the Menu Window. The Curves menu is displayed in the Menu Window.
2. Click on Tangent in the Curves menu. You can also click on 2D Curves in the Create pull-down menu and then click on Tangent Arc in the submenu. The Tangent menu is displayed with a Dynamic option that is toggled on by default, allowing you to preview the arc before entering the final point. You are prompted to “Select line or arc from which to draw”.
3. Move your cursor over the entity that will be tangent to the arc you’re drawing and click to select it. You can also object snap to an entity at a particular point. For more information on object snapping, see “Object Snapping” in the “Drawing Tools” chapter. If you’ve selected the wrong entity, click on NewLine in the Tangent menu (or simply right-click using your mouse) to deselect the entity; you can then select another entity for your tangent. You are prompted to “Select ending point of arc”.
4. Select the end point of the arc by clicking in the Drawing Area or using coordinate entry. For more information on coordinate entry, see “Drawing Using Coordinate Entry” in “The Drawing Board” chapter. The arc is drawn. Notice your cursor is still attached to the arc. You can continue drawing tangent arcs by entering end points. Right-click to detach your cursor from the last arc you drew.

Drawing Circles By Radius

You can draw a circle by defining its radius.

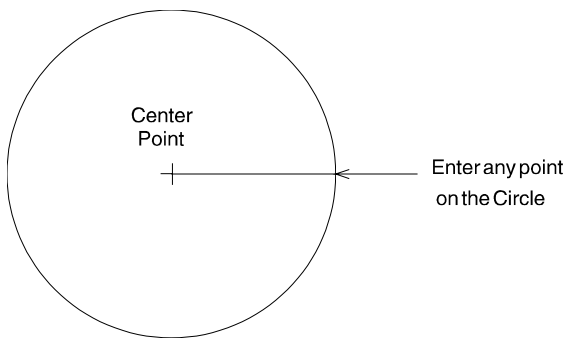


Figure 9.10: A circle defined by selecting a center point and a point on the circle

➔ To draw a circle based on its radius:

1. Click on Curves in the Edit menu in the Menu Window. The Curves menu is displayed in the Menu Window.
2. Click on RadCirc in the Curves menu. You can also click on 2D Curves in the Create pull-down menu and then click on Radius Circle in the submenu. The RadCirc menu is displayed with a Dynamic option that is toggled on by default, allowing you to preview the circle before you select the last point. You are prompted to “Select center of circle”.
3. Select the center point of the circle by clicking in the Drawing Area or using coordinate entry. For more information on coordinate entry, see “Drawing Using Coordinate Entry” in “The Drawing Board” chapter. Notice the radius (and therefore the circle) changes size as you move your mouse. You are prompted to “Select any point on circle”.
4. Select a point on the circle by clicking in the Drawing Area or using coordinate entry. The circle is drawn.

Drawing Circles By Diameter

You can draw a circle by defining its diameter.

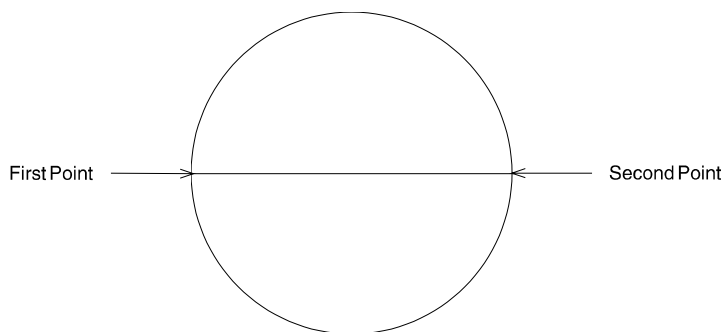


Figure 9.11: A circle defined by selecting two points opposite each other on the circle

➔ To draw a circle based on its diameter:

1. Click on Curves in the Edit menu in the Menu Window. The Curves menu is displayed in the Menu Window.

2. Click on DiaCirc in the Curves menu. You can also click on 2D Curves in the Create pull-down menu and then click on Diameter Circle in the submenu. The DiaCirc menu is displayed with a Dynamic option that is toggled on by default, allowing you to preview the circle before you select the last point. You are prompted to “Select first point on circle”.
3. Select the first point on the circle by clicking in the Drawing Area or using coordinate entry. Notice the diameter (and therefore the circle) changes size as you move your mouse. You are prompted to “Select second point on circle”.
4. Select the second point on the circle by clicking in the Drawing Area or using coordinate entry. The circle is drawn.

Drawing Circles By Points

You can draw a circle by defining three points on the circle.

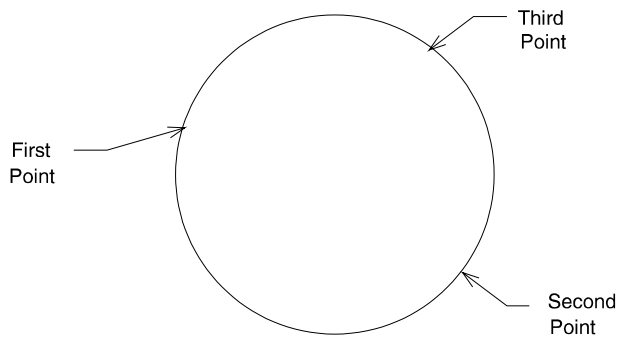


Figure 9.12: A circle defined by selecting three points

➔ To draw a circle by selecting points:

1. Click on Curves in the Edit menu in the Menu Window. The Curves menu is displayed in the Menu Window.
2. Click on 3PtCirc in the Curves menu. You can also click on 2D Curves in the Create pull-down menu and then click on 3-Point Circle in the submenu. The 3PtCirc menu is displayed with a Dynamic option that is toggled on by default, allowing you to preview the circle before you select the last point. You are prompted to “Select first point on circle”.
3. Select the first point on the circle by clicking in the Drawing Area or using coordinate entry. For more information on coordinate entry, see “Drawing Using Coordinate Entry” in “The Drawing Board” chapter. You are prompted to “Select second point on circle”.
4. Select the second point on the circle by clicking in the Drawing Area or using coordinate entry. Notice the circle changes size as you move your mouse. You are prompted to “Select third point on circle”.
5. Select the third point on the circle by clicking in the Drawing Area or using coordinate entry. The circle is drawn.

Drawing Ellipses

You can draw ellipses by simply selecting two diagonally opposed points, as though you were selecting two diagonal corners of a rectangle.

➔ To draw an ellipse:

1. Click on Curves in the Edit menu in the Menu Window. The Curves menu is displayed in the Menu Window.
2. Click on Ellipse in the Curves menu. The Ellipse menu is displayed with a Dynamic option that is toggled on by default, allowing you to preview the ellipse before you select the last point. You are prompted to “Select first corner of ellipse rectangle”.
3. Select the first point for the ellipse by clicking in the Drawing Area or using coordinate entry. For more information on coordinate entry, see “Drawing Using Coordinate Entry” in “The Drawing Board” chapter. Notice the ellipse changes shape and size as you move your mouse. You are prompted to “Select second corner of ellipse rectangle”.
4. Select the second point for the ellipse by clicking in the Drawing Area or using coordinate entry. The ellipse is drawn.

Drawing Curves Using Control Points

You can draw more complex curves by selecting control points that affect how the curves are drawn.

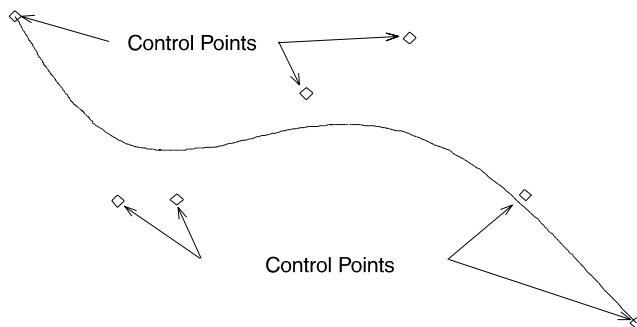


Figure 9.13: A Bezier curve with selected control points

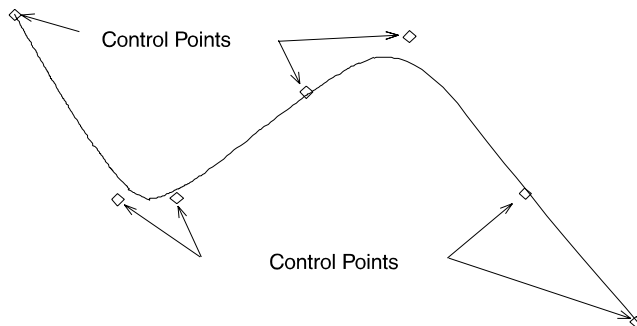


Figure 9.14: A B-spline curve with selected control points

➔ To draw a Bezier curve:

1. Click on Curves in the Edit menu in the Menu Window. The Curves menu is displayed in the Menu Window.
2. Click on Bezier in the Curves menu. You can also click on 2D Curves in the Create pull-down menu and then click on Bezier Curve in the submenu. The Bezier menu is displayed with a Dynamic option that is toggled on by default, allowing you to preview the curve as you select control points. You are prompted to “Select Bezier control point”.
3. Select a control point for the curve by clicking in the Drawing Area or using coordinate entry. For more information on coordinate entry, see “Drawing Using Coordinate Entry” in “The Drawing Board” chapter.
4. Continue selecting control points; you must select at least three but no more than eight points.
5. Right-click to detach your cursor from the curve. The Bezier curve is drawn. If you enter all eight control points, your cursor is detached automatically and the curve is drawn.

The formula used to create B-spline curves in DataCAD is specific to Third Order splines.

➔ To draw a B-spline curve:

1. Click on Curves in the Edit menu in the Menu Window. The Curves menu is displayed in the Menu Window.
2. Click on B-Spline in the Curves menu. You can also click on 2D Curves in the Create pull-down menu and then click on B-Spline Curve in the submenu. The B-Spline menu is displayed with a Dynamic option that is toggled on by default, allowing you to preview the curve as you select control points. You are prompted to “Select B-Spline control point”.
3. Select a control point for the curve by clicking in the Drawing Area or using coordinate entry. For more information on coordinate entry, see “Drawing Using Coordinate Entry” in “The Drawing Board” chapter.
4. Continue selecting control points; you must select at least three but no more than eight points.
5. Right-click to detach your cursor from the curve. The B-spline curve is drawn. If you enter all eight control points, your cursor is detached automatically and the curve is drawn.

Drawing Curves Using Surveyor’s Data

You can draw curves based on surveyor’s data that you enter. Based on this data and the options you use, DataCAD can even extrapolate additional data. For instance, if you enter the radius point of the curve, the central angle of the curve and the starting point of the curve, DataCAD can calculate the coordinates of the ending point and draw the curve. For more information on the options noted in Figure 9.15, see “CurvData Menu Options” on the following page.

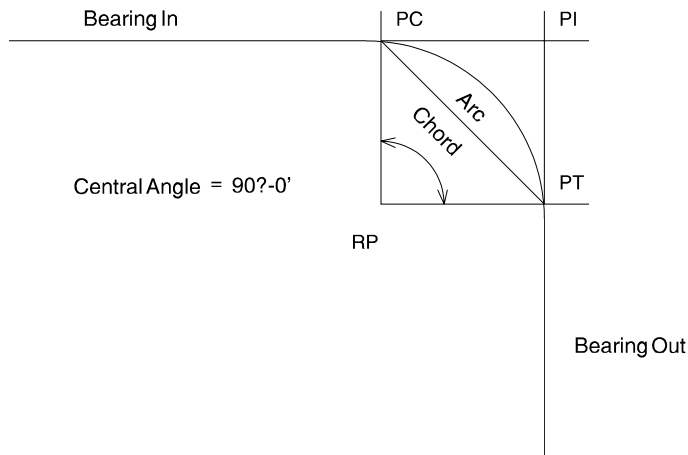


Figure 9.15: An arc drawn by entering surveyor's data

➔ To draw a curve using surveyor's data:

1. Click on Curves in the Edit menu in the Menu Window. The Curves menu is displayed in the Menu Window.
2. Click on CurvData in the Curves menu. You can also click on 2D Curves in the Create pull-down menu and then click on Curve from Data in the submenu. The CurvData menu is displayed.
3. Enter the surveyor's data using CurvData menu options. Once enough information has been entered to draw the curve, an Add option is displayed in the CurvData menu.
4. Click on Add. The curve is drawn.

CurvData Menu Options

RP	Defines the center, or radius point, of the curve
PC	Defines the starting point, or the point of curve, of the curve
PT	Defines the ending point, or point of tangency, of the curve
PI	Defines the point of intersection, where the tangent line intersects the starting point (PC) of the curve
Angle	Defines the central angle about the radius point
Radius	Defines the radius of the curve, or the distance between the radius point and the starting point (PC) or the ending point (PT) of the curve
Tangent	Defines the distance from the starting point (PC) or ending point (PT) to the intersection of the tangent lines to these two points
Chord	Defines the straight-line distance between the starting point (PC) and the ending point (PT) of the curve
ArcLnth	Defines the distance along the curve from the starting point (PC) to the ending point (PT)
BrngIn	Defines the bearing of the line going into the starting point (PC) of the curve
BrngOut	Defines the bearing of the line going away from the ending point (PT) of the curve
BrngPC	Defines the bearing of the line from the radius point to the starting point (PC) of the curve
BrngPT	Defines the bearing of the line from the radius point to the ending point (PT) of the curve

Drawing Polylines

A *polyline* is a continuous series of lines and/or arcs that are treated by DataCAD as a single entity. You can use the RectAngl option to quickly draw rectangular polylines or draw polylines, segment by segment. Polygons drawn this way can be open or closed depending on whether you toggle Closed off or on.

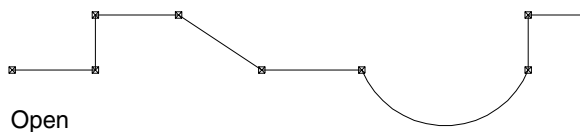
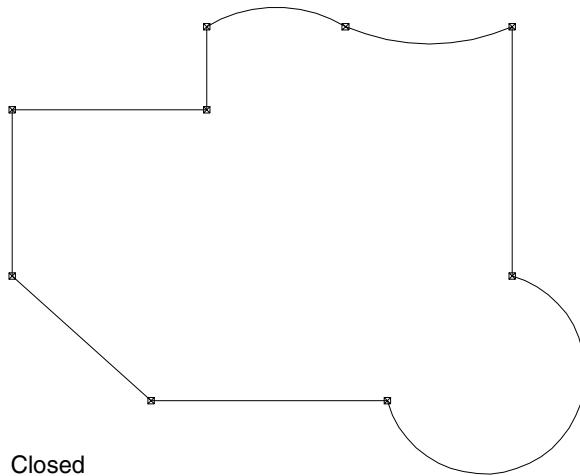


Figure 9.16: Examples of a closed polyline (drawn with Closed toggled on) and an open polyline (drawn with Closed toggle off)

Closed polylines can be hatched; see the “Hatch” chapter for more information about hatching. If your polyline (no matter how irregular) is not closed, you cannot hatch it. In addition, you can measure the area or perimeter of a closed polyline or find its centroid; see “Measuring Areas, Perimeters, and Volumes” in the “Drawing Tools” chapter for more information.

➔ To draw a rectangular polyline:

1. Click on Curves in the Edit menu in the Menu Window.
2. Click on Polyline in the Curves menu.
3. Click on RectAngl in the Polyline menu. You are prompted to “Select first corner of rectangle”.
4. Click in the Drawing Area or object snap to a point to select the first corner of the polyline. Notice that the rectangle stretches as you move your cursor. You are prompted to “Select second corner of rectangle”.
5. Click in the Drawing Area or object snap to a point to select the second corner of the polyline. The polyline is drawn.

➔ To draw a polyline, segment by segment:

1. Click on Curves in the Edit menu in the Menu Window.

2. Click on Polyline in the Curves menu. Closed is toggled on by default, allowing you to draw a polyline that will automatically be closed when you're finished selecting points. You are prompted to "Select first point on polyline".
3. Select the first point for the polyline by clicking in the Drawing Area or using coordinate entry. For more information on coordinate entry, see "Drawing Using Coordinate Entry" in "The Drawing Board" chapter. The options 2PtArc and 3PtArc are displayed in the Polyline menu. You are prompted to "Select next point on polyline".
4. Add line segments, 2-point arcs, or 3-point arcs to your polyline:
 - To select the second point of a line segment, click in the Drawing Area, use coordinate entry, or object snap to a point on an entity.
 - To draw a 2-point arc, click on 2PtArc in the Polyline menu. Select a center point for the arc, and then select an end point. Right-click to exit the 2-point arc function.
 - To draw a 3-point arc, click on 3PtArc in the Polyline menu. Select a second end point for the arc (with the first end point being the point you selected in the previous step), and then select any other point on the arc. Right-click to exit the 3-point arc function.

The option Tangent is displayed in the Polyline menu after you finish drawing the first line segment or after you right-click to exit the arc functions. You are prompted to "Select next point on polyline".

5. Draw a tangent arc in your polyline by clicking on Tangent in the Polyline menu. Move your cursor until the arc displayed is the one you want drawn, and then click to draw the arc.
6. Continue drawing line segments or arcs, as outlined in steps 4 and 5, until your polyline is complete.
7. Click Backup any time to erase the last line segment or arc of your polyline. You can click Cancel at any time to erase the polyline you're drawing.
8. Right-click to finish the polyline. If Closed is toggled on, the polyline is automatically closed. If Closed is toggled off, your cursor is detached from the last point of the polyline.

➔ To draw a polyline that is both closed and covered:

1. Click on Curves in the Edit menu in the Menu Window.
2. Click on Polyline in the Curves menu. Closed is toggled on by default in the Polyline menu.
3. Toggle Covered on in the Polyline menu.
4. Draw the polyline. Right-click when you are finished.
5. Select Object Viewer from the View pull-down menu to look at the covered polyline from all angles.

Contour Search

Contour search (ContSrch) is an option that is available in the Curves, Polyline, Hatch, and Boundary menus. With this tool, you can easily create outlines for fills and hatches. It automatically searches for the boundaries of a closed shape and creates a polyline around it. For example, contour search can find the edges of your walls in plan view so that you can add a hatch pattern to your drawing.

➔ To search for the contour of a closed polyline:

1. Make sure you have a closed polyline in your drawing window.
2. Select Curves from the Edit menu and click on Polyline. The Polyline menu appears.
3. Click on ContSrch from the Polyline menu. You are prompted to “Select point inside/outside contour.”

Hint: You can press (K) to change the color if you want the contour to be obvious.

4. Click an entity along the boundary of the closed polyline or entity. DataCAD creates a closed polyline that corresponds to the original closed polyline. Alternatively, you can place the cursor within a closed group of lines. DataCAD searches for the boundary and draws a closed polyline around the perimeter.

You can also select Hatch from the Utility menu, select Boundary from the Hatch menu, and click on ContSrch. Then, click on the boundary of the closed polyline or entity so that DataCAD can draw around its perimeter.

Editing Polylines

You can edit any polyline in your drawing. You can add or delete a vertex, convert line segments to curves and curves to line segments, move a vertex, convert the entire polyline into lines and arcs, modify arcs in the polyline, convert lines and arcs into a polyline, or open or close a polyline. A *vertex* is the point where two segments of the polyline intersect, usually marked with a point.

Converting Polylines

You can convert a polyline to standard lines and curves, and conversely you can convert lines and arcs into a single polyline. You can also convert an open polyline to a closed one. A polyline is considered a single entity by DataCAD. You cannot convert a single line or arc that is part of a polyline; you must convert the entire polyline.

➔ To convert a polyline into standard lines and arcs:

1. Click on Polyline in the Curves menu. The Polyline menu is displayed.
2. Click on Edit in the Polyline menu. The Edit menu is displayed.
3. Click on Explode in the Polyline menu. You are prompted to “Select polyline to explode into lines/arcs”.
4. Move your mouse over one of the lines or vertices of the polyline and click. The polyline is exploded.

- ➔ To convert lines and arcs into a polyline:
 1. Click on Polyline in the Curves menu. The Polyline menu is displayed.
 2. Click on Edit in the Polyline menu. The Edit menu is displayed.
 3. Click on Generate in the Polyline menu. You are prompted to “Select start point of polyline”.
 4. Toggle AndCopy on to retain the original lines and arcs and draw a polyline on top of them; toggle AndCopy off to convert the lines and arcs to a polyline.
 5. Move your mouse over one of the lines or vertices of the polyline and click. Dots are drawn at each vertex of the new polyline, and its perimeter, area, and volume are displayed in the Message Area.
- ➔ To open or close a polyline:
 1. Click on Polyline in the Curves menu. The Polyline menu is displayed.
 2. Click on Edit in the Polyline menu. The Edit menu is displayed.
 3. Click on Close in the Edit menu. You are prompted to “Select polyline to be closed”.
 4. Move your mouse over the polyline and click. A line segment is added to the polyline, closing it.
- ➔ To convert a closed shape to a polyline:
 1. Click on Polyline in the Curves menu. The Polyline menu is displayed.
 2. Click on ContSrch in the Polyline menu. The ContSrch menu is displayed, and you are prompted to “Select point inside/outside contour”.
 3. Click inside the area you want to convert, or click outside the area. If contour search finds an area enclosed by lines, a polyline is created, tracing those lines. If contour search cannot find a completely enclosed area, the message “Contour is not closed” is displayed; you must search again or use another method to define the boundary.
- ➔ To convert a closed polyline to a polygon or slab:
 1. Select DCAD 3D from the Edit menu. This displays the 3DEdit menu.
 2. Click on Explode. The Explode menu appears.
 3. Select ToPgons if you want to convert the polyline to a polygon; alternatively, select ToSlab if you want to convert the polyline to a slab. These are mutually exclusive toggle switches. “Select entity to <EXPLODE>” appears in the message area.
 4. Click on the closed polygon you want to convert.

Editing Polyline Vertices

You can add, delete, or move a vertex on a polyline. The MoveCnr option lets you move a vertex freely; the DynCnr option preserves the angle of the existing vertex while letting you move it.

➔ To add a vertex to a polyline:

1. Click on Polyline in the Curves menu. The Polyline menu is displayed.
2. Click on Edit in the Polyline menu. The Edit menu is displayed.
3. Click on AddCnr in the Edit menu. You are prompted to “Select edge to which to add the corner”.
4. Toggle Info on at the bottom of the Edit menu to display the area and perimeter of the polyline, *as you position the new vertex*.
5. Move your mouse over the line or arc segment of the polyline that you want to add the vertex to, and click. Notice your cursor is now attached to the segment; this point will be the new vertex.
6. Move your mouse to position the new vertex and click, or use coordinate entry to enter the vertex position. The new vertex is added and the polyline is modified.
7. Repeat steps 4 and 5 to continue adding vertices to polylines in your drawing. To exit this function, click on another Edit menu option, or right-click until the Polyline menu is displayed again in the Menu Window.

➔ To delete a vertex from a polyline:

1. Click on Polyline in the Curves menu. The Polyline menu is displayed.
2. Click on Edit in the Polyline menu. The Edit menu is displayed.
3. Click on DelCnr in the Edit menu. You are prompted to “Select corner to be deleted”.
4. Move your mouse over the vertex that you want to delete, and click. The vertex is deleted, and the polyline is modified.
5. Repeat the previous step to continue deleting vertices from polylines in your drawing. To exit this function, click on another Edit menu option, or right-click until the Polyline menu is displayed again in the Menu Window.

➔ To move a vertex on a polyline:

1. Click on Polyline in the Curves menu. The Polyline menu is displayed.
2. Click on Edit in the Polyline menu. The Edit menu is displayed.
3. Click on MoveCnr in the Edit menu. You are prompted to “Select corner to be moved”.
4. Toggle Info on at the bottom of the Edit menu to display the area and perimeter of the polyline, *as you reposition the vertex*.
5. Move your mouse over the vertex that you want to move, and click. Your cursor is now attached to the vertex you selected.
6. Move your mouse to reposition the vertex and click; you can also use coordinate entry to precisely position the vertex. The new vertex is moved and the polyline is modified.

7. Repeat steps 4 and 5 to continue moving vertices on polylines in your drawing. To exit this function, click on another Edit menu option, or right-click until the Polyline menu is displayed again in the Menu Window.

➔ To move a vertex dynamically:

1. Click on Polyline in the Curves menu. The Polyline menu is displayed.
2. Click on Edit in the Polyline menu. The Edit menu is displayed.
3. Click on DynCrnr in the Edit menu. You are prompted to "Select corner to be moved".
4. Toggle Info on at the bottom of the Edit menu to display the area and perimeter of the polyline, *as you reposition the vertex*.
5. Move your mouse over the vertex that you want to move, and click. Your cursor is now attached to the vertex you selected.
6. Move your mouse to reposition the vertex and click; you can also use coordinate entry to precisely position the vertex. For more information on coordinate entry, see "Drawing Using Coordinate Entry" in "The Drawing Board" chapter. The new vertex is moved and the polyline is modified.
7. Repeat steps 4 and 5 to continue moving vertices on polylines in your drawing. To exit this function, click on another Edit menu option, or right-click until the Polyline menu is displayed again in the Menu Window.

Editing Polyline Arcs

➔ To edit an arc in a polyline:

1. Click on Polyline in the Curves menu. The Polyline menu is displayed.
2. Click on Edit in the Polyline menu. The Edit menu is displayed.
3. Click on EditArc in the Polyline menu. You are prompted to "Select arc to be modified".
4. Toggle Info on at the bottom of the Edit menu to display the area and perimeter of the polyline, *as you reposition the arc*.
5. Move your mouse over the arc you want to edit and click. Notice that your cursor is now attached to the arc.
6. Move your mouse to change the arc and then click. The new arc is drawn and the polyline is modified. You can also use coordinate entry to edit the arc. For more information on coordinate entry, see "Drawing Using Coordinate Entry" in "The Drawing Board" chapter.
7. Repeat steps 4 and 5 to continue editing arcs on the polyline. To exit this function, click on another Edit menu option, or right-click until the Polyline menu is displayed again in the Menu Window.

Editing Polyline Edges

You can drag a polyline's edge to a new position or delete an edge entirely.

➔ To reposition a line in a polyline:

1. Click on Polyline in the Curves menu. The Polyline menu is displayed.
2. Click on Edit in the Polyline menu. The Edit menu is displayed.
3. Click on DragEdge in the Polyline menu. You are prompted to "Select an edge to drag".
4. Toggle Info on at the bottom of the Edit menu to display the area and perimeter of the polyline, *as you reposition the edge*.
5. Move your mouse over the edge you want to edit and click. Notice that your cursor is now attached to the line.
6. Move your mouse to change the line's position and then click. The new edge is drawn and the polyline is modified. You can also use coordinate entry to edit the edge. For more information on coordinate entry, see "Drawing Using Coordinate Entry" in "The Drawing Board" chapter.
7. Repeat steps 4 and 5 to continue editing edges on the polyline. To exit this function, click on another Edit menu option, or right-click until the Polyline menu is displayed again in the Menu Window.

➔ To delete an edge from a polyline:

1. Click on Polyline in the Curves menu. The Polyline menu is displayed.
2. Click on Edit in the Polyline menu. The Edit menu is displayed.
3. Click on Del Edge in the Polyline menu. You are prompted to "Select edge to be deleted".
4. Move your mouse over the edge you want to delete and click. The edge is deleted.
5. Repeat the previous step to continue deleting edges on the polyline. To exit this function, click on another Edit menu option, or right-click until the Polyline menu is displayed again in the Menu Window.

Converting Edges and Arcs

You can convert an edge (or line) in a polyline to an arc, or conversely, convert an arc to a straight edge. Both of these actions can be done with a single menu option: the Arc<>Edge option simply converts any polyline segment you click on. So if you click on an arc, this function will automatically change it to a straight line segment; and if you click on an edge, it will convert it to an arc, which you can then define by dragging it into position.

➔ To convert an arc to a straight edge or an edge to an arc:

1. Click on Polyline in the Curves menu. The Polyline menu is displayed.
2. Click on Edit in the Polyline menu. The Edit menu is displayed..
3. Click on Arc<>Edge in the Polyline menu. You are prompted to "Select edge/arc to be modified".

4. Toggle Info on at the bottom of the Edit menu to display the area and perimeter of the polyline, *as you convert the edge or arc*.
5. Move your mouse over the edge or arc that you want to convert and click.
 - If you clicked on an arc, it is converted into a straight edge.
 - If you clicked on an edge, it is converted into an arc; move your mouse to position the arc and click.
6. Repeat the previous step to continue converting arcs or edges. To exit this function, click on another Edit menu option, or right-click until the Polyline menu is displayed again in the Menu Window.

Editing Polyline Corners

You can connect any two nonparallel edges in a polyline with either an arc or a line. The Fillets option connects two edges with an arc, automatically adjusting the line segments to match the arc radius and, if indicated, cleaning up the line extensions to form a smooth curve. Chamfer clips the corners of rectangular groups, connecting two joined sides with a line. By default, DataCAD uses a distance of 1'-0" for the first and second chamfer distance.

➔ To edit the vertices of polylines:

1. Select Curves from the Edit menu; then, choose Polyline.
2. Click on Edit in the Polyline menu; then, choose one of these options, depending on what type of vertex you want:
 - Fillets – Clicking on Radius lets you control the fillet.
 - Chamfer – Clicking on Distnces and selecting two values lets you control the chamfer.
3. Select the first edge of the polyline you want to fillet or chamfer. Then, click on the second.

➔ To connect two lines with an arc:

1. Choose Fillets from the Cleanup menu. The following options are displayed:
 - Radius Sets the fillet radius. To join two nonparallel, nonintersecting lines, accept the standard radius of zero. This extends the lines to intersect. After you set the radius, DataCAD saves it with the drawing file until you modify it again.
 - Clip Trims (erases) the line segments extending past the intersection of the selected lines; set Clip prior to selecting the entities to fillet
 - LyrSrch Scans all layers during selection operations when toggled on
2. Select the first line to trim. For more information on using selection menus, see "Selection Menus" in "The Drawing Board" chapter.

3. Select the second line to trim. DataCAD creates a fillet at the intersection.

➔ To connect two lines with a line:

1. Choose Chamfer from the Cleanup menu. The following options appear:
 - Distnces Sets the first and second chamfer distance
 - Clip Trims (erases) the line segments extending the intersection of the selected lines to chamfer
 - LyrSrch Scans all layers during selection operations when toggled on
2. Select the first line to trim. For more information on using selection menus, see “Selection Menus” in “The Drawing Board” chapter.
3. Select the second line trim. A chamfer is created at the intersection of the two lines.

Measuring Polylines

You can measure the area or perimeter of a closed polyline. If you are measuring land, you can use the Acres option. In addition, you can find the closed polyline’s centroid.

➔ To measure the area or perimeter of a closed polyline:

1. Select Curves from the Edit menu; then, choose Polyline and click on Edit.
2. Click on Area/Per.
3. Click on Select in the Area/Per menu and click on the closed polyline you want to measure.
4. Select the appropriate toggles. The Perim+ and Perim- options are mutually exclusive as are the Area+ and Area- toggles.
 - Acres – displays the area within the closed polyline as acreage.
 - Perim+ - displays the perimeter of the closed polyline.
 - Perim- - subtracts the perimeter of a void contained within a closed polyline. This is useful with windows and doors in elevations.
 - Area+ - displays the area of the closed polyline.
 - Area- - subtracts the area of the void(s) contained within a closed polyline.
5. Toggle on ContSrch.
6. Click on the closed polyline you want to measure. DataCAD traces the contour of the polyline and displays the measurement information in the Message Area.

➔ To find the centroid of an irregular closed polyline:

1. Select Curves from the Edit menu; then, choose Polyline and click on Edit.
2. Click on Area/Per to display the Area/Per menu.

3. Select ContSrch and click on the closed polyline.
4. Click on ShowIt to simply display a temporary crosshair or choose AddIt to put an actual dot in the center of your closed polyline.

Voids in Polylines

You can add voids to or erase voids from polylines as well as convert voids to polygons. To add a polyline void, you must first draw the closed master polyline (to which you want to add the void); then, draw the closed polyline that represents the void itself. For example, in a building elevation that you want to hatch, you would create a master polyline around the perimeter of the elevation. Then, you would draw closed polylines around each of the windows and doors; these would become voids or areas that you wouldn't want to hatch.

➔ To add a void to a polyline:

1. Click on Polyline in the Curves menu. The Polyline menu is displayed.
2. Draw your master, closed polyline. Then, draw the closed polyline(s) that will become your void(s) within that master polyline.
3. Click on Voids in the Polyline menu. You are prompted to "Select master polyline to process voids".
4. Click on the closed master polyline that you want to put the void in. DataCAD puts dashed lines around that shape. You are prompted to "Select entity to select polylines to convert to voids".
5. Make sure Entity and AddVoid are toggled on in the Voids menu.
6. Click on the polyline that will represent the void. The void is added. Dashed lines surround the void.
7. Continue selecting other closed polylines within the master if necessary.

You could easily hatch the polyline with voids. Just select Hatch from the Utility menu; click Pattern, choose one of the options, and press (Enter); then click Begin. The area within the master polyline is hatched, but the voids are not. Alternatively, you could fill the master polyline with solid fills or bitmaps.

➔ To erase a void polyline:

1. Click on Polyline in the Curves menu. The Polyline menu is displayed.
2. Click on Voids in the Polyline menu. "Select master polyline to process voids" appears in the Message Window.
3. Click on the master polyline that contains the void(s) you want to erase. The Voids menu appears.
4. Toggle ErasVoid on. You are prompted to "Select polyline to delete from drawing".
5. Click on a void to erase it. If you want to eliminate more voids, click on them, too.

➔ To convert a void to a polygon:

1. Click on Polyline in the Curves menu. The Polyline menu is displayed.

2. Click on Voids in the Polyline menu.
3. Toggle ConvVoid on. You are prompted to “Select void to convert to a polygon”.
4. Click on a void to convert it.

Drawing Stairs

The Stairs tool in the DCAD_AEC macro creates basic 2D single- and double-run stairs.

➔ To draw stairs:

1. Click on DCAD_AEC in the Macros pull-down menu. The DCAD_AEC menu is displayed in the Menu Window. If DCAD_AEC is not listed in the Macros pull-down menu, see “Customizing the Macros Menu” for details on how to add it to the menu. The Stairs menu is displayed in the Menu Window.
2. Click on Stairs in the DCAD_AEC menu.
3. Click on Up Only to draw stairs going up, click on Dn Only to draw stairs going down, or click on UpAndDn to draw bidirectional stairs.
4. Click on Up Left to draw stairs going up on the left side of the stair well, or click on Up Rght to draw stairs going up on the right side of the stair well.
5. Customize stair options if necessary:
 - To set the number of stair risers, click on NumRiser. Use the value menu or type a number, and press . For more information on using value menus, see “Value Menus” in “The Drawing Board” chapter.
 - To set the distance from the railing to the first riser, click on Offset. Use the value menu or type a number, and press .
 - To set the stair width including newel posts and stringers, click on StrWidth. Use the value menu or type a number, and press .
 - To set the width of the stair treads, click on TrdWidth. Use the value menu or type a number, and press .
 - To set the nosing overhang, click on Nosing. Use the value menu or type a number, and press .
 - To set the color of the stairs, click on Color and use the color menu. For more information on setting colors, see “Color Menus” in “The Drawing Board” chapter.
6. Click on Rails to customize the stair railings. Railing options are displayed in the Menu Window. For details on railing options, see “Stair Railing Options” below. You are prompted to “Select railing option”.
7. Click on a railing option to change the setting.

8. Click on StrWell to customize the stair well. Stair well options are displayed in the Menu Window. For details on stair well options, see “Stair Well Options” on the following page. You are prompted to “Select stair well option”.
9. Click on a stair well option to change the setting.
10. Click on SectnLin to customize the section lines that are drawn when UpOnly or UpAndDn is toggled on. Section line options are displayed in the Menu Window. For details on section line options, see “Section Line Options” on the following page. You are prompted to “Select stair variable option”.
11. Click on a section line option to change the setting. You are prompted to “Enter starting location of stair”.
12. Click in the Drawing Area or use coordinate entry, and press **[Enter]**. For more information on coordinate entry, see “Drawing Using Coordinate Entry” in “The Drawing Board” chapter. Notice that your cursor is now attached to the point you just entered. You are prompted to “Enter point along stairs”.
13. Enter a point along the stair line by clicking in the Drawing Area or using coordinate entry, and press **[Enter]**. You are prompted to “Point to inside of stair well”.
14. Click to indicate the inside of the stair well. The stairs are drawn.

Stair Railing Options

DrwRail	Toggle on to draw railings
Diameter	Set the railing diameter; use the value menu or type a number, and press [Enter]
Clearnce	Set the clearance from the wall (the space between the railing and the wall); use the value menu or type a number, and press [Enter]
RunOut	Set the railing runout (the extension past the last tread); use the value menu or type a number, and press [Enter]
Color	Set the color of the railings using the color menu. For more information on setting colors, see “Color Menus” in “The Drawing Board” chapter.

Stair Well Options

DrwWell	Toggle on to draw stair wells
LandLnth	Set the stair landing length; use the value menu or type a number, and press [Enter]
ChseWdth	Set the chase width (the distance between stair wells); use the value menu or type a number, and press [Enter]
Color	Set the color of the stair wells using the color menu

Section Line Options

GapWidth	Set the width of the gap between section lines; use the value menu or type a number, and press [Enter]
LinAngle	Set the angle of section lines; use the value menu or type a number, and press [Enter]
LinColor	Set the color of section lines using the color menu
DashSpchg	Set the dashed line spacing; use the value menu or type a number, and press [Enter]

Drawing Columns

With the Columns menu in the DCAD_AEC macro, you can draw a single column or multiple columns in straight or curved rows. Single columns are simply symbols inserted into your drawing. Multiple columns are defined on a rectangular or radial grid and drawn with column grid lines that are finished with labels identifying the ends of each grid line.

Drawing a Single Column

Single columns can be added by simply opening a template and choosing a column symbol to add to your drawing.

➔ To draw a single column:

1. Click on DCAD_AEC in the Macros pull-down menu. The DCAD_AEC menu is displayed in the Menu Window.
2. Click on Columns in the DCAD_AEC menu. If DCAD_AEC is not listed in the Macros pull-down menu, see “Customizing the Macros Menu” for details on how to add it to the menu. The Columns menu is displayed in the Menu Window.
3. Click on OneColumn in the Columns menu. A dialog box is displayed, prompting you to “Enter template filename”.
4. Click on a template of column symbols to open it or change to another folder to find the template you want. For more information about using templates and symbols, see the “Templates and Symbols” chapter.
5. Click on Open or press **[Enter]**. The dialog box closes, and a template window opens on the right side of the Drawing Area.
6. Click on the column symbol you want to use. Notice that a box representing the symbol is now attached to your cursor.
7. Click in the Drawing Area to add the symbol to your drawing. You can also use coordinate entry to place the symbol. For more information on coordinate entry, see “Drawing Using Coordinate Entry” in “The Drawing Board” chapter.

Drawing Rectangular Grids of Columns

One way to draw multiple columns is to define a rectangular grid. Columns are then placed according to the grid. Rectangular grids are defined by the number of column rows in both the X and Y directions.

➔ To draw a rectangular grid of columns:

1. Click on DCAD_AEC in the Macros pull-down menu. The DCAD_AEC menu is displayed in the Menu Window.
2. Click on Columns in the DCAD_AEC menu. The Columns menu is displayed in the Menu Window.
3. Click on RectGrid in the Columns menu. The Rectnglr menu is displayed in the Menu Window with all the options for creating rectangular grids.

4. Define the number of columns in the grid by clicking on X Rows to set the number of columns in the X direction or click on Y Rows to set the number of columns in the Y direction. A value menu is displayed; type a value or use the value menu, and press **[Enter]**. For more information on using value menus, see “Value Menus” in “The Drawing Board” chapter.
5. Click on X Spcg to set the distance between rows in the X direction, or click on Y Spcg to set the distance between rows in the Y direction. A value menu is displayed; type a value or use the value menu, and press **[Enter]**. Keep in mind that when the grid is rotated 90°, the “X direction” refers to the vertical and the “Y direction” refers to the horizontal. See Rotation under “Rectangular Grid Options” below for details on rotating the column grid.
6. Choose the column symbol for the grid by clicking on SymbName. A dialog box is displayed, prompting you to “Enter symbol filename”.
7. Click on a column symbol name to use that symbol; or change to another folder to find the symbol you want.
8. Click on Open or press **[Enter]**. The dialog box closes.
9. Use other options in the Rectnglr menu to customize how columns, the column grid, and the labels are drawn. See “Rectangular Grid Options” below for details. You are prompted to “Enter grid starting point or choose grid option”.
10. Enter the starting point for the grid by clicking in the Drawing Area or using coordinate entry. For more information on coordinate entry, see “Drawing Using Coordinate Entry” in “The Drawing Board” chapter. The columns are drawn.

Rectangular Grid Options

Keep in mind that when the column grid is rotated 90°, the X direction is vertical and the Y direction is horizontal.

X Rows	Set the number of rows in the X direction of the grid using the value menu
Y Rows	Set the number of rows in the Y direction of the grid using the value menu
X Spcg	Set the distance between the Y rows using the value menu
Y Spcg	Set the distance between the X rows using the value menu
SymbName	Choose the symbol for the columns you want to draw
Numbring	Customize numbering of grid rows as displayed in grid labels; toggle Lft2Rht on to number Y rows of the grid from left to right, or toggle it off to number Y rows from right to left; toggle Top2Bot on to letter X rows top to bottom, or toggle it off to letter X rows from bottom to top
Controls	Displays toggles for the top, bottom, left, and right sides of the grid: toggle GridLbl on to draw labels for that side of the grid; toggle ExtLine on to extend grid lines for that side
Rotation	Rotate the column grid, the column symbols or the label text; to rotate the grid, click on Rotation, click on ColmGrid, enter a rotation angle using the value menu, and press [Enter] ; to rotate column symbols, click on Rotation, click on Columns, enter a rotation angle using the value menu, and press [Enter] ; to rotate label text, toggle Text on (you can't set a text rotation angle). For more information on using value menus, see “Value Menus” in “The Drawing Board” chapter.
Layers	Displays options you can use to set the layers to draw column grid entities on: click on GridLayr and then click on the layer to draw the grid lines on; click on SymLayr and then click on the layer to draw the label symbols on; click on ColLayr and then click on the layer to draw the column symbols on

Colors	Displays three options you can use to set the colors to draw column grid entities in; click on GridColr and use the color menu to choose a color for the grid lines; click on SymColr and use the color menu to choose a color for the label symbols; click on ChrColr and use the color menu to choose a color for the label characters
LnFactor	Set the spacing within the linetype that you're using to draw the grid lines
LnExtnsn	Set the distance that the grid lines extend past the outer column; useful in conjunction with LblOffst, which sets the labels' extension line lengths
LblOffst	Set the distance of the lines extending from the labels to the end of the extension lines of the grid
CharSize	Set the size of the label text; for example, if you are plotting at $1/8'' = 1'$ scale and you want text to plot $1/4''$ high on the sheet, set this option to 2' (2 feet at $1/8''$ scale equals $1/4''$)
CharName	Set the starting number for the X rows and the starting letter for the Y rows
CharFont	Set the font to be used for label text; the font dialog box is displayed, listing all DataCAD fonts (normally found in your DATACAD\CHR folder)

Drawing Radial Grids of Columns

One way to draw multiple columns is to define a radial grid. Columns are then placed according to the grid. Radial grids are defined by the number of concentric column rows around a grid origin and the number of rows radially from the grid origin.

➔ To draw a radial grid of columns:

1. Click on DCAD_AEC in the Macros pull-down menu. The DCAD_AEC menu is displayed in the Menu Window.
2. Click on Columns in the DCAD_AEC menu. The Columns menu is displayed in the Menu Window.
3. Click on RadlGrid in the Columns menu. The Radial menu is displayed with all the options for creating circular grids. See "Radial Grid Options" later in this chapter for details on these options.
4. Define the number of concentric column rows in the grid (concentric circles around the grid origin) by clicking on CurvRows to set the number of concentric circles in the grid. A value menu is displayed; type a value or use the value menu, and press **[Enter]**.
5. Set the number of radial rows in the grid by clicking on RadlRows. A value menu is displayed; type a value or use the value menu, and press **[Enter]**. For more information on using value menus, see "Value Menus" in "The Drawing Board" chapter.
6. Set the distance between the curved rows (set in step 4) by clicking on CurvSpcg. A value menu is displayed; type a value or use the value menu, and press **[Enter]**.
7. Determine the angle between radial rows using any of three different methods:
 - Toggle Fill on and toggle IncAngl off. The angle between radial rows is calculated automatically based on where you place the first and last radial rows.

- Toggle both Fill and IncAngl on. When you click on IncAngl, a value menu is displayed, which you can use to enter the total included angle between the first and last radial rows. The angle between the remaining radial rows is calculated automatically.
 - Toggle Fill off and click on SepAngl. A value menu is displayed, which you can use to enter the separation angle between each radial row.
8. Choose the column symbol for the grid by clicking on SymbName. A dialog box is displayed, prompting you to “Enter symbol filename”.
 9. Click on a column symbol name to use that symbol; or change to another folder to find the symbol you want.
 10. Click on Open or press **[Enter]**. The dialog box closes.
 11. Use other options in the Radial menu to customize how columns, the column grid, and the labels are drawn. See “Radial Grid Options” for details and set these options as necessary. You are prompted to “Enter grid origin or choose grid option”.
 12. Enter the origin of the grid by clicking in the Drawing Area or using coordinate entry. For more information on coordinate entry, see “Drawing Using Coordinate Entry” in “The Drawing Board” chapter. You are prompted to “Indicate position of first column.”
 13. Enter the coordinates of the first column by clicking in the Drawing Area or using coordinate entry. If you set either the included angle (IncAngl) or the separation angle (SepAngl) in step 6, the columns are drawn. If you toggled Fill on as described in the first option in step 6, then you are prompted to “Indicate orientation of last radial row”.
 14. Position the last radial row by clicking in the Drawing Area or using coordinate entry. The columns are drawn.

Radial Grid Options

CurvRows	Set the number of curved rows, or concentric circles, around the grid origin
RadlRows	Set the number of radial rows extending out from the grid origin
CurvSpceg	Set the distance between curved rows
SepAngl	Set the (separation) angle between radial rows; available only when Fill is toggled off
IncAngl	Set the (included) angle between the first and last radial rows; available only when Fill is toggled on
Fill	Toggle on to automatically calculate the angle between radial rows
SymbName	Choose the symbol for the columns you want to draw
Numbring	Customize numbering of grid rows as displayed in grid labels: toggle Clkwise on to number the radial rows of the grid in a clockwise direction, or toggle it off to number radial rows counterclockwise; toggle In2Out on to letter the curved rows from the center of the grid outward, or toggle In2Out off to letter curved rows from the outermost row inward
Controls	Displays toggles for the first and last radial rows: toggle GridLbl on to draw labels for the first or last radial row; toggle ExtLine on to draw grid lines for the first or last radial row; toggle Columns on to draw column symbols in the first or last radial row
Rotation	Rotate column symbols or label text: to rotate column symbols, click on Rotation, click on Columns, enter a rotation angle using the value menu, and press [Enter] ; to rotate label text, toggle Text on (you can’t specify a text rotation angle). For more information on using value menus, see “Value Menus” in “The Drawing Board” chapter.

Layers	Displays three options you can use to set the layers to draw column grid entities on: click on GridLayr and then click on the layer to draw the grid lines on; click on SymLayr and then click on the layer to draw the label symbols on; click on ColLayr and then click on the layer to draw the column symbols on
Colors	Displays three options you can use to set the colors to draw column grid entities in: click on GridColr and use the color menu to choose a color for the grid lines; click on SymColr and use the color menu to choose a color for the label symbols; click on ChrColr and use the color menu to choose a color for the label characters
LnFactor	Set the spacing within the linetype you're using to draw the grid lines
LnExtnsn	Set the distance of the grid lines extend past the outer column; useful in conjunction with LblOffst, which sets the labels' extension line lengths. The LnExtnsn option is useful to extend the grid lines so that grid labels do not overlap another entity in your drawing.
LblOffst	Set the distance of the lines extending from the labels to the end of the extension lines of the grid
CharSize	Set the size of the label text; for example, if you are plotting at $1/8'' = 1'$ scale and you want text to plot $1/4''$ high on the sheet, set this option to 2' (2 feet at $1/8''$ scale equals $1/4''$)
CharName	Set the starting number for the radial rows and the starting letter for the curved rows. The CharName option is useful when you construct a composite grid in which the first columns are not necessarily "1" or "A".
CharFont	Set the font to be used for label text; the font dialog box is displayed, listing all DataCAD fonts (normally found in your DATACAD\CHR folder)

Drawing Elevators

With the Elevator option in the DCAD_AEC macro, you can draw a single elevator or banks of elevators. Start by customizing the capacities, doors, shaft walls, and cabs of the elevator banks, then select points to draw the elevators.

➔ To draw elevators:

1. Click on DCAD_AEC in the Macros pull-down menu. The DCAD_AEC menu is displayed in the Menu Window.
2. Click on Elevator in the DCAD_AEC menu. The Elevator menu is displayed; use the Num Cab, Capacity, Cab, Walls, Doors, ByFront, and ByBack options to customize your elevators.
3. Click on Num Cab. A value menu is displayed, and you are prompted to "Enter number of elevator cabs".
4. Type the number of elevator cabs you want in the bank of elevators or use the value menu to enter the number of cabs, and press .
5. Click on Capacity. A list of capacity values in pounds is displayed in the Menu Window. You are prompted to "Select standard elevator capacity".
6. Click on the capacity value. Based on the capacity value you choose, DataCAD automatically resizes the elevator dimensions to match those found in Architectural Graphic Standards; you can, however, modify these dimensions using the Cab, Walls, and Doors options. You are returned to the Elevator menu.
7. Click on Cab to customize the elevator cab. Cab options are displayed in the Menu Window. You are prompted to "Select cab variable to change".
8. Click on a cab option to change the setting. For all cab options except Color, a value menu listing clearance distances is displayed; when you click on Color, a color menu is displayed.

9. Use the value menu to enter a clearance distance for the cab option or type a new value, and press **[Enter]**; use the color menu to select a color for the elevator cab lines.
10. Click on Walls to customize the elevator shaft walls. Walls options are displayed in the Menu Window. For details on Walls options, see “Elevator Shaft Wall Options” below. You are prompted to “Select wall variable to change”.
11. Change wall dimensions by clicking on Front, Back, Left, Right, or Between. A value menu listing wall dimensions is displayed. Use the value menu to enter a dimension for the wall option or type a new value, and press **[Enter]**.
12. Click on Doors to customize the elevator doors. Door options are displayed in the Menu Window. For details on Doors options, see “Elevator Door Options” below. You are prompted to “Select door variable to change”.
13. Click on a door option to change the setting.
14. Position the elevator bank by defining the front or the back wall. When you toggle ByFront on, the two points you select in the following steps define the outside face of the front elevator shaft wall; when you toggle ByBack on, the two points you select will define the outside face of the back elevator shaft wall. Only one of the options, ByFront and ByBack, can be toggled on at any one time. If ByFront is toggled on, you are prompted to “Enter one front corner of elevator bank”. If ByBack is toggled on, you are prompted to “Enter one back corner of elevator bank”.
15. Begin drawing the elevator banks. To select the corner of the elevator bank, click in the Drawing Area or use coordinate entry. For more information on coordinate entry, see “Drawing Using Coordinate Entry” in “The Drawing Board” chapter. (The elevator bank will be drawn starting at this corner.) Your cursor is now attached to this point, and you are prompted to “Indicate direction of elevator bank”.
16. Notice the “direction” of the elevator bank or the bearing of the row of elevators. Position the line now attached to your cursor so that it’s parallel to the front and back of the elevator bank, and click. You are prompted to “Point to back side of elevator bank” if ByFront is toggled on; you are prompted to “Point to front side of elevator bank” if ByBack is toggled on.
17. Click anywhere towards the front or back of the bank, as prompted. The bank of elevators is drawn.

Elevator Cab Options

ClearFrt	Set the clearance distance between the front of the elevator cab and the inside face of the elevator shaft wall
ClearBck	Set the clearance distance between the back of the elevator cab and the inside face of the elevator shaft wall
ClearLft	Set the clearance distance between the left side of the elevator (as you face the back of the cab) cab and the inside face of the elevator shaft wall

ClearRht	Set the clearance distance between the right side of the elevator cab and the inside face of the elevator shaft wall
ClearBtw	Set the clearance distance between elevator cabs when the separating shaft wall between the cabs is not drawn
CabWidth	Set the width of the elevator cab using the value menu
CabDepth	Set the depth of the elevator cab using the value menu
Color	Set the color of the elevator cab lines using the color menu. For more information on setting colors, see “Color Menus” in “The Drawing Board” chapter.

Elevator Shaft Wall Options

Front	Set the width of the front wall of the elevator shaft using the value menu
Back	Set the width of the back wall of the elevator shaft using the value menu
Left	Set the width of the left wall of the elevator shaft using the value menu
Right	Set the width of the right wall of the elevator shaft using the value menu
Between	Set the width of the wall between elevator cabs using the value menu
DrwBack	Toggle on to draw the back wall of the elevator shaft when the elevator banks are drawn; when toggled off, the back wall of the elevator shaft is not drawn and the side walls of the elevator shaft are extended a distance equal to half the width of the back wall (in other words, side walls are drawn to the center line of the back wall). Toggling DrwBack off allows for easy creation of back-to-back elevator banks, since two halves of a back wall put together equal one party wall between back-to-back cabs.
DrwCntr	Toggle on to draw the elevator shaft wall between elevators when the elevator banks are drawn; when toggled off, these center walls are not drawn. Toggling DrwCntr on increases the center-to-center distance between elevators.
Color	Set the color of the elevator shaft walls using the color menu

Elevator Door Options

None	Toggle on to draw elevators without doors
Single	Toggle on to draw elevators with single doors
Double	Toggle on to draw elevators with double doors
Lap	Toggle on to draw elevators with overlapping doors
AtFront	Toggle on to draw doors at the front of the elevator bank. Toggle both AtFront and AtBack on to draw elevators with doors at both the front and the back of the elevator bank. Toggling both options off is the equivalent of toggling None on: no elevator doors are drawn.
AtBack	Toggle on to draw doors at the back of the elevator bank
Alternat	Toggle on to alternate drawing overlapping doors, with the left panel outside and the left panel inside, on every other elevator; when the door is offset from the center of the cab (using the Offset option), toggling Alternat on causes the first elevator to be drawn with the doors offset to the right, and the next to be drawn with the doors offset to the left, creating the effect of pairs of elevators based on the proximity of their doors
Width	Set the width of the elevator doors using the value menu
Thicknss	Set the thickness of the elevator doors using the value menu
Offset	Elevator doors are drawn off-center by the distance specified. The <i>offset distance</i> is the distance along the front wall from the center of the cab to the center of the elevator door. An offset distance of 1'-0" center the door one foot to the right of the center of the cab, while an offset distance of -1'-0" centers the door one foot to the left of the center of the cab.
Color	Set the color of the elevator doors using the color menu. For more information on setting colors, see “Color Menus” in “The Drawing Board” chapter.

Other Ways to Draw

Drawing lines and entering points directly in the Drawing Area are the most common ways of drawing. However, DataCAD has other tools that can significantly increase your productivity and even the accuracy of your drawings.

The Copy and Mirror options in the Edit menu are two such tools that can add entities to your drawing with just a few steps, freeing you from drawing similar things over and over.

DataCAD even offers a tool that can be used to sketch your ideas before you begin drawing. With the FreeHand tool in the Utility menu, you can sketch in a variety of line widths and linetypes.

In this chapter:

- ⊕ Copying entities
- ⊕ Making multiple copies of entities
- ⊕ Mirroring entities
- ⊕ Sketching

Copying Entities

There are two ways to copy DataCAD entities. The Copy option in the Edit pull-down menu (grouped with Cut and Paste) is a standard Windows command and can be used to quickly copy entities to other DataCAD drawings or to other Windows applications. For details on how to use Cut, Copy, and Paste, see “Basic Editing Tools” in “The Drawing Board” chapter. The Copy option in the Edit menu in the Menu Window (also available just below the Move option in the Edit pull-down menu) duplicates entities at a specified distance within the same drawing. With this option, you can:

- copy entities at a distance and angle that you set;
- copy entities by dragging them
- copy entities to a different layer in your drawing; or
- make multiple copies of selected entities, arranged in rectangular or circular arrays

To copy entities in the Z direction, use the AndCopy option in the Move menu. See “Moving Entities” in the “Editing Drawings” chapter for details..

Copying Entities a Specified Distance and Angle

When you use Copy in the Edit menu in the Menu Window, you are prompted to enter two points. The distance between these two points will be the distance the copy is drawn from the original selected entity; the angle defined by these two points will be the angle the copy is drawn from the original selected entity.

➔ To copy an entity a specified distance and angle:

1. Click on Copy in the Edit menu in the Menu Window, or click on Copy (located just below the Move option) in the Edit pull-down menu. The Copy menu is displayed in the Menu Window, and you are prompted to “Select first point of the distance to copy”. To use the same distance and angle as the last time you used Copy, click on PrevDist.
2. Click in the Drawing Area or use coordinate entry to enter the first point. A directional arrow is displayed; notice that moving your mouse changes the length and angle of the arrow. You are prompted to “Select second point of the distance to copy”. The points you select in steps 2 and 3 can be selected anywhere on the screen; they indicate the distance and angle to copy only, and **not** the beginning and ending points of the copy.
3. Click in the Drawing Area or use coordinate entry to enter the second point. For more information on using coordinate entry, see “Drawing Using Coordinate Entry” in “The Drawing Board” chapter. A selection menu as well as other Copy options are displayed in the Menu Window.
4. Click on a selection method in the menu. For more information on using selection menus, see “Selection Menus” in “The Drawing Board” chapter. You are prompted to select the entities you want to copy.

5. Select the entities; they are copied in the drawing. You can continue selecting entities to make additional copies at the same distance and angle.
6. Copy entities at the same distance and angle but in the opposite direction by clicking on Invert and then selecting the entities to be copied (see Figure 10.1).
7. Enter a new distance and angle by clicking on NewDist and continuing with step 2 above.

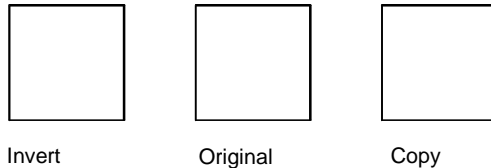


Figure 10.1: Inverting a copy

Copying Entities by Dragging

Instead of entering a specific distance and angle to copy entities, you can simply drag the copy any distance and at any angle.

➔ To copy entities by dragging them:

1. Click on Copy in the Edit menu. The Copy menu is displayed in the Menu Window.
2. Click on Drag in the Copy menu. The Drag menu is displayed in the Menu Window.
3. Toggle LyrSrch on if the entities you want to copy are on a different layer than the active layer.
4. Drag complex entities more smoothly by setting the maximum number of lines to be displayed while dragging. Click on MaxLines and use the value menu to enter the number of lines. Press **[Enter]** or right-click to return to the Drag menu. When the entities you selected to drag contain more lines than the MaxLines setting, a rectangular box representing the selected entities is displayed when you begin to drag them. The entities reappear after you select the new location for them. When you set MaxLines to 0, the rectangular box is always displayed while dragging.
5. Toggle Multi on to make multiple selections of entities to be copied.
6. Click on a selection method in the menu. You are prompted to select the entities you want to copy.
7. Select the entities to be copied. You are prompted to select a point to drag from.
8. Check the button for Multi. If it is toggled off, skip to step 9. If you toggled Multi on in step 5, click on Begin and continue with step 9.

9. Click on a point in the Drawing Area; you can also use coordinate entry or object snapping to select this point. For more information on using coordinate entry, see “Drawing Using Coordinate Entry” in “The Drawing Board” chapter.
10. Move your mouse to drag the entities to their new position and click to place them. The entities are copied in the drawing.

Copying Entities at the Perpendicular

You can copy an entity a specific distance from and perpendicular to the original. While you can't copy an entity at any angle, as you can with the Copy function described above, this is a very straightforward way to quickly make one entity into an entire row.

➔ To copy an entity perpendicular to the original:

1. Click on Geometry in the Tools pull-down menu, and then click on Offset. The Offset menu is displayed in the Menu Window.
2. Set the copy distance using one of the following methods:
 - You can define the distance just as you did in steps 2 and 3 in the “Copying Entities a Specified Distance and Angle” on the previous page. Click in the Drawing Area or use coordinate entry to select two points to define the distance.
 - You can toggle Dynamic on; you'll be able to see the position of the copy before you place it in your drawing.
 - You can click on PerpDist to display a value menu. Use the value menu or type distance and press Enter.
3. Toggle LyrSrch on if the entities you want to copy are on a different layer than the active layer.
4. Click on the entity you want to copy. The selected entity is displayed in dashed lines. If you toggled Dynamic on in step 2, a copy of the entity is attached to your cursor. You are prompted with “Offset to which side?”
5. Click to one side or the other of the original entity. The copy is placed.
6. Continue copying entities by repeating steps 4 and 5; or if Dynamic is toggled on, simply repeat step 5. To redefine the offset distance, click on NewDist and continue with step 2 above.

Copying to Another Layer

When you have more than one layer in your drawing, the ToLayer option is displayed in the Copy menu. ToLayer is used to copy entities from one layer to another in your drawing. You can copy entities in the same position as the original layer (no distance and angle entered) or you can copy entities to another layer at a specified distance and angle. For more information on creating layers, see “Creating Layers” in the “Before You Draw” chapter.

➔ To copy entities to another layer without changing their positions:

1. Click on Copy in the Edit menu in the Menu Window, or click on Copy (located just below Move) in the Edit pull-down menu. The Copy menu is displayed in the Menu Window.
2. Click on ToLayer in the Copy menu. A list of all the layers in your drawing is displayed in the Menu Window, and you are prompted to "Select layer to copy to".
3. Click on the layer you'd like to copy entities to, or type the name of the layer, and press **[Enter]**. The ToLayer menu is displayed in the Menu Window. Use Match and Filter to select or find the layer you want to copy to. For details on how to use these options, see "Selecting Layers Without Using Layer Names" and "Searching for Layers" in the "Before You Draw" chapter.
4. Click on a selection method in the menu. You are prompted to select the entities you want to copy.
5. Select the entities; they are copied to the layer you chose in step 3. You can continue selecting entities to make additional copies.
6. Choose another layer to copy entities to by clicking on NewLyr and continuing with step 3 above.

➔ To copy entities to another layer at a specific distance and angle:

1. Click on Copy in the Edit menu in the Menu Window, or click on Copy (located just below the Move option) in the Edit pull-down menu. The Copy menu is displayed in the Menu Window, and you are prompted to "Select first point of the distance to copy". To use the same distance and angle as the last time you used Copy, click on PrevDist.
2. Click in the Drawing Area or use coordinate entry to enter the first point. A directional arrow is displayed; notice that moving your mouse changes the length and angle of the arrow. You are prompted to "Select second point of the distance to copy". The points you select in steps 2 and 3 can be selected anywhere on the screen; they indicate the distance and angle to copy only, and **not** the beginning and ending points of the copy.
3. Click in the Drawing Area or use coordinate entry to enter the second point. A selection menu as well as other Copy options are displayed in the Menu Window.
4. Click on ToLayer in the Copy menu. A list of all the layers in your drawing is displayed in the Menu Window, and you are prompted to "Select layer to copy to".
5. Click on the layer you'd like to copy entities to, or type the name of the layer, and press **[Enter]**. A selection menu as well as other Copy options are displayed in the Menu Window. For more information on using selection menus, see "Selection Menus" in "The Drawing Board" chapter.
6. Click on a selection method in the menu. You are prompted to select the entities you want to copy.

7. Select the entities; they are copied to the layer you chose in step 5. You can continue selecting entities to make additional copies.
8. Choose another layer to copy entities to if necessary. Just click on NewLyr and continue with step 5 above.

Copying Using Arrays

You can make multiple copies of an entity in a one-dimensional, rectangular, or circular array. A one-dimensional array simply makes multiple copies in one direction. In a circular array, an entity is copied a specified number of times at a specified angle of separation around a center of rotation.

A rectangular array repeats an entity a specified number of times at a specified angle. You can copy an array of elements in the X direction, the Y direction, or in both directions at the same time. The number of X and Y repetitions determines the total number of entities in the array. An array of one X repetition and one Y repetition has only one entity, the original. A two-by-two array has a total of four entities. (See figures 10.2 and 10.3.) You can also change the angle of the X and Y axes to create a rectangular array at the specified angle.

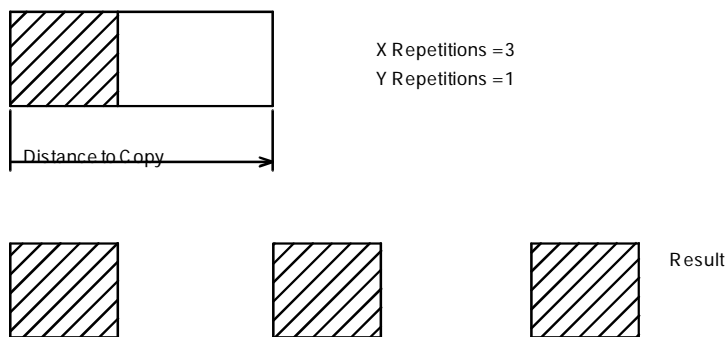


Figure 10.2: Creating a rectangular array in only the X direction. You can also use Array in the Copy menu to make multiple copies in only one direction. See the instructions for copying an entity in a one-dimensional array below.

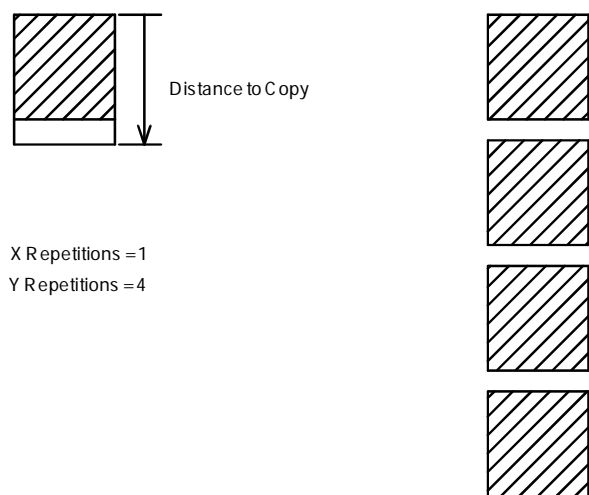


Figure 10.3: Creating a rectangular array in only the Y direction. You can also use Array in the Copy menu to make multiple copies in only one direction. See the instructions for copying an entity in a one-dimensional array below.

➔ To copy an entity in a one-dimensional array:

1. Click on Copy in the Edit menu in the Menu Window, or click on Copy (located just below the Move option) in the Edit pull-down menu. The Copy menu is displayed in the Menu Window, and you are prompted to “Select first point of the distance to copy”.
2. Click in the Drawing Area or use coordinate entry to enter the first point. A directional arrow is displayed; notice that moving your mouse changes the length and angle of the arrow. You are prompted to “Select second point of the distance to copy”. The points you select in steps 2 and 3 can be selected anywhere on the screen; they indicate only the distance and direction of each copy in the array, and **not** the beginning and ending points of the copy.
3. Click in the Drawing Area or use coordinate entry to enter the second point. A selection menu as well as other Copy options are displayed in the Menu Window.
4. Click on Array in the Copy menu. A value menu is displayed, and you are prompted to “Enter the number of repetitions”.
5. Use the value menu or type a number, and press Enter. You are prompted to select the entities you want to copy.
6. Select the entities; they are copied in the array you defined. You can continue selecting entities to make additional copies.

➔ To copy an entity in a rectangular array:

1. Click on Copy in the Edit menu in the Menu Window, or click on Copy (located just below the Move option) in the Edit pull-down menu. The Copy menu is displayed in the Menu Window.
2. Click on RectArry in the Copy menu. The RectArry menu is displayed in the Menu Window, and you are prompted to “Select first point of the distance to copy”. Toggle Counter on in the RectArry menu to display the number of entities added each time you copy.
3. Click in the Drawing Area or use coordinate entry to enter the first point. Two bi-directional arrows are displayed, representing the X and Y axes. Notice that moving your mouse changes the length and direction of the arrows; the length and direction of the arrows indicates the distance and angle you want to copy entities at. You are prompted to “Select second point of the distance to copy”. The points you select in steps 3 and 4 can be selected anywhere on the screen; they indicate only the distance and direction of each copy in the array, and **not** the beginning and ending points of the copy.
4. Click in the Drawing Area or use coordinate entry to enter the second point. A value menu is displayed in the Menu Window, and you are prompted to “Enter number of X repetitions”. For more information on using coordinate entry, see “Drawing Using Coordinate Entry” in “The Drawing Board” chapter.
5. Use the value menu or type a number, and press Enter. You are prompted to “Enter number of Y repetitions”.

6. Use the value menu or type a number, and press **Enter**. The RectArray menu is displayed in the Menu Window. For more information on using value menus, see “Value Menus” in “The Drawing Board” chapter.
7. Click on a selection method in the menu. You are prompted to select the entities you want to copy.
8. Select the entities; they are copied in the array you defined. You can continue selecting entities to make additional copies.
9. Click on NewArray to define another rectangular array and continue with step 3 above. Or click exit to return to the Edit menu.

Keep in mind that an array is a separate group from the original copied entity. If you use Group in the selection menu and select an entity in an array, the entire array will be copied.

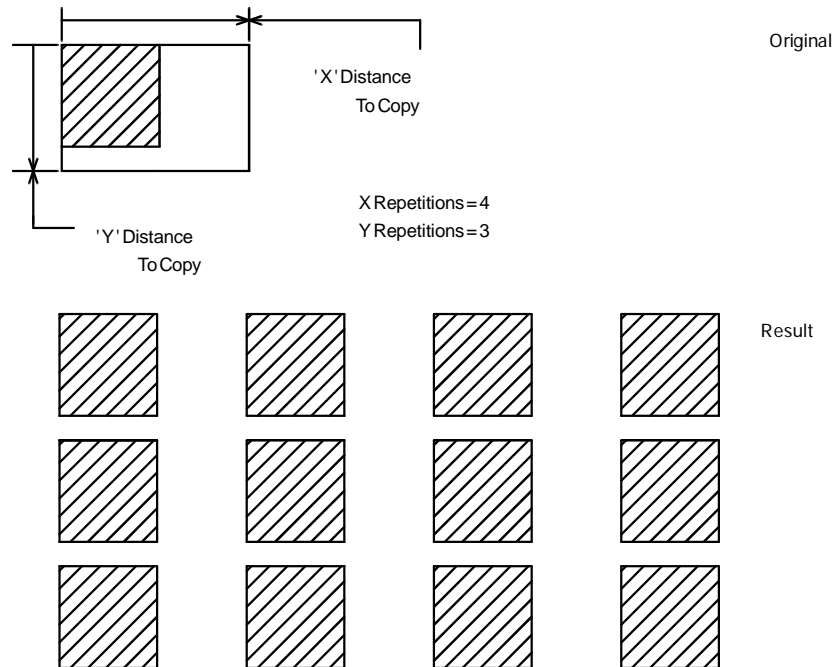


Figure 10.4: Creating a rectangular array

➔ To copy an entity in a rectangular array set at an angle:

1. Click on Copy in the Edit menu in the Menu Window, or click on Copy (located just below the Move option) in the Edit pull-down menu. The Copy menu is displayed in the Menu Window.
2. Click on RectArray in the Copy menu. The RectArray menu is displayed in the Menu Window.
3. Click on Angular in the RectArray menu. You are prompted to “Select first point of the distance to copy”.

4. Enter the angle for the X axis for the copy by defining the beginning and end points of a directional arrow. The points you select for directional arrows can be selected anywhere on the screen; they indicate only the distance and direction of each copy in the array, and **not** the beginning and ending points of the copy. Click in the Drawing Area or use coordinate entry to enter the beginning point. A directional arrow is displayed; notice that moving your mouse changes the angle of the arrow. You are prompted to "Select vector indicating angle of array". For more information on using coordinate entry, see "Drawing Using Coordinate Entry" in "The Drawing Board" chapter.
5. Click in the Drawing Area or use coordinate entry to enter the end point of the arrow and, therefore, define the angle. Two bi-directional arrows are displayed, representing the X and Y axes. Notice that moving your mouse changes the length and direction of the arrows; the length and direction of the arrows indicates the distance and angle you want to copy entities at. You are prompted to "Select the second point of the distance to copy".
6. Click in the Drawing Area or use coordinate entry to enter the second point. A value menu is displayed in the Menu Window, and you are prompted to "Enter number of X repetitions". For more information on using value menus, see "Value Menus" in "The Drawing Board" chapter.
7. Use the value menu or type a number, and press . You are prompted to "Enter number of Y repetitions".
8. Use the value menu or type a number, and press . The RectArray menu is displayed in the Menu Window. Toggle Counter on in the RectArray menu to display the number of entities added each time you copy.
9. Click on a selection method in the menu. You are prompted to select the entities you want to copy.
10. Select the entities; they are copied in the array you defined. You can continue selecting entities to make additional copies.
10. Click on NewArray to define another rectangular array and continue with step 3 above. Or click exit to return to the Edit menu.

Keep in mind that an array is a separate group from the original copied entity. If you use Group in the selection menu and select an entity in an array, the entire array will be copied.

➔ To copy an entity in a circular array:

1. Click on Copy in the Edit menu in the Menu Window, or click on Copy (located just below the Move option) in the Edit pull-down menu. The Copy menu is displayed in the Menu Window.
2. Click on CircArray in the Copy menu. The CircArray menu is displayed in the Menu Window, and you are prompted to "Select center point of array". For more information on circular array options, see "CircArray Menu Options" later in this chapter.

3. Click in the Drawing Area or use coordinate entry to enter the center point. You are prompted to “Select center of object”.
4. Click in the Drawing Area or use coordinate entry to enter the center of the entities you’re copying. A value menu is displayed in the Menu Window, and you are prompted to “Select angle between items (+=CCW, -=CW)”. “+=CCW” means that entering a positive angle will copy the array in a counterclockwise direction; “-=CW” means that entering a negative angle will copy the array clockwise.
5. Use the value menu or type a number, and press . Another value menu is displayed, and you are prompted to “Enter number of objects in array”. For more information on using value menus, see “Value Menus” in “The Drawing Board” chapter.
6. Use the value menu or type a number, and press . The CircArray menu is displayed in the Menu Window.
7. Click on a selection method in the menu. You are prompted to select the entities you want to copy.
8. Select the entities; they are copied in the array you defined. You can continue selecting entities to make additional copies. When you are finished, click exit to return to the Edit menu.

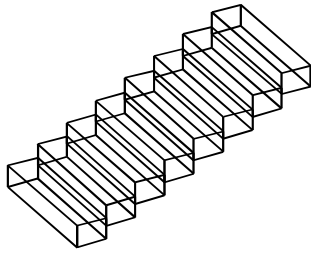
Keep in mind that an array is a separate group from the original copied entity. If you use Group in the selection menu and select an entity in an array, the entire array will be copied.

More About Using Z Elevations in Rectangular Arrays

The Z elevation can be increased or decreased incrementally with each repetition in a rectangular array. In the RectArray menu, the ZIncrX option sets the Z elevation in the X direction of an array, while the ZIncrY option sets the Z elevation in the Y direction of an array. To set an incremental Z elevation for an array that copies an entity in both the X and Y directions, set both options.

The ZIncrX and ZIncrY options are set using value menus. For more information on using value menus, see “Value Menus” in “The Drawing Board” chapter.

You can use the ZIncrX and ZIncrY to generate such things as three-dimensional stairs with only a few keystrokes (see Figure 10.5). You can also generate several three-dimensional floors of a building by setting ZIncrX or ZIncrY to the floor-to-floor height for the building and copying a rectangular array with an X and Y distance of zero. This creates an array of floors, stacked one above the other.



X Repetitions = 1 Z Incr X = 0
 Y Repetitions = 8 Z Incr Y = 8"

Figure 10.5: Creating 3D stairs with the incremental Z elevation options

More About CircArray Menu Options

You can change array settings, set a Z elevation for the array, and even rotate the entity copies using the following CircArray menu options:

NewCentr	Sets a new center for the array you are copying; click on NewCentr, select a new center point for the array, and then select the entity to copy
ObjCent	Sets a new center for the entity you are copying; click on ObjCent, select a new center point for the entity, and then select the entity to copy
Sep.Angl	Sets a new angle of separation between each entity in the array; click on Sep.Angl, enter a new angle, and press [Enter]
No.Object	Sets a new number of repetitions for the array; click on No.Object, enter a new number, and press [Enter]
Z-Incr	Sets the Z elevation with each repetition in the array; click on Z-Incr, enter a positive value to increase the elevation or a negative value to decrease it, and press [Enter] (use this option to draw 3D circular stairs)
Rotate	Toggle on to rotate each copy in the array by an angle equal to the angle of separation; toggle off to leave the orientation of each copy in the array the same as the original entity

Mirroring Entities

In some projects, being able to copy and mirror part of your drawing would save you a lot of time. For instance, it would be far more efficient to draw only one side of a hotel corridor and mirror that side to create the full corridor than it would be to draw both sides. The Mirror option in the Edit menu allows you to do just that, mirroring geometry across a reflection line that you define.

➔ To mirror entities:

1. Click on Mirror in either the Edit menu in the Menu Window or the Edit pull-down menu. The Mirror menu is displayed in the Menu Window, and you are prompted to "Select first point along the line of reflection". You can use the same reflection line as the last time you used Mirror. Instead of defining a new reflection line, click on PrevLine in the Mirror menu. The PrevLine option is only available if you have used Mirror during the current drawing session.
2. Click in the Drawing Area or use coordinate entry to select the first point of the reflection line. You are prompted to "Select second point along the line of reflection".

3. Click in the Drawing Area or use coordinate entry to select the second point of the reflection line. A temporary, dashed line representing the reflection line is drawn, and the Mirror menu is displayed in the Menu Window.
4. Determine if the reflection line is not correct. If it is, click on NewLine and repeat steps 2 and 3.
5. Mirror *and* copy the selected entities across the reflection line by toggling AndCopy on. Toggle AndCopy off to mirror and *move* selected entities across the reflection line.
6. Toggle FixText on to mirror the position of text but not its orientation. When FixText is toggled off, text is mirrored and rotated 180°. (See Figure 10.6)
7. Click on a selection method in the Mirror menu. You are prompted to select the entities you want to mirror.
8. Select the entities; they are mirrored across the reflection line you defined. You can continue selecting entities to mirror. When you are finished, click exit to exit the Mirror menu and return to the Edit menu.

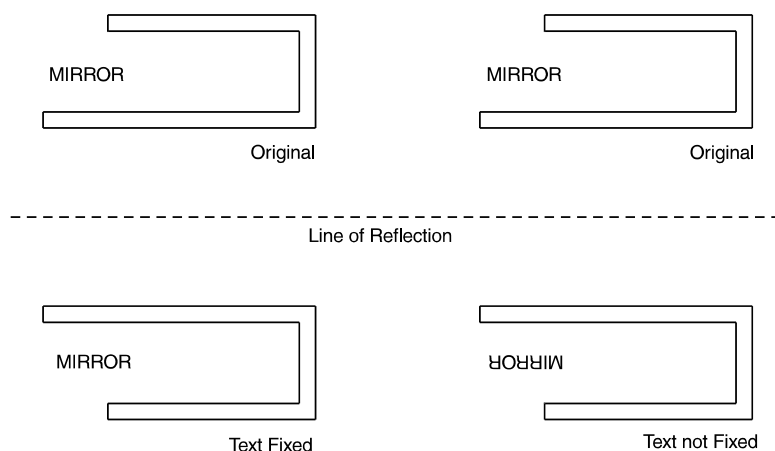


Figure 10.6: Mirroring text with FixText toggled on (left) and toggled off (right)

Sketching Your Plan

You can use the FreeHand option to sketch lines in varying widths and linetypes. When using linetypes such as dotted or dashed, you can give a rendered or weighted quality to the lines by varying the speed that you move the cursor across the Drawing Area as you draw. Sketching with FreeHand works best if you toggle both grid snapping and ortho mode off. Press **[X]** to toggle grid snapping off, and press **[O]** to toggle ortho mode off.

➔ To sketch:

1. Click on FreeHand in the Utility menu in the Menu Window. The FreeHand menu is displayed in the Menu Window.
2. Toggle Lines on to draw lines, toggle 3DLines on to draw three-dimensional lines, or toggle PolyLine on to draw polylines. Of the Lines, 3DLines, and PolyLine options, only one can be toggled on at any given time.

3. Toggle Closed on to automatically close the shape when you finish drawing.
4. Click in the Drawing Area and then move your mouse to draw.
5. Click to disconnect your cursor from the line when your sketch is complete. You can continue drawing additional shapes or click Exit to return to the Utility menu.
- 6.

Drawing Tools

11

There are many tools in DataCAD to aid in drawing. Reference points can be set anywhere in your drawing, giving you a precise way to position new entities. Object snapping, too, can help you add entities to existing geometry quickly and accurately.

You can also identify and measure entities in your drawing to retrieve information, such as what layer the entity is drawn on, it's linetype and line weight, it's length or angle, or even its area or perimeter measurements.

You can even use the LyrUtil macro to recover your work if your drawing file somehow becomes corrupted.

In this chapter:

- ⊕ Setting reference points
- ⊕ Object snapping
- ⊕ Identifying entities
- ⊕ Measuring entities
- ⊕ Dividing entities
- ⊕ Editing entity attributes
- ⊕ Recovering corrupted drawings

Setting Reference Points

A *reference point* is used as a starting point for drawing. In DataCAD, there are two kinds of reference points: floating and fixed.

Floating reference points are useful when you use relative coordinate entry methods to draw. They can be set at any time, regardless of what menu is displayed or what you're drawing, by using the reference point shortcut noted in the sidebar. Floating reference points are temporary and can be displayed and snapped to, but they can't be plotted.

To see how floating reference points are used when drawing, see the "Tutorial: Drawing a Schoolhouse" chapter, specifically the instructions on drawing interior walls and adding windows and doors to the schoolhouse.

Fixed reference points are useful when you use absolute coordinate entry methods to draw. Absolute coordinate entry uses absolute zero as a fixed point of reference: every time you enter a point using absolute coordinate entry, the distances you enter are measured from absolute zero. With a fixed reference point, you can use another point in your drawing, exactly as you would use absolute zero.

➔ To set a floating reference point:

Click on Reference point in the Tools pull-down menu. The RefPnt menu is displayed in the Menu Window.

Shortcut: Press `[]` (located in the upper-left corner of most keyboards, just below `[Esc]`) to quickly access the RefPnt menu and set a reference point. After selecting a point, you are automatically returned to the menu you were working in before you set the point.

2. Click on DrwMarks to display a small "x", representing the reference point.
3. Click in the Drawing Area, use coordinate entry, or object snap to a point in your drawing to select the reference point. The reference point is marked with a small "x". Reference points are displayed until you refresh your drawing by pressing `[Esc]`.

➔ To set a fixed reference point:

1. Click on Settings in the Utility menu. The Settings menu is displayed in the Menu Window.
2. Click on FixedRef in the Settings menu. You are prompted to "Select fixed reference point using cursor".
3. If you'd like the fixed reference point marked with a small "x", toggle DrwMarks on in the Settings menu.
4. Click in the Drawing Area, use coordinate entry, or object snap to a point in your drawing to select the reference point. If DrwMarks was toggled on in the previous step, the reference point is marked with a small "x". Reference points are displayed until you refresh your drawing by pressing `[Esc]`.

5. Right-click to exit the Settings menu. You can now continue drawing, using your fixed reference point.
6. Use absolute zero again if necessary. Instead of your fixed reference point, click on Settings in the Utility menu, toggle FixedRef off and then on again (click on the option twice), and then click on AbsZero. Right-click to exit the Settings menu, and continue drawing.

More About Setting Reference Points

Reference points can give you a quick way to draw very precisely. For example, to insert a window in a wall 3'-0" from the corner of a room, you could create a reference point at that corner and then, using relative Cartesian for coordinate entry, simply enter the distance from the reference point, which in this case is 3'-0".

➔ To use a reference point, using this example:

1. Click on Input Mode in the Tools pull-down menu and make sure relative Cartesian is checked.
2. Click on Window in the Create pull-down menu, or click Windows in the Edit/Architct menu in the Menu Window. The Windows menu is displayed.
3. Press []; the RefPnt menu is displayed in the Menu Window.

Shortcut: To set a reference point using the mouse, press [Ctrl] and click. To set a reference point at a specific point on an entity by object snapping to it, press [Ctrl] and click the middle mouse button or press [Ctrl] + [Shift] and click the left mouse button.

4. Toggle DrwMarks on to display the reference point.
5. Object snap to the inside left corner of the wall to place your reference point exactly at the corner intersection. You are prompted to "Select one jamb of window". For more information on object snapping, see "Object Snapping" in this chapter.
6. Press [Spacebar] to activate coordinate entry. You are prompted to "Enter relative X-distance".
7. Type 3 and press [Enter]. You are prompted to "Enter relative Y-distance".
8. Type 0 and press [Enter]. Your cursor is connected to a point on the wall that is 3'-0" from the corner of the room. You are prompted to "Select second jamb of window".
9. Press [Spacebar] to activate coordinate entry, and enter the X and Y coordinates for the other side of the window. You are prompted to "Select any point on the outside of the wall".
10. Click anywhere outside the wall. The window is drawn, exactly 3'-0" from the corner of the room.

Using reference points with direction-distance coordinate entry is even easier and faster.

➔ To use a reference point and direction-distance mode, using the same example:

1. Click on Input Mode in the Tools pull-down menu and make sure Direction-Distance is checked.
2. Click on Window in the Create pull-down menu, or click Windows in the Edit/Architct menu in the Menu Window. The Windows menu is displayed.
3. Press []; the RefPnt menu is displayed in the Menu Window.
4. Toggle DrwMarks on to display the reference point.
5. Object snap to the inside left corner of the wall to place your reference point exactly at the corner intersection. You are prompted to “Indicate the reference angle”, which is the direction that will be used in the following steps.
6. Make sure Ortho mode is on, then drag your cursor to the right and click. You are prompted to “Enter one jamb of window”.

Shortcut: To toggle Ortho mode on or off, press [O].

7. Press [Spacebar], type 3, and press [Enter]. You are prompted to “Select second jamb of window”.
8. Drag your cursor to the right. Press [Spacebar], type the width of the window, and press [Enter]. You are prompted to “Select any point on the outside of the wall”.
9. Click anywhere on the exterior side. The window is inserted in the wall.

Object Snapping

Object snapping is a precise way of selecting a point on an existing entity or selecting a snap point that you’ve set. Depending on what drawing function is in progress, this point may simply be entered to complete the task, or your cursor may be attached to that point so you can begin drawing from that point.

There are several ways you can object snap to drawing entities and snap points. You can use one method or any combination of the object snapping methods described in this chapter. See the following pages for detailed instructions on how to use the various object snapping methods.

ObjSnap Menu Options

Most object snapping options are in the ObjSnap menu; however, a related function called Get-Snap also uses snapping to help you draw. See “Using Get-Snap” later in this chapter for details.

None Toggles all object snap methods off

Set, Quick, FastSym, and Fast3D can increase object snapping speed. See “More About Increasing Snapping Speed” on the following page.

FastSym	Toggle on to snap only to symbol insertion points rather than end points within the symbol
Fast3D	Toggle on to snap to the center marker of circular 3D entities
MissDistt	Set the distance your cursor can be from a point while still being able to snap to it
LyrSnap	Toggle on to snap to entities on all displayed layers; toggle off to snap to entities only on the active layer
SrchHch	Toggle on to snap to hatch lines; toggle off to ignore hatching when object snapping and increase snapping speed
Quick	Toggle on to snap to the first entity you drew; toggle off to snap to the nearest point according to the snapping methods you have toggled on
SelSet	Toggle on to snap only to entities in the active selection set
Aperture	Toggle on to display a box representing the miss distance. Toggle Aperture on to make sure the point you want to snap to is inside the miss distance.

More About Snapping Points

You can add snapping points anywhere in your drawing; these points are displayed as dots in the Drawing Area but can't be plotted.

➔ To add a snapping point to your drawing:

1. Go to the Utility menu in the Menu Window and click on Measures.
2. Click on SnapPnt in the Measures menu.
3. Click in the Drawing Area or use coordinate entry to enter the snapping point. The point is displayed in the Drawing Area.

More About Using Your Mouse for Snapping

If you're using a three-button mouse, set your middle mouse button to Middle Button; do not set it to Double Click, because you will not be able to use it for object snapping. If your middle mouse button doesn't seem to be snapping to points, check your mouse settings.

➔ To check your mouse settings:

1. Close DataCAD if it is running. From the Windows Start menu, click on Settings and then on Control Panel.
2. Double-click on Mouse in the Control Panel window to open the Mouse Properties dialog box.
3. Click on the Devices tab and make sure you have a three-button mouse driver installed.
4. Click on the Buttons tab and make sure the left button is set to Click Select, the middle button is set to Middle Button, and the right button is set to Alternate Menu/Click Select. If there is no Middle Button setting, set the middle mouse button to Unassigned or Default.
5. Click on OK to close the dialog box.

More About Increasing Snapping Speed

There are several options you can use to increase object snapping speed. Use SelSet, SrchHch, and FastSym to more precisely target the geometry you want to snap to.

The Fast3D option can also increase snapping speed when your drawing has many circular 3D entities, such as 3D arcs, cylinders, cones, domes, tori, or contour curves. When you toggle on Fast3D and then select entities using the area or fence selection method, only those entities whose center axis markers or control points are within the defined area or fence are selected.

Snapping to the Nearest Point

You can snap to the point nearest your cursor, regardless of what other object snap options are toggled on. For instance, even if you don't have MidPnt toggled on, it is still possible to snap to the midpoint of a line using the Nearest option, as long as that midpoint is the point nearest to your cursor.

➔ To snap to the nearest point:

1. Go to the Utility menu in the Menu Window and click on ObjSnap. You can also click on Options in the Tools pull-down menu, and then click on Object Snap in the submenu. The ObjSnap menu is displayed. Press [PageDown] to zoom in on your drawing and place your cursor more accurately.
2. Click on Nearest in the ObjSnap menu.
3. Move your cursor close to the point you'd like to snap to, and press [N] or click the middle mouse button. The nearest point on any entity is snapped to.

Snapping to Dividing Points

You can divide an entity in a specified number of sections, with each section separated by a snapping point. These snapping points are only visible if DrwMarks is toggled on in the Settings menu and after you snap to them.

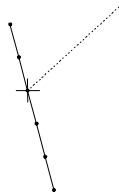


Figure 11.1: Snapping to a point on a divided entity

➔ To snap to a dividing point:

1. Go to the Utility menu in the Menu Window and click on ObjSnap. You can also click on Options in the Tools pull-down menu, and then click on Object Snap in the submenu. The ObjSnap menu is displayed.
2. Click on No.Pnts in the ObjSnap menu. A value menu is displayed in the Menu Window.
3. Set the number of divisions for an entity by use the value menu or typing a number. Then press [Enter].
4. Move your cursor close to the point you'd like to snap to, and press [N] or click the middle mouse button. The nearest division point is snapped to.

Snapping to End Points

You can snap to the end points of entities in your drawing.

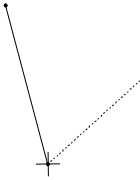


Figure 11.2: Snapping to an end point of a line

➔ To snap to the nearest end point:

1. Go to the Utility menu in the Menu Window and click on ObjSnap. You can also click on Options in the Tools pull-down menu, and then click on Object Snap in the submenu. The ObjSnap menu is displayed. You can also snap to end points of lines in unexploded symbols and nested symbols. For more information on symbols, see the “Templates and Symbols” chapter.
2. Click on EndPnt in the ObjSnap menu.
3. Move your cursor close to the point you’d like to snap to, and press **[N]** or click the middle mouse button. The nearest end point is snapped to.

Snapping to the Midpoint of a Line

You can snap to the nearest midpoint of a line.

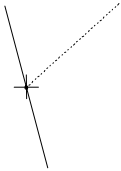


Figure 11.3: Snapping to the midpoint of a line

➔ To snap to the nearest midpoint of a line:

1. Go to the Utility menu in the Menu Window and click on ObjSnap. You can also click on Options in the Tools pull-down menu, and then click on Object Snap in the submenu. The ObjSnap menu is displayed.
2. Click on MidPnt in the ObjSnap menu.
3. Move your cursor close to the point you’d like to snap to, and press **[N]** or click the middle mouse button. The nearest midpoint is snapped to.

Snapping to Arc and Circle Centers

You can snap to the center point of an arc or circle.

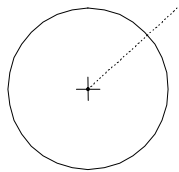


Figure 11.4: Snapping to the center point of a circle

➔ To snap to the nearest center point:

1. Go to the Utility menu in the Menu Window and click on ObjSnap. You can also click on Options in the Tools pull-down menu, and then click on Object Snap in the submenu. The ObjSnap menu is displayed.
2. Click on Center in the ObjSnap menu.
3. Move your cursor close to the point you'd like to snap to, and press **[N]** or click the middle mouse button. The nearest center point of any arc or circle point is snapped to.

Snapping to Quadrant Points

You can snap to points marking the north, south, east, or west quadrants of a circle or arc.

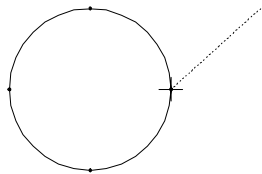


Figure 11.5: Snapping to a quadrant point on a circle

➔ To snap to the nearest quadrant point:

1. Go to the Utility menu in the Menu Window and click on ObjSnap. You can also click on Options in the Tools pull-down menu, and then click on Object Snap in the submenu. The ObjSnap menu is displayed.
2. Click on Quadrant in the ObjSnap menu.
3. Move your cursor close to the point you'd like to snap to, and press **[N]** or click the middle mouse button. The nearest quadrant point is snapped to.

Snapping to Intersections

You can snap to the intersection of lines, arcs, and circles.

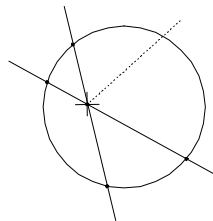


Figure 11.6: Snapping to an intersection of two lines

➔ To snap to the nearest intersection point:

1. Go to the Utility menu in the Menu Window and click on ObjSnap. You can also click on Options in the Tools pull-down menu, and then click on Object Snap in the submenu. The ObjSnap menu is displayed.
2. Click on Intsect in the ObjSnap menu.
3. Move your cursor close to the point you'd like to snap to, and press **[N]** or click the middle mouse button. The nearest intersection point is snapped to.

Snapping to a Perpendicular

You can use perpendicular snapping to draw a line that is perfectly perpendicular to another line, arc, or circle.

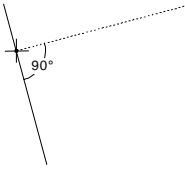


Figure 11.7: Snapping to a point that forms a perpendicular to a line

➔ To draw a line perpendicular to another entity:

1. Go to the Utility menu in the Menu Window and click on ObjSnap. You can also click on Options in the Tools pull-down menu, and then click on Object Snap in the submenu. The ObjSnap menu is displayed.
2. Click on Perpend in the ObjSnap menu.
3. Click in the Drawing Area or use coordinate entry to enter the first end point of the line.
4. Move your cursor close to the entity you'd like to draw the line perpendicular to, and press **[N]** or click the middle mouse button to snap to it. The nearest perpendicular point on that entity is snapped to.

Snapping to Tangents

You can use tangent snapping to draw a line that is tangent to an arc or circle.

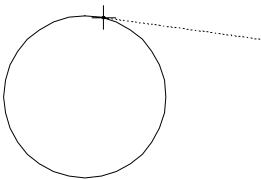


Figure 11.8: Snapping to a point that forms a tangent with a circle

➔ To draw a line that is tangent to an arc or circle:

1. Click on Options in the Tools pull-down menu, and then click on Object Snap in the submenu. The ObjSnap menu is displayed.
2. Click on Tangent in the ObjSnap menu.

3. Click in the Drawing Area or use coordinate entry to enter the first end point of the line.
4. Move your cursor close to the arc or circle you'd like to draw the line tangent to, and press **[N]** or click the middle mouse button to snap to it. The nearest tangent point on the arc or circle is snapped to.

Using Super Snap

Super Snap allows you to input distances relative to other objects in the drawing. Instead of inputting a distance, you can simply object snap to a point; the distance between your original point and the point you snapped to is automatically entered. So you can quickly create, move, copy, or stretch entities that align with other objects without having to draw guidelines or markers first.

➔ To use snapping to enter a horizontal or vertical distance:

1. Enter a point in your drawing. This may be one end point of a wall or the first point in the distance to move an entity.
2. Press and hold down **[Shift]** and object snap to a point in your drawing. The Get-Snap pop-up menu is displayed.
3. Click on Get X to enter a horizontal distance; click on Get Y to enter a vertical distance. The distance is entered.

➔ To add to or subtract from the Get X distance:

1. Enter a point in your drawing. This may be one end point of a wall or the first point in the distance to move an entity.
2. Press and hold down **[Shift]** and object snap to a point in your drawing. The Get-Snap pop-up menu is displayed.
3. Click on Get X + Distance to enter a horizontal distance; click on Get Y + Distance to enter a vertical distance. A value menu is displayed, and you are prompted to "Enter a relative distance".
4. Enter a positive value to add to the distance between your original point and the point you snapped to; enter a negative value to subtract from the distance between your original point and the point you snapped to. The distance is entered.

➔ To use snapping to define an angle:

1. Enter a point in your drawing. This may be one end point of a wall or the first point in the distance to move an entity.
2. Press **[Shift]** and object snap to the point that lies along the angle. The Get-Snap pop-up menu is displayed.
3. Click on Get Angle from the pop-up menu. A value menu is displayed, and you are prompted to "Enter relative distance".
4. Use the value menu or type a length for your line, and press **[Enter]**. The line is created at the referenced angle and length.

➔ To use snapping to define an angle, and then add or subtract from it:

1. Enter a point in your drawing. This may be one end point of a wall or the first point in the distance to move an entity.
2. Press **[Shift]** and object snap to the point that lies along the angle. The Get-Snap pop-up menu is displayed.
3. Click on Get Angle + Angle in the pop-up menu. A value menu is displayed, and you are prompted to "Enter angle from the origin".
4. To enter the angle to add to the reference angle, use the value menu or type an angle, and press **[Enter]**. The default is 90°. You are prompted to "Enter relative distance".
5. To enter the length of the line, use the value menu or type a value, and press **[Enter]**. The line is created at the referenced angle and length.

Identifying Entities

You can review the attributes of any entity in your drawing.

➔ To identify an entity:

1. Click on Identify in the Tools pull-down menu. The Identify menu is displayed in the Menu Window.
2. Click on the entity in your drawing that you want to identify. The selected entity is temporarily displayed as dashed lines, and a list of its attributes is displayed in the Menu Window. It may be necessary to change the view of your drawing, so that you have a clear view of the entity you want to identify. Click on 3DViews in the Identify menu to display the 3DViews menu and change the view of your drawing, choose a view from the View pull-down menu, or simply zoom in on your drawing.
3. Change the current settings for linetype, layer, line color, line weight, overshoot, and line spacing to match those of the entity you identified in step 2 by clicking on SetAll. The settings are changed; any entities you subsequently draw will automatically have those attributes, until you change them again.

More About Entity Attributes

The attributes displayed when you identify an entity can vary, depending on the type of entity. For example, if you identify a line, the following information might be displayed:

LINE (entity type)	Click on the entity type to display all of that entity's attributes in the Message Area.
Walls (layer entity is on)	
White (entity color)	
Solid (linetype)	
Spacing (line spacing)	Click on Spacing, LineWgt, Oversht, Z-Base, or Z-Hgt to display that information in the Message Area.
LineWgt (line weight)	
Oversht (line overshoot)	
Z-Base (z-base of line)	
Z-Hgt (z-height of line)	

If you select text, the following information might be displayed:

TEXT (entity type)
Text (layer entity is on)
Red (entity color)
Romans (font)
Size (text size)

Click on Size, Angle, Weight, Slant, or Aspect to display that information in the Message Area.

Angle (text angle)
Weight (text weight)
Slant (text slant)
Aspect (text aspect ratio)

Measuring Entities

You can measure the length or angle of entities or measure the distance between two points without dimensioning them. You can also calculate areas and perimeters as well as the total length of lines in your drawing. And most measurements can be added as text to your drawing.

Measuring Lengths and Angles

You can take a variety of measurements of lines, arcs, and angles in your drawing.

➔ To measure the length of a line:

1. Go to the Tools pull-down menu, click on Measure, and then click on Line. You are prompted to “Select line”.
2. Click on the line in your drawing that you want to identify. The line is displayed as a dashed line and the measurement is displayed in the Message Area.
3. Add this measurement as text to your drawing by clicking on ToDrwing, setting the text options as necessary, and clicking in the drawing where you want to place the text.

➔ To measure the angle of a line:

1. Go to the Tools pull-down menu, click on Measure, and then click on Line Angle. You are prompted to “Select line”.
2. Click on the line. The angle measurement is displayed in the Message Area.
3. Add this measurement as text to your drawing by clicking on ToDrwing, setting the text options as necessary, and clicking in the drawing where you want to place the text.

➔ To measure the distance between two points in your drawing:

1. Go to the Tools pull-down menu, click on Measure, and then click on Point to Point. You are prompted to “Select first point of distance to measure”.
2. Click on the first point, object snap to a point, or use coordinate entry to select a point. You are prompted to “Select second point of distance to measure”.

3. Click on the second point, object snap to a point, or use coordinate entry to select a point. The measurement is displayed in the Message Area.
4. Add this measurement as text to your drawing by clicking on ToDrwing, setting the text options as necessary, and clicking in the drawing where you want to place the text.

➔ To measure the diameter of a curve:

1. Go to the Tools pull-down menu, click on Measure, and then click on Diameter. You are prompted to "Select curve".
2. Click on the curve in your drawing that you want to identify. The curve is displayed as a dashed line and the measurement is displayed in the Message Area.
3. Add this measurement in text to your drawing by clicking on ToDrwing, setting the text options as necessary, and clicking in the drawing where you want to place the text.

➔ To measure the radius of a curve:

1. Go to the Tools pull-down menu, click on Measure, and then click on Radius. You are prompted to "Select curve".
2. Click on the curve in your drawing that you want to identify. It's displayed in a dashed line and the measurement is displayed in the Message Area.
3. Add this measurement as text to your drawing by clicking on ToDrwing, setting the text options as necessary, and clicking in the drawing where you want to place the text.

➔ To measure the chord length of an arc::

1. Go to the Tools pull-down menu, click on Measure, and then click on Chord. You are prompted to "Select curve".
2. Click on the curve in your drawing that you want to identify. It's displayed in a dashed line and the measurement is displayed in the Message Area.
3. Add this measurement as text to your drawing by clicking on ToDrwing, setting the text options as necessary, and clicking in the drawing where you want to place the text.

➔ To measure the arc length of a curve:

1. Go to the Tools pull-down menu, click on Measure, and then click on Arc Length. You are prompted to "Select curve".
2. Click on the curve in your drawing that you want to identify. It's displayed in a dashed line and the measurement is displayed in the Message Area.
3. Add this measurement as text to your drawing by clicking on ToDrwing, setting the text options as necessary, and clicking in the drawing where you want to place the text.

➔ To measure the circumference of a circle:

1. Go to the Tools pull-down menu, click on Measure, and then click on Circumference. You are prompted to “Select curve”.
2. Click on the circle in your drawing that you want to identify. It’s displayed in a dashed line and the measurement is displayed in the Message Area.
3. Add this measurement as text to your drawing by clicking on ToDrwing, setting the text options as necessary, and clicking in the drawing where you want to place the text.

➔ To measure the included angle between two lines:

1. Go to the Tools pull-down menu, click on Measure, and then click on Inclusive Angle. You are prompted to “Select first line of angle to measure”.
2. Click on the first line. You are prompted to “Select second line of distance to measure”.
3. Click on the second line. The measurement is displayed in the Message Area.
4. Add this measurement as text to your drawing by clicking on ToDrwing, setting the text options as necessary, and clicking in the drawing where you want to place the text.

➔ To measure the excluded angle between two lines:

1. Go to the Tools pull-down menu, click on Measure, and then click on Exclusive Angle. You are prompted to “Select first line of angle to measure”.
2. Click on the first line. You are prompted to “Select second line of distance to measure”.
3. Click on the second line. The measurement is displayed in the Message Area.
4. Add this measurement as text to your drawing by clicking on ToDrwing, setting the text options as necessary, and clicking in the drawing where you want to place the text.

Measuring the Total Length of Lines

You can calculate the total length of lines in a drawing according to color, linetype, and line weight. You can use the Color, Linetype, and Weight filters individually or in any combination.

➔ To measure total length of lines:

1. Go to the Tools pull-down menu, click on Measure, and then click on Takeoffs. The Takeoffs menu is displayed in the Menu Window.
2. Decide which lines you want to include:

- To include all lines of a specific color in the measurement, click on Color in the Takeoffs menu, and then select a color from the color menu. For more information on using color menus, see “Color Menus” in “The Drawing Board” chapter.
 - To include all lines of a specific type, click on Linetype in the Takeoffs menu, and then select a linetype from the menu.
 - To include all lines of a specific weight, click on Weight in the Takeoffs menu, and then select a weight from the value menu. For more information on using value menus, see “Value Menus” in “The Drawing Board” chapter.
 - To include lines on all displayed layers, toggle LyrSrch on. Toggle it off to include only those lines on the active layer.
3. Click on Begin. The measurement is displayed in the Message Area.
 4. Add this measurement as text to your drawing by clicking on ToDrwing; choose to add the area, the perimeter, or both to your drawing; set the text options as necessary; and click in the drawing where you want to place the text.

Measuring Areas, Perimeters, and Volumes

You can calculate the area, perimeter, and volume of an existing polyline or circle in your drawing, or you can define a temporary polyline to measure an area or arc in your drawing.

- ➔ To measure the area, perimeter, and volume of an existing polyline or circle:
1. Go to the Tools pull-down menu, click on Measure, and then click on Area/Perimeter. The Area/Per menu is displayed in the Menu Window.
 2. Click on ScaleTyp; a list of scale types is displayed in the Menu Window. This ScaleTyp setting is independent from the drawing scale type set in the Settings/ ScaleTyp
 3. Click on a scale type to select it, and right-click to exit the scale type list. If you selected Decimal, InchDec, Meters, Centimtr, Millimtr, DIN, or AS1100 for the scale type, the AreaPrec option is available in the scale type menu. Use this option to set the number of digits of precision to be displayed for area calculations.
 4. Toggle AddSnap on to add a snapping point to your drawing at each vertex of the polyline you select.
 5. Click on Select. You are prompted to “Select polyline or circle for area/perimeter calculation”.
 6. Click on the polyline or circle you want to measure. The area and perimeter measurements are displayed in the Message Area, and additional Area/Per menu options are displayed in the Menu Window. If you select Arch, Engr, Decimal, Inch/Frc, or Inch/Dec for the scale type, the Acres option is available in the Area/ Per menu, which you can toggle on to display area units in acres.

7. Display the centroid of the polyline by clicking on Centroid; then click on ShowIt to display a cross marking the centroid of the polyline, or click on AddIt to add a snapping point at the centroid. Right-click to return to the Area\Per menu.
 8. Add or subtract the next area or perimeter measurement from the total. Click on Perim+ to add the next perimeter measurement, click on Perim- to subtract the next perimeter measurement, click on Area+ to add the next area measurement, or click on Area- to subtract the next area measurement. Repeat step 6 as necessary to add or subtract an area or perimeter measurement. Click on Volume in the Area/Per menu to display the volume of the selected polyline. If you select Decimal, InchDec, Meters, Centi-mtr, Millimtr, DIN, or AS1100 for the scale type, AreaPrec is available in the scale type menu. Click on AreaPrec and use the value menu to set the number of digits of precision to be displayed for area calculations.
 9. Add the measurement as text to your drawing by clicking on ToDrwing; choose to add the area, the perimeter, the volume, or both area and perimeter to your drawing; set the text options as necessary; and click in the drawing to place the text.
- ➔ To measure the area, perimeter, and volume of a polyline you define:
1. Go to the Tools pull-down menu, click on Measure, and then click on Area/Perimeter. The Area/Per menu is displayed in the Menu Window.
 2. Click on ScaleTyp. A list of scale types is displayed in the Menu Window.
 3. Click on a scale type to select it, and then right-click to exit the scale type list. If you select Arch, Engr, Decimal, Inch/Frc, or Inch/Dec for the scale type, the Acres option is available in the Area/ Per menu, which you can toggle on to display area units in acres.
 4. Toggle AddSnap on to add a snapping point to your drawing at each vertex of the polyline you select.
 5. Define the polyline to measure by clicking in the Drawing Area or using object snap to precisely select the first point for the polyline. You are prompted to "Select next point on polyline".
 6. Continue selecting points as necessary, until the area you want to measure is enclosed by the polyline. You can also trace around arcs. See "More About Tracing Polylines" on the following pages for details.
 7. Click on Close in the Area/Per menu to automatically connect the last point on the polyline you selected with the first one. Your cursor is detached from the polyline, and the area and perimeter measurements are displayed in the Message Area. Additional Area/Per menu options are displayed in the Menu Window. Click on Volume in the Area/Per menu to display the volume of the selected polyline.
 8. Display the centroid of the polyline if necessary. Click on Centroid; then click on ShowIt to display a cross marking the centroid of the polyline, or click on AddIt to add a snapping point at the centroid. Right-click to return to the Area\Per menu.

9. Add or subtract the next area or perimeter measurement from the total. Click on Perim+ to add the next perimeter measurement, click on Perim- to subtract the next perimeter measurement, click on Area+ to add the next area measurement, or click on Area- to subtract the next area measurement. Repeat step 6 and 7 as necessary to add or subtract an area or perimeter measurement. To measure a rectangular area or perimeter, simply click on RectAngl in the Area/Per menu, and enter two, diagonally opposite points.
10. Add this measurement as text to your drawing by clicking on ToDrwing; choose to add the area, the perimeter, the volume, or both area and perimeter to your drawing; set the text options as necessary; and click in the drawing to place the text.

More About Tracing Polylines

Besides tracing straight sides for your polylines, you can also trace around 2-point arcs, 3-point arcs, and tangent arcs. The 2PtArc, 3PtArc, and Tangent options are available in the Area/Per menu after you select the first point for your polyline.

➔ To trace 2-point arcs:

1. Begin your polyline anywhere on the shape and continue it until you select one end point of the 2-point arc.
2. Click on 2PtArc in the Area/Per menu. You are prompted to "Select center of arc".
3. Click on the center of the arc, or object snap to it. You are prompted to "Select end point of arc".
4. Click or object snap to the other end point of the arc. For more information on how to object snap, see "Object Snapping" earlier in this chapter.
5. Repeat steps 3 and 4 to immediately trace around another 2-point arc, or right-click to exit the 2PtArc menu.
6. Continue tracing your polyline, and click on Close when finished. The area and perimeter measurements are displayed in the Message Area.

➔ To trace 3-point arcs:

1. Begin your polyline anywhere on the shape and continue it until you select one end point of the 3-point arc.
2. Click on 3PtArc in the Area/Per menu. You are prompted to "Select second end point of arc".
3. Click on the second end point of the arc, or object snap to it. You are prompted to "Select any point on the arc".
4. Click anywhere on the arc.
5. Repeat steps 3 and 4 to immediately trace around another 3-point arc, or right-click to exit the 3PtArc menu.

6. Continue tracing your polyline, and click on Close when finished. The area and perimeter measurements are displayed in the Message Area.

➔ To trace tangent arcs:

1. Begin your polyline anywhere on the shape, making sure you trace the line the arc was drawn tangent to *before* you reach the first end point of the tangent arc.
2. Once you've selected the first end point of the tangent arc, click on Tangent in the Area/Per menu. You are prompted to "Select ending point of arc".
3. Click on the other end point of the arc, or object snap to it.
4. Repeat step 3 to immediately trace around another tangent arc, or right-click to exit the Tangent menu.
5. Continue tracing your polyline, and click on Close when finished. The area and perimeter measurements are displayed in the Message Area.

Working with Entities

The Geometry menu has three options that may help you work with the entities in your drawing. You can mark divisions or intersections of entities as well as rotate the snap grid and even the X and Y axes to match an entity.

Dividing Entities

You can calculate equal divisions of lines and arcs and mark those divisions with snap points. This feature is useful for such things as dividing a stair run into equal tread lengths.

You can also divide a line using the Golden Section proportion. *Golden Section* divides a line into two segments, one longer than the other, so that the smaller section is to the larger section as the larger is to the whole. The exact ratio is 1:1.618, a proportion considered harmonic by ancient Greeks.

➔ To divide a line:

1. Click on Geometry in the Tools pull-down menu, and then click on Divide. The Divide menu is displayed in the Menu Window.
2. Click on Divisions. A value menu is displayed, and you are prompted to "Enter the number of divisions".
3. Use the value menu or type a number, and press .
4. Decide how you want to divide the line. If you simply want to divide the line into equal parts, skip to step 5. However, to divide the line into sections of a specific distance as well, click on Div+Dist. A value menu is displayed, and you are prompted to "Enter distance between division points". Use the value menu or type a number, and press . You are prompted to "Select first point".
5. Click in the Drawing Area to select the first end point of the line, or use object snap to select it. You are prompted to "Select second point".

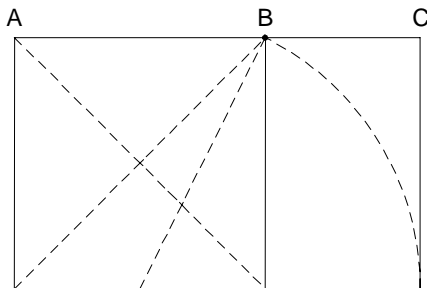
- Click in the Drawing Area to select the second end point of the line, or use object snap to select it. The line is divided, and a snap point is displayed at each division.

➔ To divide an arc, circle, or line:

- Click on Geometry in the Tools pull-down menu, and then click on Divide. The Divide menu is displayed in the Menu Window.
- Click on Divisions. A value menu is displayed, and you are prompted to "Enter the number of divisions".
- Use the value menu or type a number, and press **Enter**.
- Decide how you want to divide the line, arc, or circle. If you simply want to divide the entity into equal parts, skip to step 5. However, to divide the entity into sections of a specific distance as well, click on Div+Dist. A value menu is displayed and you are prompted to "Enter distance between division points". Use the value menu or type a number, and press **Enter**.
- Click on Entity. You are prompted to "Select line, arc, or circle to divide".
- Select the entity you want to divide by clicking on it in the Drawing Area. The line is divided, and a snap point is displayed at each division.

➔ To divide a line using Golden Section:

- Click on Geometry in the Tools pull-down menu, and then click on Divide. The Divide menu is displayed in the Menu Window.
- Click on GoldSect. You are prompted to "Select first point".
- Select the two end points of the line or simply select the line itself. To do this, click on Entity. You are prompted to "Select line, arc, or circle to divide". When you select the two end points of a line, the longer segment starts at the first end point. When you select the line using Entity, the longer segment is on the side of the line that you clicked on.
- Click in the Drawing Area to select the entity you want to divide. The line is divided, and a snap point is displayed at the division between the two segments.



Golden Section:

Line divided at point B, where the ratio of AB : AC is equal to 1 : 1.618

Figure 11.9: Dividing a line using Golden Section

Marking Line Intersections

You can identify the precise point where two lines intersect and mark the intersection with a snapping point.

➔ To mark the intersection of two lines:

1. Click on Geometry in the Tools pull-down menu, and then click on Intrsect. The Intrsect menu is displayed in the Menu Window.
2. Determine if the two lines are on different layers. If so, toggle LyrSrch on. You are prompted to “Select first line”.
3. Click on one of the lines to select it. You are prompted to “Select second line”.
4. Click on the other line to select it. The intersection is marked with a snapping point.

Matching the Snap Grid to an Entity

You can rotate the snap grid to match the angle of an entity, making it easier to draw. This does not rotate the X and Y axes.

➔ To match the snap grid to an entity:

1. Click on Geometry in the Tools pull-down menu, and then click on Tangents. The Tangents menu is displayed in the Menu Window. DataCAD automatically changes to orthographic (plan) view when you use Tangents.
2. Set the number of cursor snap angles by clicking on TanDivs. A value menu is displayed in the Menu Window, and you are prompted to “Enter the number of cursor snap angles”.
3. Use the value menu or type a number, and press . You are prompted to “Select the line to draw tangent to”.
4. Select an entity to match its angle or select two lines and align the snap grid with the line that bisects the angle between them. You can also select an arc or circle; the cursor is rotated to match the angle of the entity *at the point you selected*.
 - To match an entity’s angle, click on a line to select it. The cursor and snap grid are rotated to match the angle of the line.
 - To match a bisected angle between two lines, click on Bi-sect. Click on the two lines to select them. The cursor and snap grid are rotated to match the angle of the line bisecting the selected entities (see Figure 11.11).
 - To align the X and Y axes with the tangent entity, toggle DistSync on in the Utility/Settings menu (see Figure 11.10).

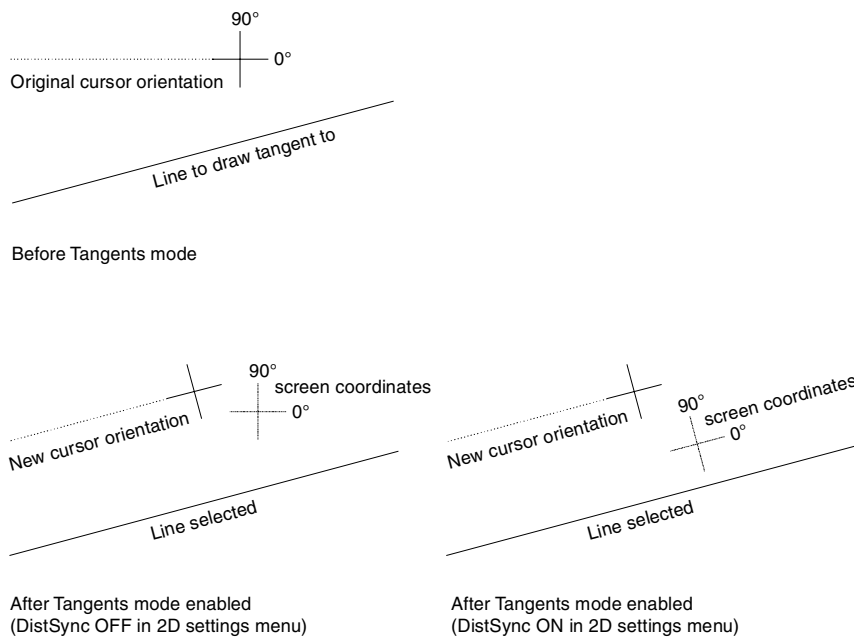
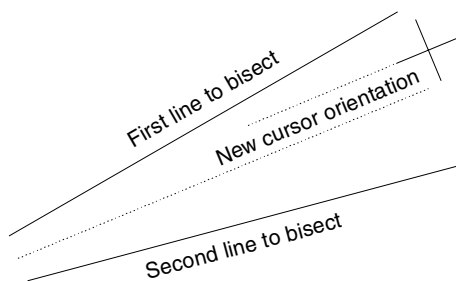


Figure 11.10: Rotating the cursor and snap grid with Tangents, both with and without DistSync toggled on



Tangents mode using Bisect

Figure 11.11: Rotating cursor and snap grid to match bisecting angle

Layer Utility

You can use the Layer Utility macro to recover a corrupted drawing file or to release excess memory being held in a drawing file. Excess memory can be tied up in a drawing file that has undergone extensive editing. Using Layer Utility is a two-step process:

- First, save the layers of the problem drawing file to a layer file. You can save all the layers in the drawing or only selected ones.
- Second, create a new drawing and load the layers from the layer file into the new drawing. Corrupted drawing information and excess memory are not saved or transferred to the new drawing. It is very important that you load the layers into a *new* drawing file and not into an existing drawing file.

➔ To save drawing layers:

1. Click on LyrUtil in the Macros pull-down menu. The LyrUtil menu is displayed in the Menu Window. If LyrUtil is not listed in your Macros menu, click Configure in the Macros menu and add it to the Macros in Menu list.
2. Click on LyrSave in the LyrUtil menu. The LyrSave menu is displayed in the Menu Window.
3. Determine if you defined go-to-views for your drawing. If so, toggle Views on to save them.
4. Determine if you have multi-scale plotting details defined for your drawing. If so, toggle Details on to save them.
5. Save your layers to a layer file using one of the following:
 - To save all the layers in the drawing, click on SaveAll.
 - To save only those layers that are toggled on (currently being displayed), click on SaveOn.
 - To change the layers that are toggled on without leaving Layer Utility, click on Lyr Menu in the LyrSave menu. The Layers menu is displayed in the Menu Window. For details on using the Layers menu, see the “Before You Draw” chapter.
6. Look at the dialog box. Enter the name of the layer file and click Open. The layer file is saved.

➔ To load a layer file into a new drawing file:

1. Create a drawing, as described in “The Drawing Board” chapter.
2. Click on LyrUtil in the Macros pull-down menu. The LyrUtil menu is displayed in the Menu Window.
3. Click on LyrLoad in the LyrUtil menu. A dialog box is displayed.
4. Click on the name of the layer file you want to load, and click Open. The layer file is loaded into your new drawing. You are prompted with “Would you like to delete this session’s control and layer files”.
5. Click Yes in the Menu Window to delete the layer (.LYS) file; click No to retain the layer file.

➔ To delete layer files you no longer need:

1. Click on LyrUtil in the Macros pull-down menu. The LyrUtil menu is displayed in the Menu Window.
2. Click on Del Lyrs in the LyrUtil menu. A dialog box is displayed.
3. Click on the layer file you want to delete, and then click Open. The layer file is deleted.

More About Layer Utility

The file size of a drawing is increased when entities are added to the drawing, but it is not reduced in size when they are deleted. This results in a file with potentially large amounts of unused memory. Layer Utility releases this memory from the file, shrinking drawing file size.

Layer Utility can also recover unaffected data in a corrupted file and rebuild the drawing file. Corrupted drawing files are usually caused by power or hardware failure while DataCAD is running. When a drawing file is corrupted, information is distorted or cannot be accessed.

Editing Drawings

12

The Edit menu contains several tools you can use to change the visible characteristics of entities as well as their position in the drawing. Wall and line intersections can be cleaned, breaks can be welded, and lines can be trimmed to another entity or to an intersection in your drawing.

In addition to repairing drawing mistakes, you can also use a number of features to edit entities. Not only can you change such attributes as the color or linetype of an entity, you can also move, rotate, or stretch an entity or even an entire room.

In this chapter:

- ⊕ Cleaning intersections
- ⊕ Welding lines and walls
- ⊕ Trimming lines
- ⊕ Changing entity attributes
- ⊕ Moving entities
- ⊕ Rotating entities
- ⊕ Erasing entities
- ⊕ Enlarging entities
- ⊕ Stretching entities
- ⊕ Clipping entities

Trimming, Welding, and Cleaning

There are several tools you can use to trim corners and lines, weld lines and walls, and clean up intersections in your drawing.

Trimming Corners

You can connect any two non-parallel lines with either an arc or a line. The Fillets option connects two lines with an arc and can then trim them to make a smooth curve. The Chamfer option connects two lines with a line and can then trim the two lines to the chamfer. These two lines do not have to be connected prior to using Fillets or Chamfer. They can be two lines that form the corner of a room, or they can be two completely separate lines in your drawing.

➔ To connect two lines with an arc:

1. Click on Cleanup in the Edit menu in the Menu Window. The Cleanup menu is displayed in the Menu Window.
2. Click on Fillets in the Cleanup menu. The Fillets menu is displayed in the Menu Window.
3. Change the fillet radius if necessary. The default setting for the fillet radius is 1'-0." To change the radius of the fillet arc, click on Radius. A value menu is displayed.
4. Use the value menu or type a radius, and press **Enter**. The radius remains at this setting for this drawing until you change it. In fact, this setting is saved with the drawing, so that every time you open the drawing the Fillets radius is automatically set to the value you enter in these steps.
5. Toggle Clip on or off. To trim the ends of the selected lines to the ends of the arc, toggle Clip on. To draw the arc but leave the ends of the lines unaffected, toggle Clip off (see Figure 12.1).
6. Connect lines on two different layers by toggling LyrSrch on. You are prompted to "Select first line to fillet."
7. Click on one of the lines to select it. You are prompted to "Select second line to fillet."

Click on the other line to select it. The two lines are connected by an arc.

You can use Fillets to extend two nonparallel, nonintersecting lines until they intersect. Simply set the Radius to 0, and select the two lines. The lines are lengthened until they intersect.

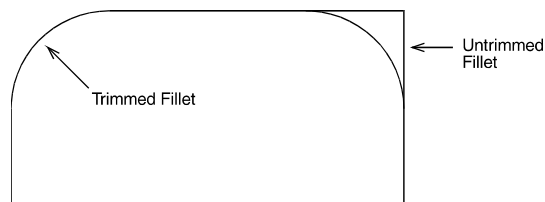


Figure 12.1: Lines trimmed to fillet (Clip on) and lines left untrimmed (Clip off)

➔ To connect two lines with a line:

1. Click on Cleanup in the Edit menu in the Menu Window. The Cleanup menu is displayed in the Menu Window.
2. Click on Chamfer in the Cleanup menu. The Chamfer menu is displayed in the Menu Window.
3. Set the chamfer distance for each line by clicking on Distnces. The *chamfer distance* is the distance from the intersection of the two lines to the chamfer point on each line. A value menu is displayed. You are prompted to "Enter first chamfer distance." See Figure 12.2 for an illustration of chamfer distances and how they relate to the lines you select.
4. Use the value menu or type a chamfer distance for the first line you will select, and press **[Enter]**. Another value menu is displayed. You are prompted to "Enter second chamfer distance."
5. Use the value menu or type a chamfer distance for the second line you will select, and press **[Enter]**.
6. Toggle Clip on or off. To trim the ends of the selected lines to the ends of the chamfer line, toggle Clip on. To draw the chamfer but leave the ends of the lines unaffected, toggle Clip off.
6. Connect lines on two different layers by toggling LyrSrch on. You are prompted to "Select first line to chamfer."
7. Click on one of the lines to select it. You are prompted to "Select second line to chamfer."

Click on the other line to select it. The two lines are connected by a line.

Lines may be shortened or lengthened to meet the ends of the chamfer.

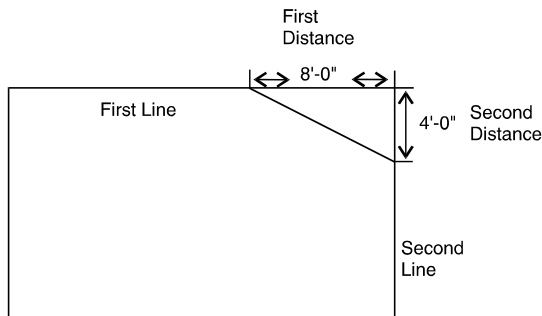


Figure 12.2: The first chamfer distance entered corresponds with the first line you select, and the second chamfer distance entered corresponds with the second line you select.

Trimming Entities

Trimming entities involves either shortening or lengthening selected lines or arcs. You can trim existing entities to a temporary line that you draw, you can trim two lines so they intersect cleanly, or you can specify not only what entities you want trimmed, but exactly how to trim them.

➔ To trim entities so they end at a line you specify:

1. Click on Line Cleanup in the Edit pull-down menu and then click on 1 Line Trim in the submenu. The 1LnTrim menu is displayed in the Menu Window.
2. Select a *trim line*, a line that entities will be trimmed to. Entities in your drawing will either be shortened or lengthened to meet the trim line. This line can either be an existing line in your drawing or it can be a temporary line that you draw.
 - To use an existing line in your drawing as the trim line, click on Entity in the 1LnTrim menu. Toggle LyrSrch on if your trim line is on a different layer. Click on the line in your drawing to select it as the trim line. Skip to step 4.
 - To draw a temporary trim line, click in the Drawing Area or use coordinate entry to enter the first end point of your trim line. Click in the Drawing Area or use coordinate entry to enter the second end point of your trim line. A dashed line representing the trim line is displayed.
3. Click anywhere on the outside of the trim line; this will be the side that will be trimmed (see Figure 12.3).
4. Choose a selection method, and then select the entities you want to trim. The entities are trimmed to the trim line. For more information on using selection menus, see “Selection Menus” in “The Drawing Board” chapter.

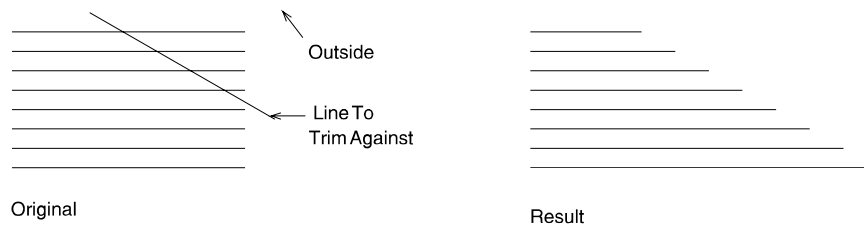


Figure 12.3: Trimming entities to a trim line

➔ To trim two lines to their intersection:

1. Click on Line Cleanup in the Edit pull-down menu and then click on 2 Line Trim in the submenu. The 2LnTrim menu is displayed in the Menu Window.
2. Toggle LyrSrch on if the lines you want to trim are on different layers. You are prompted to “Select first line to trim.”
3. Click on one of the lines you want to trim. You are prompted to “Select second line to trim.”

- Click on the other line you want to trim. The lines are trimmed at their intersection. The lines you select do not have to intersect; if they don't, they will be lengthened when they are trimmed.

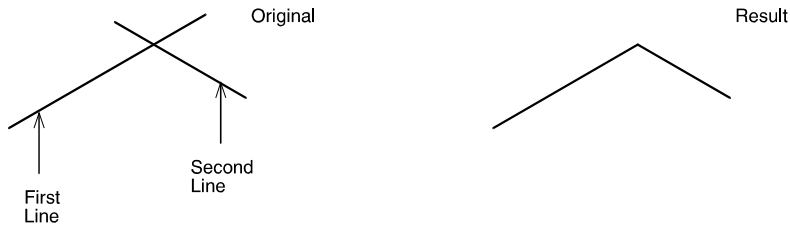


Figure 12.4: Trimming two intersecting lines

➔ To specify the entities to be trimmed and how to trim them:

- Click on Line Cleanup in the Edit pull-down menu and then click on Free Trim in the submenu. The FreeTrim menu is displayed in the Menu Window. FreeTrim only trims to or between selected entities; therefore, you must select not only the entity you want to trim but also all the entities that it intersects with.
- Choose a selection method, and then select the entities you want to trim as well as any intersecting entities. All selected entities are displayed in dashed lines.
- Click on Begin in the FreeTrim menu. You are prompted to “Select entity section to be trimmed.”
- Click on the line or arc segment that you want trimmed (removed). DataCAD searches for the nearest intersections and trims to the intersection points (see Figure 12.5).

Continue selecting segments to be trimmed or exit the FreeTrim menu by clicking on Exit or right-clicking once.

If you get unexpected results after using FreeTrim, click on Undo in the Edit pull-down menu.

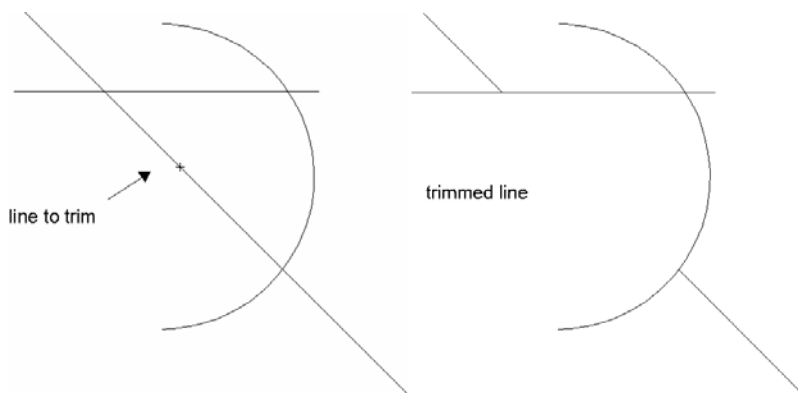


Figure 12.5: Selecting the line segment you want trimmed (removed)

Welding Lines and Walls

You can repair lines and walls that have breaks in them, making a single, straight, continuous line or wall.

➔ To weld lines:

1. Click on Line Cleanup in the Edit pull-down menu and then click on Weld Line in the submenu. The WeldLine menu is displayed in the Menu Window.
2. Toggle LyrSrch on if the lines you want to weld together are on different layers. You are prompted to "Select first line to weld."
3. Click on one of the lines you want to weld. You are prompted to "Select second line to weld."
4. Click on the other line you want to weld. The lines are welded. You are prompted with "Is this correct?"
5. Click on Yes if the weld is correct. Click on No if it is not; the lines are returned to their original state.

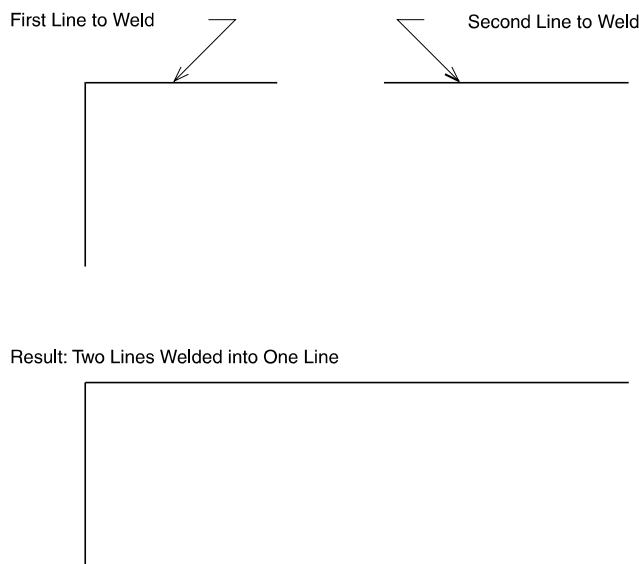


Figure 12.6: Two lines welded into a single line

➔ To weld walls:

1. Click on Wall Cleanup in the Edit pull-down menu and then click on Weld Wall in the submenu. The WeldWall menu is displayed in the Menu Window. If you have already cut the walls by inserting a window or door, you must first erase the window or door before you can weld the walls.
2. Toggle LyrSrch on if the walls you want to weld together are on different layers. You are prompted to "Select first point on box around wall opening to weld."
3. Select the walls to be welded by drawing a rubberband box around the two wall ends you want to weld. There must be only two wall ends within the rubberband box in order for the walls to be welded. To select the first corner of this box, click above and to the left of the wall ends you want to weld. You are prompted to "Select second point on box around wall opening to weld."

4. Move your mouse diagonally to stretch the rubberband box around the two wall ends you want to weld and click. The walls are welded. If the two walls are not in line with each other, an error message will be displayed, saying that the two walls can't be welded.

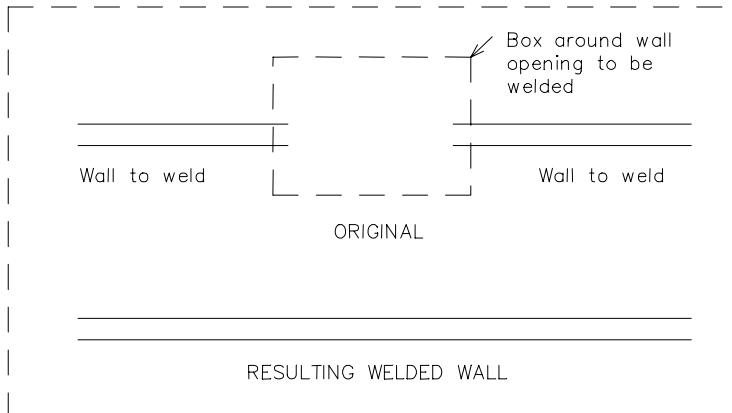


Figure 12.7: Welding two wall ends

Cleaning Wall Intersections

You can clean the wall intersections of 2-line, 3-line, or 4-line walls that meet in an L, T, or X shape.

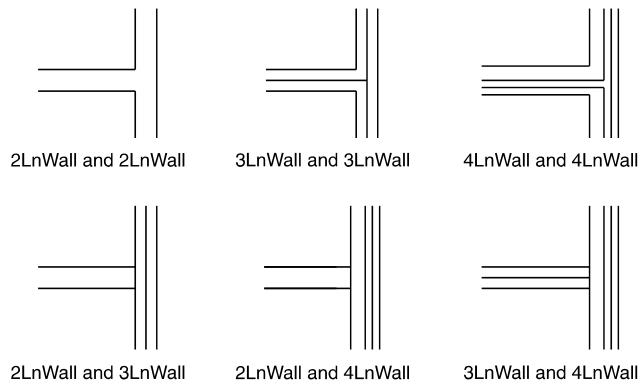


Figure 12.8: T-shaped intersections with different wall types

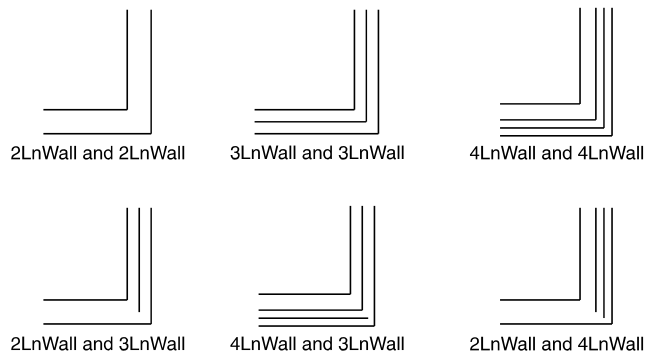


Figure 12.9: L-shaped intersections with different wall types

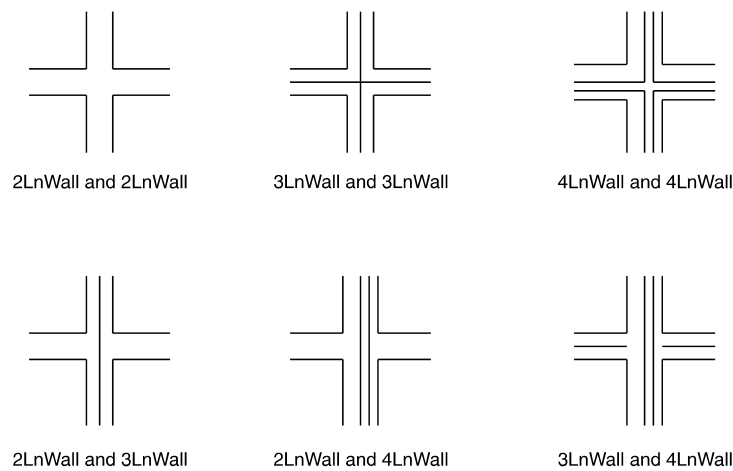


Figure 12.10: X-shaped intersections with different wall types

➔ To clean wall lines meeting in a T intersection:

1. Click on Wall Cleanup in the Edit pull-down menu and then click on T-Intersection in the submenu. The TIntsct menu is displayed in the Menu Window.
2. Toggle LyrSrch on if the wall intersection you want to clean is on a different layer than the active layer. You are prompted to “Select first corner around T intersect (wall line ends only).”
3. Select the intersection to be cleaned by drawing a rubberband box around it. To select the first corner of the box, click above and to the left of the intersection. You are prompted to “Select second corner around T intersect (wall line ends only).”
4. Move your mouse diagonally to stretch the rubberband box around the intersection, and click. You are prompted to “Point to wall line to trim to.” Make sure you include only one intersection within the rubberband box.
5. Click on the wall line to trim to (see Figure 12.11). The intersection is cleaned.

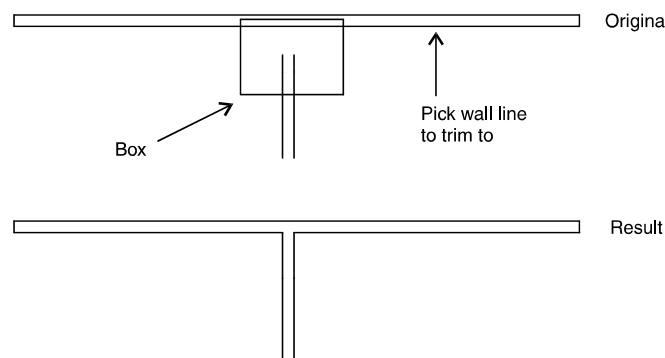


Figure 12.11: Cleaning a T-shaped intersection

➔ To clean wall lines meeting in a L intersection:

1. Click on Wall Cleanup in the Edit pull-down menu and then click on L-Intersection in the submenu. The LIntsct menu is displayed in the Menu Window.
2. Toggle LyrSrch on if the wall intersection you want to clean is on a different layer than the active layer. You are prompted to "Select first corner around L intersect (wall line ends only)."
3. Select the intersection to be cleaned by drawing a rubberband box around it. To select the first corner of the box, click above and to the left of the intersection you want to clean. You are prompted to "Select second corner around L intersect (wall line ends only)."
4. Move your mouse diagonally to stretch the rubberband box around the intersection, and click. The intersection is cleaned.

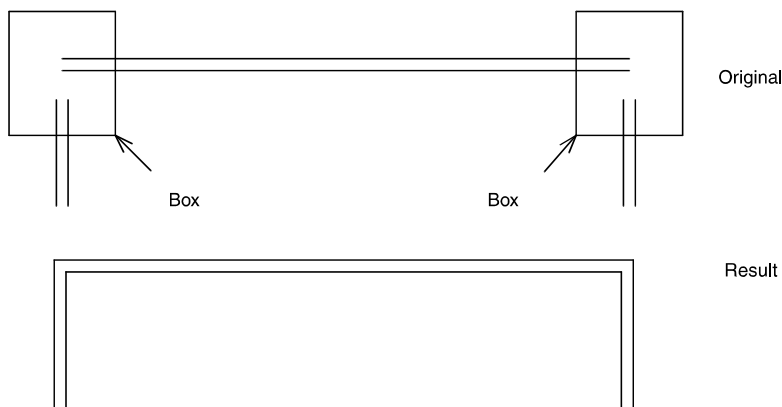


Figure 12.12: Cleaning L-shaped intersections

➔ To clean wall lines meeting in a X intersection:

1. Click on Wall Cleanup in the Edit pull-down menu and then click on X-Intersection in the submenu. The XIntsct menu is displayed in the Menu Window.
2. Toggle LyrSrch on if the wall intersection you want to clean is on a different layer than the active layer. You are prompted to "Select first corner around X intersect (0 line ends)."
3. Select the intersection to be cleaned by drawing a rubberband box around it. To select the first corner of the box, click above and to the left of the intersection you want to clean. You are prompted to "Select second corner around X intersect (0 line ends)."
4. Move your mouse diagonally to stretch the rubberband box around the intersection and click. The intersection is cleaned.

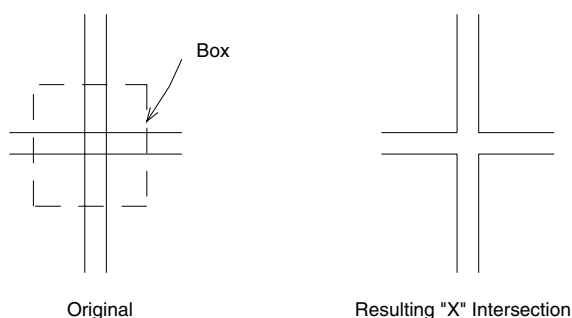


Figure 12.13: Cleaning an X-shaped intersection

Changing Entities

You can change an entity's attributes, such as linetype, line weight, color, line spacing, overshoot, Z-base, and Z-height. You can also change the size, weight, slant, aspect, and font of text.

In addition to attributes, some entities have information associated with them, and this information, too, can be edited. There's even a macro that you can use to convert lines to walls or walls to lines.

Changing Entity Attributes

The linetype, line weight, line spacing, overshoot, Z-base, and Z-height attributes can all be changed using value menus. Line spacing settings apply only to non-solid linetypes. Use a color menu to change an entity's color.

➔ To change entity attributes:

1. Click on Change in either the Edit pull-down menu or the Menu Window. The Change menu is displayed in the Menu Window.
2. Toggle LyrSrch on if the entities you want to change are on a different layer than the active layer.
3. Click on the attribute you want to change: LineType, LineWgt (line weight), Spacing (line spacing), OverSht (line overshoot), Z-Base, or Z-Hgt (Z-height). A value menu is displayed.
4. Use the value menu or type a number, and press **Enter**. For more information on using value menus, see "Value Menus" in "The Drawing Board" chapter.
5. Change the color of the entity by clicking on Color. Select the color you want to use from the color menu. For more information on using color menus, see "Color Menus" in "The Drawing Board" chapter.
6. Choose the selection method you want to use. You are prompted to select the entities you want to change.
7. Select the entities you want to change. The attributes of these entities are modified.

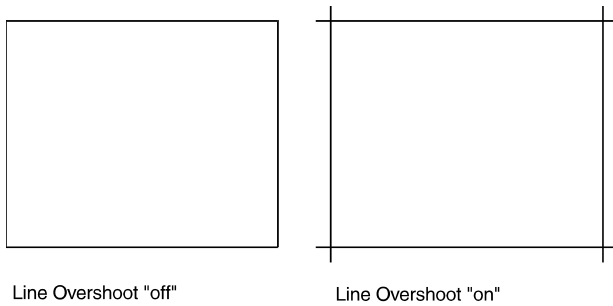


Figure 12.14: Changing the overshoot of lines

➔ To change entity attributes to match those of another entity:

1. Click on Change in either the Edit menu or the Menu Window. The Change menu is displayed in the Menu Window.
2. Click on Match.
3. Toggle LyrSrch on if the entities you want to change are on a different layer from the entities you want to match.
4. Choose which attributes to change by toggling on any of the following options: LineType, LineWgt, Color, Spacing, OverSht, Z-Base, and Z-Hgt. Or click on All to toggle all of these attribute options on.
5. Choose the selection method you want to use. You are prompted to select the entities you want to change. For more information on selection menus, see “Selection Menus” in “The Drawing Board” chapter.
6. Select the entities you want to change. You are prompted to select the entity whose attributes you want to match.
7. Select the entity to match. The attributes of the entities you selected in step 6 are modified.

Changing Text Attributes

You can change the attributes of a single line of text or of all text in your drawing, depending on what selection method you use.

➔ To change the attributes of text:

1. Click on Change in the Edit menu. The Change menu is displayed in the Menu Window.
2. Click on Text in the Change menu.
3. Toggle LyrSrch on if the text you want to change is on a different layer than the active layer.
4. Change the appearance of text in your drawing. You can change the size, weight, slant, aspect, contents, capitalization, font name, and matching characteristics in the Change\Text menu. For more information on text parameter settings, see the “Text, Notes, and Labels” chapter.

- To change the font, click on FontName, then select a font in the Font Name list box. Keep in mind that the fonts displayed under Font Name are determined by the Font Type selection in the lower-left corner of the dialog box. You can display true-type fonts (TTF), DataCAD fonts (CHR), or both in the list.
 - To change the text's size, use the Size button and type the new value. Alternatively, you can click FontName to open the Set Text Parameters dialog box; then enter the new size in the Height field or select a value from the Height list box.
 - To change the angle of the lines of text, type an angle value or click on one in the Angle list box.
 - To change the height of each character relative to its width, type an aspect ratio value or click on one in the Aspect list box.
 - To change the slant of text when using a DataCAD font, type a slant value or click on one in the CHR Slant list box.
 - To change the line weight of text when using a DataCAD font, type a weight value or click on one in the CHR Weight list box.
 - To change the font style when using a true-type font, click on a style in the TTF Style list box.
 - To change the text properties when using a true-type font, select Fill, Outline, or Both and then set the colors for the Fill and Outline, as necessary.
5. Click on OK to save the text parameter changes and close the dialog box.
 6. Change the text itself by clicking on Contents. (Otherwise, skip to step 9.) You are prompted to select the text you want to change.
 7. Choose a selection method from the menu, and then select the text. It is displayed in the Message Area. For more information on selection menus, see "Selection Menus" in "The Drawing Board" chapter.
 8. Type the new text and press .
 9. Choose the selection method you want to use. You are prompted to select the text you want to change.
 10. Select the text you want to change. The attributes of the text are modified.
- ➔ To change text attributes to match those of another text block:
1. Click on Change in the Edit menu. The Change menu is displayed in the Menu Window.
 2. Click on Match..
 3. Toggle LyrSrch on if the text you want to change is on a different layer from the text you want to match.
 4. Choose which attributes to change by toggling on any of the following options: Size, Weight, Slant, Aspect, and FontName. Or click on All to toggle all of these attribute options on.

5. Choose the selection method you want to use. You are prompted to select the text you want to change.
6. Select the text you want to change. You are prompted to select the text whose attributes you want to match.
7. Select the text to match. The attributes of the text you selected in step 6 are modified.

Editing Entity Information

Most of the standard DataCAD entities don't have any additional information attached to them, but a few do, such as associative hatching and entities that are hyperlinked to go-to-views. You can edit this information using the FormAtr macro.

➔ To edit entity attributes:

1. Click on FormAtr in the Macros pull-down menu. The FormAtr menu is displayed in the Menu Window. If FormAtr is not listed in your Macros menu, click Configure in the Macros menu and add it to the Macros in Menu list.
2. Click on Entities in the FormAtr menu. You are prompted to "Select entity to edit attribute fields."
3. Click on the entity. The entity's information is displayed in the Drawing Area.
4. Press **[Tab]** to move the edit box to the text you want to change.
5. Type the new text and press **[Enter]**. The information is changed.
6. Repeat steps 4 – 5 to continue editing, or press **[Esc]** to exit editing and return to your drawing.

Converting Lines to Walls

Using the Lin2Wall macro, you can convert lines to 2-line, 3-line, or 4-line walls. You can only convert lines; you can't convert curves. Alternatively, you can convert walls to lines. This tool can be useful when, for instance, you draw three lines and then want DataCAD to recognize those lines as a 3-line wall, so that it will be properly cut when you insert doors or windows.

➔ To convert lines or walls:

1. Click on Lin2Wall in the Macros pull-down menu. The Lin2Wall menu is displayed in the Menu Window. If Lin2Wall is not listed in your Macros menu, click Configure in the Macros menu and add it to the Macros in Menu list.
2. Toggle LyrSrch on if the lines/walls you want to convert are on a different layer than the active layer.
3. Choose your conversion result:
 - To convert walls to lines, click on Lines.
 - To convert lines to 2-line walls, click on 2LnWall.

- To convert lines to 3-line walls, click on 3LnWall.
- To convert lines to 4-line walls, click on 4LnWall.
- 4. Choose the selection method you want to use. You are prompted to select the entities you want to change.
- 5. Select the entities you want to change. The entities are modified, although their actual appearance on screen won't change. See "More About Converting Lines to Walls" below for details.

When you use Lin2Wall to convert lines with doors or windows in them to walls, use the Entity selection method to ensure that all lines are converted to walls.

More About Converting Lines to Walls

When you convert lines to walls in your drawing, they will no longer be recognized as separate lines; DataCAD will instead see them as a wall. If you've drawn three lines that you now want to convert into a single 3-line wall, simply choose the 3LnWall option and select each of the lines. When you select a line to convert it, its appearance isn't changed and it's not redrawn on screen; rather its **Attrib** setting is changed so that DataCAD recognizes each line as part of a 3-line wall. If you use Identify after you've converted the lines, you'll see the **Attrib** setting displayed in the Message Area. Single lines and lines in 2-line walls have an **Attrib** setting equal to 5; lines in 3-line walls have **Attrib**=36; and lines in 4-line walls have **Attrib**=63.

Moving Entities

You can move an entity to a different position in your drawing by either specifying the distance and angle to move the entity or by dragging the entity to its new position. You can also move entities to another layer in your drawing.

If an entity is linked to others in a group or selection set, it remains linked when you move it.

You can use object snapping to align a point on the entity you move with a point on another entity in your drawing. Simply snap to the entity to move at the point you want to align, and then snap to the other entity at the point to align. The first entity is moved and aligned at the points you snapped to.

Moving Entities to Absolute Zero

➔ To move entities to absolute zero:

1. Select Input Mode from the Tool menu. Set the input mode to Absolute Cartesian.
2. Recalculate the extents of the drawing by clicking the R in the Navigation Pad. Then, press (PgUp) to zoom out.
3. Select Move from the Edit menu.
4. Select the first point near the center of the drawing.
5. Respond to the prompt for the second point of the distance to move by pressing (Spacebar); this allows you to use coordinate entry. Press (Enter) twice to accept the coordinates 0, 0 as the distance from absolute zero.

6. Draw an area box around the entire drawing, making sure that LyrSrch is on and that no layers are locked.

Moving Entities by Specific Distance and Angle

When you use Move, you are prompted to enter two points. The distance between these two points will be the distance the entity is moved from the original position; the angle defined by these two points will be the angle of the move from the original position.

➔ To move an entity a specified distance and angle:

1. Click on Move in the Edit menu. The Move menu is displayed in the Menu Window, and you are prompted to "Select first point of the distance to move."
2. Click in the Drawing Area or use coordinate entry to enter the first point. A directional arrow is displayed; notice that moving your mouse changes the length and angle of the arrow. You are prompted to "Select second point of the distance to move." To use the same distance and angle as the last time you used Move, click on PrevDist.
3. Click in the Drawing Area or use coordinate entry to enter the second point. A selection menu as well as other Move options are displayed in the Menu Window. The points you select in steps 2 and 3 can be selected anywhere on the screen; they indicate the distance and angle to move only, and **not** the beginning and ending points of the move.
4. Move the selected entities in the Z direction if necessary. Click on MoveZ, and use the value menu to enter the distance to move the entities in the Z direction. Press **[Enter]** or right-click to return to the Move menu.
5. Toggle LyrSrch on if the entities you want to move are on a different layer than the active layer.
6. Toggle AndCopy on to copy the selected entities the specified distance and angle.
7. Click on a selection method in the menu. You are prompted to select the entities you want to move. For more information on using selection menus, see "Selection Menus" in "The Drawing Board" chapter.
8. Select the entities; they are moved in the drawing. You can continue selecting entities to move them at the same distance and angle.
 - To move entities at the same distance and angle but in the opposite direction, click on Invert and then select the entities to be moved.
 - To enter a new distance and angle, click on NewDist and continue with step 2 above.

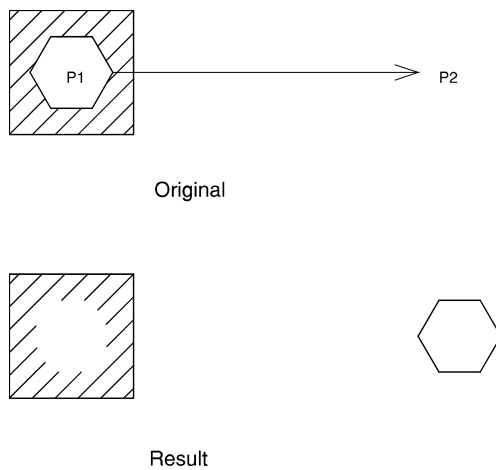


Figure 12.15: Specifying the distance and angle to move a polygon

Moving Entities by Dragging

One advantage to dragging entities is that you have the option to select multiple entities before moving them.

➔ To move entities by dragging them:

1. Click on Move in the Edit menu. The Move menu is displayed in the Menu Window.
2. Click on Drag in the Move menu. The Drag menu is displayed in the Menu Window.
3. Toggle LyrSrch on if the entities you want to move are on a different layer than the active layer.
4. Toggle AndCopy on to copy the selected entities as you drag them.
5. Change the maximum number of lines to be displayed if your entities are complex and you want a smoother drag. Click on MaxLines and use the value menu to enter the number of lines. Press **[Enter]** or right-click to return to the Drag menu. When the entities you selected to drag contain more lines than the MaxLines setting, a rectangular box representing the selected entities is displayed when you begin to drag them. The entities reappear after you select the new location for them. When you set MaxLines to 0, the rectangular box is always displayed while dragging.
6. Toggle Multi on to make multiple selections of entities to be moved.
7. Click on a selection method in the menu. You are prompted to select the entities you want to move.
8. Select the entities to be moved. You are prompted to select a point to drag from.
9. Skip to step 10 if Multi is toggled off. If you toggled Multi on in step 6, click on Begin and continue with step 10.

10. Click on a point in the Drawing Area; you can also use coordinate entry or object snapping to select this point. For more information on using coordinate entry, see “Drawing Using Coordinate Entry” in “The Drawing Board” chapter.
11. Move your mouse to drag the entities to their new position and click to place them. The entities are moved in the drawing.
12. Continue dragging and placing copies of the selected entities if AndCopy is toggled on.

Moving Entities to Another Layer

When you have more than one layer in your drawing, the ToLayer option is displayed in the Move menu. ToLayer is used to move entities from one layer to another in your drawing. You can move entities in the same position as the original layer (no distance and angle entered) or you can move entities to another layer at a specified distance and angle. For more information on creating layers, see “Creating Layers” in the “Before You Draw” chapter.

➔ To move entities to another layer without changing their positions:

1. Click on Move in the Edit menu. The Move menu is displayed in the Menu Window.
2. Click on ToLayer in the Move menu. A list of all the layers in your drawing is displayed in the Menu Window, and you are prompted to “Select layer to move to.”
3. Click on the layer you’d like to move entities to or type the name of the layer and press **[Enter]**. The ToLayer menu is displayed in the Menu Window. Use Match and Filter to select or find the layer you want to move to. For details on how to use these options, see “Selecting Layers Without Using Layer Names” and “Searching for Layers” in the “Before You Draw” chapter.
4. Toggle LyrSrch on if the entities you want to move are on a different layer than the active layer.
5. Click on a selection method in the menu. You are prompted to select the entities you want to move.
6. Select the entities; they are moved to the layer you chose in step 3. You can continue selecting entities to move them.
7. Choose another layer to move entities to by clicking on NewLyr and continuing with step 4 above.

➔ To move entities to another layer at a different distance and angle:

1. Click on Move in the Edit menu. The Move menu is displayed in the Menu Window and you are prompted to “Select first point of the distance to move.” To use the same distance and angle as the last time you used Move, click on PrevDist.

The points you select in steps 2 and 3 can be selected anywhere on the screen; they indicate the distance and angle to move only, **not** the beginning and ending points of the move.

2. Click in the Drawing Area or use coordinate entry to enter the first point. A directional arrow is displayed; notice that moving your mouse changes the length and angle of the arrow. You are prompted to “Select second point of the distance to move.”
3. Click in the Drawing Area or use coordinate entry to enter the second point. A selection menu as well as other Move options are displayed in the Menu Window. For more information on using coordinate entry, see “Drawing Using Coordinate Entry” in “The Drawing Board” chapter.
4. Click on ToLayer in the Move menu. A list of all the layers in your drawing is displayed in the Menu Window. You are prompted to “Select layer to move to.”
5. Click on the layer you’d like to move entities to, or type the name of the layer, and press **[Enter]**. A selection menu as well as other Move options are displayed in the Menu Window.
6. Toggle LyrSrch on if the entities you want to move are on a different layer than the active layer.
7. Click on a selection method in the menu. You are prompted to select the entities you want to move. For more information on using selection menus, see “Selection Menus” in “The Drawing Board” chapter.
8. Select the entities; they are moved to the layer you chose in step 5. You can continue selecting entities to move them.
9. Choose another layer to move entities to by clicking on NewLyr and continuing with step 6 above.

Rotating Entities

You can turn an entity around a rotation center and position it at a specific angle. Or you can dynamically rotate an entity, allowing you to preview the rotated position before placing the entity.

➔ To rotate an entity:

1. Click on Rotate in the Edit menu. The Rotate menu is displayed in the Menu Window, and you are prompted to “Select center of rotation.”
2. Click in the Drawing Area or use coordinate entry to enter the center of rotation. A selection menu as well as other Rotate options are displayed in the Menu Window.
3. Toggle LyrSrch on if the entities you want to rotate are on a different layer than the active layer.
4. Enter an angle of rotation by clicking on NewAngle. A value menu is displayed. The default angle of rotation is 90°. Use the value menu or type an angle, and press **[Enter]**.
5. Toggle AndCopy on if you want to both rotate and copy the selected entities.
6. Click on a selection method in the menu. You are prompted to select the entities you want to rotate.

7. Select the entities; they are rotated in the drawing. You can continue selecting entities to rotate them at the same angle and around the same center.
 - To rotate entities around the same rotation center but at the opposite angle, click on Invert and then select the entities to be rotated.
 - To return to the original angle, click on Invert again.
 - To enter a new center of rotation, click on NewCentr and continue with step 2 above.

➔ To rotate an entity dynamically:

1. Click on Rotate in the Edit menu. The Rotate menu is displayed in the Menu Window, and you are prompted to "Select center of rotation."
2. Click in the Drawing Area or use coordinate entry to enter the center of rotation. A selection menu as well as other Rotate options are displayed in the Menu Window.
3. Toggle LyrSrch on if the entities you want to rotate are on a different layer than the active layer.
4. Click on NewAngle to enter an angle of rotation. A value menu is displayed. Use the value menu or type an angle and press .
5. Toggle AndCopy on to both rotate and copy the selected entities.
6. Click on Dynamic to preview the rotation before you place the entity. The MaxLines and Multi options are displayed.
7. Set the maximum number of lines to be displayed while rotating for smoother rotating of complex entities. Click on MaxLines and use the value menu to enter the number of lines. Press or right-click to return to the Rotate menu.
8. Toggle Multi on to select multiple entities to be rotated.
9. Click on a selection method in the menu. You are prompted to select the entities you want to rotate.
10. Select the entities; they are rotated in the drawing. You can continue selecting entities to rotate them at the same angle and around the same center.
 - To rotate entities around the same rotation center but at the opposite angle, click on Invert and then select the entities to be rotated.
 - To enter a new center of rotation, click on NewCentr and continue with step 2 above.

More About Setting an Angle of Rotation

Not only can you set the angle of rotation using the NewAngle value menu, but you can also use other options when you click on Match in the value menu to get exactly the angle you want. You can:

- click on an entity to match its angle of rotation

- select two points which define an angle of rotation to match
- enter an angle of rotation and then use the inverse of that angle
- enter an angle of rotation and then use the complement of that angle
- enter an angle of rotation and then use the supplement of that angle

For instance, when your original angle is 30°, clicking on Invert in the Match menu gives you an angle of rotation of -30°. The complement angle is 60° and the supplement angle is 150°.

Stretching Entities

You can increase or decrease the length of an entity — or even an entire room --- while maintaining connections to adjacent entities. For example, you can stretch a wall or closet without breaking it from connected walls.

➔ To stretch an entity:

1. Click on Stretch in the Edit menu. The Stretch menu is displayed in the Menu Window, and you are prompted to “Select first point of the distance to stretch.”

Click in the Drawing Area or use coordinate entry to enter the first point. A directional arrow is displayed; notice that moving your mouse changes the length and angle of the arrow. You are prompted to “Select second point of the distance to stretch.”

To increase the possible angles of the directional arrow, press ☐ to toggle ortho mode off.

3. Click in the Drawing Area or use coordinate entry to enter the second point. A selection menu as well as other Stretch options are displayed in the Menu Window.
4. Toggle LyrSrch on if any of the entities you want to stretch are on a different layer than the active layer.
5. Click on a selection method in the menu. You are prompted to select the entities you want to stretch.
6. Select the entities; they are stretched in the drawing. You can continue selecting entities to stretch them at the same distance and angle.
 - To stretch entities at the same distance and angle but in the opposite direction, click on Invert and then select the entites to be stretched.
 - To enter a new stretch distance and angle, click on NewDist and continue with step 2 above.

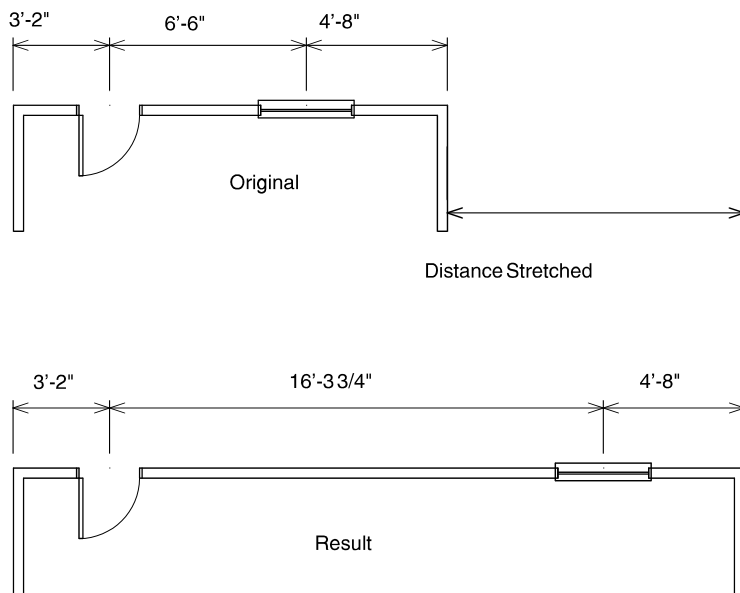


Figure 12.16: Stretching a wall

Enlarging Entities

You can increase or decrease the size of an entity by entering specific enlargement values for the X, Y, and Z axes.

➔ To enlarge an entity:

1. Click on Enlarge in the Edit menu. The Enlarge menu is displayed in the Menu Window, and you are prompted to "Select center of enlargement."
2. Click in the Drawing Area or use coordinate entry to enter the center of enlargement. A selection menu as well as other Enlarge options are displayed in the Menu Window, and the enlargement factor and line spacing setting are displayed in the Message Area.
3. Click on EnlrgeZ to enter a Z elevation for the center of enlargement. Use the value menu to enter a Z elevation and press **[Enter]**.
4. Toggle LyrSrch on if the entities you want to enlarge are on a different layer than the active layer.
5. Click on Enlrgmnt in the Enlarge menu to change one or all of the enlargement factors. The XEnlgmt, YEnlgmt, ZEnlgmt, LineFact, and SetAll options are displayed.
6. Click on any enlargement factor option to display a value menu that you can use to set these factors:
 - To enlarge entities along the X axis, click on XEnlgmt.
 - To enlarge entities along the Y axis, click on YEnlgmt.
 - To enlarge entities along the Z axis, click on ZEnlgmt.

- To set the line spacing enlargement factor for non-solid linetypes, click on LineFact. The LineFact setting affects how linetypes like Dashed are enlarged. A setting matching those of the X, Y, and Z enlargement factors will maintain the linetype's spacing. The LineFact setting cannot be applied to the geometry of referenced files.
 - To set all of these options to the same setting and enlarge the entity equally in all directions, click on SetAll.
7. Use the value menu or type an enlargement setting, and press Enter.
 8. Right-click to return to the Enlarge menu.
 9. Toggle AndCopy on if you want to both enlarge and copy the selected entities.
 10. Click on a selection method in the menu. You are prompted to select the entities you want to enlarge.
 11. Select the entities; they are enlarged in the drawing. You can continue selecting entities to enlarge them by the same factor and around the same center.
- To enlarge entities around the same enlargement center but at different enlargement factors, click on Invert and then select the entities to be enlarged. Invert mathematically inverts all of the current enlargement factors and displayed the new factors in the Message Area. For example, if your enlargement factor is 2.0, clicking on Invert sets it to 0.5.
 - To enter a new center of enlargement, click on Center and continue with step 2 above.

Erasing Entities

You can remove entities from your drawing by simply using the selection menu in the Erase menu to select the entities. You can also erase a specific segment of a line, circle, or curve.

All entities in an array are identified as a single group. To erase entities within an array, use the Entity or Area selection method.

➔ To erase an entity:

1. Click on Erase in the Edit menu. The Erase menu is displayed in the Menu Window.
2. Toggle LyrSrch on if the entities you want to erase are on a layer other than the active layer.
3. Click on a selection method in the menu. You are prompted to select the entities you want to erase.
4. Select the entities; they are erased from the drawing. You can continue selecting entities to erase them.

Avoid selecting an entity where it crosses another entity because it may be difficult for DataCAD to know which entity you want to select. Zoom in on your drawing or change your view to select entities more precisely.

5. Right-click when you're finished erasing entities to exit the Erase menu.

➔ To erase a selected piece of an entity:

1. Click on Erase in the Edit menu. The Erase menu is displayed in the Menu Window.
2. Click on Partial in the Erase menu. The Partial menu is displayed.

Toggle LyrSrch on if the entities you want to erase are on a layer other than the active layer. You are prompted to "Select entity to modify."

Click on the entity. It is displayed in dashed lines to indicate its selection, and you are prompted to "Select first point of line segment to remove."

You cannot partially erase an ellipse, B-spline or Bezier curve, polyline, or contour.

5. Object snap or click on one point of the segment you want to remove. You are prompted to "Select second point of line segment to remove."
6. Object snap or click on the other end point of the segment you want to remove. When erasing the end of an entity, select the second point slightly beyond the end of the segment to avoid leaving a single point at the end of the line. If you're erasing part of a line, the line segment you selected is removed; skip to step 8. If you're erasing part of a curve or circle, you are prompted to "Select point on arc to clip out."
7. Click between the two points you selected to remove that segment of the curve or circle, or click outside the two points you selected to remove the rest of the curve or circle.
8. Repeat steps 4 – 7 as necessary to continue erasing entities. When you're finished erasing entities, right-click to exit the Erase menu.

Editing Multiple Entities

You can associate entities in DataCAD to simplify editing groups of entities, making your drawing process more efficient and productive. Simple linking groups selected entities together, while selection sets associate entities and groups based on their attributes.

Linking Entities

You can combine entities, groups, areas of your drawing, or even multiple selection sets into a single group for editing purposes.

Caution: When you link entities, DataCAD moves all entities in the group to the same layer. By definition, all entities in a group must be on the same layer.

➔ To link entities:

1. Click on LinkEnts in the Edit menu in the Menu Window, or click on Link Entities in the Tools pull-down menu. The LinkEnts menu is displayed in the Menu Window.
2. Toggle LyrSrch on if the entities you want to link are on different layers.

3. Click on a selection method in the menu. You are prompted to select the entities you want to link.
4. Click on the entities you want to link; they are displayed in dashed lines as you select them.
5. Right-click when you're finished selecting entities. This links the selected entities.
6. Click on Exit or right-click again to exit the LinkEnts menu.

➔ To unlink entities:

1. Click on LinkEnts in the Edit menu in the Menu Window, or click on Link Entities in the Tools pull-down menu. The LinkEnts menu is displayed in the Menu Window.
2. Toggle LyrSrch on if the entities you want to link are on different layers.
3. Toggle Unlink on in the LinkEnts menu.
4. Click on a selection method in the menu. You are prompted to select the entities you want to unlink.
5. Click on the entities you want to unlink; they are displayed in dashed lines as you select them.
6. Right-click when you're finished selecting entities. This removes the selected entities from the group.
7. Click on Exit or right-click again to exit the LinkEnts menu.

Working with Selection Sets

Selection sets are temporary storage containers, used to quickly identify and edit certain entities in your drawing. There are eight selection sets available in DataCAD.

Unlike linking entities, selection sets do not regroup entities; in fact, entities in a selection set will maintain their association with their group. Instead, you can identify entities by particular attributes, associate those entities with a selection set, and then select *all* of those entities at once by using the SelSet option in the selection menus. Instead of using the usual selection methods when moving or copying entities, for instance, you can specify a selection set to edit.

➔ To display the entities in a selection set:

1. Click on EditSets in the Edit menu in the Menu Window, or click on Selection Sets in the Tools pull-down menu. The EditSets menu is displayed in the Menu Window.
2. Click on Show in the EditSets menu. All eight selection sets available in DataCAD are displayed in the Menu Window.
3. Click on the selection set you want to view. All entities in that selection set are displayed in dashed lines; if there are no entities in that selection set, nothing in your drawing will be displayed in dashed lines.

➔ To add entities to a selection set:

1. Click on EditSets in the Edit menu in the Menu Window, or click on Selection Sets in the Tools pull-down menu. The EditSets menu is displayed in the Menu Window.
2. Click on AddTo in the EditSets menu. All eight selection sets available in DataCAD are displayed in the Menu Window. You can add entities to a selection set automatically. See “Automatically Adding to Selection Sets” later in this chapter for more information.
3. Click on the selection set you want to add entities to. All entities already in that selection set are displayed in dashed lines, and the AddTo menu is displayed in the Menu Window.

If you need to delete an entity from a selection set, click on DelFrom in the AddTo menu to quickly access the DelFrom menu.

4. Click on Mask. Four entity attribute types are displayed in the Menu Window: Entity, Color, LineType, and Weight. These attribute type settings are set to All by default, as noted in the Message Area. You can change any or all of these settings. These settings apply not only to the AddTo menu but also to the DelFrom menu.

Besides using Mask, you can add entities to a selection set by simply clicking on them; they are displayed in dashed lines as you select them.

- To include only certain types of entities in the selection set, click on Entity. A list of entity types is displayed. Click on every entity type that you want to add to the selection set. Right-click when you’re finished selecting.
 - To include only certain entity colors in the selection set, click on Color. A color menu is displayed. Click on every color for entities that you want to add to the selection set. Right-click when you’re finished selecting.
 - To include only certain linetypes in the selection set, click on LineType. A list of linetypes is displayed. Click on the linetype that you want to add to the selection set.
 - To include only certain entity line weights in the selection set, click on Weight. A value menu is displayed. Use the value menu to enter the line weight of entities you want to add to the selection set, and press **Enter**.
5. Right-click to return to the AddTo menu.
 6. Toggle LyrSrch on if the entities you want to add to the selection set are on different layers.
 7. Click on Area or Fence in the selection menu. You are prompted to select the entities you want to add to the selection set. It may be beneficial to simply enclose the entire drawing in the area or fence box, so that every entity that matches attribute settings is added to the selection set. For more information on how to use Area or Fence, see “Selection Menus” in “The Drawing Board” chapter.
 8. Select the entities you want to add to the selection set. All selected entities matching the mask settings are displayed in dashed lines.

9. Right-click to exit the AddTo menu and return to the EditSets menu.

➔ To delete entities from a selection set:

1. Click on EditSets in the Edit menu in the Menu Window, or click on Selection Sets in the Tools pull-down menu. The EditSets menu is displayed in the Menu Window.
2. Click on DelFrom in the EditSets menu. All eight selection sets available in DataCAD are displayed in the Menu Window.
3. Click on the selection set you want to delete entities from. All entities in that selection set are displayed in dashed lines, and the DelFrom menu is displayed in the Menu Window.

If you need to add an entity to a selection set, click on AddTo in the DelFrom menu to quickly access the AddTo menu.

4. Click on Mask. Four entity attribute types are displayed in the Menu Window: Entity, Color, LineType, and Weight. These attribute type settings are set to All by default, as noted in the Message Area. You can change any or all of these settings. These settings apply not only to the DelFrom menu but also to the AddTo menu.

Besides using Mask, you can also delete entities from a selection set by simply clicking on them; they are displayed in their assigned linetype as you select them.

- To choose the type of entity you want to delete from the selection set, click on Entity. A list of entity types is displayed. Click on every entity type that you want to delete from the selection set. Right-click when you're finished selecting.
 - To choose the color of entities you want to delete from the selection set, click on Color. A color menu is displayed. Click on every color for entities that you want to delete from the selection set, or click on Match and then click on entities of the color you want to delete. Right-click when you're finished selecting.
 - To choose the linetype of entities you want to delete from the selection set, click on LineType. A list of linetypes is displayed. Click on the linetype that you want to delete from the selection set.
 - To choose the line weight of entities you want to delete from the selection set, click on Weight. A value menu is displayed. Use the value menu to enter the line weight of entities you want to delete from the selection set, and press **[Enter]**.
5. Right-click to return to the DelFrom menu.
 6. Toggle LyrSrch on if the entities you want to delete from the selection set are on different layers.
 7. Click on Area or Fence in the selection menu. You are prompted to select the entities you want to delete from the selection set. For more information on how to use Area or Fence, see "Selection Menus" in "The Drawing Board" chapter.

8. Select the entities you want to delete from the selection set. All selected entities matching the mask settings are displayed in their normal linetype; entities still in the selection set remain displayed in dashed lines.
9. Right-click to return to the EditSets menu.

To edit a selection set, the set must be active. See “Automatically Adding to Selection Sets” above for details on how to activate a selection set.

More About Selection Sets

Selection sets let you select entities in your drawing based on particular attributes of those entities. Say, for instance, you want to change all yellow text in your drawing to green. The complexity of your drawing combined with a lot of yellow text would make it time-consuming and even difficult to use the standard selection methods to identify the entities in your drawing that you want to change to green. However, masking by Entity (text) and Color (yellow) and then using the Area selection method to select the entire drawing quickly adds all yellow text to the selection set. To change the color of the yellow text to green, simply select Green from the color menu in the Change menu, and then click on SelSet; the text color is changed.

Clearing Selection Sets

Because there are only eight selection sets available, you may on occasion need to reuse one that you’ve already defined. Before you can redefine a selection set, you must first delete its current contents.

➔ To clear a selection set:

1. Click on EditSets in the Edit menu in the Menu Window, or click on Selection Sets in the Tools pull-down menu. The EditSets menu is displayed in the Menu Window.
2. Click on Clear in the EditSets menu. All eight selection sets available in DataCAD are displayed in the Menu Window, and you are prompted to “Select selection set to clear.”
3. Click on the selection set you want to clear. You are prompted to verify whether you want to clear that selection set.
4. Click Yes to clear the selection set; the selection set is cleared. Click No to cancel the clear function.

Naming Selection Sets

You can name your selection sets to help you better identify them.

Shortcut: To activate a selection set, press **[Alt]** + a number 1 through 8, which corresponds to the number of the selection set. For instance, press **[Alt]** + **[1]** to make SelSet1 active; press **[Alt]** + **[1]** again to deactivate selection set 1.

➔ To name a selection set:

1. Click on EditSets in the Edit menu in the Menu Window, or click on Selection Sets in the Tools pull-down menu. The EditSets menu is displayed in the Menu Window.

2. Click on NameSet in the EditSets menu. All eight selection sets available in DataCAD are displayed in the Menu Window, and you are prompted to “Select selection set to rename.”
3. Click on the selection set you want to name. You are prompted to “Enter new name.”
4. Type a name for the selection set and press **[Enter]**. The selection set is named.
5. Repeat steps 3 – 4 to continue naming sets. Right-click when finished to return to the EditSets menu.

Automatically Adding to Selection Sets

Another way to add entities to a selection set is to append them as you draw them. Simply set the active selection set and then toggle Append on in the EditSets menu.

➔ To automatically add entities to a selection set:

1. Click on EditSets in the Edit menu in the Menu Window, or click on Selection Sets in the Tools pull-down menu. The EditSets menu is displayed in the Menu Window.
2. Click on SetActiv in the EditSets menu. All eight selection sets available in DataCAD are displayed in the Menu Window, and you are prompted to “Select selection set to set active.”
3. Click on the selection set you want to activate. A dollar sign (\$) is displayed before the selection set’s name in the menu window, and the active selection set name is displayed in the Status Area.
4. Right-click to return to the EditSets menu.
5. Toggle Append on in the EditSets menu. Any entities you draw will now be automatically added to the active selection set.

Clipping and Cropping

The ClipIt macro can be used to erase, move, or copy 2D and 3D lines, 2D arcs, or 2D circles. ClipIt is especially useful for editing hatched areas in elevation drawings or for copying large portions of your drawing. Entities that are clipped or cropped retain such attributes as linetype and line color. Circles, 2D arcs, and 2D lines all retain their Z-base and Z-height settings. The Z values for the new end points for 3D lines are recalculated based on the slope of the line.

ClipIt only works in orthographic (plan) view. So if you are using a perspective view, DataCAD will automatically switch to plan view for ClipIt functions and then switch back to perspective view when you exit ClipIt.

Caution: Make sure you leave the ClipIt macro by clicking the Exit button in the menu window. This macro creates a temporary file that it uses as an undo buffer. If you exit the ClipIt macro by right-clicking, this temporary undo buffer is not deleted. That can lead to file corruption in future drawing sessions. When you click on the Exit button to leave the ClipIt macro, you delete the temporary undo buffer.

You can choose whether to clip or crop the selected portion of your drawing. *Clipping* selects entities inside the selection boundary, while *cropping* selects entities outside the selection boundary.

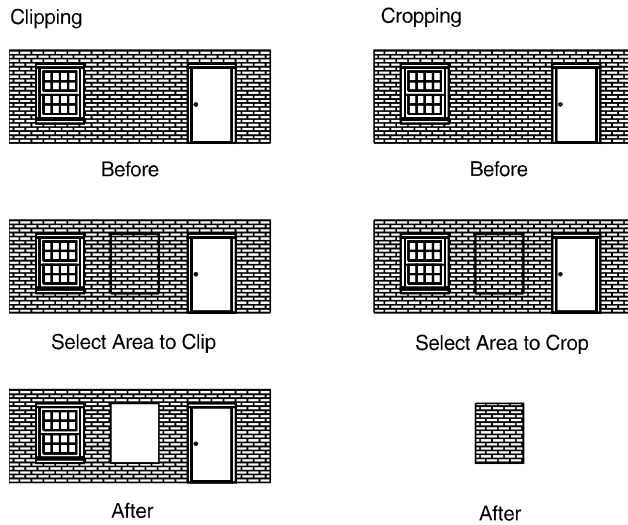


Figure 12.17: Clipping vs. cropping

You can also choose to erase the selection from your drawing, move the selection to another place in your drawing, or copy the selection to another place in your drawing.

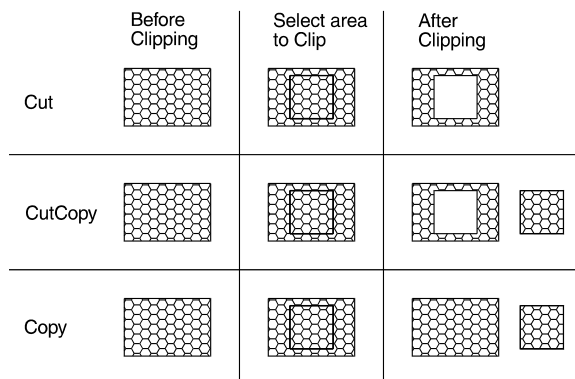


Figure 12.18: Different results from cutting, cutting and copying, and copying

➔ To clip or crop entities in your drawing:

1. Click on ClipIt in the Macros pull-down menu. The ClipIt menu is displayed in the Menu Window. If ClipIt is not listed in your Macros menu, click on Configure in the Macros menu and add ClipIt to the Macros in Menu list.
2. The ClipIt *boundary* separates a selected area from the rest of your drawing, allowing you to select entities either inside or outside of it. Toggle one of four boundary types on:
 - Area lets you draw a rectangular boundary by selecting two diagonal corners of the rectangle.
 - Fence lets you draw a polygonal boundary with up to 36 sides that can be concave or convex as long as its lines don't cross. If you select less than 36 points for the boundary, right-click to finish the polygon.

- RadCirc lets you draw a circle by defining its radius (select its center point and a point on the circle).
- DiaCirc lets you draw a circle by defining its diameter (select two points opposite each other on the circle).

Generally, ClipIt processes entities most quickly when the boundary is a rectangle or a simple polygon with only three or four sides. A circular boundary will process more slowly, and a more complex polygon will have the slowest processing time.

3. Click on Cut to erase entities, click on CutCopy to move entities, or click on Copy to copy entities. You can clip or crop entities from multiple layers, but you can only copy entities to the active layer.
4. Click on Clip to select entities inside the boundary you will draw or click on Crop to select entities outside the boundary.
5. Click on 2D Line, 3D Line, 2D Arc, or Circle to specify the type of entity you want to select. You can select one or any combination of these four entity types.
6. Toggle Boundary on to leave the outline of the boundary in your drawing. (See the “After Clipping” examples for Cut and CutCopy in Figure 12.18 for an example of a boundary outline.)
7. Toggle LyrSrch on to clip or crop entities on all layers that are toggled on; toggle LyrSrch off to clip or crop entities on only the active layer.

To decrease processing time, click on Layers in the ClipIt menu and toggle off all the layers that don't have entities that you want to clip or crop. This Layers option accesses the Layers menu in DataCAD without exiting ClipIt.

8. Select points as prompted to draw the boundary. After you select the last point for the boundary:
 - If you toggled Cut on, selected entities are clipped or cropped. Skip to step 11.
 - If you toggled CutCopy or Copy on, you are prompted to “Enter point to drag from.”
9. Click in the Drawing Area to enter the point where your cursor will attach to the copy. You are prompted to “Enter point of insertion on [the active layer].”
10. Place the copy in your drawing. To do this, move your mouse and click in the Drawing Area, use coordinate entry, or object snap to an existing entity. Repeat this step to continue placing copies.
11. Repeat steps 2 – 10 to continue clipping or cropping, or click on Exit to close the ClipIt macro.

If your drawing is very complex, the percentage of entities clipped/cropped (*not* the percentage of processing time) is displayed in the Message Area. When processing is complete, the total time is also displayed.

You can change the layer to copy to – *without changing the active layer setting* – by clicking on ToLayer in the ClipIt menu and selecting the layer you want to copy to. To create new layers to copy to, use the Layers option in the ClipIt menu to quickly access the Layers menu without exiting ClipIt.

Text, Notes, and Labels

You can add notes, labels, and other text to your drawing using DataCAD's Text tool. You can control text appearance and justification and even save these settings as a text style that you can use in any drawing. To place text in your drawing, simply click in the Drawing Area to select a justification point or define an area to fit the text into. With either method, you can position notes quickly and precisely.

Text can be imported into your drawing or even exported from it. DataCAD uses text (.TXT) files in each of these processes.

You can use Windows TrueType fonts to enter text in your drawings.

You can also add room, door and window labels using some of DataCAD's macros. Descriptive attributes can even be assigned to door and window labels and later applied to schedules and reports.

In this chapter:

- ⊕ Drawing text and arrows
- ⊕ Editing text
- ⊕ Importing and exporting text
- ⊕ Using TrueType fonts
- ⊕ Drawing room labels
- ⊕ Drawing door and window labels

Drawing Text

Text can be entered and edited anywhere in your plan. DataCAD has 23 text fonts you can use, and you can add your own fonts to your DATACAD\CHR folder. You can also use any true-type fonts (TTF) that you have installed on your computer.

When the Text menu is displayed, the text cursor replaces the standard drawing crosshairs. This cursor appearance represents the current text height, angle, slant, and aspect ratio settings. To change the appearance of text already in your drawing, you must select Change\Text\FontName to open the Set Text Parameters dialog box. This lets you change text parameters. See “Changing Text Attributes” in the “Editing Drawings” chapter.

➔ To add text to your drawing:

1. Click on Text in the Create pull-down menu, or click on Text in the Edit menu in the Menu Window. The Text menu is displayed in the Menu Window.
2. Customize the appearance of text by clicking on each setting option in the Text menu or clicking on FontName to change all the settings at once:
 - Click on a font in the Font Name list box. Keep in mind that the fonts displayed under Font Name are determined by the Font Type selection in the lower-left corner of the dialog box. You can display true-type fonts (TTF), DataCAD fonts (CHR), or both in the list. You are limited to using 32 fonts in a drawing session.
 - To set the text’s actual size in your drawing, type a height value or click on one in the Height list box. With TxtScale toggled on, text is entered in the drawing relative to the current plot scale setting. For example, if you set Height to 1” and the plotting scale to ¼”, text entered in your drawing would appear 4’ high.
 - To set the angle of the lines of text, type an angle value or click on one in the Angle list box.
 - The *aspect ratio* of text is the height relative to the width of each character. An aspect ratio of 1 represents the normal character style, while an aspect ratio of 5 produces text that is taller and narrower. To set the height of each character relative to its width, type an aspect ratio value or click on one in the Aspect list box.
 - When using a DataCAD font, you can set the angle of each character of text, as measured from the base of the character to its top. Type a slant value or click on one in the CHR Slant list box.

- When using a DataCAD font, you can set a text weight from 1 to 99, with “1” being the default weight setting. To triple the weight of text, for instance, set Weight to 3. To set the line weight of the text, type a weight value or click on one in the CHR Weight list box. Another way to produce heavy text on a drawing is to place it on a separate layer and then define a unique color for that layer. When you plot the drawing, you can assign a thicker pen to that layer. See “Using Pen Tables” in the “Printing Your Drawing” chapter for details on how to set pens for plotting.
 - When using a true-type font, you can select a font style. Click on a style in the TTF Style list box.
 - When using a true-type font, you can set the properties for the font using the TTF Properties options. Select Fill, Outline, or Both and then set the colors for the Fill and Outline, as necessary.
3. Click on OK to save the text parameter changes and close the dialog box.
 4. Set the space between lines of text by clicking on Factor. Use the value menu or type a factor for your text, and right-click to return to the Text menu. DataCAD calculates the amount of space between lines of text by multiplying the text height by the factor.
 5. Toggle Dynamic on to enter text directly into the drawing as you type it. Toggle Dynamic off to enter the text only in the Message Area; you will have to press **[Enter]** to add the text to the drawing.

Hint: With Dynamic on, your cursor must remain in the drawing window if you want carriage return to work when you press (Enter).

6. Toggle TxtScale on to size your text by using the Size setting relative to the plot scale. If you toggle TxtScale off, the text size displays and plots, according to the Height setting in the text parameters dialog box.
7. Toggle AllCaps on to enter your text in capital letters; toggle AllCaps off to enter text as you typed it.
8. Decide on how you want to justify the text. To left justify text, toggle Left on; to center justify text, toggle Center on; to right justify text, toggle Right on.
9. Click in the Drawing Area or use coordinate entry to select the point where text should be entered in your drawing. This point is the point of justification for the text you enter.
10. Type the text. If you toggled Dynamic on in step 5, the text is displayed in the drawing as you type; if Dynamic is toggled off, the text is displayed in the Message Area and you must press **[Enter]** to add the text to the drawing.

Text can include many other characters in addition to those found on the keyboard. These extended characters are typed by pressing **[Alt]** + a character number. See the Appendix for a complete list of extended characters.

11. Click on Exit in the Menu Window or right-click when you’re finished entering text.

Justifying Text

You can change the justification of text after you enter it in your drawing. Text can be justified to the left, center, or right of a justification line that you define.

➔ To justify text in your drawing:

1. Click on Text in the Create pull-down menu, or click on Text in the Edit menu in the Menu Window. The Text menu is displayed in the Menu Window.
2. Click on Justify in the Text menu. You are prompted to “Select first point on justification line”.
3. Click in the Drawing Area or use coordinate entry to enter the first point of the justification line. You are prompted to “Select second point on justification line”.
4. Click in the Drawing Area or use coordinate entry to enter the second point of the justification line. The Justify menu is displayed in the Menu Window.
5. Decide how you want to justify the text. To left justify text, toggle Left on; to center justify text, toggle Center on; to right justify text, toggle Right on.
6. Click on a selection method in the menu. You are prompted to select the text you want to justify. For more information on using selection menus, see “Selection Menus” in “The Drawing Board” chapter.
7. Select the text you want to justify. The text is aligned with the justification line, according to the justification setting you chose in step 5.
8. Continue selecting text to justify it to the same justification line; click on NewLine to define a new justification line and continue with step 3 above; or right-click to return to the Text menu.

Fitting Text into Defined Areas

Another way to enter text in your drawing is to define the height and width of the area where you’d like to place a line of text. The text size and aspect ratio are automatically calculated so that the text you type fits into the area you defined.

➔ To fit text into an area:

1. Click on Text in the Create pull-down menu, or click on Text in the Edit menu in the Menu Window. The Text menu is displayed in the Menu Window.
2. Click on FitText in the Text menu. You are prompted to “Select text start point”.
3. Click in the Drawing Area or use coordinate entry to enter the point where you want the text to begin. You are prompted to “Select text end point”.
4. Click in the Drawing Area or use coordinate entry to enter the point where you want the text to end. You are prompted to “Select point to mark text height”.

5. Click in the Drawing Area or use coordinate entry to enter a point; this point and the end point you selected in step 4 define the height of the area. You are prompted to “Enter text”.
6. Type the line of text that you want to fit in the defined area.
7. Press **[Enter]**. The aspect ratio and size is calculated, and the text you typed is fit into the area you defined. The Aspect and Size settings in the Text menu are actually changed to the values calculated in step 7. To return them to their original settings before FitText was used, click on Reset in the Text menu.
8. Enter additional lines of text using the same aspect ratio and size calculated in step 7 by simply typing the additional text.
9. Right-click to return to the Text menu.

Importing and Exporting Text

You can export text in your drawing to a text (.TXT) file as well as import a text file into your drawing.

➔ To import text into your drawing:

1. Click on Text File in the Insert pull-down menu. A dialog box is displayed, listing all text files in your DATACAD\PLT folder.
2. Click on the text (.TXT) file you want to import to select it, and then click on Open. The dialog box closes, and the FromFile menu is displayed in the Menu Window.
3. Use the menu options (as described earlier in this chapter in the instructions on adding text to your drawing) to customize the appearance of the text you want to import. You are prompted to “Position text using cursor”.
4. Click in the Drawing Area or use coordinate entry to enter the upper-left point of the block of text being imported. The text is added to the drawing.

WrapLen option uses a value menu to set the number of characters allowed per line; characters over this number are wrapped to the next line.

5. Continue selecting points to copy the imported text to other parts of your drawing, or right-click to return to the Text menu.

➔ To export text from your drawing:

1. Click on Export in the File pull-down menu, and then click on TXT in the submenu. A dialog box is displayed, prompting you to “Enter the name of the text file to create”.
2. Type a name for the text file that your text will be exported to, and then click on Save. The dialog box closes, and a selection menu is displayed in the Menu Window.
3. Click on a selection method in the menu. You are prompted to select the text you want to add to the text file. For more information on using selection menus, see “Selection Menus” in “The Drawing Board” chapter.

4. Select the text you want to export. The text is exported to the text file you created in step 2.
5. Continue selecting text to export to the text file, or right-click to display the dialog box again and save another text file. If you don't need to save another text file, click Cancel to close the dialog box and display the Text menu in the Menu Window.

Using Text Styles

You can save your Text menu settings as a text style, and then use them to quickly apply those settings before adding text to a drawing.

➔ To save a text style:

1. Click on Text in the Create pull-down menu, or click on Text in the Edit menu in the Menu Window. The Text menu is displayed in the Menu Window.
2. Click on TxtStyl in the Text menu. The TxtStyl menu is displayed in the Menu Window.
3. Click on SaveCurr in the TxtStyl menu. A list of existing text styles is displayed in the Menu Window, including the default style. You are prompted to "Enter name to save current text style to".
4. Type a name for your new text style and press **[Enter]**. The text style is saved, and you are returned to the TxtStyl menu.

➔ To use a text style:

1. Click on Text in the Create pull-down menu, or click on Text in the Edit menu in the Menu Window. The Text menu is displayed in the Menu Window.
2. Click on TxtStyl in the Text menu. The TxtStyl menu is displayed in the Menu Window.
3. Click on Load in the TxtStyl menu. A list of text styles is displayed in the Menu Window, and you are prompted to "Enter name of text style to load".
4. Click on a text style in the list. Settings for Text menu options are changed to those of the text style you chose; any text you draw now will use those settings.

➔ To delete a text style:

1. Click on Text in the Create pull-down menu, or click on Text in the Edit menu in the Menu Window. The Text menu is displayed in the Menu Window.
2. Click on TxtStyl in the Text menu. The TxtStyl menu is displayed in the Menu Window.
3. Click on Delete in the TxtStyl menu. A list of existing text styles is displayed in the Menu Window, and you are prompted to "Enter name of text style to delete".

4. Click on the name of the text style you want to delete. The text style is deleted.
5. Return to the TxtStyl menu by either clicking on Exit in the Menu Window or right-clicking.

➔ To display the settings for a text style:

1. Click on Text in the Create pull-down menu, or click on Text in the Edit menu in the Menu Window. The Text menu is displayed in the Menu Window.
2. Click on TxtStyl in the Text menu. The TxtStyl menu is displayed in the Menu Window.
3. Click on ShowVals in the TxtStyl menu. A list of text styles is displayed in the Menu Window, and you are prompted to "Enter name of text style to show".
4. Click on the name of the text style you want to view. The values of the selected text style are displayed in the Message Area.
5. Return to the TxtStyl menu by clicking on Exit or right-clicking.

Drawing Text Arrows

You can draw arrows from text in your drawing to the parts of the drawing the text refers to. The size, style, aspect, weight, and color of the arrowheads are fully customizable.





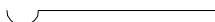
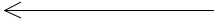




	
OPEN	4:1
	
CLOSED	3:1
	
BRIDGE	2:1
	
DOT	1:1
	
TICK	0.5:1
Arrowhead styles	Variable aspect ratios

Figure 13.1: Examples of different styles and aspect ratios for text arrowheads

➔ To draw a text arrow:

1. Click on Text in the Create pull-down menu, or click on Text in the Edit menu in the Menu Window. The Text menu is displayed in the Menu Window.
2. Click on Arrows in the Text menu. The Arrows menu is displayed in the Menu Window.
3. Choose an arrowhead style. Click on Style in the Arrows menu. A list of five arrowhead styles is displayed. (See Figure 13.2 for examples.)

4. Click on the name of the arrowhead style you want to use, and right-click to return to the Arrows menu. For Open and Closed arrowheads, Size sets the length of the arrowhead base. For Bridge arrowheads, Size sets the radius of the curve of the arrowhead. For Dot arrowheads, Size sets the diameter of the dot. For Tick arrowheads, Size sets the length of the tick mark.
5. Indicate the size of the arrowhead. The *size* of the arrowhead is the measurement of its widest point. Arrowhead size is relative to text size; if the Size setting in the Text menu is 1 and the Size setting in the Arrows menu is 3, the arrowhead size will be three times the height of your text. To set the size of the arrowhead, click on Size. A value menu is displayed.
6. Use the value menu or type a number for the size, and press **[Enter]**. This setting, “Relative arrow size”, is displayed in the Message Area whenever you are in the Arrows menu.
7. Set the aspect ratio for arrowheads by clicking on Aspect in the Arrows menu. A value menu is displayed.
8. Use the value menu or type a number for the aspect ratio, and press **[Enter]**. (See Figure 13.2 for examples of aspect settings.) An ANSI-standard arrowhead has an aspect ratio of 6.
9. Set the line weight for arrowheads by clicking on Weight. A value menu is displayed.
10. Use the value menu or type a number for the line weight, and press **[Enter]**.
11. To set the color for arrowheads, click on Color. A color menu is displayed. For more information on using color menus, see “Color Menus” in “The Drawing Board” chapter.
12. Use the color menu to select a color for arrowheads. You are prompted to “Draw arrow”.
13. Draw arrows. Arrows are drawn from the end of the arrow to the point of the arrowhead. Click in the Drawing Area or use coordinate entry to enter the end of the arrow.
14. Continue entering points to draw bends or angles in the arrow if necessary. The last point you enter will be the point of the arrowhead.
15. Right-click to complete the arrow; the arrowhead is drawn, according to the settings specified in the steps above.

Using TrueType Fonts Instead of .CHR Fonts

In addition to single and multi-stroke, vector-based DataCAD .CHR fonts, you can use Windows TrueType fonts in your drawings. You can convert TrueType fonts to 3D for signage in architectural models. If you are using an earlier version than DataCAD 10, TrueType fonts will appear in your drawing as DataCAD CHR fonts.

➔ To use TrueType fonts:

1. Select Text from the Edit menu. Then, click on FontName in the Text menu. The Set Text Parameters dialog box appears.

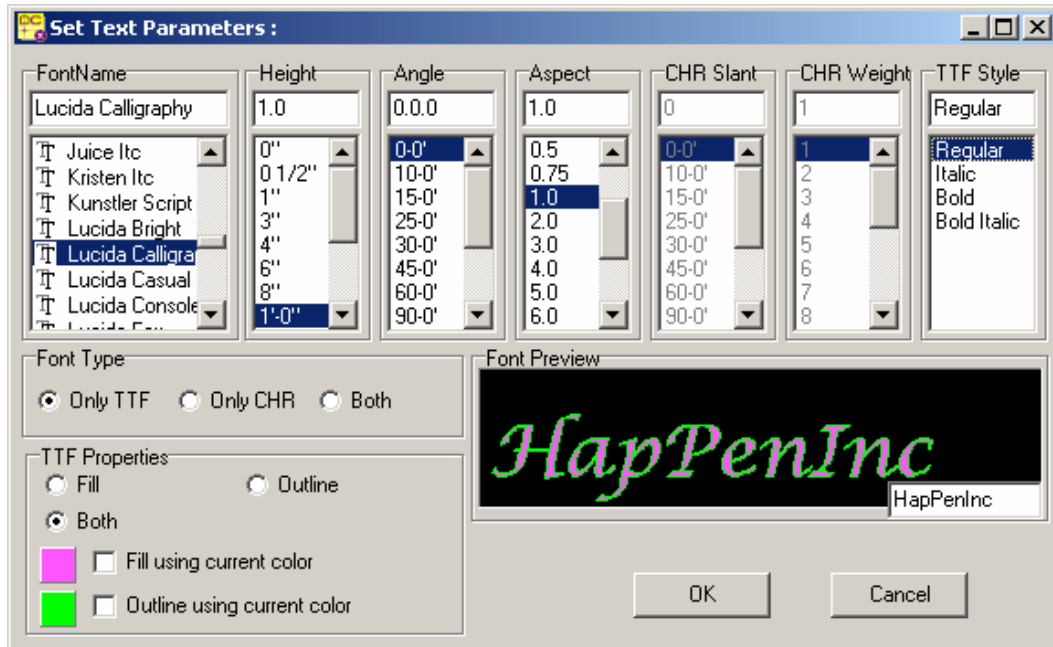


Figure 13.2: Control the appearance of TrueType fonts

2. Type something in the small box at the lower right corner of the Font Preview area. For example, in Figure 13.3, HapPenInc is the name of the Hap company that manufactures pens. The larger display shows what the text will look like, depending on the font you highlight in the FontName section of this dialog box.
3. Go to the Font Type section and click Only TTF if you want to display just the TrueType fonts in the Font Name section of this dialog box.
4. Change the settings in the Font Name, Height, Angle, Aspect, CHR Slant, CHR Weight, and TTF Style sections until the sample in the Font Preview window is correct.
5. Control the color of the font's fill and outline in the TTF Properties section. Color choices are displayed in the Font Preview window.
 - To choose colors, click Fill, Outline, or Both in the TTF Properties section.
 - To use the active layer's color for the fill and/or outline, check the Fill using current color and/or the Outline using current color boxes.
 - To specify your own colors for the fill and/or outline, uncheck the boxes. Then click on the color swatch to the left of the unchecked box to display the Color Palette dialog box, where you can either select a standard color or mix your own. For more information on using the color palette, see "Color Menus" in "The Drawing Board" chapter and "Customizing Colors" in the "Before You Draw" chapter.

6. Click OK when you are satisfied with your text parameters. When you return to the Drawing Window, the cursor changes to a text tool and you are prompted to “Select point to position text.”
7. Click where you want the text to appear in your drawing.
8. Type the text in your drawing. Notice that all your font settings are in effect.
9. Right-click when you are finished typing text.



Figure 13.3: Results of TrueType font decisions in the drawing

➔ To convert TrueType fonts to 3D:

1. Go to the 3DEdit menu and select Explode.
2. Click on TTF2Plin in the Explode menu to convert the TrueType fonts to covered polylines.

You can now create 3D signage by exploding the polylines to slabs. To do this, use the ToSlab option in the Explode menu.

Drawing Labels

In addition to drawing standard text, you can add labels to rooms, doors, and windows using some of DataCAD's macros. Descriptive attributes can even be assigned to door and window labels.

Drawing Room Labels

You can customize the appearance of both the text and the border of the label. Room labels can have only one line of text per label.

➔ To draw a room label:

1. Click on DCAD_AEC in the Macros pull-down menu. The DCAD AEC menu is displayed in the Menu Window.
2. Click on RmLabels in the DCAD AEC menu. The RoomLbl menu is displayed.

3. Indicate which of the three types of borders you want to use for the label. Click on Box to draw a rectangle around the text; click on Capsule to draw a rectangle with rounded corners around the text; click on None to draw labels with no border at all. Of the Box, Capsule, and None options, only one can be toggled on at any one time.
4. Customize the appearance of label text with several options that use value menus:
 - To set the label text's size, click on TextSize. Use the value menu or type a size for your label text, and right-click to return to the RoomLabl menu. If TxtScale is on in the Text menu, label text size will be determined relative to the current plotting scale.
 - To set the angle of the lines of label text, click on TextAngl. Use the value menu or type an angle for your label text, and right-click to return to the RoomLabl menu.
 - To set the line weight of the label text, click on TextWgt. Use the value menu or type a size for your text, and right-click to return to the RoomLabl menu.
 - To set the angle of each character of label text, as measured from the base of the character to its top, click on TextSlnt. Use the value menu or type an angle for text characters, and right-click to return to the RoomLabl menu.
 - To set the height of each character relative to its width, click on TextAspt. Use the value menu or type an aspect ratio for your text, and right-click to return to the RoomLabl menu. The *aspect ratio* of text is the height relative to the width of each character. An aspect ratio of 1 represents the normal character style, while an aspect ratio of 5 produces text that is taller and narrower.
5. Choose a font to use for label text by clicking on TextFont. A dialog box is displayed, listing all the fonts available in your DATACAD\CHR folder. Click on a font to select it, and click on Open.
6. Insert a symbol as part of your room labels by clicking on AddSymb. A dialog box is displayed. If necessary, change to your DATACAD\SYM folder, to list all your symbol folders. Open a symbol folder, select the symbol to be used with labels, and click on Open. The AddSymb option appears depressed, indicating that a symbol will be used with labels. To stop using a symbol in room labels, click on AddSymb again to turn that option off. The AddSymb option appears raised.
7. Set the text color for room labels by clicking on TextColr. Use the color menu to choose a color. If necessary, right-click to return to the RoomLabl menu. For more information on using color menus, see "Color Menus" in "The Drawing Board" chapter.
8. Set the border color for room labels by clicking on BrdrColr. Use the color menu to choose a color. If necessary, right-click to return to the RoomLabl menu.
9. Place the room label by clicking in the Drawing Area or using coordinate entry to enter a point that will be the bottom left corner of the label. You are prompted to "Enter label text".

10. Type the text for the room label and press Enter. The label is added to your drawing.
11. Repeat steps 9 – 10 to continue adding labels to your drawing, or right-click to exit the RoomLabl menu.

Drawing Door Labels

You can add labels (or “tags”) for doors in your drawing.

Drawing Window Labels

You can add labels (or “tags”) for windows in your drawing.

Dimensions

14

Dimensions in DataCAD can measure the distance between two points, the diameter or radius of curves, or the angle between two entities. You have full control over how the dimension text, lines, and arrows look as well as where they are placed in your drawing.

Stringline and baseline dimensions can be drawn by simply selecting a series of points; they can even be drawn automatically, with the distance between every end point along a wall measured and added to the series of dimensions.

You can even draw associative dimensions. These dimensions are linked to the entity they measure; whenever that entity changes size, the dimension is automatically updated.

In this chapter:

- ⊕ Dimension text and line styles
- ⊕ Adding dimensions to your drawing
- ⊕ Editing dimensions
- ⊕ Associative dimensioning

Drawing Linear Dimensions

Linear dimensions, the most common type of dimensions, measure the distance between two points. You can draw standard stringline or baseline linear dimensions, which require you to select the end points for each dimension, or you can automatically create a stringline or baseline dimension by simply selecting the two end points of the entire series of dimensions. In addition, you can edit dimensions or explode them into separate entities.

➔ To dimension between two points in your drawing:

1. Click on Dimension in the Create pull-down menu, and then click on Linear Dimension in the submenu. The Linear menu is displayed in the Menu Window.
2. Choose one of the following options to position dimensions:

Of the Horizntl, Vertical, Aligned, and Rotated options, only one can be toggled on at any time.

- To draw horizontal linear dimensions, toggle Horizntl on.
- To draw vertical linear dimensions, toggle Vertical on.
- To draw linear dimensions that are parallel to the dimensioned entity, toggle Aligned on.
- To set the angle to draw linear dimensions, click on Rotated and use the value menu to enter a rotation angle. For more information on using value menus, see “Value Menus” in “The Drawing Board” chapter.

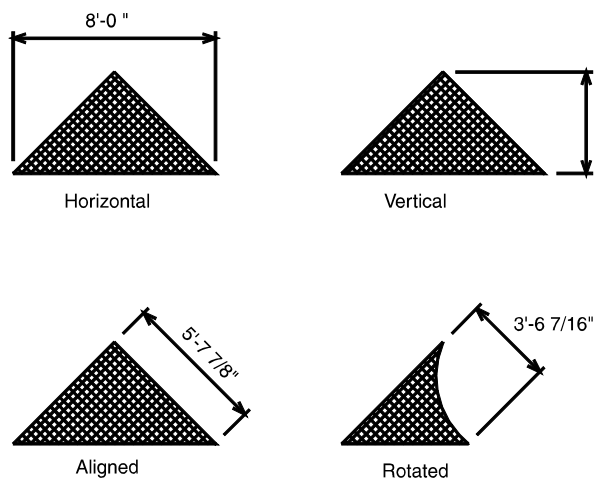


Figure 14.1: Examples of the four available positions for linear dimensions

3. Decide whether you want to use associative dimensions. *Associative dimensions* are connected to the entity they measure; so if you were to stretch that entity, its dimension would automatically be updated. Toggle Assoc on if you want to draw associative dimensions. Another advantage to using associative dimensions is that if you change drawing scales (from architectural to engineering, for example), all associative dimensions would automatically be updated with the new scale.
4. Customize the way dimension text is drawn by clicking on TextStyl and setting the options as necessary. (See “Text Style Options for Linear Dimensions” on the following pages for details.)
5. Customize the way dimension lines are drawn by clicking on DimStyl and setting the options as necessary. (See “Line Style Options for Linear Dimensions” on the following pages for details.)
6. Customize the way dimension arrows are drawn by clicking on ArroStyl and setting the options as necessary. (See “Arrow Style Options for Linear Dimensions” on the following pages for details.)
7. Respond to the prompt to “Select first point of distance to dimension”. For greater accuracy, object snap to one end point of the distance you want to dimension. You can also select this point by clicking in the Drawing Area or using coordinate entry. You are prompted to “Select second point of distance to dimension”. For more information on how to object snap to an entity, see “Object Snapping” in the “Drawing Tools” chapter.
8. Object snap to the second end point of the distance you want to dimension. The dimension lines are displayed; notice if you move your mouse cursor to the other side of the entity you’re dimensioning, the dimension lines flip to that side. You are asked, “Dimension which side of line?”
9. Click to one side of the line to place the dimension on that side. If the dimension text fits between the extension lines, the dimension is drawn; skip to step 12 to continue.

If the dimension text doesn’t fit between the extension lines, only the dimension lines are drawn, and you’re prompted with “Text does not fit. Enter new text position”.
10. Place dimension text that doesn’t fit between the extension lines by using any of the following options:

You can use any combination of the Rotate, DrwLeadr, and TextStyl options to customize dimension text.

- Click on Rotate and use the value menu to set an angle for the text; click in the Drawing Area to place the text.
- Click on DrwLeadr to draw a leader from the dimension line to the text. Click in the Drawing Area or use coordinate entry to draw the leader, and then right-click to place the dimension text.
- Click on TextStyl to change the text style settings so that the text will fit between the extension lines; click in the Drawing Area to place the text.

11. Draw stringline dimensions from the dimension you just drew by clicking on StrngLin. To draw baseline dimensions, click on Baseline. Follow the prompts to draw the series of dimensions. Using StrngLin or Baseline to dimension along an entire wall of your plan can save you time because you only have to enter the second point of each dimension. Whether this series of dimensions is stringline or baseline depends on the settings in the AutoStyl menu.
 12. Finish drawing the dimension series. Then right-click to return to the Linear menu.
 13. Complete drawing stringline dimensions. Then click on Overall in the Linear menu to dimension the entire stringline.
 14. Continue with step 8 above to enter additional dimensions.
- ➔ To automatically create a series of dimensions:
1. Click on Dimension in the Create pull-down menu, and then click on Linear Dimension in the submenu. The Linear menu is displayed in the Menu Window.
 2. Indicate which of these four options you want use to position dimensions:
 - To draw horizontal linear dimensions, toggle Horizntl on.
 - To draw vertical linear dimensions, toggle Vertical on.
 - To draw linear dimensions that are parallel to the dimensioned entity, toggle Aligned on.
 - To set the angle to draw linear dimensions, click on Rotated and use the value menu to enter a rotation angle.
- See Figure 14.2 for examples of these options.
3. Decide whether you want to use associative dimensions. *Associative dimensions* are connected to the entity they measure; so if you were to stretch that entity, its dimension would automatically be updated. Toggle Assoc on if you want to draw associative dimensions. Associative dimensions can be exploded, or changed to standard dimensions that aren't connected to the entity. See "Exploding Dimensions" later in this chapter for more information.
 4. Customize the way dimension text is drawn by clicking on TextStyl and setting the options as necessary. (See "Text Style Options for Linear Dimensions" on the following pages for details.)
 5. Customize the way dimension lines are drawn by clicking on DimStyl and setting the options as necessary. (See "Line Style Options for Linear Dimensions" on the following pages for details.)
 6. Customize the way dimension arrows are drawn by clicking on ArroStyl and setting the options as necessary. (See "Arrow Style Options for Linear Dimensions" on the following pages for details.)

7. Customize the way automatic dimensions are drawn by clicking on AutoStyl and setting the options as necessary. (See “Automatic Dimension Style Options for Linear Dimensions” on the following pages for details.)
8. Click on AutoDim. You are prompted to “Select first end point of line to dimension along”.
9. Object snap to one end point of the distance you want to dimension for greater accuracy. You can also select this point by clicking in the Drawing Area or using coordinate entry. You are prompted to “Select second end point of line to dimension along”.
10. Object snap to the second end point of the distance you want to dimension. The dimension lines are displayed; notice if you move your mouse cursor to the other side of the entity you’re dimensioning, the dimension lines flip to that side. You are asked, “Dimension which side of line?”
11. Click to one side of the line to place the dimension on that side. The series of dimensions is drawn.
12. Continue using automatic dimensioning by going back to step 9 above.

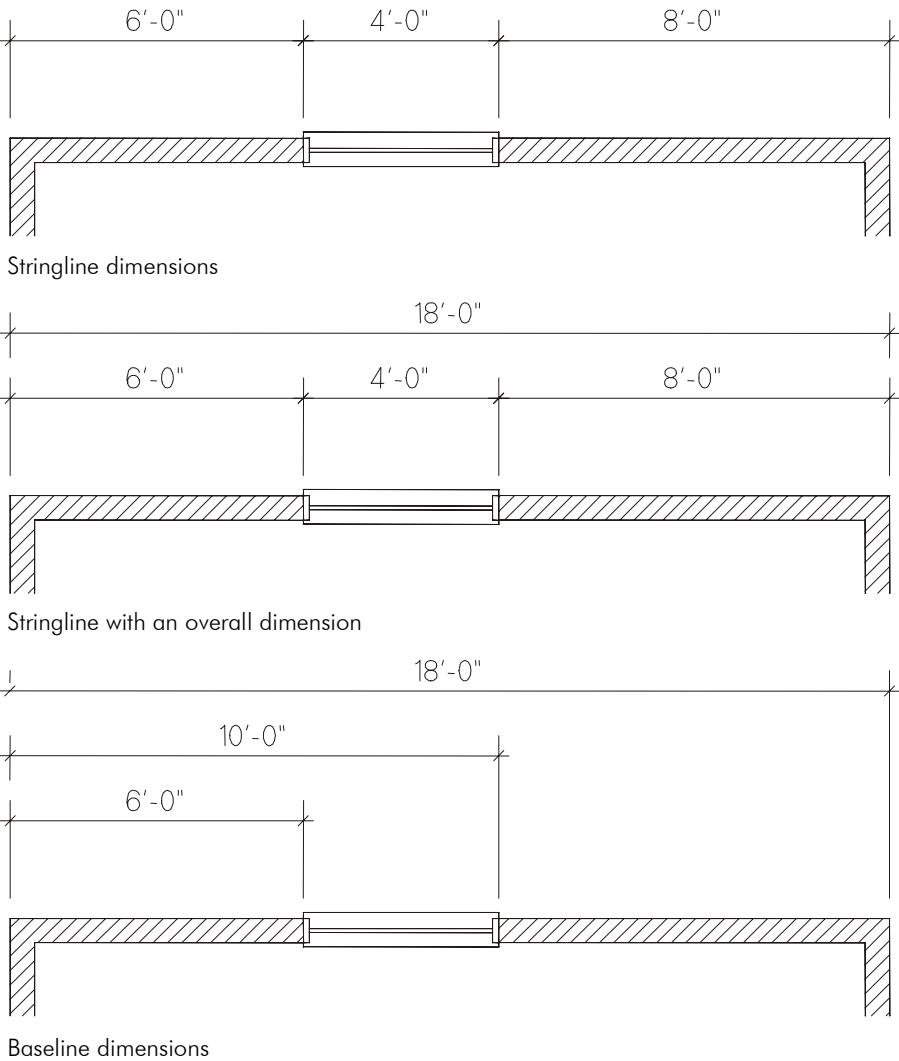


Figure 14.2: Examples of stringline, stringline with overall, and baseline dimensions

Text Style Options for Linear Dimensions

Use the following options in the TextStyl menu to customize dimension text styles:

TextSize	Set the dimension text’s actual size in your drawing. The TextSize, Weight, Slant, Aspect, Color, TxtScale, and FontName options work exactly like equivalent options in the Text menu. See the “Text” chapter for more information.
For more information on using value menus, see “Value Menus” in “The Drawing Board” chapter. The decimal separator for all dimensions and numbers is determined by the Decimal Symbol setting. In your Windows Start menu, click on Set-tings, click on Control Panel, click on Regional Settings, and click on the Number tab.	
Weight	Set the line weight of dimension text, from 1 to 99 with “1” being the default weight setting; to triple the weight of text, for instance, set Weight to 3
Slant	Set the angle of each character of text, as measured from the base of the character to its top
Aspect	Set the height of each character relative to its width; an aspect ratio of 1 represents the normal character style, while an aspect ratio of 5 produces text that is taller and narrower
TxtScale	Toggle TxtScale on to size your text by using the Size setting relative to the plot scale; toggle TxtScale off to size your text size based only on the TextSize setting

InHorz	Toggle on to place dimension text horizontally in your drawing (independent of the orientation of the dimension lines); the dimension line is “broken” to accommodate the text
OutHorz	Toggle on to place dimension text horizontally in your drawing (independent of the orientation of the dimension lines) and at the end of the leader; this option is only used with the DrwLeader option, which is available only if the text is too large to fit between the dimension’s extension lines
Above	Toggle on to draw the dimension text parallel to and “above” the dimension line; if InHorz is also toggled on, then the dimension text will be drawn horizontally but the dimension line will not be “broken”
Offset	Set the distance of text from the dimension line
Auto	Toggle on to automatically place text, according to the TextStyl menu settings; toggle off to place the text for each dimension manually, using Rotate, DrwLeadr, or TextStyl as described in the instructions for drawing dimensions earlier in this chapter; when Auto is off, right-clicking to place the dimension text will place the text as if Auto were toggled on
Color	Set the color of dimension text. For more information on using color menus, see “Color Menus” in “The Drawing Board” chapter.
FontName	Set the font used for dimension text (see Figure 14.3)

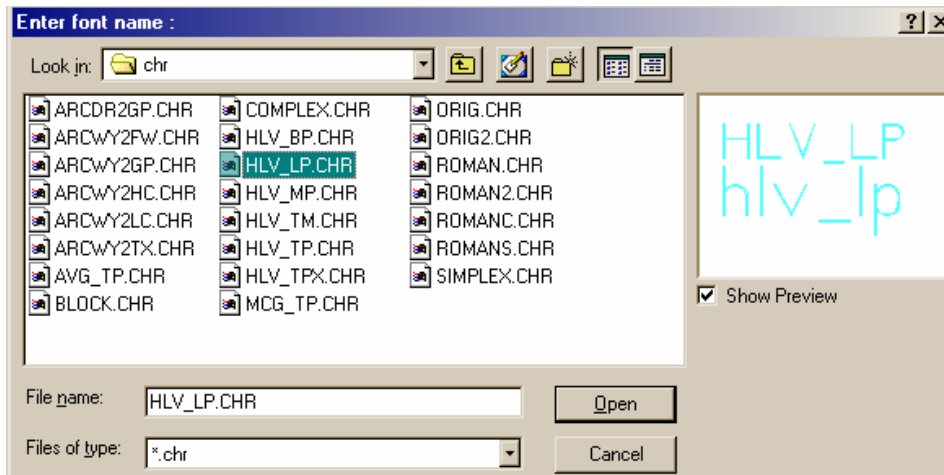


Figure 14.3: Selecting a font, with a preview of the selection

Line Style Options for Linear Dimensions

Each dimension in a string-line shares its extension lines with the next dimension in the series, so you see only a single line displayed on screen. But when you plot your drawing, your plotter reads two lines – the second extension line for one dimension and the first extension line for the following dimension. If your plotter prints both these lines, extension lines in a string-line can appear to have a double thickness. To avoid this, toggle either Line 1 or Line 2 off when you draw stringline dimensions.

Use the following options in the DimStyl menu to customize dimension line styles:

Line 1	Toggle on to draw the first extension line
Line 2	Toggle on to draw the second extension line
Offset	Set the distance between the entity being dimensioned and the beginning of the extension lines of the dimension (see Figure 14.4)
Overlap	Set the distance extension lines are drawn past dimension lines (see Figure 14.4)
Increment	Set the distance between dimension lines when drawing baseline dimensions; this option has no effect on stringline dimensions unless set to 0, which results in a running dimension without the first arrow (see Figure 14.4)

Overrun	Set the distance dimension lines are drawn past extension lines; a setting of 0 results in the dimension line ending at the extension lines (see Figure 14.4)
FixdDist	Toggle on to draw dimension lines a fixed distance away from the entity being dimensioned; toggle off to enter the distance yourself when you click to select which side of the entity to draw the dimension on (see Figure 14.4)
DINStd	Toggle on to use the DIN Standard, set by the German Institute for Standardization (see Figure 14.5)
Rounding	Set rounding for dimension text; toggle RoundIt on and choose from rounding up or down or banker's rounding; click on Precision to set the fractional rounding value for rounding up or down
Limits	Set the upper and lower limit for the size of entities to dimension (see Figure 14.6)
Tolerance	Set positive and negative geometric tolerances (see Figure 14.6)

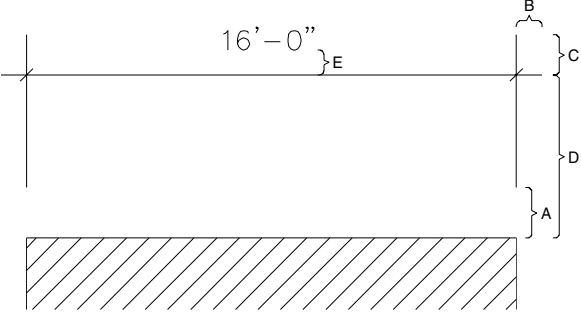


Figure 14.4: Dimension settings: A = Offset, B = Overrun, C = Overlap, D = Increment (with FixdDist toggled on), E = Text Offset

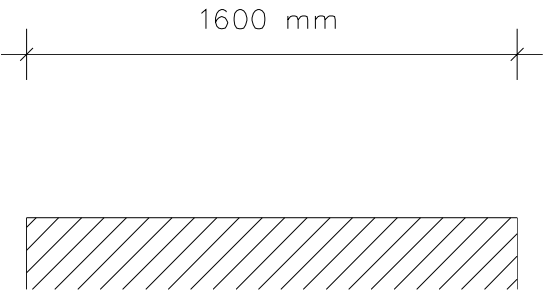


Figure 14.5: DIN standard dimension

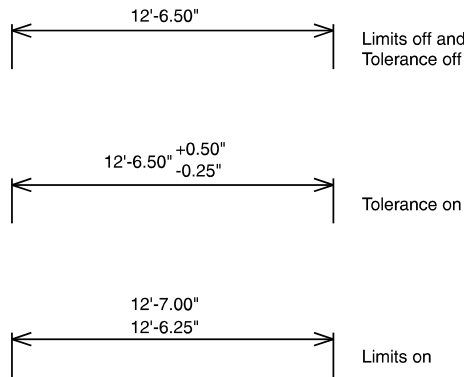


Figure 14.6: Dimensions with Limit and Tolerance settings

Arrow Style Options for Linear Dimensions

Use the following options in the ArroStyl menu to customize dimension arrow styles:

Of the three options Arrows, TickMrks, and Dots, only one can be toggled on at a time.

Arrows Toggle on to draw arrows at the ends of dimension lines

TickMrks	Toggle on to draw tick marks at the ends of dimension lines
Dots	Toggle on to draw dots at the ends of dimension lines
Size	Set the size of dimension arrows, tick marks, or dots; this setting is relative to the dimension text size, so if Size is set to 2, dimension arrows will be drawn twice the size of dimension text. The dimension text size is set using the TextSize option in the Linear/TextStyl menu.
Weight	Set the line weight of tick marks; not available if Arrows or Dots is toggled on
Aspect	Set the aspect ratio of the dimension arrows; this aspect ratio setting is the length of the arrowhead divided by the height, so a setting of 1 draws arrowheads with a 90° angle; not available when TickMrks or Dots is toggled on
Color	Set the color of arrows, dots, or tick marks. For more information on using color menus, see “Color Menus” in “The Drawing Board” chapter.

Automatic Dimension Style Options for Linear Dimensions

Use the following options in the AutoStyl menu to customize automatic dimension styles:

Baseline	Toggle on to draw a series of dimensions that all have one end point in common
StrngLin	Toggle on to draw a series of dimensions, each of which uses the last end point of the previous dimension as the first end point of the next dimension

Of the two options OnlyPnts and NoPnts, only one can be toggled on at a time.

OnlyPnts	Toggle on to dimension to center points only; useful for dimensioning window and door openings
NoPnts	Toggle on to dimension to every center and end point that is greater than the minimum distance to dimension (set with the MinDist option)
MinDist	Set the minimum distance to dimension; a setting of zero (0) results in all distances, no matter how small, being dimensioned
MissDist	Set the search distance for points to dimension; the miss distance is perpendicular to the line you want to dimension. For more information on using value menus, see “Value Menus” in “The Drawing Board” chapter.
LyrSrch	Toggle on if any of the entities you want to dimension are on layers other than the active layer
DirDim	Toggle on to automatically go directly to the AutoDim menu when you click on Linear Dimension in the Create pull-down menu

Editing Linear Dimensions

You can edit any linear dimensions in your drawing by changing the text styles, lines styles, or arrow styles of the dimension. You can also change the position of dimension text.

➔ To edit linear dimension text, lines, or arrows:

1. Click on Dimension in the Create pull-down menu, and then click on Linear Dimension in the submenu. The Linear menu is displayed in the Menu Window.
2. Click on Change in the Linear menu. The Change menu is displayed in the Menu Window.
3. Go to the Change menu and click on TextStyl to change the dimension text settings, click on DimStyl to change the dimension line settings, or click on ArroStyl to change the dimension arrow settings. The TextStyl, DimStyl, and ArroStyl options in the Change menu are the same as those in the Linear menu. See “Text Style Options for Linear Dimensions”, “Line Style Options for Linear Dimensions”, and “Arrow Style Options for Linear Dimensions” earlier in this chapter for details.

4. Click on a selection method in the Change menu. You are prompted to select the dimension you want to change.
5. Click on the dimension you want to edit. The dimension is changed to match the text, line, and arrow style changes you made in step 3.

➔ To move dimension text:

1. Click on Dimension in the Create pull-down menu, and then click on Linear Dimension in the submenu. The Linear menu is displayed in the Menu Window.
2. Click on Change in the Linear menu. The Change menu is displayed in the Menu Window.
3. Click on TxtPostn in the Change menu. You are prompted to “Select associative dimension to change text position”.
4. Click on the dimension text you want to move. A box representing the text is connected to your cursor, and the Rotate option is displayed in the menu.
5. Rotate the text before repositioning it in your drawing by clicking on Rotate and using the value menu to enter a rotation angle. Otherwise, continue with step 6.
6. Move your cursor to the new location and click to place the text in its new position, or use coordinate entry to enter a new location for the text. For more information on using coordinate entry, see “Drawing Using Coordinate Entry” in “The Drawing Board” chapter.
7. Move dimension text back to its default position by clicking on Default and then clicking on the dimension text. The text is returned to its default angle and position.

Exploding Linear Dimensions

While you can freely edit non-associative dimensions, there may also be times that you need to edit an associative dimension measurement. Associative dimensions are connected to the entity they measure, so these measurements automatically adjust when the entity changes size. To edit an associative dimension, you first must *explode* the dimension, or disconnect it from the entity it's associated with.

When you explode dimensions, their appearance is not affected.

➔ To explode an associative dimension:

1. Click on Dimension in the Create pull-down menu, and then click on Linear Dimension in the submenu. The Linear menu is displayed in the Menu Window.
2. Click on Explode in the Linear menu. The Explode menu is displayed in the Menu Window.
3. Click on a selection method in the Explode menu. You are prompted to select the dimension you want to explode.

4. Click on the associative dimension(s) you want to explode. The dimensions you selected are *exploded*, or disassociated from the entities they measure.

Drawing Dimensions for Angles and Curves

You can dimension the angle between two lines as well as the diameter or radius of a curve.

➔ To dimension the angle between two lines:

1. Click on Dimension in the Create pull-down menu, and then click on Angular Dimension in the submenu. The Angular menu is displayed in the Menu Window.
2. Customize the way dimension text is drawn by clicking on TextStyl and setting the options as necessary. (See “Text Style Options for Angle and Curve Dimensions” on the following pages for details.)
3. Customize the way dimension lines are drawn by clicking on DimStyl and setting the options as necessary. (See “Line Style Options for Angle and Curve Dimensions” on the following pages for details.)
4. Customize the way dimension arrows are drawn by clicking on ArroStyl and setting the options as necessary. (See “Arrow Style Options for Angle and Curve Dimensions” on the following pages for details.)
5. Toggle LyrSrch on if the lines that form the angle you want to dimension are on different layers. You are prompted to “Select first line”.
6. Click on one of the lines in your drawing that forms the angle you want to dimension. The line is displayed as dashed to indicate its selection. You are prompted to “Select second line”.
7. Click on the other line in your drawing that forms the angle you want to dimension. The line is displayed as dashed to indicate its selection. Notice, too, that your cursor is connected to a curve representing the angle dimension line; moving your cursor changes the position of the dimension line.
8. Position the angle dimension line and click to place it in your drawing. You are prompted to “Enter angle”.
9. Press to accept the actual angle measurement, or type an angle and press . The dimension line is drawn, and a small box representing the dimension text is connected to your cursor.
10. Place the dimension text next to the dimension line by positioning your cursor and clicking. To place the text in its default position, simply right-click. The dimension is completed.

➔ To dimension a curve’s diameter:

1. Click on Dimension in the Create pull-down menu, and then click on Diameter Dimension in the submenu. The Diameter menu is displayed in the Menu Window.

2. Customize the way dimension text is drawn by clicking on TextStyl and setting the options as necessary. (See “Text Style Options for Angle and Curve Dimensions” on the following pages for details.)
3. Customize the way dimension lines are drawn by clicking on DimStyl and setting the options as necessary. (See “Line Style Options for Angle and Curve Dimensions” on the following pages for details.)
4. Customize the way dimension arrows are drawn by clicking on ArroStyl and setting the options as necessary. (See “Arrow Style Options for Angle and Curve Dimensions” on the following pages for details.)
5. Toggle LyrSrch on if the curve you want to dimension is on another layer. You are prompted to “Select arc or circle to dimension”.
6. Click on the curve you want to dimension. You are prompted to “Enter diameter”.
7. Press to accept the actual diameter measurement, or type a diameter and press . The dimension is drawn.

➔ To dimension a curve’s radius:

1. Click on Dimension in the Create pull-down menu, and then click on Radius Dimension in the submenu. The Radius menu is displayed in the Menu Window.
2. Customize the way dimension text is drawn by clicking on TextStyl and setting the options as necessary. (See “Text Style Options for Angle and Curve Dimensions” on the following pages for details.)
3. Customize the way dimension lines are drawn by clicking on DimStyl and setting the options as necessary. (See “Line Style Options for Angle and Curve Dimensions” on the following pages for details.)
4. Customize the way dimension arrows are drawn by clicking on ArroStyl and setting the options as necessary. (See “Arrow Style Options for Angle and Curve Dimensions” on the following pages for details.)
5. Toggle LyrSrch on if the curve you want to dimension is on another layer. You are prompted to “Select arc or circle to dimension”.
6. Click on the curve you want to dimension. You are prompted to “Enter radius”.
7. Press to accept the actual radius measurement, or type a radius and press . The dimension is drawn.

Text Style Options for Angle and Curve Dimensions

Use the following options in the TextStyl menu to customize dimension text styles:

TextSize	Set the dimension text’s actual size in your drawing. The TextSize, Weight, Slant, Aspect, Color, TxtScale, and FontName options work exactly like equivalent options in the Text menu. See the “Text” chapter for more information.
Weight	Set the line weight of dimension text, from 1 to 99 with “1” being the default weight setting; to triple the weight of text, for instance, set Weight to 3. For more information on using value menus, see “Value Menus” in “The Drawing Board” chapter.

Slant	Set the angle of each character of text, as measured from the base of the character to its top
Aspect	Set the height of each character relative to its width; an aspect ratio of 1 represents the normal character style, while an aspect ratio of 5 produces text that is taller and narrower
TxtScale	Toggle TxtScale on to size your text by using the Size setting relative to the plot scale; toggle TxtScale off to size your text size based only on the TextSize setting
Auto	Toggle on to automatically place text, according to the TextStyl menu settings; toggle off to place the text for each dimension manually, using Rotate, DrwLead, or TextStyl as described in the instructions for drawing dimensions earlier in this chapter
Color	Set the color of dimension text
FontName	Set the font used for dimension text (See Figure 14.3)

The decimal separator for all dimensions and numbers is determined by the Decimal Symbol setting. In your Windows Start menu, click on Set-tings, click on Control Panel, click on Regional Settings, and click on the Number tab.

Line Style Options for Angle and Curve Dimensions

Use the following options in the DimStyl menu to customize dimension line styles:

Offset	Set the distance between the entity being dimensioned and the beginning of the extension lines of the dimension. See Figure 14.4 for an example of Offset and Overlap settings.
Overlap	Set the distance extension lines are drawn past dimension lines
CntrMark	Toggle on to draw center marks for diameter and radius dimensions
Rounding	Set rounding for dimension text; toggle RoundIt on and choose from rounding up or down or banker's rounding; click on Precsion to set the fractional rounding value for rounding up or down
Limits	Set the upper and lower limit for the size of entities to dimension (see Figure 14.6)
Tolerance	Set positive and negative geometric tolerances (see Figure 14.6)

Arrow Style Options for Angle and Curve Dimensions

While the Arrows, TickMrks, and Dots toggles are displayed only in the ArroStyl menu for linear dimensions, they affect all four dimension types: Linear, Angular, Diameter, and Radius. Use the following options in the ArroStyl menu to customize dimension arrow styles:

Size	Set the size of dimension arrows, tick marks, or dots; this setting is relative to the dimension text size, so if Size is set to 2, dimension arrows will be drawn twice the size of dimension text
Weight	Set the line weight of tick marks; not available if Arrows or Dots is toggled on in the ArroStyl menu for linear dimensions
Aspect	Set the aspect ratio of the dimension arrows; this aspect ratio setting is the length of the arrowhead divided by the height, so a setting of 1 draws arrowheads with a 90° angle; not available when TickMrks or Dots is toggled on in the ArroStyl menu for linear dimensions
Color	Set the color of arrows, dots, or tick marks. For more information on using color menus, see "Color Menus" in "The Drawing Board" chapter.

Hatch patterns can add detail as well as visual information to your drawings. In DataCAD, there are two types of hatch, standard and associative. Standard hatch is a group of entities inserted in your drawing according to Hatch menu settings. Associative hatch, however, is a single entity and will update itself as necessary when you edit your drawing. Both types of hatch require you to define polyline boundaries for the hatch pattern.

In this chapter:

- ⊕ Drawing hatch
- ⊕ Hatch boundaries
- ⊕ Associative hatching

Drawing Hatch Patterns

You can fill areas of your drawing with repetitive patterns, called *hatch patterns*, to distinguish between building materials or to highlight parts of your drawing. Standard hatch, however, can increase file size and refresh times dramatically. To control file size as well as save time while editing your drawing, use associative hatch. See the Appendix for a list of the hatch patterns that come with DataCAD.

Drawing Standard Hatch

The lines in hatch patterns are separate entities that are linked as a group, so you can move or edit the entire area of hatch using the Group option in selection menus. However, you can also edit individual lines in the hatch pattern using the Entity selection method.

➔ To draw a standard hatch pattern:

1. Click on Hatch in the Create pull-down menu, or click on Hatch in the Utility menu in the Menu Window. The Hatch menu is displayed in the Menu Window.
2. Click on HtchType to set the hatching type. The HtchLine menu is displayed.
3. Toggle Normal on to alternate hatching within defined boundaries, toggle OutMost on to hatch only the first boundary, or toggle Ignore on to hatch within the outermost boundary and ignore all other boundaries (see Figures 15.1, 15.2, and 15.3 for examples).

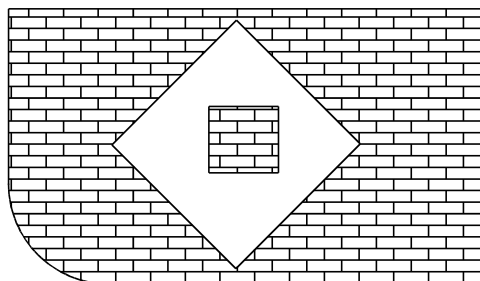


Figure 15.1: Hatching using the Normal hatching type

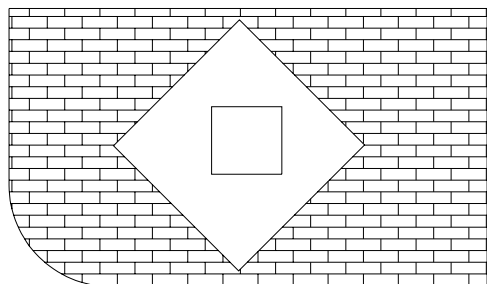


Figure 15.2: Hatching using the OutMost hatching type

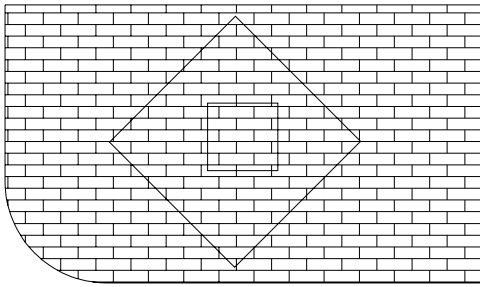


Figure 15.3: Hatching using the Ignore hatching type

4. Right-click to return to the Hatch menu.
5. Click on Pattern. A list of hatch patterns is displayed in the Menu Window, along with a preview window. Notice that as you move your cursor over each hatch pattern name in the list, an example of the highlighted hatch pattern is displayed in the preview window. Click on **ScrlFwd** to scroll forward through the list of patterns; click on **ScrlBack** to scroll back through the list.
6. Choose a pattern by clicking on its name in the list. You are returned to the Hatch menu.
7. Change the scale that the hatch will be drawn in by clicking on Scale. A value menu is displayed. The Scale setting in the Hatch menu affects only the scale of the hatch pattern; it does not change the drawing scale. The pattern Line, for example, is based on $1/32''$ increments. To draw the Line pattern with 4" spacing, set Scale to 128. ($128 * 1/32'' = 4''$).
8. Use the value menu or type a scale, and press **Enter**.
9. Change the angle of the pattern by clicking on Angle. A value menu is displayed.
10. Use the value menu or type an angle, and press **Enter**. (The default setting is 0° .)
11. Set the point where the hatch pattern will begin by clicking on Origin. You are prompted to "Select hatch pattern origin".
12. Click in the Drawing Area, use coordinate entry, or object snap to an entity to select the origin for the hatch pattern.
13. Define a boundary for the hatch pattern. A *boundary* is a polyline that defines the area of your drawing that will be hatched. Click on Boundary. The Polyline menu is displayed, and you are prompted to "Select first point on boundary to hatch to". To define a rectangular boundary, simply click on Boundary and then **RectAngl** in the Polyline menu, and enter two, diagonally opposite points.
14. Select the first point of the boundary by clicking in the Drawing Area, using coordinate entry, or object snapping to an entity. The options **2PtArc** and **3PtArc** are displayed in the Polyline menu. You are prompted to "Select next point on boundary to hatch to". For more information on coordinate entry, see "Drawing Using Coordinate Entry" in "The Drawing Board" chapter.
15. Add a line segment, 2-point arc, or 3-point arc to your polyline boundary:

- To select the second point of a line segment, click in the Drawing Area, use coordinate entry, or object snap to a point on an entity.
- To add a 2-point arc to the boundary, click on 2PtArc in the Polyline menu. Select a center point for the arc, and then select an end point. Right-click to exit the 2-point arc function.
- To add a 3-point arc to the boundary, click on 3PtArc in the Polyline menu. Select a second end point for the arc (with the first end point being the point you selected in the previous step), and then select any other point on the arc. Right-click to exit the 3-point arc function.

The option Tangent is displayed in the Polyline menu, and you are prompted to “Select next point on boundary to hatch to”.

You can click Backup any time to erase the last line segment or arc of your polyline. You can click Cancel at any time to erase the boundary you’re drawing.

16. Draw a tangent arc in your polyline by clicking on Tangent in the Polyline menu. Move your cursor until the arc displayed is the one you want drawn, and then click to draw the arc.
 17. Continue selecting points for line segments or arcs, as outlined in steps 15 and 16, until your polyline boundary is complete.
 18. Click on Close in the Polyline menu. The Hatch menu is displayed in the Menu Window.
 19. Click on Begin to hatch the area within the boundary. The hatch pattern is drawn. Click on Clear to delete the boundary and return all hatch settings to their defaults.
- ➔ To hatch a master polyline that contains voids (or areas that you do not want to hatch):
1. Create the master polyline. This will contain the hatch pattern. For details on creating the master polyline, see “Voids in Polylines” in the “Drawing Other Geometry” chapter.
 2. Create the polylines that will serve as voids (areas that will not be hatched).
 3. Click on Voids in the Polyline menu. You are prompted to “Select master polyline to process voids”.
 4. Click on the closed master polyline that you want to put the void in. DataCAD puts dashed lines around that shape. You are prompted to “Select entity to select polylines to convert to voids”.
 5. Make sure Entity and AddVoid are toggled on in the Voids menu.
 6. Click on the polyline that will represent the void. The void is added. Dashed lines surround the void.
 7. Continue selecting other closed polylines within the master if necessary.
 8. Select Hatch from the Utility menu.

9. Click Pattern and choose one of the options. Make other adjustments to the pattern if necessary. For example, you may want to change the scale or the angle.
10. Press (Enter) when you are satisfied that the pattern, scale, angle, and hatch type shown in the Message Window meet your needs.
11. Click the polyline you created in steps 1 through 7. This selects the polyline.
12. Click on Begin. The hatch pattern you selected appears within the master polyline boundary; however, the voids you created are empty.

➔ To hatch a closed polyline with contour search:

1. Select Hatch from the Utility menu.
2. Use Pattern and select one of the options. Make other adjustments (such as changing the scale or the angle). Press (Enter) when you are satisfied with your selections.
3. Choose Boundary from the Hatch menu.
4. Click on ContSrch in the Boundary menu. You are prompted to “Select point inside/outside contour.”
5. Click on the boundary of the closed polyline you want to use.
6. Right-click to return to the Hatch menu.
7. Select Begin. DataCAD applies the hatch pattern you selected within the closed polyline you identified with ContSrch.

Using SPB Fills Instead of Standard Hatch Patterns

Instead of using the standard vector-based hatch patterns (saved as .PAT files), you can use solid, pattern, and bitmap (SPB) fills. Unlike hatch patterns, solid fills will display and print with the same density, no matter what the viewed or plotted scale is.

➔ To use solid, pattern, and bitmap fills:

1. Click on Display in the Utility menu and make sure Fill On and Bmp On are activated in the Display menu. Then, right-click to return to the Utility menu. In the Status Window, there is now an upper-case F in SWOHLUDFB, indicating that Fill is on (lower-case f means it's off).
2. Select Hatch from the Utility menu.
3. Choose SPB-Fill from the Hatch menu.
4. Click on Settings to choose the colors and patterns you want in your drawing. The Select Fill Color/Pattern dialog box appears.

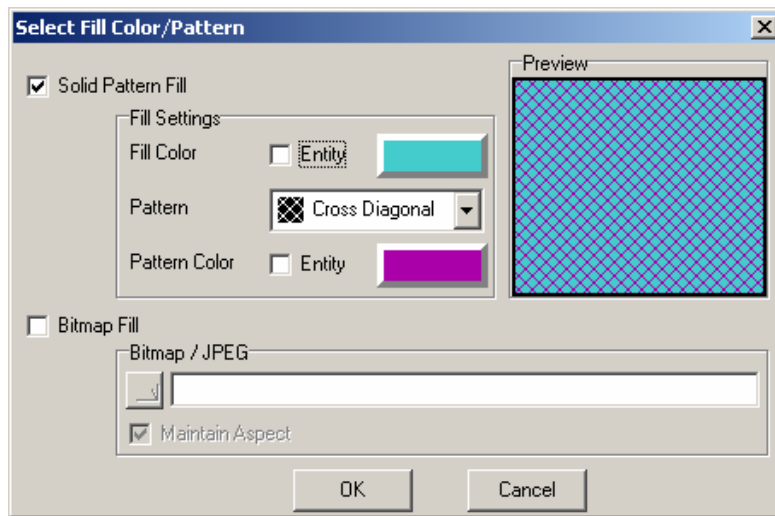


Figure 15.4: Control the solid, pattern, and bitmap (SPB) fills.

5. Make decisions about the solid, pattern, and/or bitmap fills for your drawing. Your choices are displayed in the Preview window of the Select Fill Color/Pattern dialog box.
 - Click in the box beside Solid Pattern Fill or Bitmap Fill. These are mutually exclusive.
 - Use the Fill Settings options to make decisions about solid pattern details.
 - Control the fill and pattern colors by clicking on the rectangular swatch and making your selections from the Color Palette dialog box. For more information, see “Customizing Colors” in the “Before You Draw” chapter.
 - Select the pattern you prefer from the drop-down box. The outline boundary, pattern, and fill may have different colors. Your choices are shown in the Preview window.
 - Check the Entity box if you want the fill to display with the color of the polyline entity in the drawing.
 - Use the Bitmap/JPEG options to make decisions about bitmap fills.
 - Click the file folder button to select the .BMP or .JPG image you want to use.
 - Uncheck the Maintain Aspect box if you want to change the X/Y aspect of the original image.
6. Click OK on the Select Fill Color/Pattern dialog box when you are satisfied with your choices. The dialog box disappears.
7. Click on an existing polyline area that you want to fill. If none exists yet, select Boundary from the SolidFil menu and draw one.

You can control whether solid fills and bitmaps print behind or in front of lines by using the Print first or Print last options in the Pen Table. For more information about using the Pen Table, see “Using Pen Tables” in the “Printing Your Drawing” chapter.

Drawing Associative Hatch

Associative hatch treats hatch patterns as a single entity, instead of a group of entities as standard hatch does. You can associatively hatch only 3D polygons and 2D polylines. When you use associative hatch:

- You can stretch walls in your drawing, and the hatch will update automatically.
- You can rotate your drawing without changing the angle of the hatch pattern.
- Your drawing file size is usually smaller because the hatch pattern is a single entity instead of a group of entities.
- You can't edit individual lines in the hatch pattern.
- The hatch is drawn the same color as your entity. If you use standard hatch (Associat toggled off), the hatch is drawn in the current layer's color.
- You have different hatch type options: OutLine draws an outline around the hatching, while NoOutLin does not.

➔ To draw associative hatching:

1. Click on Hatch in the Create pull-down menu, or click on Hatch in the Utility menu in the Menu Window. The Hatch menu is displayed in the Menu Window.
2. Click on HtchType to set the hatching type. The HtchLine menu is displayed.
3. Toggle Outline on to draw an outline around the hatching; toggle NoOutLin on to draw only the hatching.
4. Continue with step 5 in the instructions on drawing a standard hatch pattern on the previous pages.

After the hatching is drawn, you can click on DelAssoc to remove the associative hatching, without clearing the boundary you traced. Click Clear to delete the boundary and return the hatch settings to their defaults.

7.

Symbols, Images, and Objects

A *symbol* is a group of lines and arcs that is treated as a single entity. Symbols in DataCAD can be organized and displayed in templates. With DataCAD's Template menu, you can enter symbols in your drawing, revise symbol descriptions as well as the symbols themselves, and even create new symbols from geometry in your drawing. Each symbol can have an unlimited amount of information associated with it. And one of the biggest advantages to using symbols in your drawings, instead of separate lines and arcs, is that you can use this information to generate a variety of schedules and estimates.

In addition to symbols, you can also import bitmap (.BMP) and JPEG (.JPG) images into your drawings or insert o2c objects.

In this chapter:

- ⊕ Using templates
- ⊕ Creating templates
- ⊕ Working with symbols
- ⊕ Creating symbols
- ⊕ Importing BMP and JPEG images
- ⊕ Inserting o2c objects

Inserting Symbols in Your Drawing

There are two methods for inserting symbols into your drawing. Using the Insert pull-down menu is faster, but using the Template menu offers you more options for working with your symbols.

➔ To insert a symbol into your drawing using the Insert menu:

1. Click on Symbol in the Insert pull-down menu. A dialog box is displayed, listing all symbol folders in your DATACAD\SYM folder.

Shortcut: Press [""] (the double quotation key) to insert a symbol.

2. Double-click on a symbol folder to open it and display those symbols.
3. Click on the name of the symbol you want to use; if Show Preview is checked, a preview of the selected symbol is displayed in the preview window on the right side of the dialog box. The symbol preview displays in either orthographic (plan) or isometric view, depending on either the Symbol Preview setting on the Misc tab of the Program Preferences dialog box, available from the Tools pull-down menu, or the Isometric View checkbox in the Insert Symbol dialog box.
4. Click on Open. The dialog box closes, and the Insert/Symbol menu is displayed in the Menu Window. Notice that a box representing the extents of the symbol is now attached to your cursor at the symbol's insertion point.
5. Change the symbol's size if necessary. Before you place the symbol in your drawing, you can increase or decrease the size of an entity by entering specific enlargement values for the X, Y, and Z axes:
 - To resize the symbol while maintaining its original scale, increase the XEnlgmt, YEnlgmt, and ZEnlgmt values equally
 - To adjust the size of the symbol along the X axis, click on XEnlgmt. Use the value menu or type an enlargement factor, and press [Enter].
 - To adjust the size of the symbol along the Y axis, click on YEnlgmt. Use the value menu or type an enlargement factor, and press [Enter].
 - To adjust the size of the symbol along the Z axis, click on ZEnlgmt. Use the value menu or type an enlargement factor, and press [Enter].
6. Rotate the symbol either dynamically positioning it or by setting a specific rotation angle for the symbol. The symbol is rotated around the insertion point, the point where the symbol is attached to your cursor.
 - To dynamically rotate the symbol into position when you place it in your drawing, toggle DynamRot on.
 - To set a specific rotation angle, toggle DynamRot off. Use the value menu or type an angle, and press [Enter].
7. Convert the symbol into standard and separate lines and arcs when you place it in your drawing by toggling Explode on.

8. Place the symbol by clicking in the Drawing Area, using coordinate entry, or object snapping to a point in your drawing. The symbol is inserted into your drawing. For details on coordinate entry, see “Drawing Using Coordinate Entry” in “The Drawing Board” chapter. For details on object snapping, see “Object Snapping” in the “Drawing Tools” chapter.
9. Repeat the previous step to continue placing the symbol in your drawing, or right-click to exit the menu.

➔ To insert a symbol into your drawing using the Template menu:

1. Right-click until the Utility menu is displayed in the Menu Window.
2. Click on Template in the Utility menu. A dialog box is displayed.

Shortcut: Press **[T]** anytime to open the Template menu.

3. Locate the template you’d like to use and click on it to select it. The templates that come with DataCAD are in your DATACAD\TPL folder.
4. Click on Open. The template is displayed on the right side of the Drawing Area, and the Template menu is displayed in the Menu Window.
5. Rotate the symbol into position either dynamically or by setting a specific rotation angle. The symbol is rotated around the insertion point, the point where the symbol is attached to your cursor.
 - To dynamically rotate the symbol into position when you place it in your drawing, toggle DynamRot on.
 - To set a specific rotation angle, toggle DynamRot off. Use the value menu or type an angle, and press **[Enter]**.
6. Change the symbol size if necessary. Before you place a symbol in your drawing, you can increase or decrease the size of an entity by entering specific enlargement values for the X, Y, and Z axes:
 - To enlarge entities along the X axis, click on XEnlgmt. Use the value menu or type an enlargement factor, and press **[Enter]**.
 - To enlarge entities along the Y axis, click on YEnlgmt. Use the value menu or type an enlargement factor, and press **[Enter]**.
 - To enlarge entities along the Z axis, click on ZEnlgmt. Use the value menu or type an enlargement factor, and press **[Enter]**.
 - To set the line spacing enlargement factor for non-solid linetypes, click on LineFact. Use the value menu or type a line factor, and press **[Enter]**. The LineFact setting affects how linetypes like Dashed are enlarged. A setting matching those of the X, Y, and Z enlargement factors will maintain the linetype’s spacing.
 - To set all of these options to the same setting and enlarge the entity equally in all directions, click on SetAll. Use the value menu or type an enlargement factor, and press **[Enter]**.

7. Decide if you need to place symbols above or below the current Z-base. When placing tree symbols on a 3D site plan for example, you can change the Z-Offset option to enter the trees at the proper elevation. To set the vertical distance above or below the correct Z-base where you'd like to insert your symbols, click on Z-Offset. Use the value menu or type an offset value, and press **Enter**. Enter a positive value to place the symbol above the current Z-base; enter a negative value to place the symbol below the current Z-base.
8. Decide if you want to change the symbol. Normally, symbols are placed in your drawing as a single entity, so DataCAD doesn't recognize the individual lines and arcs that make up the symbol. To be able to edit a symbol in your drawing, you must explode (or convert) the symbol into separate lines and arcs. To convert the symbol into separate lines and arcs when you place it in your drawing, toggle Explode on. (Note that this will only explode those symbols entered into your drawing *after* Explode is toggled on. To explode symbols already in your drawing, you must use the SymExp macro, as described in the "Editing Symbols" section later in this chapter. Exploded symbols appear the same as standard symbols, but DataCAD recognizes only the individual lines and arcs that make up the symbol, and not necessarily their relationship to each other.
9. Select a symbol from the template by moving your cursor over the symbol you want to use and clicking. A box, representing the extents of the symbol, is attached to your cursor at the symbol's insertion point. The symbol's insertion point is marked by a small "x" once you place the symbol in your drawing.

You can object snap to any point on a symbol to select an insertion point other than the one associated with the symbol definition. This is especially useful if you want to align a symbol with another entity in your drawing. Simply snap to the symbol in the template and then snap to the entity in your drawing to place the symbol.

10. Place the symbol in your drawing by clicking in the Drawing Area, using coordinate entry, or object snapping to a point in your drawing. The symbol is inserted into your drawing.
11. Repeat the previous step to continue placing the symbol in your drawing, or right-click to detach from the symbol.
12. Open a different template if necessary by clicking on NewFile and selecting another template file from the dialog box.
13. Close the template window by clicking on TempOff.

More About Templates and Symbols

Each template is a graphic representation of the template (.TPL) file and is displayed with a number of divisions, each containing a symbol. The template file itself is a simple text file containing the name of the file, the number of rows and columns in the template, and the paths to the symbols associated with that template. When you select a template, the symbols associated with that template file are displayed in the template window.

Inserting Plumbing Symbols

Not only can you insert plumbing symbols in your drawing, but you can also design lines of multiple plumbing fixtures and toilet partitions. The **Furnitur** and **Mechanic** options in the **DCAD_AEC** macro open a dialog box, allowing you to choose a template to display.

➔ To insert a plumbing symbol:

1. Click on **DCAD_AEC** in the **Macros** pull-down menu. The **DCAD_AEC** menu is displayed in the **Menu Window**. If **DCAD_AEC** is not listed in the **Macros** pull-down menu, see “Customizing the Macros Menu” for details on how to add it to the menu.
2. Click on **Plumbing** in the **DCAD_AEC** menu. The **Plumbing** menu is displayed in the **Menu Window**. You are prompted to “Enter first point along fixture wall”.
3. Define the wall where you will place plumbing fixtures. To enter one end point of the wall your plumbing fixtures will be drawn along, click in the **Drawing Area**, use coordinate entry, or object snap to a point in your drawing. You are prompted to “Enter second point along fixture wall”.
4. Enter the other end point of the wall by clicking in the **Drawing Area**, using coordinate entry, or object snapping to a point in your drawing. You are prompted to “Point to side of wall on which to add fixtures”.
5. Click on the side of the wall where the plumbing fixtures will be drawn. Several fixture options are displayed in the **Menu Window**.
6. Add fixtures such as handicapped stalls, standard stalls, urinals, and sinks to your drawing. Click on **PlumbSym** to display the standard template dialog box and insert plumbing symbols using the **Template** menu, as described earlier in this chapter. To determine the number of each, click on the “+” and “-” buttons under the **HC Stall**, **Stall**, **Urinal**, and **Sink** headings. The total number of each fixture, along with the available space left along the wall, is displayed in the **Message Area**. You can only add a fixture if available space allows.
7. Customize the dimensions and location of each type of fixture:
 - To customize handicapped stalls, click on **HC Stall**. The **HC Stall** menu is displayed in the **Menu Window**. Use the options, as described in “Handicapped Stall Options” below, to customize handicapped stall fixtures.
 - To customize standard stalls, click on **Stall**. The **Stall** menu is displayed in the **Menu Window**. Use the options, as described in “Standard Stall Options” below, to customize stall fixtures.
 - To customize urinals, click on **Urinal**. The **Urinal** menu is displayed in the **Menu Window**. Use the options, as described in “Urinal Options” below, to customize urinal fixtures.
 - To customize sinks, click on **Sink**. The **Sink** menu is displayed in the **Menu Window**. Use the options, as described in “Sink Options” below, to customize sink fixtures.

8. Draw the plumbing fixtures according to the Plumbing menu options you've set by clicking on Begin. The fixtures are drawn, beginning from the second wall end point you entered. Handicapped stalls are always drawn first, followed by standard stalls, urinals, and finally sinks. You can't customize the order in which fixtures are drawn.

Handicapped Stall Options

DataCAD's toilet symbols are in your DATACAD\SYM\ TOILETS folder.

Number	Set the number of stalls to draw (use the value menu to set the number or click on +HC Stl and -HC Stl in the Plumbing menu). For more information on using value menus, see "Value Menus" in "The Drawing Board" chapter.
Length	Set the length of the stall; the <i>length</i> is measured from the wall behind the fixture to the inside of the partition containing the stall door
Width	Set the width of the stall; the <i>width</i> is measured from the inside of one partition wall to the inside of the other
OpgWidth	Set the width of the stall opening; if a door is drawn, this option also sets the width of the door
Doors	Toggle on to draw doors on stalls; toggle off to draw stalls without doors

The HingeRt and SwingIn options are only available if Doors is toggled on.

HingeRt	Toggle on to draw doors hinged on the right side of stalls (as you face the stall); toggle off to draw doors hinged on the left side of stalls
SwingIn	Toggle on to draw doors that swing into stalls; toggle off to draw doors that swing out from stalls
ToiltSym	Open dialog box and select a custom toilet symbol to use when drawing handicapped stalls
ResetDef	Reset all HC Stall menu settings to their defaults

Standard Stall Options

DataCAD's toilet symbols are in your DATACAD\SYM\ TOILETS folder.

Number	Set the number of stalls to draw (use the value menu to set the number or click on +Stall and -Stall in the Plumbing menu). For more information on using value menus, see "Value Menus" in "The Drawing Board" chapter.
Length	Set the length of the stall; the <i>length</i> is measured from the wall behind the fixture to the inside of the partition containing the stall door
Width	Set the width of the stall; the <i>width</i> is measured from the inside of one partition wall to the inside of the other
OpgWidth	Set the width of the stall opening; if a door is drawn, this option also sets the width of the door
Doors	Toggle on to draw doors on stalls; toggle off to draw stalls without doors

The HingeRt and SwingIn options are only available if Doors is toggled on.

HingeRt	Toggle on to draw doors hinged on the right side of stalls (as you face the stall); toggle off to draw doors hinged on the left side of stalls
SwingIn	Toggle on to draw doors that swing into stalls; toggle off to draw doors that swing out from stalls
ToiltSym	Open dialog box and select a custom toilet symbol to use when drawing stalls
ResetDef	Reset all Stall menu settings to their defaults

Urinal Options

Number	Set the number of urinals to draw (use the value menu to set the number or click on +Urinal and -Urinal in the Plumbing menu). For more information on using value menus, see "Value Menus" in "The Drawing Board" chapter.
Spacing	Set the distance between urinal partitions; if partitions are not drawn, the distance is measured from the center of one urinal to the center of the next

Partitn	Toggle on to draw partitions between urinals; toggle off to draw urinals without partitions (a partition is always drawn between the last urinal and the first sink)
PrtnSize	Set the length of partitions; the <i>length</i> is measured from the wall out to the edge of the partition; PrtnSize is only available when the Partitn option is toggled on
UrnlSym	Open dialog box and select a custom urinal symbol to use when drawing urinals. DataCAD's urinal symbols are in your DATACAD\SYM\ TOILETS folder.
ResetDef	Reset all Urinal menu settings to their defaults

Sink Options

Number	Set the number of sinks to draw (use the value menu to set the number or click on +Sink and -Sink in the Plumbing menu)
Spacing	Set the distance from the center of one sink to the center of the next; only available if FitSinks is toggled off. Because the FitSinks option uses all remaining space on the fixture wall, the Space Left display in the Message Area always reads 0 when FitSinks is toggled on, no matter how many sinks are added or subtracted.
FitSinks	Toggle on to override the Spacing setting and automatically calculate the spacing between sinks; you can add sinks until they butt up against one another but this option will not allow them to overlap; toggle off to use the Spacing setting
CodeMin	Toggle on to set a minimum distance of 1'-6" between the center of the last sink and inner edge of the wall; toggle off to automatically set this to half the center-to-center sink spacing
SinkSym	Open dialog box and select a custom sink symbol to use when drawing sinks. DataCAD's sink symbols are in your DATACAD\SYM\ SINKS folder.
ResetDef	Reset all Sink menu settings to their defaults

Drawing Electrical Symbols and Ceiling Grids

Along with inserting electrical symbols in your drawing, you can also create ceiling grids of any size and orientation.

➔ To insert electrical symbols:

1. Click on DCAD_AEC in the Macros pull-down menu. The DCAD_AEC menu is displayed in the Menu Window. If DCAD_AEC is not listed in the Macros pull-down menu, see "Customizing the Macros Menu" for details on how to add it to the menu.
2. Click on Electric in the DCAD_AEC menu. The Electric menu is displayed in the Menu Window.
3. Click on ElecSymb in the Electric menu. If no template is currently being displayed, a dialog box is displayed.
4. Continue with step 3 of the instructions on inserting a symbol using the Template menu on page 341.

➔ To create a ceiling grid:

1. Click on DCAD_AEC in the Macros pull-down menu. The DCAD_AEC menu is displayed in the Menu Window.
2. Click on Electric in the DCAD_AEC menu. The Electric menu is displayed in the Menu Window.
3. Click on CeilGrid in the Electric menu. The CeilGrid menu is displayed. You can also use the CeilGrid menu to draw a floor grid that you can place furniture symbols on. Be sure to toggle Z-base on in the CeilGrid menu.

4. Draw the ceiling grid at either Z-base or Z-height. To draw the ceiling grid at the current Z-base, toggle Z-Base on in the CeilGrid menu. To draw the grid at the current Z-height, toggle Z-Hite on.
5. Define a boundary for the ceiling grid. Click on Boundary to display the Boundary menu in the Menu Window.
6. Use any 3D polygon as a boundary for your grid. If you already have a 3D polygon in your drawing that you can use for the boundary, you can select it; if you don't, you can create a boundary for your ceiling grid.
 - To use a 3D polygon already in your drawing, click on Select. A selection menu is displayed, and you are prompted to select a ceiling boundary. Choose a selection method from the selection menu, and then select the 3D polygon you want to use for your boundary. Skip to step 10. For more information on using selection menus, see "Selection Menus" in "The Drawing Board" chapter.
 - To create a boundary for your grid, click on Create. The Create menu is displayed in the Menu Window. You are prompted to "Enter the first corner of the ceiling boundary".
7. Click in the Drawing Area, use coordinate entry, or object snap to an entity in your drawing to select the first point for the boundary. You are prompted to "Enter the next corner of the ceiling boundary". You can add your boundary to a specific selection set as well as set the color, linetype, line width, and line spacing of the boundary line. For more information about boundary settings, see Boundary Menu Settings on the following pages.
8. Click in the Drawing Area, use coordinate entry, or object snap to an entity in your drawing to select the next point on the boundary.
9. Repeat the previous step until your boundary is complete. You can click Backup any time to erase the last line segment or arc of your boundary. You can click Cancel at any time to erase the boundary you're drawing.
10. Right-click or click on Close in the Create menu to finish your boundary. You are returned to the Boundary menu.
11. Right-click to exit the Boundary menu and return to the CeilGrid menu.
12. Click on Grids to display the Grids menu. For more information about grid settings, see "Grids Menu Settings" on the following pages.
13. Set the distance between grid lines in the X direction by clicking on X-Size in the Grids menu. A value menu is displayed.
14. Use the value menu or type a distance, and press Enter.
15. Set the distance between grid lines in the Y direction by clicking on Y-Size in the Grids menu. A value menu is displayed.
16. Use the value menu or type a distance, and press Enter.
17. Rotate the grid by clicking on RotAngle in the Grids menu. A value menu is displayed.
18. Use the value menu or type an angle, and press Enter.

19. Notice if you've selected a polygon for your boundary and you want to trace over the polygon to graphically show the boundary, toggle DoBound on in the Grids menu to display the grid boundary when you draw the grid. If you created a boundary in step 6, toggle DoBound off.

You can add your grid to a specific selection set as well as set the color, linetype, line width, and line spacing of the grid lines.

20. Draw the grid by entering a point in the Drawing Area where you want the grid to start. Click in the Drawing Area, use coordinate entry, or object snap to an entity in your drawing to enter the grid starting point. The grid is drawn within the boundary. You can enter the starting point anywhere in the Drawing Area, but the grid will actually be drawn only within the boundary you defined earlier.
21. Right-click to exit the Grids menu and return to the CeilGrid menu.
22. Add lighting fixtures to your ceiling grid. Click on Fixtures in the CeilGrid menu to display the Fixtures menu. For more information about fixture settings, see "Fixtures Menu Settings" on the following pages.
23. Use a symbol for the lighting fixtures by toggling UseSymb on; to use a polygon to represent the fixtures, toggle UsePoly on. (A different set of options will be available, depending on which option you toggle on.)
24. Define the symbol or polygon to be used for your fixtures. Of the two options, UseSymb and UsePoly, only one can be toggled on at any given time.
 - If you toggled UseSymb on, click on SymbNam in the Fixtures menu. A dialog box is displayed, and you are prompted to "Enter file name of symbol to use". Select a symbol to use for the lighting fixtures, and click on Open.
 - If you toggled UsePoly on, click on PolySize. A list of rectangular sizes followed by a list of circular sizes are displayed in the Menu Window. Click on a size for the polygon, and then right-click to return to the Fixtures menu. Use the CustRect option to define a custom-sized rectangle, or use the Cust Rnd option to define a custom circular polygon.
25. Enter the fixtures at an angle in your grid by clicking on RotAngle and using the value menu to enter a rotation angle.
26. Toggle Cutout on to automatically erase the grid lines that would otherwise run through the middle of the fixture symbol or polygon.
27. Enter each fixture by its corner by toggling ByCornr on; to enter each fixture by its center, toggle ByCentr on.
28. Notice that a circle or rectangle representing the fixture is now attached to your cursor. Click in the Drawing Area, use coordinate entry, or object snap to a point on your grid to draw a fixture.
29. Repeat the previous step to enter additional fixtures on your ceiling grid.

Click on Cleanup in the CeilGrid menu to use the Cleanup menu without exiting the DCAD_AEC macro.

Boundary Menu Settings

Select	Select an existing boundary or any 3D polygon to act as a boundary for your ceiling grid
Create	Draw a boundary for the ceiling grid; enter up to 36 sides in a single boundary
Delete	Delete any existing boundaries or 3D polygons
Sel Set	Choose the selection set to add a boundary to; only the first boundary added to a selection set can have a grid drawn in it
ShwBoun	Displays the number of polygons in that selection set
Clear SS	Delete all boundaries from that selection set. The ClearSS option doesn't delete the actual polygons from your drawing; rather this option simply disassociates the polygons from the selection set.
Color	Set the color of boundaries
LineType	Set the linetype of boundaries
LineWdth	Set the width of boundary lines. For more information on using value menus, see "Value Menus" in "The Drawing Board" chapter.
LinSpncg	Set the spacing of boundary lines (only affects linetypes with spaces in them, such as Dashed)

Grids Menu Settings

X-Size	Set the distance between grid lines in the X direction
Y-Size	Set the distance between grid lines in the Y direction
RotAngle	Set the rotation angle of the grid
DoBound	Toggle on to draw the grid boundary when you draw the grid
Grid SS	Set the selection set to add the grid to
ShowGrid	Display the number of lines in the current selection set in the Message Area
Append	Toggle on to add multiple grids to the same selection set; toggle off to replace the previous grid drawn in the current selection set with a new grid. If Append is toggled off and you insert lighting fixtures into overlapping grids, only the last grid drawn will be affected by such settings as Cutout.
Color	Set the color of grids. For more information on using color menus, see "Color Menus" in "The Drawing Board" chapter.
LineType	Set the linetype of grids
LineWdth	Set the width of grid lines. For more information on using value menus, see "Value Menus" in "The Drawing Board" chapter.
LinSpncg	Set the spacing of grid lines (only affects linetypes with spaces in them, such as Dashed)

Fixtures Menu Settings

UseSymb	Toggle on to use a symbol to represent fixtures
UsePoly	Toggle on to use a polygon for fixtures
SymbName	Choose the symbol to use for fixtures; available only when UseSymb is toggled on
PolySize	Choose a rectangular or round polygon to represent fixtures; available only when UsePoly is toggled on
RotAngle	Set the rotation angle for fixtures
Cutout	Toggle on to cut the ceiling grid and erase grid lines that would otherwise display over the fixture
ByCornr	Toggle on to insert fixtures by their corners
ByCentr	Toggle on to insert fixtures by their centers
Grid SS	Set the selection set to add the grid to
ShowGrid	Display the number of lines in the current selection set in the Message Area
ObjSnap	Shortcut to the ObjSnap menu. For more information on object snapping, see "Object Snapping" in the "Drawing Tools" chapter.
Color	Set the color of fixtures
LineType	Set the linetype of fixtures
LineWdth	Set the width of fixture lines
LinSpncg	Set the spacing of fixture lines (only affects linetypes with spaces in them, such as Dashed)

Inserting Stamps

You can choose from more than a dozen pre-formatted date-, time-, and file-related stamps to insert into your drawing. The examples shown in the stamp selection menu are based on the current date, time, and file settings as they will appear in the drawing. The values are updated when the display is regenerated and when the drawing is printed. Since a stamp is really an ordinary DataCAD text entity that contains a special formatting string, it has all the font, sizing, and style options of ordinary DataCAD text strings. You can further customize how you want stamps to appear in your drawing by doing a change/text/content and adding text around the special strings. Stamp text can be used in XREFs and symbols. The values update automatically, making stamps very flexible.

➔ To add a stamp to your drawing:

1. Select Stamp from the Insert pull-down menu.
2. Select the stamp you want to use from the extended menu.

Inserting o2c Objects

Thousands of o2c objects -- including people, cars, furniture, and landscape elements -- are available for you to import into DataCAD drawings. Or you can import your own o2c objects, such as your company logo.

➔ To insert an o2c object:

1. Click on o2c Object in the Insert pull-down menu. A dialog box is displayed, prompting you to "Select o2c file to import".
2. Locate the o2c object you want to insert into your drawing, and click on it to select it.
3. Click on Open. The dialog box closes, and a box representing the extents of the o2c object is now attached to your cursor.
4. Place the object by clicking in the Drawing Area, using coordinate entry, or object snapping to a point in your drawing. The o2c object is inserted into your drawing. For details on coordinate entry, see "Drawing Using Coordinate Entry" in "The Drawing Board" chapter. For details on object snapping, see "Object Snapping" in the "Drawing Tools" chapter.
5. Repeat the previous step to continue placing the object in your drawing, or right-click to exit the menu.

Importing BMP and JPEG Images

You can import both bitmap (.BMP) and JPEG (.JPG) files into your DataCAD drawing. Import any number of black and white, grayscale, and 8- to 32-bit color images. You can trace scanned drawings, display site maps, or show digital photos of existing building facades when you import bitmaps into your drawing.

Imported bitmaps are referenced, not inserted into your drawing file. That means your drawing file size will not increase.

➔ To import an image:

1. Click on Bitmap in the Insert pull-down menu. The Select Image dialog box is displayed.
3. Click on the .BMP or .JPG file you want to insert in your drawing and click Open. The Bitmap menu is displayed in the Menu Window, and you are prompted to “Enter first point of bitmap rectangle”.
4. Toggle OutLine on if you want to draw a visible polyline boundary around the inserted image. Toggle NoOutLin on if you do not want a visible boundary around the image. These are mutually exclusive toggles.
5. Toggle the FixRatio option on to preserve the original dimensions of the image; toggle it off to dynamically stretch the X and/or Y ratios while you are inserting the bitmap. After you insert the image, you cannot change the ratio if FixRatio was turned on; therefore, for future flexibility, you may want to turn FixRatio off.

While working in your drawing, you can toggle off Bmp On in the Display menu to decrease drawing refresh times by not displaying all the bitmaps in your drawing. Toggle it on again to display all images in your drawing as well as allow printing of them. Notice that SWOTHLUDFB in the Status Window contains an upper-case letter B if BmpOn is active; but it turns to a lower-case b if BmpOn is not active.

4. Enter the first corner of the image’s extents by clicking in the Drawing Area, using coordinate entry, or object snapping to an existing point in your drawing. Notice that if you move your cursor diagonally away from that point, a rectangular box (representing the extents of your image) stretches with it. You are prompted to “Enter second point of bitmap rectangle”.
6. Enter the corner diagonal from the first corner by clicking in the Drawing Area, using coordinate entry, or object snapping to an existing point in your drawing. The image is inserted in your drawing. If Bmp On is toggled off in the Utility/Display menu, only a rectangular box representing the extents of the image is displayed in your drawing. To display your image, toggle Bmp On in the Display menu.
7. Use the Calibrate options in the Enlarge menu to scale bitmaps to real-world size. For more information about this, see “Calibrating Enlargements” in this chapter.

You can control whether bitmaps or JPEGs print behind or in front of lines by using the Print first or Print last options in the Pen Table. For more information about using the Pen Table, see “Using Pen Tables” in the “Printing Your Drawing” chapter.

Calibrating Enlargements

You can enlarge or reduce imported bitmap and vector graphics to real-world scale. For example, if you import a site map, you can use the Calibrate Distance function to enlarge the plan to real-world dimensions.

- ➔ To calibrate imported bitmaps or vector graphics and enlarge to real-world size:
1. Select Enlarge from the Edit menu. You are prompted to "Select CENTER of enlargement."
 2. Click on the graphic you want to resize. Options appear in the Enlarge menu.
 3. Select Enlrgmnt to display the calibration options.
 4. Click on CalDist.
 5. Show the length of an entity or a known distance in the bitmap by clicking on your imported graphic at the first point to begin drawing a measuring line. Extend the line and click again when you reach the end of the entity or known distance in the graphic. You are prompted to "Enter new distance." The current distance follows this prompt.
 6. Replace that value with what it should be in the "real world." For example, the current value may be 0.8, but in reality, it should be 6 feet; therefore, you would type 6 in the input field. Press (Enter). DataCAD calculates the enlargement factor and displays it in the Message Window.
 7. Select the entity or entities that you want to enlarge by that factor.

CalDist automatically calculates an equal X/Y enlargement factor. Instead of CalDist, you can use the CalX-Y, CalX, or CalY options to enlarge entities independently or unequally in the X and Y directions.

Drawing Arrows

You can draw arrows of varying widths using the Arrow macro.

- ➔ To draw an arrow:
1. Click on Arrow in the Macros pull-down menu. The Arrow menu is displayed in the Menu Window. If ARROW is not listed in the Macros pull-down menu, see "Customizing the Macros Menu" for details on how to add it to the menu.
 2. Click on Width in the Arrow menu. A value menu is displayed.
 3. Use the value menu or type a width for the arrow, and press . You are prompted to "Enter first point of arrow centerline".
 4. Enter the two end points of the arrow. Click in the Drawing Area, use coordinate entry, or object snap to an entity to enter the base of the arrow. Notice that the length and direction of the arrow changes as you move your cursor. You are prompted to "Enter second point of arrow centerline".
 5. Click in the Drawing Area, use coordinate entry, or object snap to an entity to enter the tip of the arrowhead. The arrow is drawn.

Creating and Editing Templates

Templates are a very effective way to organize a library of predrawn items, which you can use in many different projects. You can, for instance, create a template with symbols that you use frequently or one that supports office standards. See “More About Creating Templates” for details on defining information fields.

Each template has information fields defined specifically for that template. Whenever you add a symbol to that template, you can enter information describing that symbol in each field. There are six standard information fields that are defined for every template, but you can define an unlimited number of additional fields.

➔ To create a template:

1. Click on Template in the Utility menu; or if you’re already in the Template menu, click on NewFile. A dialog box is displayed.
2. Type a name for your new template, and click on Open or press **[Enter]**. The dialog box closes, and you are prompted with “Create new file?” For more information about establishing a naming system for template files, see “More About Naming Templates” later in this chapter.

Toggle AutoPath on in the Template menu to automatically create a folder in your SYM\ folder with the same name as the template file you’re creating. This allows you to easily keep track of the symbols associated with your new template.

3. Click on Yes in the Menu Window to create the template. You are prompted with “Field name”.
4. Notice that all templates have six standard fields automatically defined for them. You can either define additional information fields, or simply create the template with only the standard six fields. To use only the standard fields, right-click and skip to step 7. To enter an additional field, type a name for the field (up to 20 characters) and press **[Enter]**. A list of field data types is displayed in the Menu Window, and you are prompted to “Select field type”.
5. Notice that there are three field types available: text, number, and dollar. To create a field that you can enter letters and numbers into, click on Text. To create a field that you can enter an integer into, click on Number. To create a field that you can enter a real number with two fixed decimal places, click on Dollar. Numbers entered in a text field cannot be used in any calculations.
6. Repeat steps 4 – 5 to continue entering information fields.
7. Right-click when you’ve completed entering fields. The template is created and displayed at the right side of the DataCAD window, and the Template menu is displayed in the Menu Window. For information on how to add symbols to your new template, see “Creating Symbols later in this chapter.”

More About Creating Templates

When you create a template, DataCAD automatically defines six information fields:

Field Name	Field Type
Item nam	Text
Manufact	Text
Model No	Text
Remark 1	Text
Remark 2	Text
Cost	Dollar

You can create an unlimited number of additional information fields when you create a template. This is the *only* time you can define information fields; once the template is created, you can't add another field or delete a field, nor can you change the field's name or type.

While you can't change the fields themselves, you can always edit any information you've entered into a field. See "Editing Information Fields" later in this chapter for more information. All fields you create can accept up to 80 characters of information.

More About Naming Templates

As you add more and more templates to your library, it becomes increasingly important to name those templates in an organized, easily recognizable way. You can simply give your templates longer, more descriptive names, such as "office chairs.tpl" or "living room.tpl". Template names with spaces or more than 8 characters may not be compatible with older versions of DataCAD. The total name including path cannot exceed 80 characters. Or you can devise a naming system like the following one:

- Sweet's catalog section number
- title of template
- chronological number of template

For example, if you're creating a template of door jamb details, and three similar templates already exist, you could name this fourth template 08JAMB04.TPL: "08" is the catalog number, "JAMB" is the title of the template, and "04" tells you it's the fourth template of door jambs.

Editing Templates

Templates you create in DataCAD have a standard number of divisions: three columns and 12 rows. Once you've created a template, you can change the number of divisions to suit your needs. For information on editing the symbols in a template, see "Editing Symbols" later in this chapter.

The number of divisions in a template doesn't have to equal the number of symbols associated with that template. You can have more divisions than symbols, in which case some of the divisions in your template will be empty. You can also have fewer divisions than symbols, so that not all symbols associated with that template will be displayed at once. Displaying fewer symbols will also increase refresh times.

➔ To change the number of divisions in a template:

1. Create your template. Then click on Divisions in the Template menu. You are prompted with “Number of X-divisions in template”.
2. Use the value menu or type the number of columns for your template, and press **[Enter]**. You are prompted with “Number of Y-divisions in template”.
3. Use the value menu or type the number of rows for your template, and press **[Enter]**. The template is updated with the new divisions, and you are returned to the Template menu.

Creating and Editing Symbols

Creating and editing symbols is a very effective way to build a library of predrawn items that you can use in many different projects. You can, for instance, draw a lighting fixture, create a symbol from the fixture geometry and add it to a template, and then very quickly place several instances of that fixture in your drawing.

To quickly determine the exact path and folder name of a symbol, click on SymName in the Template menu and then click on the symbol.

You can also enter descriptive information about each symbol, and then edit that information. You can even replace symbols in your drawing with others in your symbol library.

➔ To create a symbol and add it to a template:

1. Click on Template in the Utility menu. The Template menu is displayed in the Menu Window.

Shortcut: Press **[T]** to display the Template menu.

2. Click on SaveSymb. A dialog box is displayed, prompting you to “Enter name of symbol file to be saved. Symbol folders are stored in your DATACAD\SYM folder.
3. Make sure the folder that’s open in the dialog box is the one you want to save the symbol in. If it’s not, change to the proper folder.
4. Type a name for your symbol and click on Save. The SaveSymb menu is displayed in the Menu Window. Use SaveSymb to select either symbols already in your drawing or geometry you want to make into a symbol; both types of selection will be added to the currently displayed template.
5. Choose a selection method and then select the parts of your drawing you want to make into a symbol. You are prompted to “Select reference point for symbol”.
6. Select the reference, or insertion. You can point for the symbol, click in the Drawing Area, use coordinate entry, or object snap to a point in your drawing. The entities you selected are copied into the first open space in the template; if there are no open spaces in the currently displayed template, the symbol can’t be displayed in that template until you increase the number of template divisions. The first field, called Item Name, is displayed in the Message Area.

7. Enter a name for your symbol. You are prompted to enter information for additional fields.
8. Continue to enter symbol information for each field, or press **Enter** to skip a field and continue with the next. To skip all remaining fields, right-click. The dialog box is displayed again.
9. Continue with step 3 above to save additional symbols.

More About Naming Symbols

As you add more and more symbols to your templates, it becomes increasingly important to name those symbols in an organized, easily recognizable way. You can simply give your symbols longer, more descriptive names, such as “office chair.sym” or “kitchen sink.sym”. Or you can devise a naming system like the following one:

- Sweet’s catalog section number
- title of symbol
- chronological number of symbol

For example, if you’re creating a door jamb symbol, and two similar symbols already exist, you could name this third symbol 08JAMB03.SM3: “08” is the catalog number, “JAMB” is the title of the symbol, and “03” tells you it’s the third door jamb symbol. Symbol names with spaces or more than 8 characters may not be compatible with older versions of DataCAD. The total name including path cannot exceed 80 characters.

Editing Symbols

To edit symbols, you must first *explode*, or convert, them into individual lines and arcs. You can do this when you insert the symbol into your drawing or after the symbol has been inserted. Once the symbol has been edited, you can reform the symbol lines and arcs into a single symbol entity and copy it back into the template and then update the symbols already in your drawing.

➔ To explode a symbol when you place it in your drawing and edit it:

1. Click on Template in the Utility menu. A dialog box is displayed, allowing you to choose a template to open.
2. Click on the template you want to use, and click on Open. The template is displayed on the right side of the Drawing Area, and the Template menu is displayed in the Menu Window.
3. Toggle Explode on in the Template menu.
4. Click on the symbol in the template that you want to insert into your drawing.
5. Place the symbol in your drawing. You can click in the Drawing Area, use coordinate entry, or object snap to an entity in your drawing. The symbol is copied to your drawing and automatically exploded into separate entities.

6. Edit the symbol geometry as necessary. For more information on editing lines and arcs, see the “Editing Drawings” chapter.
 7. Copy your changes back to the template. Click on Redefine in the Template menu. You are prompted to “Point to symbol in template window to redefine”.
 8. Click on the symbol *in the template* that you just edited in the Drawing Area. A selection menu is displayed, and you are prompted to select the geometry you want to be your new symbol.
 9. Choose a selection method, and then select the geometry for the symbol. Your selection is highlighted with dashed lines. If necessary, continue selecting geometry until all entities that you want to be included in the new symbol are highlighted.
 10. Click on Continue to go on with the symbol definition process. You are prompted to “Select a reference point for symbol”.
 11. Click in the Drawing Area or use coordinate entry to select the point that will be used to insert the symbols in drawings. The edits to the symbol are copied to the template.
 12. Update all instances of that symbol in your drawing. Click on Re-Load in the Template menu. You are prompted to “Point to symbol in template window to re-load from disk”.
 13. Click on the symbol in the template that you want to reload, or click on All in the Re-Load menu to reload all symbols in your drawing.
- ➔ To explode a symbol already in your drawing and edit it:
1. Click on SymExp in the Macros pull-down menu. A selection menu is displayed in the Menu Window. If SYMEXP is not listed in your Macros menu, click Configure in the Macros menu and add SYMEXP to the Macros in Menu list.
 2. Choose a selection method, and then select the symbol(s) you want to edit. The number of symbols selected and the number of entities they were exploded into is displayed in the Message Area. For more information on using selection menus, see “Selection Menus” in “The Drawing Board” chapter.
 3. Edit the symbol geometry as necessary.
 4. Load your changes to this symbol back into its template if necessary. Then, open the appropriate template and continue with step 7 in the previous instructions.

Editing Information Fields

You can edit information associated with symbols by selecting a symbol in a template or one that’s already in your drawing. Use the Reports option in the Template menu to extract symbol information for all symbols used in your drawing, and create specification reports or cost estimates. See the “Reports and Estimates” chapter for more information.

➔ To edit symbol information:

1. Open the template that the symbol (whose information you want to edit) is associated with.
2. Click on EditFlds in the Template menu.
3. Click on the symbol, either in the template or in your drawing, whose information you want to edit. The fields for that symbol are displayed in the Menu Window.
4. Click on the field you want to edit.
5. Type the new information for that field and press **[Enter]**.
6. Repeat steps 4 –5 to continue changing field information as necessary.
7. Edit information for another symbol by right-clicking and continuing with step 3 above. Right-click twice to exit back to the Template menu.

➔ To edit symbol information:

1. Click on FormAtr in the Macros pull-down menu. The FormAtr menu is displayed in the Menu Window. If FormAtr is not listed in your Macros menu, click Configure in the Macros menu and add it to the Macros in Menu list.
2. Click on Symbol in the FormAtr menu. You are prompted to “Select symbol to edit attribute fields”.
3. Click on the symbol. The symbol’s information is displayed in the Drawing Area.
4. Press **[Tab]** to move the edit box to the text you want to change.
5. Type the new text and press **[Enter]**. The information is changed.
6. Repeat steps 4 – 5 to continue editing.
7. Press **[Esc]** to exit editing and return to your drawing. You are prompted with “Do you wish to update symbol file [symbol path and filename]?”
8. Click on Yes to update the symbol information; click on No to cancel the changes.

Replacing Symbols in Your Drawing

You can replace one symbol in your drawing with another and choose whether to replace one instance of that symbol or all instances throughout your drawing. This is useful, for instance, if you’ve entered one type of bathroom sink throughout your drawing and you need to replace it with another type.

➔ To replace one symbol with another symbol:

1. Click on Template in the Utility menu. The Template menu is displayed in the Menu Window.
2. Click on Replace in the Template menu. The Replace menu is displayed in the Menu Window, and you are prompted to “Point to the symbol to replace”.

3. Click on the symbol in your drawing that you want to replace. You are prompted to “Point to symbol to replace with”.
 4. Choose the new symbol that will replace the original by clicking on a symbol in a template or on one that’s already in your drawing. A selection menu is displayed in the Menu Window.
 5. Replace only particular instances of the symbol you selected in step 3 or replace all instances of that symbol. To choose particular instances of the symbol, choose a selection method from the menu, and continue with the next step. To replace all instances of the symbol, click on All; all instances of the symbol are immediately replaced, and you can skip to step 7.
 6. Select each instance of the symbol that you want to replace. Each symbol is replaced as you select it.
 7. Finish replacing symbols. Then, right-click to return to the Template menu.
- ➔ To replace several different symbols with another symbol:
1. Click on Template in the Utility menu. The Template menu is displayed in the Menu Window.
 2. Click on Replace in the Template menu. The Replace menu is displayed in the Menu Window, and you are prompted to “Point to the symbol to replace”.
 3. Click on AnySymb in the Replace menu. You are prompted to “Point to symbol to replace with”.
 4. Choose the new symbol that will replace the original by clicking on a symbol in a template or on one that’s already in your drawing. A selection menu is displayed in the Menu Window.
 5. Select each symbol you want to replace or replace *every* symbol in your drawing. To choose each symbol, choose a selection method from the menu, and continue with the next step. To replace all symbols, click on All; all symbols in your drawing are immediately replaced.
 6. Select the symbols to be replaced. Each symbol is replaced as you select it.
 7. Finish replacing symbols. Then, right-click to exit the Replace menu and return to the Template menu.

Deleting Symbols

You can erase symbols from your drawing or delete them from a template.

- ➔ To erase a symbol:
1. Click on Erase in the Edit menu. The Erase menu is displayed in the Menu Window.
 2. Toggle LyrSrch on if the symbols you want to erase are on a layer other than the active layer.
 3. Click on a selection method in the menu. You are prompted to select the entities you want to erase.

4. Select the symbols; they are erased from the drawing. You can continue selecting symbols to erase them. Avoid selecting a symbol where it crosses another entity, because it may be difficult for DataCAD to know which entity you want to select. Zoom in on your drawing or change your view to select symbols more precisely.
5. Finish erasing symbols. Then, right-click to exit the Erase menu.

➔ To delete a symbol from a template:

1. Click on Template in the Utility menu. A dialog box is displayed, allowing you to choose a template to open.
2. Click on the template with the symbol you want to delete, and click on Open. The template is displayed on the right side of the Drawing Area, and the Template menu is displayed in the Menu Window.
3. Click on DelSymb in the Template menu. You are prompted to “Select symbol to delete from this template file”.
4. Click on the symbol you want to delete from the template. You are asked to confirm your selection.
5. Click on Yes to delete the symbol; click on No to cancel the function.
6. Continue selecting symbols to delete them from the template, or right-click to exit the DelSymb menu.

More About Erasing Symbols

When you erase symbols from your drawing, it's important to *purge* the symbol from your drawing as well. When you add a symbol to your drawing, DataCAD associates that symbol with your drawing. This association is not deleted when you erase the symbol and continues to take up drawing file space. To delete the association as well, you must use the PurgeSym option in the Template menu.

➔ To purge symbol associations from your drawing:

1. Click on PurgeSym in the Template menu. You are prompted with “Are you sure you want to purge unused symbols?”
2. Click on Yes to purge the associations with symbols that have been deleted; click on No to cancel the function.

Borders and Title Blocks

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You can draw drawing sheet borders and title blocks manually, using DataCAD's Polygons menu, or you can use the Drawing Sheet tool in the DCAD_AEC macro to automate the process.

In this chapter:

- ⊕ Drawing sheet borders
- ⊕ Adding title blocks

Drawing Borders and Title Blocks

Borders and title blocks can be added to your drawing in two ways: either follow the procedure outlined in the tutorial in this manual, or use the Drawing Sheet tool in the DCAD_AEC macro.

➔ To use the Drawing Sheet tool:

1. Click on DCAD_AEC in the Macros pull-down menu. The DCAD_AEC menu is displayed in the Menu Window. If DCAD_AEC is not listed in the Macros pull-down menu, see “Customizing the Macros Menu” for details on how to add it to the menu.
2. Click on DwgSheet in the DCAD_AEC menu. The DwgSheet menu is displayed in the Menu Window.
3. Notice the Drawing Sheet tool. It calculates the size of the border sheet by using the plotting scale and paper size you will use when you print your drawing. To enter the plotting scale, click on Scale. A value menu is displayed, and you are prompted to “Select the plotter output scale you will be using.”
4. Use the value menu to enter the plotting scale you will use.
5. Enter the sheet size you will use when plotting by clicking on PaperSiz. A menu listing sheet sizes is displayed, and you are prompted to “Select the sheet size you will be plotting to.”
6. Click on a sheet size in the list; or to define a custom sheet size, click on Custom and then use the value menus to enter the plotting width and height.
7. Right-click to return to the DwgSheet menu.
8. Define the title block. To specify the location of the title block on the sheet, click on TitleLoc.
9. Choose one of the following locations for your title block:
 - Click on Top to place the title block on the top of the sheet.
 - Click on Bottom to place it on the bottom of the sheet.
 - Click on Left to place it on the left side of the sheet.
 - Click on Right to place it on the right side of the sheet.
 - Click on None to draw the sheet border without a title block.
10. Set the number of boxes you want the title block to contain by clicking on TitleDiv. A value menu is displayed, and you are prompted to “Enter number of title block divisions across.” For more information on using value menus, see “Value Menus” in “The Drawing Board” chapter.
11. Use the value menu or type a number, and press . You are prompted to “Enter number of title block divisions down”.
12. Use the value menu or type a number, and press . You are returned to the DwgSheet menu.

13. Set the height of the title block by click on TitleSiz. A value menu is displayed, and you are prompted to “Enter height of drawing sheet title block”.
14. Use the value menu or type a size, and press . You are returned to the DwgSheet menu.
15. Set the number of detail blocks by clicking on DtailDiv. A value menu is displayed, and you are prompted to “Enter number of detail block divisions across”.
16. Use the value menu or type a number, and press . You are prompted to “Enter number of detail block divisions down”.
17. Use the value menu or type a number, and press . You are returned to the DwgSheet menu.
18. Set the height of detail title blocks by clicking on DtailTtl. A value menu is displayed, and you are prompted to “Enter the height of detail title block”.
19. Use the value menu or type a height, and press . You are returned to the DwgSheet menu.
20. Add a snapping point at the center of your sheet border by toggling AddSnap on. Toggle AddSnap off to draw sheet borders without the center snapping point.
21. Draw the border and title block. You must select the center point of the drawing sheet border. Click in the Drawing Area, use coordinate entry, or object snap to an entity to select the center point. The sheet border is drawn.
22. Add common architectural symbols to your title block. Click on DwgSymb, open the DRAWING.TPL file in the TPL\DWG folder in your DataCAD folder, and insert symbols as necessary. For more information on how to insert symbols, see “Inserting Symbols in Your Drawing” in the “Templates & Symbols” chapter.

8.

Elevations and Sections

With the EZ Tools macro, you can parametrically draw residential or commercial elevations and sections quickly and easily. This set of tools eliminates the need for the tedious linework and calculations otherwise required to create basic elevations. All angles, lines, offsets, and line lengths are done automatically, leaving you free to concentrate on the drawing aesthetics.

The Elevations tool assists in drawing exterior and interior residential elevations, including door, window, and cabinet elevations, while the Sections tool draws cabinet outline sections.

In this chapter:

- ⊕ Drawing exterior elevations
- ⊕ Drawing door elevations
- ⊕ Drawing window elevations
- ⊕ Drawing cabinet elevations
- ⊕ Drawing cabinet sections

Drawing Exterior Elevations

Exterior elevations are defined by selecting two points along the line of the subfloor. The exterior elevation will be drawn relative to those two points, based on the Exterior menu settings.

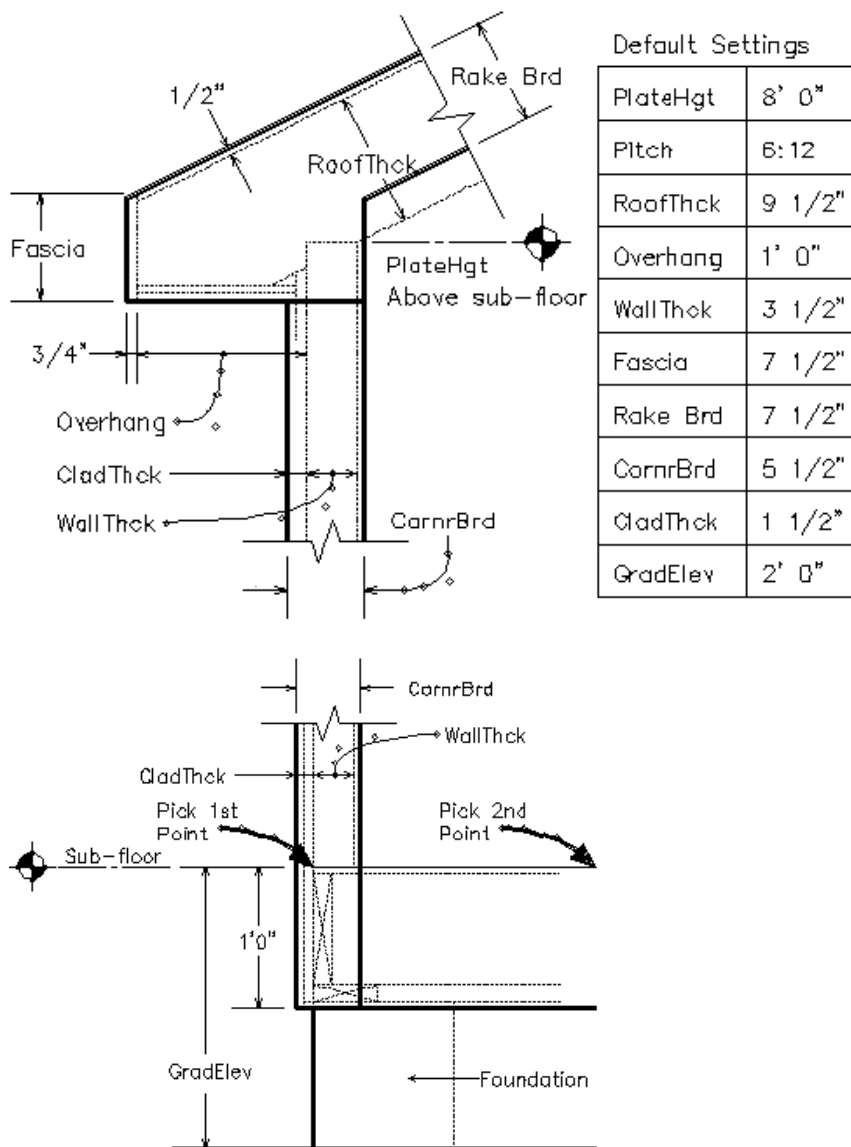


Figure 18.1: A sample exterior elevation with default exterior elevation settings noted. EZ Tools draws only those lines represented above by extra line thickness.

➔ To draw an exterior elevation:

1. Click on EZTOOLS in the Macros pull-down menu. The EZ Tools menu is displayed in the Menu Window. If EZTOOLS is not listed in the Macros pull-down menu, click Configure in the Macros menu and add EZTOOLS to the Macros in Menu list.
2. Click on Elev in the EZ Tools menu. The Elev menu is displayed in the Menu Window.

3. Click on Exterior in the Elev menu. The Exterior menu is displayed in the Menu Window.
4. Choose between a full or partial elevation. Full exterior elevations are drawn by assuming that the points you will select are on two opposite walls of the structure. Partial elevations are drawn by assuming that the first point you select will be on a wall and the second point will be at the mid-point of the structure. To draw a full elevation, toggle Full on; to draw a partial elevation, toggle Partial on. Of the two options, Full and Partial, only one can be toggled on at any given time.
5. Set the roof style for your elevation by clicking on Style. The Style menu is displayed in the Menu Window, and you are prompted to "Select roof style".
6. Toggle Gable on to draw a gable roof in your elevation; toggle Hip on to draw a hip roof in your elevation; toggle Boston on to draw a Boston-style roof in your elevation; or toggle Shed on to draw a shed roof in your elevation. Of the four options, Gable, Hip, Boston, and Shed, only one can be toggled on at any given time. You are returned to the Elev menu.
7. Set the height of the top plate on which the rafters rest by clicking on PlateHgt. A value menu is displayed, and you are prompted to "Enter plate height". The default PlateHgt setting is 8'-0". For more information on using value menus, see "Value Menus" in "The Drawing Board" chapter.
8. Use the value menu or type a height, and press .
9. Set the rise of the roof for every one foot of run by clicking on Pitch. A value menu is displayed, and you are prompted to "Enter rise per foot for roof".
10. Use the value menu or type a pitch value, and press . The default Pitch setting is 6".
11. Notice that the roof thickness is drawn relative to the plate height, and the roof line is always drawn an additional 1/2" from the top surface of the rafter. To set the width of the rafters, click on RoofThck. A value menu is displayed, and you are prompted to "Enter the roof thickness".
12. Use the value menu or type a thickness, and press . The default RoofThck setting is 9 1/2".
13. Notice that the roof overhang is measured from the wall exterior framing surface to the back surface of the fascia. To set the length of the roof overhang, click on Overhang. A value menu is displayed, and you are prompted to "Enter the roof overhang".
14. Use the value menu or type a length, and press . The default Overhang setting is 1'-0".
15. Notice that the wall thickness is drawn relative to the first point you will select to draw the elevation. To set the wall thickness, click on WallThck. A value menu is displayed, and you are prompted to "Enter the wall thickness".

16. Use the value menu or type a thickness, and press . The default WallThck setting is 3 1/2".
17. Notice that the fascia width is measured across the back surface of the fascia; the fascia thickness is set to 3/4" and can't be changed. To set the fascia width, click on Fascia. A value menu is displayed, and you are prompted to "Enter the width of the fascia".
18. Use the value menu or type a width, and press . The default Fascia setting is 7 1/2".
19. Notice that the rake board width is measured from the roof line. To set the rake board width, click on Rake Brd. A value menu is displayed, and you are prompted to "Enter the width of the rake board".
20. Use the value menu or type a height, and press . The default Rake Brd setting is 7 1/2".
19. Notice that the corner board width is measured relative to the exterior wall line. To set the corner board width, click on CornrBrd. A value menu is displayed, and you are prompted to "Enter the width of the corner board".
20. Use the value menu or type a width, and press . The default CornrBrd setting is 5 1/2".
19. Notice that the thickness of the cladding is measured from the wall framing out to the exterior wall line. To set the cladding thickness, click on CladThck. A value menu is displayed, and you are prompted to "Enter the cladding thickness".
20. Use the value menu or type a thickness, and press . The default CladThck setting is 1 1/2".
19. Notice that the grade elevation is measured relative to the subfloor. To set the grade elevation, click on GradElev. A value menu is displayed, and you are prompted to "Enter sub-floor distance above the grade".
20. Use the value menu or type a distance, and press . The default GradElev setting is 2'-0". The distance between the top surface of the subfloor and the bottom edge of the cladding is 1'-0" and can't be changed. For best results, set the grade elevation to 1'-0" or greater.
21. Set the color of the outside lines of the elevation by clicking on Out Colr. A color menu is displayed, and you are prompted to "Select outer color".
22. Use the color menu to set the outside line color. The default Out Colr setting is White.
23. Set the color of the inside lines of the elevation by clicking on In Colr. A color menu is displayed, and you are prompted to "Select inner color". The default In Colr setting is Red.
24. Use the color menu to set the inside line color.

25. Select the two points along the top surface of the subfloor. The first point should be located on the outside surface of the wall framing. To select the first point, click in the Drawing Area, use coordinate entry, or object snap to a point in your drawing. You are prompted to select the other side of the elevation.
26. Select the second point, click in the Drawing Area by using coordinate entry, or object snapping to a point in your drawing. The elevation is drawn, aligned with the two points you just selected. You can click on Defaults at any time to restore all default settings in the Exterior menu.

Drawing Door Elevations

Door elevations are defined by selecting two points along the subfloor line. The door elevation will be drawn relative to those two points, based on the Door menu settings.

➔ To draw a door elevation:

1. Click on EZTOOLS in the Macros pull-down menu. The EZ Tools menu is displayed in the Menu Window. If EZTOOLS is not listed in the Macros pull-down menu, click Configure in the Macros menu and add EZTOOLS to the Macros in Menu list.
2. Click on Elev in the EZ Tools menu. The Elev menu is displayed in the Menu Window.
3. Click on Door in the Elev menu. The Door menu is displayed in the Menu Window.
4. To draw single doors, toggle Single on; to draw double doors, toggle Double on. Of the two options, Single and Double, only one can be toggled on at any given time. The Single and Double toggles do not apply to Garage A, Garage B, or Sliding door styles.
5. Set the door style for your elevation by clicking on Style. The Style menu is displayed in the Menu Window, and you are prompted to "Pick style of door to draw".

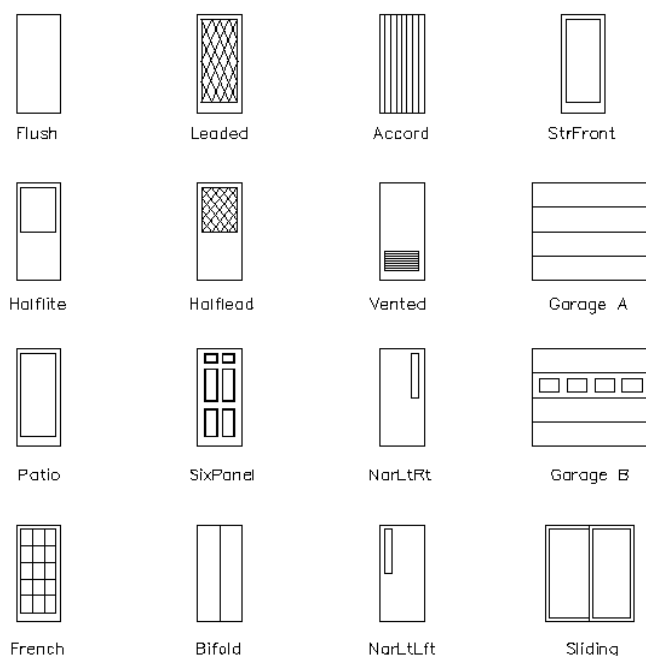


Figure 18.2: Door styles available for door elevations

- Click on a door style in the list. You are returned to the Door menu. See “Door Styles for Elevations” below for details.

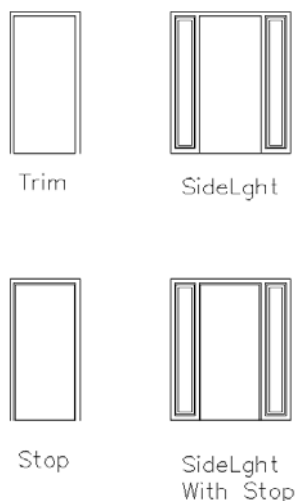


Figure 18.3: Door elevation settings

- Set the head height of the door by clicking on Head Hgt. A value menu is displayed, and you are prompted to “Enter door head height”.
- Use the value menu or type a height, and press **Enter**.
- Draw door trim in your elevation by toggling Trim on; to draw the door sidelight, toggle SideLght on. The Stop option is displayed. Of the two options, Trim and SideLght, only one can be toggled on at any given time. The default setting for Trim is 2 1/2”, the style for the SideLght option is automatically set, and the Stop option default is 3/4”.
- Toggle Stop on to draw the door stop if you toggled either Trim or SideLght on in the previous step.

11. Set the trim color by clicking on TrimColr. A color menu is displayed, and you are prompted to “Select the trim color”.
12. Use the color menu to set the trim color. The default TrimColr setting is White.
13. Set the door color by clicking on DoorColr. A color menu is displayed, and you are prompted to “Select the door color”.
14. Use the color menu to set the door color. The default DoorColr setting is Red.
15. Enter the first point for the door elevation. You can click in the Drawing Area, use coordinate entry, or object snap to a point in your drawing. You are prompted to enter a point for the other side of the door.
16. Enter the second point for the door. You can click in the Drawing Area, use coordinate entry, or object snap to a point in your drawing. The door elevation is drawn. You can click on Defaults at any time to restore all default settings in the Door menu.

Door Styles for Elevations

Flush	Flush door style
HalfLght	Half light door style
Patio	Patio door style
French	French door style
Leaded	Leaded glass door style
HalfLead	Half leaded glass door style
SixPanel	Six panel door style
Bifold	Bifold door style
Accord	Accordian door style
Vented	Vented door style
NarLtRt	Door drawn with a narrow light on the right side of the door
NarLtLft	Door drawn with a narrow light on the left side of the door
StorFrnt	Storefront door style
Garage A	Garage door style, with four panels; Single and Double toggles do not apply to this style
Garage B	Garage door style, with four panels and four windows in the second panel from the top; Single and Double toggles do not apply to this style
Sliding	Sliding door style; Single and Double toggles do not apply to this style

Drawing Window Elevations

Window elevations are defined by selecting two points along the subfloor line. The window elevation will be drawn relative to those two points, based on the Window menu settings.

➔ To draw a window elevation:

1. Click on EZTOOLS in the Macros pull-down menu. The EZ Tools menu is displayed in the Menu Window. If EZTOOLS is not listed in the Macros pull-down menu, click Configure in the Macros menu and add EZTOOLS to the Macros in Menu list.
2. Click on Elev in the EZ Tools menu. The Elev menu is displayed in the Menu Window.
3. Click on Window in the Elev menu. The Window menu is displayed in the Menu Window.

4. Draw single windows by toggling Single on; draw double windows by toggling Double on. Of the two options, Single and Double, only one can be toggled on at any given time.
5. Set the window style for your elevation by clicking on Style. The Style menu is displayed in the Menu Window, and you are prompted to “Pick style of window to draw”. The Single and Double toggles do not apply to the casement and sliding window styles.

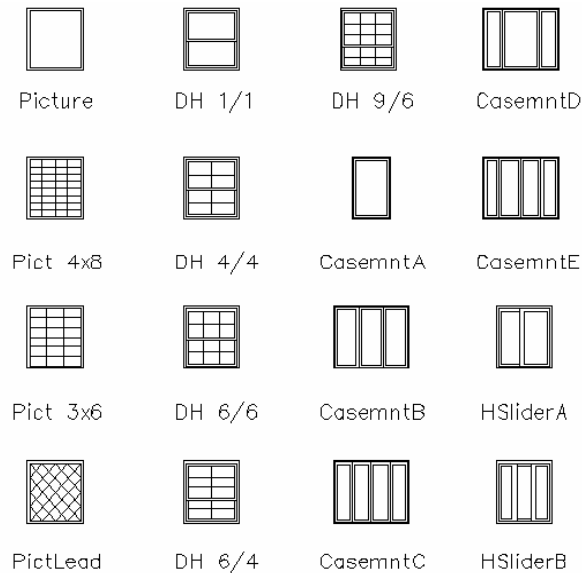


Figure 18.4: Window styles available for window elevations

6. Click on a window style in the list. You are returned to the Window menu. See “Window Styles for Elevations” below for details.
7. Set the head height of the window by clicking on Head Hgt. A value menu is displayed, and you are prompted to “Enter window head height”.
8. Use the value menu or type a height, and press **Enter**.
9. Set the sill height of the window by clicking on Sill Hgt. A value menu is displayed, and you are prompted to “Enter window sill height”.
10. Use the value menu or type a height, and press **Enter**.
11. Draw brick sills, capstones, or shutters with your window elevations. To draw brick sills, toggle BrkSillA on to draw brick sills along the bottom of the window, or toggle BrkSillB on to draw brick sills along the bottom and top of the window. To draw a bottom and top sill with a capstone, toggle Capstone on. To draw shutters, toggle ShutterA on to draw a single shutter on either side of the window, or toggle ShutterB on to draw a double shutter on either side of the window. To draw window elevations without brick sills, capstones, or shutters, click on whatever option is toggled on to toggle it off again. Of the five options BrkSillA, BrkSillB, Capstone, ShutterA, and ShutterB, only one can be toggled on at any given time.

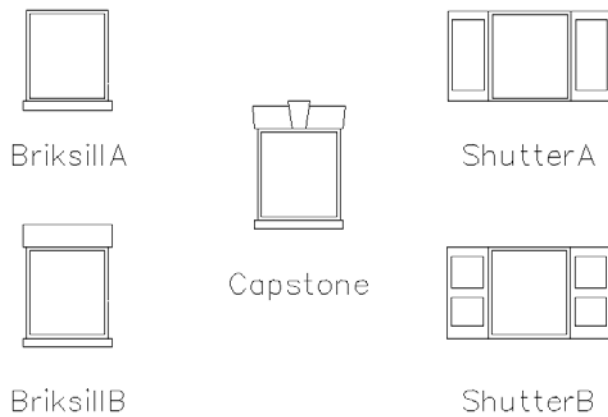


Figure 18.5: Sill and shutter styles for window elevations

12. Set the frame color by clicking on FramColr. A color menu is displayed, and you are prompted to “Select the frame color”.
13. Use the color menu to set the frame color. The default FramColr setting is White.
14. Set the window sash color by clicking on SashColr. A color menu is displayed, and you are prompted to “Select the sash color”.
15. Use the color menu to set the sash color. The default SashColr setting is Red.
16. Enter the first point for the window by clicking in the Drawing Area, using coordinate entry, or object snapping to a point in your drawing. You are prompted to enter a point for the other side of the window.
17. Enter the second point for the window by clicking in the Drawing Area, using coordinate entry, or object snapping to a point in your drawing. The window elevation is drawn. You can click on Defaults at any time to restore all default settings in the Window menu.

Window Styles for Elevations

Picture	Picture window style
Pict 4x8	Picture window style with four columns and eight rows of panes
Pct3x6	Picture window style with three columns and six rows of panes
PictLead	Picture window style with leaded glass
DH 1/1	Double-hung window style with no panes
DH 4/4	Double-hung window style with four panes in each window
DH 6/6	Double-hung window style with six panes in each window
DH 6/4	Double-hung window style with six panes in the top window and four panes in the bottom window
DH 9/6	Double-hung window style with nine panes in the top window and six panes in the bottom window
CasemntA	Casement window style with no panes; Single and Double toggles do not apply to this style
CasemntB	Casement window style with three panes of equal size; Single and Double toggles do not apply to this style
CasemntC	Casement window style with four panes of equal size; Single and Double toggles do not apply to this style
CasemntD	Casement window style with three panes, with the center pane larger and the two side panes of equal size; Single and Double toggles do not apply to this style

CasemntE	Casement window style with four panes, two center panes larger and of equal size and the two outside panes of equal size; Single and Double toggles do not apply to this style
HSliderA	Horizontal sliding window style with two panes of equal size; Single and Double toggles do not apply to this style
HSliderB	Horizontal sliding window style with three panes of equal size; Single and Double toggles do not apply to this style

Drawing Cabinet Elevations

Cabinet elevations are defined by selecting two points along the subfloor line. The cabinet elevation will be drawn relative to those two points, based on the Cabinet menu settings.

Click on Section in the Elev menu to display the Section menu and draw cabinet sections.

➔ To draw a cabinet elevation:

1. Click on EZTOOLS in the Macros pull-down menu. The EZ Tools menu is displayed in the Menu Window. If EZTOOLS is not listed in the Macros pull-down menu, click Configure in the Macros menu and add EZTOOLS to the Macros in Menu list.
2. Click on Elev in the EZ Tools menu. The Elev menu is displayed in the Menu Window.
3. Click on Cabinet in the Elev menu. The Cabinet menu is displayed in the Menu Window.
4. Draw cabinets with one door by toggling Single on; draw cabinets with double doors by toggling Double on. Of the two options, Single and Double, only one can be toggled on at any given time. The Single and Double toggles do not apply to the drawer, desk, pantry, or oven cabinet styles.
5. Set the cabinet style for your elevation by clicking on Style. The Style menu is displayed in the Menu Window, and you are prompted to "Pick style of cabinet to draw".

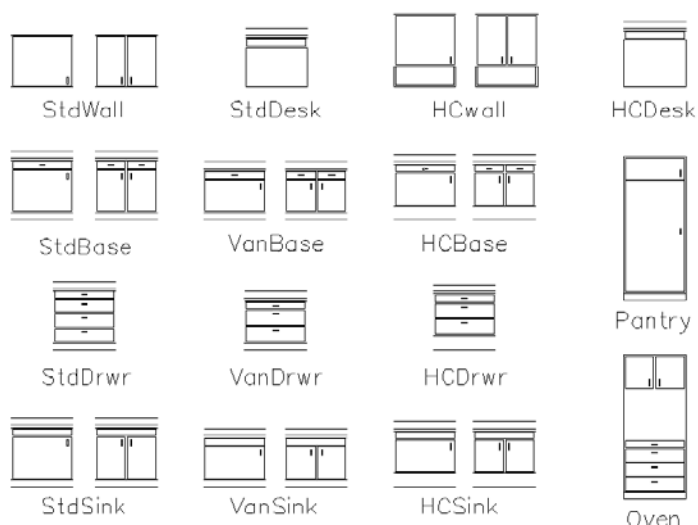


Figure 18.6: Cabinet elevation styles

6. Click on a cabinet style in the list. You are returned to the Cabinet menu. See “Cabinet Styles for Elevations” below for details.
7. Set the wall height of the cabinet by clicking on WCab Hgt. A value menu is displayed, and you are prompted to “Enter wall cabinet height”.
8. Use the value menu or type a height, and press .
9. Set the splash height of the cabinet by clicking on SplshHgt. A value menu is displayed, and you are prompted to “Enter wall splash height”.
10. Use the value menu or type a height, and press .
11. Draw cabinet pulls on the right by toggling Pull Rt on. To draw cabinet pulls on the left, toggle Pull Lft on. These options only affect cabinet elevations when Single is toggled on in the Cabinet menu; they have no effect when Double is toggled on. Of the two options Pull Rt and Pull Lft, only one can be toggled on at any given time.
12. Set the counter color by clicking on CntrColr. A color menu is displayed, and you are prompted to “Select the counter color”.
13. Use the color menu to set the counter color.
14. Set the cabinet color by clicking on Cab Colr. A color menu is displayed, and you are prompted to “Select the cabinet color”.
15. Use the color menu to set the cabinet color. The default Cab Colr setting is Red.
16. Enter the first point for the cabinet. You can click in the Drawing Area, use coordinate entry, or object snap to a point in your drawing. You are prompted to enter a point for the other side of the cabinet.
17. Enter the second point for the cabinet. You can click in the Drawing Area, use coordinate entry, or object snap to a point in your drawing. The cabinet elevation is drawn. You can click on Defaults at any time to restore all default settings in the Cabinet menu.

Cabinet Styles for Elevations

Std Wall	Standard wall cabinet style
Std Base	Standard base cabinet style
Std Drwr	Standard drawer cabinet style; Single and Double toggles do not apply to this style
Std Sink	Standard sink cabinet style
Std Desk	Standard desk cabinet style; Single and Double toggles do not apply to this style
Van Base	Vanity base cabinet style
Van Drwr	Vanity drawer cabinet style; Single and Double toggles do not apply to this style
Van Sink	Vanity sink cabinet style
HC Wall	Handicapped wall cabinet style
HC Drwr	Handicapped drawer cabinet style; Single and Double toggles do not apply to this style
HC Sink	Handicapped sink cabinet style
HC Desk	Handicapped desk cabinet style; Single and Double toggles do not apply to this style
Pantry	Pantry cabinet style; Single and Double toggles do not apply to this style
Oven	Oven cabinet style; Single and Double toggles do not apply to this style

Drawing Sections

The Section tool gives you a quick way to create an outline section cut through cabinets, baths and showers. Click on Elev in the Section menu to display the Elev menu and draw elevations.

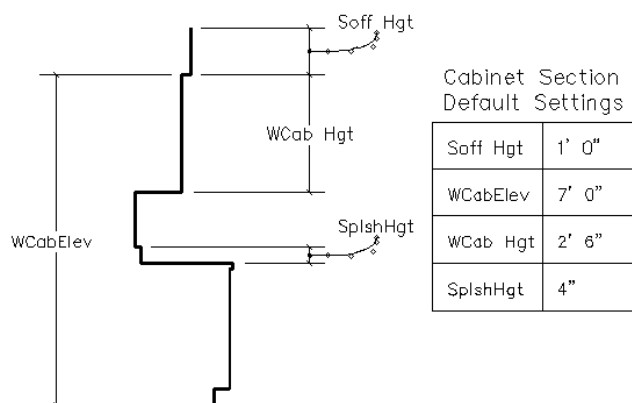


Figure 14.6: A sample cabinet section with default settings noted

➔ To draw a cabinet section:

1. Click on EZTOOLS in the Macros pull-down menu. The EZ Tools menu is displayed in the Menu Window. If EZTOOLS is not listed in the Macros pull-down menu, click Configure in the Macros menu and add EZTOOLS to the Macros in Menu list.
2. Click on Section in the EZ Tools menu. The Section menu is displayed in the Menu Window.
3. Click on Cabinet in the Section menu. The Cabinet menu is displayed in the Menu Window.
4. Set the cabinet style for your section by clicking on Style. The Style menu is displayed in the Menu Window, and you are prompted to "Select the type of section to draw". See "Cabinet Styles for Sections" below for details.
5. Click on a style in the list. You are returned to the Cabinet menu.
6. Set the soffit height of the cabinet by clicking on Soff Hgt. A value menu is displayed, and you are prompted to "Enter soffit height".
7. Use the value menu or type a height, and press **Enter**.
8. Set the wall elevation of the cabinet by clicking on WCabElev. A value menu is displayed, and you are prompted to "Enter wall cabinet elevation".
9. Use the value menu or type a value, and press **Enter**.
10. Set the wall height of the cabinet by clicking on WCab Hgt. A value menu is displayed, and you are prompted to "Enter wall cabinet height".
11. Use the value menu or type a height, and press **Enter**.
12. Set the back-splash height of the cabinet by clicking on SplshHgt. A value menu is displayed, and you are prompted to "Enter back-splash height".

13. Use the value menu or type a height, and press **Enter**.
14. Set the section color by clicking on SectColr. A color menu is displayed, and you are prompted to "Select the section color".
15. Use the color menu to set the section color. The default SectColr setting is White.
16. Set the element color (for bathtubs and showers) by clicking on ElemColr. A color menu is displayed, and you are prompted to "Select the element color".
17. Use the color menu to set the element color. The default ElemColr setting is Red.
18. Enter the first point for the cabinet section. You can click in the Drawing Area, use coordinate entry, or object snap to a point in your drawing. You are prompted to enter a point for the other side of the section.
19. Enter the second point for the cabinet section. You can click in the Drawing Area, use coordinate entry, or object snap to a point in your drawing. The cabinet section is drawn. You can click on Defaults at any time to restore all default settings in the Cabinet menu.

Cabinet Styles for Sections

Standard	Standard cabinet style
HandiCap	Handicapped cabinet style
Vanity	Vanity cabinet style
PassThru	Pass-through cabinet style
BarTop	Bar top style
Bathtub	Bathtub style
Shower	Shower style

Printing Your Drawing

Printing in DataCAD offers you many different options, so you can always get the result you need. You can set the plotting scale for your drawing, customize pen widths for plotting, and assign colors and weights to different plotter pens. You can plot your drawing, or a part of it, using a quick layout; or you can place multiple details on a single sheet using multi-scale layout. DataCAD's print preview allows you to visually check your plot before you send it to your plotter, or you can do a quick check plot to make sure your final plot will be correct. You can even choose to print a single sheet that you've set up, or you can define and then plot several sheets at once.

In this chapter:

- ⊕ Assigning colors and plotter pen widths
- ⊕ Laying out your drawing on sheets
- ⊕ Previewing your print
- ⊕ Checking your print
- ⊕ Printing details and drawings
- ⊕ Batch plotting

Printing Basics

The first time you print a drawing, you need to set several printing options, including the printer or plotter you'll use, the paper size and orientation, and the layout orientation of the drawing. These settings become part of your drawing when you save the file and will be remembered the next time you print the drawing. You can always change these settings by simply using the Print Setup option in the File pull-down menu.

➔ To print a drawing:

1. Decide if you want to print the text, dimensions, hatching, and line weights in your drawing. If you do, make sure ShowTxt, ShowDim, ShowHtch, and ShowWgt, respectively, are toggled on in the Utility/Display menu. Keep in mind that your drawing will print as it's displayed.
2. Click on Print in the File pull-down menu, or click on Plotter in the Utility menu in the Menu Window. Depending on whether you've printed this drawing before or not, one of two things may happen now:
 - If you've printed this drawing before, the Plotter menu is displayed in the Menu Window. Click on Setup in the Plotter menu to display the Printer/Plotter Settings dialog box.
 - If this is the first time you've printed this drawing, you are prompted with "A printer has not been selected for this drawing. Please click OK to open the Print Setup dialog box and assign a printer to this drawing file." Click on OK to close the message box and display the Printer/Plotter Settings dialog box.
3. In the Printer section at the top of the dialog box, use the Name drop-down box to select the printer you'd like to use.
4. Go to the Paper section of the dialog box and use the Size drop-down box to select a paper size. The paper sizes available in the Size drop-down box are read from your printer driver. If a size you want is not listed, then your printer doesn't support it or the driver may not be configured to display it. You can change the driver properties by closing DataCAD and opening the Printers dialog box from the Control Panel in Windows. Choose the next closest appropriate size, select User-Defined if listed, or use a different printer.

Notice that the Effective Plotting Area dimensions, displayed directly below the Size drop-down box, change depending on the Size of paper you choose. The *effective plotting area* is the printable area for the paper size you've selected, as defined by the printer manufacturer's driver. Any part of the drawing extending outside this area will not be printed (see Figure 19.1).

Some printer drivers, such as LaserJets and older plotters, don't support user-defined paper sizes, even though that option remains listed in the Size drop-down box. One way to check is to select User-Defined for the paper size, and enter a width and length for the paper size. If the Effective Plotting Area dimensions don't change based on the size dimensions, then the selected printer's driver doesn't support user-defined paper sizes. Select another paper size from the drop-down box.

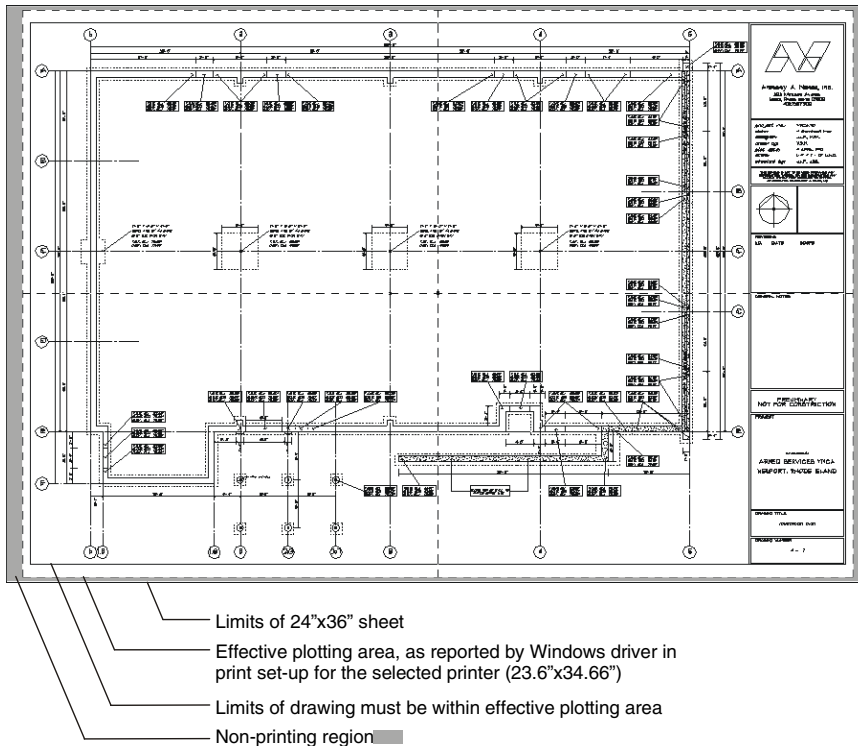


Figure 19.1: Example of printed sheet showing the edges of the sheet, the non-printing area, the effective plotting area, and the extents of the drawing

- Set the orientation for your printed sheet by clicking on Normal or Rotated.
- Go to the Copies section at the bottom of the dialog box and enter the number of copies of each sheet you'd like to print.
- Go to the DataCAD Layout Orientation section at the bottom of the dialog box. Click on Normal to print the sheet as it appears on screen or click on Rotated to rotate the sheet 90° for plotting. The Rotated option in the DataCAD Layout Orientation section rotates the geometry for the print only; it does not rotate the drawing itself.
- Click OK to close the Printer/Plotter Settings dialog box. The Plotter menu is displayed in the Menu Window.
- Change the plotting scale by clicking on Scale. A value menu is displayed, and you are prompted to "Enter new scale".
- Use the value menu to select a new scale, and press **[Enter]**.

11. Determine if you have lines in your drawing with a line weight above “1.” If so, you can change the amount of space between pen passes when printing these thicker lines (see Figure 19.2). Click on PenWidth in the Plotter menu, and use the value menu to enter a width.

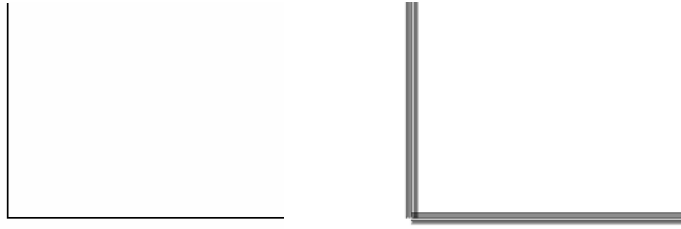


Figure 19.2: Lines drawn with a line weight of 4 and printed with a pen width setting of 12 (left) and 95 (right)

12. Use Pen Table to assign each pen in your plotter to a line color in your drawing, as well as to set the width and color density for each pen. To change Pen Table settings, click on PenTable in the Plotter menu; the Pen Table dialog box is displayed. Change the settings as necessary and click on OK. See “Using Pen Tables” later in this chapter for more information.
13. Plot your drawing or detail at a rotation angle by clicking on Rotate in the Plotter menu. Depending on the plotting history for this drawing, one of two things may happen:
 - If you haven’t plotted this drawing before, or if you used the QwkLayout option the last time you plotted it, you are prompted to “Select center of rotation”. Click in the Drawing Area to enter the center of rotation for your plot. You are prompted to “Enter angle of rotation”.
 - If you used the MltLayout option the last time you plotted the drawing, you are prompted to “Enter angle of rotation”.
14. Use the value menu or type an angle, and press **[Enter]**.
15. Lay out your drawing on a sheet by using the QwkLayout option; lay out multiple details on a sheet by using the MltLayout option. See “Laying Out Your Drawing” later in this chapter for more information on using the QwkLayout and MltLayout options.
16. Toggle Preview on to preview your plot before you actually send it to the plotter; toggle Preview off to skip the preview and send the sheet directly to the plotter. (The Preview toggle can only be used with the Plot and Partial options in the Plotter menu. See “More About Previewing Your Plot” below for details.)
17. Plot the sheet using one of the following:
 - To plot the sheet, click on Plot in the Plotter menu. If you toggled Preview off in the previous step, the sheet is plotted; if you toggled Preview on, the DataCAD Plot Previewer window opens, allowing you to preview the plot before sending it to your plotter. To interrupt a plot in progress, press **[End]**.
 - To plot the sheet to a file, click on ToFile in the Plotter menu. A dialog box is displayed. Enter a filename for your plot (.PLT) file and click on Save. Plot files are saved by default in your DATACAD\PLT folder.

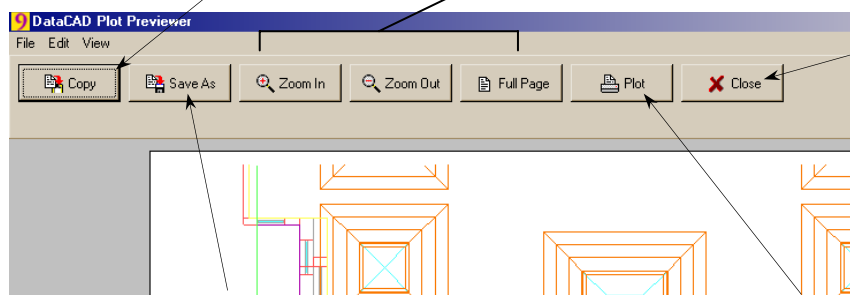
- To plot only a part of your drawing, click on Partial. Choose a selection method from the Partial menu and then select the part of your drawing you want plotted. If you toggled Preview off in the previous step, the selection is plotted; if you toggled Preview on, the DataCAD Plot Previewer window opens, allowing you to preview the selection before sending it to your plotter.

More About Previewing Your Plot

Previewing a plotting sheet before actually printing it allows you to verify that you're using the proper paper size and plotting scale, that your drawing is positioned properly, and that such things as hatching, dimensions, and line overshoot are displayed as you want them. From the DataCAD Plot Previewer window, you can change the view of your preview and then plot, copy, or save it.

Click on Copy to copy an image of the preview to the Windows clipboard. Then paste it into another Windows application.

Click on Zoom In, Zoom Out, and Full Page to view the preview at different levels of detail. You can also simply click on the preview to zoom in or press **Ctrl** or **Shift** and click to zoom out.



Click on Close to close the Plot Previewer window, without actually plotting your preview, and return to your drawing.

Click on Save As to save an image of the preview as a PDF file or a metafile. In the Save As dialog box, select either Adobe Acrobat File (.PDF) or Windows Enhanced Metafile (.EMF) from the Save as Type list box, type a name for the file, and click on Save.

Click on Plot to plot your drawing, just as it's displayed in the Plot Previewer window.

Figure 19.3: The DataCAD Plot Previewer window

Printing a Check Plot

You can also use check plots; this allows you to do a quick test print to your laser printer before sending the file to your plotter. Check plots use separate settings for the printer, paper size, and paper orientation you designated.

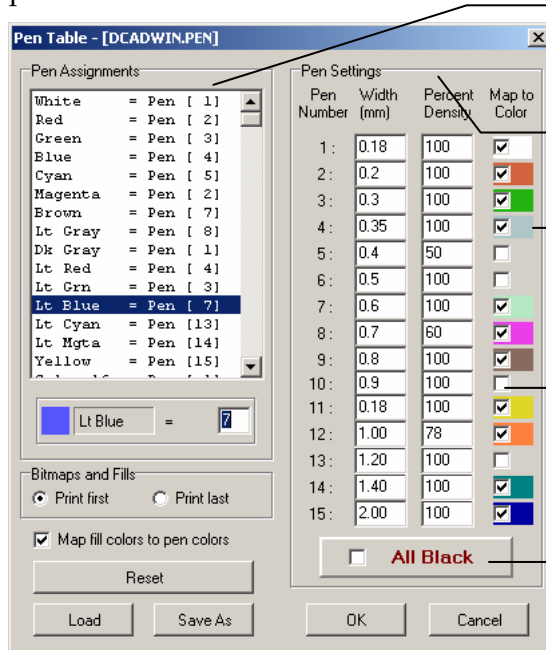
➔ To do a check plot:

1. Click on Setup in the Plotter menu. The Printer/Plotter Settings dialog box is displayed.
2. Go to the Check Plot section and click on Fit To to fit the drawing to a single page, or click on Use This Factor to scale your drawing to a particular size for the check plot. Notice that all the settings above the Check Plot section are now grayed back and the settings in the Check Plot section are now available. The FitTo option scales a previously defined layout to fit on the selected paper size. It does not automatically scale your drawing to fit on a defined paper size.

3. Select a printer or plotter to use from the drop-down box under the To This Printer label.
4. Select Fit To if necessary. Choose a page size from the drop-down box; if you selected Use This Factor, enter a percentage to scale the drawing for the check plot. If you use the Use This Factor option, your drawing and pen widths (as set in the Pen Table) are scaled.
5. Select Normal or Rotated paper orientation. This setting works exactly like the one described in “Printing Basics”.
6. Click on OK to close the Printer/Plotter Settings dialog box.
7. Continue with step 9 in the instructions under “Printing Basics” earlier in this chapter.

Using Pen Tables

Using pen tables will allow you to print the lines in your drawing in different colors than you drew them, in greyscale, or in black — or all of these in a single print!



To print lines in your drawing in a specific color and width, first assign a pen number to the on-screen line color in the Pen Assignments list box.

Next, set the desired width, density, and printed color for that pen number in the Pen Settings table.

Checking the Map to Color box prompts you to select a printed color for the pen. The color you select will be displayed in the color switch beside the checkbox.

If the Map to Color box is not checked, the pen number will print in the on-screen color.

Check All Black to override the above color settings and print all pens in black.

Figure 19.4: In the Pen Assignments list box on the left, lines drawn in Lt Blue are assigned to pen number 7. In the Pen Settings table on the right, pen number 7 has been set to plot at .6mm width and at 100% density (full color) in a grey color.

➔ To use a pen table:

1. Click on PenTable in the Plotter menu.
2. Go to the Pen Assignments list box and click on the color you want to assign to a pen. The color and its name are displayed beneath the list box, along with the pen number that color is currently assigned to.
3. Type the pen number you want to assign to that color. The pen assignment is changed in the list.
4. Repeat steps 2 – 3 to continue changing pen assignments as necessary.

5. Go to the Pen Settings section. Enter the pen width and color density for each pen that you assigned a color to.

The Percent Density setting allows you to print screened colors. A setting of 100% prints full color; entering a lesser percentage prints a screened color. This setting works only with printers that accept RGB color definitions. A LaserJet IV, for instance, can print shades of grey using screened black, but a LaserJet III can only print black.

The Percent Density and Map to Color settings are used only for color and greyscale printing, so they are only available when All Black is unchecked.

6. Print lines in your drawing in a different color than they are on screen by checking the Map to Color box for the pen assigned to the color you used in your drawing. A color dialog box is displayed.
7. Select the color you'd like to print and click on OK. The new color is displayed next to the Map to Color checkbox. The Map to Color checkboxes only affect printing; they don't change entity colors in your drawing.
8. Print all lines in black, regardless of what color they are on screen or what settings are entered in the rest of the pen table, by checking the All Black option at the bottom of the Pen Table dialog box.
9. Click on OK to save the changes you made to the pen table.

More About Using Pen Tables

Following are some examples of how pen table settings affect plotting:

- If you set the Width for Pen 1 to .25 and the Percent Density to 20% and leave Map to Color unchecked, all lines printed with Pen 1 will be printed .25mm wide in 20% of their full color (i.e., with a 20% color screen). *Full color* is a color at 100% density.
- You can produce heavy text on a print by drawing all text in a specific color, and then mapping that color to a thicker pen using the Pen Table dialog box.
- To print in gray scale, check the Map to Color option for any pen you'd like to print as gray scale. The Windows Color dialog box is displayed. Since black is chosen by default, click on OK to accept black as the color mapped to that pen. Then set the Percent Density option: to print a half-toned black color, for instance, change Percent Density to 50%. Some black and white printers will automatically produce greyscale print when Map to Color and All Black are left unchecked.
- To print lines in a different color than they are on screen, click on the Map to Color checkbox. Select a color from the Windows Color dialog box; black is selected by default. Click on OK. All lines using that pen will be reprinted in the selected color, regardless of their color on the screen. The Map to Color options only affect printing; the actual colors of the entities aren't changed.

Saving and Using Pen Table Settings

You can save a lot of time when setting up to plot your drawings if you save your Pen Table settings in a .PEN file. You can then quickly load those settings when you want to use them, instead of having to change all the Pen Table settings each time you print.

The name of the Pen Table file used in a drawing is always saved in that drawing. The default Pen Table file is DCADWIN.PEN, located in your DATACAD\SUP folder.

➔ To save pen table settings:

1. Click on PenTable in the Plotter menu. The Pen Table dialog box is displayed.
2. Change the settings as necessary.
3. Click on Save As in the Pen Table dialog box. Another dialog box is displayed, prompting you to "Save pen file as . . .".
4. Type a name for the Pen Table settings file and click on Save. The dialog box closes.
5. Click on OK in the Pen Table dialog box to close it.

➔ To use pen table settings that you've saved in a .PEN file:

1. Click on PenTable in the Plotter menu. The Pen Table dialog box is displayed.
2. Click on Load in the Pen Table dialog box. Another dialog box is displayed, prompting you to "Load pen file".
3. Click on the pen file you want to use, and click on Open. The dialog box closes, and the settings are loaded into the Pen Table dialog box.
4. Click on OK in the Pen Table dialog box to close it.

Laying Out Your Drawing

You have two options for defining plot sheets. You can quickly lay out your drawing at a single scale on a sheet, or you can place multiple details at different scales on a single sheet. Whenever you change Plotter menu settings, make sure you check the layouts you've already set up before plotting.

Using Quick Layout

As its name implies, quick layout is meant to be a fast, simple way to print all or part of your drawing on a single sheet.

➔ To quickly lay out your drawing for plotting at a single scale:

- 1. Click on QwkLayout in the Plotter menu. Notice that a grid representing the plotting area of the selected paper size is now attached to your cursor. (A dashed grid may also be displayed, representing the last defined quick layout.)

- 2. Position the grid over your drawing, so that the part you want to print is inside the grid. If your drawing is too large or small for the grid, exit the Layout menu and change the plotting scale; repeat steps 1 – 2.
- 3. Position the grid as you want it and click. The layout is defined, and you can continue with the plotting instructions given in “Printing Basics” earlier in this chapter.

Using Multi-Scale Layout

Multi-scale layout lets you place multiple details, each at any scale, on a single sheet. You can define up to 256 multi-scale sheets for plotting. Sheets can be renamed as well as have their contents deleted. See “More About Editing Details and Sheets” below for details.

Multi-scale plotting can be used with go-to-views to lay out multiple floors of a building on a single sheet. Toggle only layers for the first floor on and then save this view using the GotoView option in the View pull-down menu. Repeat for additional floors. Then place each go-to-view on the sheet using multi-scale plotting. See “Tutorial: Printing a Drawing” for an example of how to use go-to-views and multi-scale plotting together.

➔ To use multi-scale layout:

1. Click on MltLayout in the Plotter menu. The MltLayout menu is displayed.
 2. Make sure ShowAll is toggled on to display all details already on the sheet while you’re placing additional details.
 3. Select the sheet you want to work on. Click on Sheet in the MltLayout menu. A list of plotting sheets, along with a few other options, is displayed in the Menu Window.
 4. Click on the sheet you want to work on, keeping in mind you can click on ScrlFwrD and ScrlBck to scroll through all 256 sheets available to you.
 5. Right-click to return to the MltLayout menu.
 6. Now you can define how you’d like to set up your sheet layout. Click on LayoutSet in the MltLayout menu. The LayoutSet menu is displayed.
- Set the number of divisions for the sheet by clicking on LayoutDiv in the LayoutSet menu. A value menu is displayed, and you are prompted to “Enter X-divisions for layout sheet.” Use the value menu or type a number, and press **[Enter]**. A value menu is displayed, and you are prompted to “Enter Y-divisions for layout sheet.” Use the value menu or type a number, and press **[Enter]**. You are returned to the LayoutSet menu.
 - Toggle AutoCalc on in the LayoutSet menu to have DataCAD automatically recalculate the extents of a detail before you place it on the sheet. If this option is toggled off, you may need to use the ReCalc option for the layout box to accurately represent the extents of the detail.
 - Toggle Extents on in the LayoutSet menu to display only the extents of details already on the sheet. This reduces refresh time when you’re working on a sheet with several details on it. Toggle Extents off to display the details.

- Toggle Snap2Div on to make the cursor snap to the intersection of divisions you defined by clicking LayoutDiv.
 - Toggle on either GridLins or GridMrks. These are mutually exclusive toggles. GridLins uses dashed lines to represent the layout grid; GridMrks displays only the corner marks.
 - Toggle on either CC Exts or EntExts. These are mutually exclusive toggles. When you toggle on CC Exts (clip cube extents), the layout box will reflect the extents of the clip cube (which may not correspond with the extents of the entities contained within it). If you toggle on EntExts (entity extents), the Layout box will reflect the extents of the entities (which may not correspond with the extents of the clip cube surrounding them).
7. Right-click to return to the MltLayout menu.
 8. Begin placing details on the sheet. You can place the current view of your drawing or select a go-to-view that you've defined.
 - To place the current view of your drawing, click on Layout in the MltLayout menu. Notice that a box representing the extents of the current view is now attached to your cursor.
 - To change a plot layout setting, click on LayoutSet in the MltLayout menu.

If you select Layout, you can use InsPoint to select or snap to a specific point within the drawing. This becomes the insertion point for the detail. If you select Layout, you can toggle Snap2Div on so the detail can be snapped to the intersection you defined with LayoutDiv. You can also change the "handle" to position a multi-scale plotting detail on the fly by using the arrow keys during layout. If you select LayoutSet, you can use AutoCalc and Extents options as well as Snap2Div. Both Layout and LayoutSet let you set the mutually exclusive toggles GridLins or GridMrks as well as CC Exts or EntExts.

- To place a go-to-view you've defined, click on GotoView in the MltLayout menu. A list of your go-to-views is displayed in the Menu Window. Click on the go-to-view you want to place on the sheet. Notice that a box representing the extents of the go-to-view is now attached to your cursor.

Shortcut: To save go-to-views, click on either 3DViews in the Plotter menu or on the "V" in the Navigation pad to quickly access the 3DViews menu and then the GotoView menu.

9. Press **PageUp** or **PageDown** to dynamically scale the detail, then click on the sheet to place the detail. You are prompted to "Enter name of new detail".
10. Type a name for the detail and press **Enter**. If you're placing go-to-views, you can add another go-to-view or right-click to return to the MltLayout menu.
11. Right-click to return to the Plotter menu and plot your sheets.

More About Editing Details and Sheets

You can make changes to sheets and the individual details on them.

➔ To rename a sheet:

1. Click on Sheet in the MltLayout menu.
2. Click on Rename. The Name menu is displayed, and you are prompted to “Select sheet to rename”.
3. Click on the sheet in the list that you want to rename. You are prompted to “Enter new name”.
4. Type a new name for the sheet and press **Enter**. The sheet name is changed in the list.

➔ To erase all details on a sheet:

1. Click on Sheet in the MltLayout menu.
2. Click on Clear. You are prompted to “Select sheet to clear”.

Warning: You cannot undo clearing a sheet. Be certain you’re clearing the correct sheet.

3. Click on the sheet in the list that you want to clear. You are prompted with “Are you sure you wish to clear ‘[sheet name]’”.
4. Click on Yes to clear the sheet; click on No to cancel.

➔ To erase only selected details from a sheet:

1. Click on Details in the MltLayout menu. The Details menu is displayed in the Menu Window.
2. Click on Delete in the Details menu. You are prompted to “Select detail to delete from list”.
3. Go to the Menu Window and click on the detail you want to delete. It’s removed from the list and from the sheet.

➔ To move a detail on a sheet:

1. Go to the MltLayout menu, click on the name of the detail you want to move. Notice as you move your cursor over the Drawing Area, a box representing the extents of the detail is now attached to your cursor.
2. Press **PageUp** or **PageDown** to dynamically scale the detail. Then click on the sheet to place the detail on the sheet.

➔ To update a detail with your drawing’s current layer settings:

1. Click on Details in the MltLayout menu. The Details menu is displayed in the Menu Window.
2. Click on Update in the Details menu. You are prompted to “Select detail to update from list”.
3. Click on the detail you want to update. The detail is redrawn with your current layer settings.

➔ To toggle layers on or off in a detail:

1. Click on Details in the MltLayout menu. The Details menu is displayed in the Menu Window.
2. Click on LyrTogl in the Details menu. You are prompted to “Select the detail to toggle layers on and off”.
3. Click on the detail you want to change. The layers used in that detail are displayed in the Menu Window.
4. Click on layers to toggle them on or off, as necessary. The detail is updated on the sheet as you toggle layers on and off.
5. Right-click three times to return to the MltLayout menu.

➔ To change the current view of your drawing to match a detail:

1. Click on Details in the MltLayout menu. The Details menu is displayed in the Menu Window.
2. Click on MakeCurr in the Details menu. You are prompted to “Select the detail you wish to use as the current view”.
3. Click on the detail you want to use. You are returned to the Utility menu, and the view of your drawing is changed.

Batch Plotting

You can batch plot details, multi-layout sheets, and go-to-views from one drawing or from multiple drawings, all at once. Batch plotting your drawings can save your office a lot of time; instead of waiting for each drawing to finish printing before you print others, you can batch plot everything at once. You can set up a batch plot for multiple drawings and plot them all overnight, thereby freeing up valuable time during regular office hours. It isn't possible to batch plot in a background mode, so you must wait until the batch plotting process is complete before continuing your work in DataCAD.

See “Printing Basics”, “Using Pen Tables”, and “Laying Out Your Drawing” earlier in this chapter for details. Keep in mind that all go-to-views that are batch plotted will use the same quick layout that you set up using the QwkLayout option in the Plotter menu. Using multi-layout sheets is recommended for greatest control over your plotting results.

➔ To batch plot from a single drawing:

1. Set up details, multi-layout sheets, and go-to-views and change other Plotter menu settings as necessary.
2. Click on BatchPlt in the Plotter menu. The DataCAD Batch Plotting Setup dialog box is displayed.

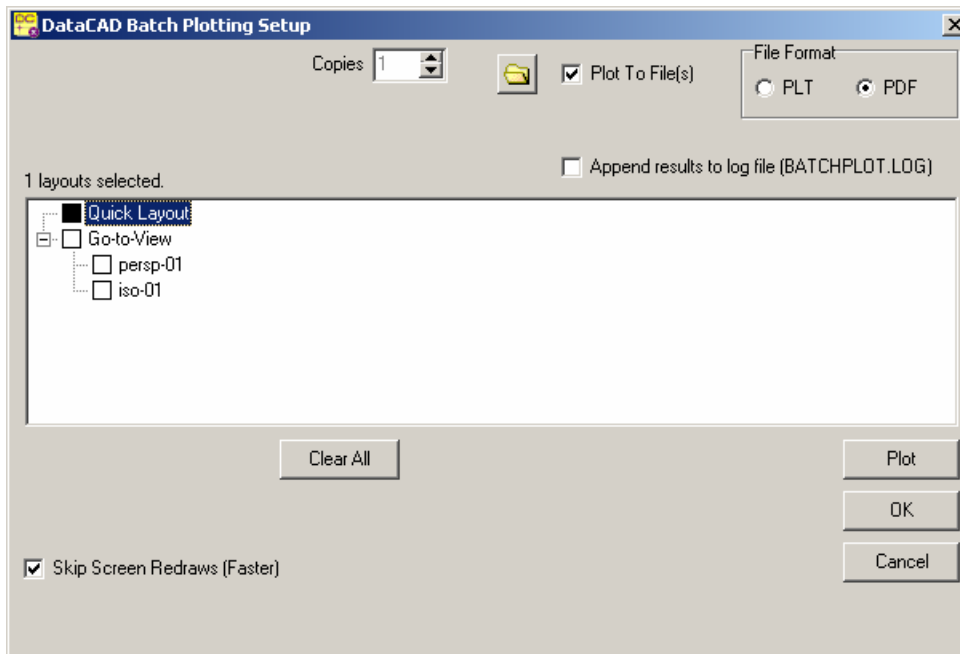


Figure 19.5: The Batch Plotting Setup dialog box

3. Notice that the list box in the Batch Plotting Setup dialog box lists any quick layouts, go-to-views, and multi-layout sheets that you've defined for the drawing. To choose the views and layouts to plot, click in the checkbox beside the view or layout. A white box indicates that view will not be plotted; a black box indicates that view will be plotted. If the checkbox beside the Go-To-Views or Multi-Layout Sheets folders is half white and half black, some of the views or sheets will be printed but not all of them. Click the Clear All button to uncheck all boxes.
4. Go to the Copies input box at the top of the dialog box. Enter the number of copies of each layout you want to plot.
5. Go to the top of the dialog box. Check the Plot to File(s) option to plot each selected layout to a file, instead of sending them directly to your plotter. These plot files will be saved to your DATACAD\PLT folder, with the following filenames:

Go-to-views:	"go-to-view" viewname.PLT
Quick layout:	drawing name – "quick layout".PLT
Multi-layout sheets:	drawing name – "sheet" sheetname.PLT
6. Go to the top of the dialog box. Check the Append Results to Log File option to copy details of the plotting process and results to a file called BATCHPLOT.LOG in your DATACAD\PLT folder. Normally, each DataCAD station that batch plots will have a separate BATCHPLOT.LOG file. However, if you set each DataCAD station to a shared network folder for plot files (.PLT) and .LOG files, then all plot results will be appended in a single .LOG file.

7. Go to the bottom of the dialog box. Check the Skip Screen Redraws option to speed up the plotting process. Checking this option skips displaying each drawing or view as it is opened; you simply won't see each view or layout displayed as it is plotted.
8. Click on Plot to begin plotting; click on OK to save these plot settings for plotting later (see instructions below for batch plotting from multiple drawings); or click on Cancel to cancel the batch plot settings and close the dialog box.

➔ To batch plot from multiple drawings:

1. Complete steps 1 – 7 in the instructions above (batch plotting from a single drawing) for *each* drawing you'd like to include in the multiple-drawing batch plot. Click on OK in step 8 to simply save the plot settings.
2. Click on Close All in the File pull-down menu to close all drawings. Make sure you click Yes to save changes for each file.
3. Click on Batch Plot in the File pull-down menu. The DataCAD Batch Plotting Setup dialog box is displayed.

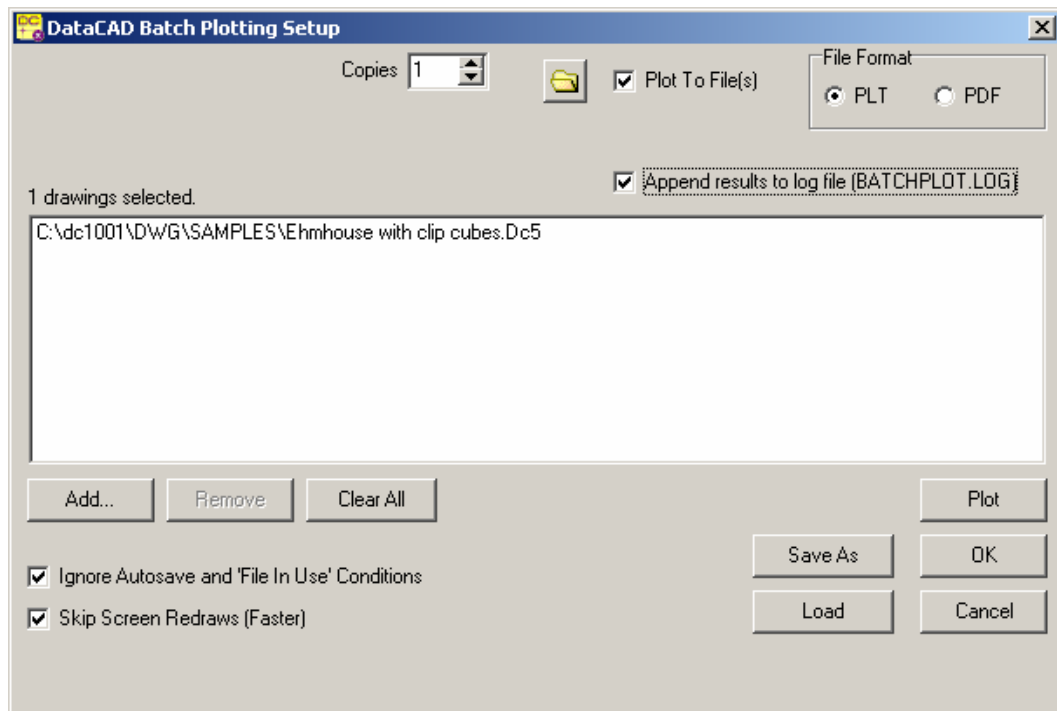


Figure 19.6: The Batch Plotting dialog box; notice the differences between this one and Figure 19.5

4. Click on Add to open the dialog box to select the drawings you want to batch plot. After you add drawings to the Batch Plotting Setup list box, you can save the group of drawings as a Project Set File (.SET); simply click on Save As, type a name for the .SET file, and click on Save. In subsequent batch plotting sessions, you can load a .SET file to quickly plot sets of drawings that are commonly plotted together, without having to add them to the Batch Plotting Setup list box each time.

5. Select the drawings that you set up for batch plotting in step 1, and click on Open. The dialog box closes and the drawing names and paths are displayed in the list box in the DataCAD Batch Plotting Setup dialog box. To remove a drawing from the list, select it and click on Remove. Click on Clear All to remove all drawings listed.
6. Enter the number of copies to be plotted. This setting overrides the number of copies setting that you entered in the Batch Plotting dialog box for each individual drawing.
7. Plot each drawing to a file instead of sending them directly to your plotter, check the Plot to File(s) option at the top of the dialog box. These plot files will be saved to your DATACAD\PLT folder, with the filename: drawing name – “sheet” sheetname.PLT. This setting overrides the Plot to File setting in each individual drawing.
8. Go to the top of the dialog box and check the Append Results to Log File option to copy details of the plotting process and results to a file called BATCHPLOT.LOG in your DataCAD\PLT folder. Normally, each DataCAD station that batch plots will have a separate BATCHPLOT.LOG file. However, if you set each DataCAD station to a shared network folder for plot files (.PLT) and .LOG files, then all plot results will be appended in a single .LOG file.
9. Go to the bottom of the dialog box and check the Skip Screen Redraws option to speed up the plotting process. Checking this option skips displaying each drawing or view as it is opened; you simply won't see each view or layout displayed as it is plotted.
10. Go to the bottom of the dialog box and check the Ignore Autosave and 'File in Use' Conditions option. This setting lets DataCAD ignore autosave and in use messages that would normally interrupt batch plotting.
11. Click on Plot to begin plotting; click on OK to save these plotting settings for plotting later; or click on Cancel to cancel the batch plot settings and close the dialog box.

Printer selection and batch plot settings are stored in each drawing, so you could batch plot each drawing to a different printer.

➔ To batch plot drawings to Adobe Acrobat's PDF format:

1. Check Plot to Files in the Batch Plotting Setup dialog box (see Figure19.5).
2. Set the File Format to PDF.

Larger firms may want to develop customized batch plotting routines using an enhanced scripting language. For more information about how to do this, contact DataCAD Technical Support.

Modeling Basics

20

DataCAD has a comprehensive set of 3D modeling tools, which can be used to complement the 2D tools. Since DataCAD is a true 3D modeling program, 2D and 3D data can coexist in the same drawing, allowing the user to combine the flexibility and strengths of each method. For example, you can develop a 2D drawing that can then be used to construct a 3D profile or perhaps a surface of revolution.

In this chapter:

- ⊕ Z-base and Z-height
- ⊕ The zero plane
- ⊕ Changing 3D settings
- ⊕ Drawing 3D lines
- ⊕ 3D snapping
- ⊕ Using markers

Z-Base and Z-Height

After using DataCAD as a 2D drafting tool, you're probably familiar with the concept of Z-base and Z-height as the "floor" and "ceiling" of entities you draw. When you use the 3D modeling tools in DataCAD, however, Z-base and Z-height no longer have to be floor and ceiling values. Instead, they're measured from the zero plane. Normally, the zero plane is parallel to the floor of your model; so in plan view, positive values for Z-base and Z-height extend toward you (out of the screen) while negative Z values extend away from you (into the screen). See "The Zero Plane" below for more information.

Z-base doesn't have to be at a lower elevation than Z-height, although you may find it easier to use them this way. Keep in mind, however, that entities are always constructed from Z-base to Z-height.

Most 3D entities have some kind of thickness associated with them; for instance, slabs have thickness while cylinders and cones have height. Because you can place the zero plane, Z-base, and Z-height in any spatial orientation with respect to your model, DataCAD can use the Z-base and Z-height settings to determine the thickness or height of a 3D entity. They can represent the front and back walls of a building, the sides of a bookcase, or any other two values that, taken together, create a depth to a drawing object.

In addition to Z-base and Z-height, you can use two auxiliary Z elevations as well: Z-User1 and Z-User2. These elevations are two extra planes in space, placed somewhere between Z-base and Z-height, that can be useful for creating 3D entities.

The Zero Plane

The *zero plane* is the plane in your model where $Z=0$. In plan view, the zero plane is fixed at the ground plane; however, in other views, you can change its position in relation to your model. It is important to note that, regardless of how its position changes relative to your model, the zero plane is always parallel to the screen. *When you change its position, the zero plane doesn't change in relation to your screen; rather it changes in relation to your model.* So when you change views of your model, you can use the following methods to change the orientation of the zero plane as well:

- Change the view of your model to orthographic (plan) view. The zero plane is changed to equal the ground plane. This method is best used when changes to your model can be made in plan view. Keep in mind that the model is temporarily displayed in plan view when you define a new elevation.
- Create a new elevation (click on New in the View/Elevation pull-down menu). The zero plane is changed to equal the elevation plane. This method is best used when changes to your model can be made in elevation view.
- Change the view of your model to a parallel view; this is often an elevation but you can use any parallel view. Use EditPlne in the 3DViews menu to change the orientation of the zero plane. See "More About the Zero Plane" below for more information on using the EditPlne and PlneSnap options.

- Change the view of your model to an isometric or similar view. Using PlneSnap in the 3DViews menu, change the orientation of the zero plane by selecting three points on the plane, such as the corners of a slab. This method is useful when you want to change the zero plane to match the plane of a roof or something similar.
- Using the Controls option in the 3DViews menu, change the view of your model to any parallel view. The zero plane is changed so that it passes through the view center.

More About Positioning the Zero Plane

As described above, you can use both EditPlne and PlneSnap to position the zero plane. Following are specific instructions on using both options.

➔ To position the zero plane using EditPlne:

1. Change to a parallel view that will allow you to draw a line indicating where you want the zero plane to be. This parallel view is usually an elevation of some kind.
2. Click on DCAD 3D in the Edit menu in the Menu Window. The 3DEdit menu is displayed.
3. Click on 3DViews in the 3DEdit menu. The 3DViews menu is displayed.
4. Click on EditPlne in the 3DViews menu. The EditPlne menu is displayed.
5. Notice that a new view center is created when you define a new zero plane. To set the Z coordinate (or height) of the view center, click on CenterZ in the EditPlne menu, enter a Z value, and press **Enter**. You are prompted to “Select a point on the edit plane”. The CenterZ setting automatically reverts back to its default of 5'-0" after you finish defining the new zero plane.
6. Click in the Drawing Area or use coordinate entry to enter the first point of the new zero plane. This point will also be the new view center. The cursor changes to a New Elevation Indicator, and you are prompted to “Indicate the direction of the edit plane”. See “More About the New Elevation Indicator Cursor” in the “Viewing Your Model” chapter.
7. Click in the Drawing Area or use coordinate entry to enter the second point of the new zero plane. The view is changed based on the plane you just defined.
8. Save this new view. Click on AddView in the EditPlne menu, type a name for the view, and press **Enter**. The view is added to your go-to-view list. Keep in mind that view names can include up to eight characters but no spaces. If your model moves off screen, click Reset in the EditPlne menu. The view is changed back to the most recent plan view.

There are a few rules when using PlneSnap that you should be aware of before you begin:

- When you select the three plane-defining points, be sure you snap to them in a clockwise direction.

- The first two points you snap to will align with the bottom of the screen in the new view; be sure to select the points in such a way that the new view will make sense to you. Keep in mind that if the new view is not what you wanted, you can always click on Reset in the PlneSnap menu to revert back to your original view.

In the following example, we need to place a skylight in the roof. To do this, we must first reposition the zero plane so it matches the plane of the roof. The easiest method for doing this is to use PlneSnap to simply snap to three points of the roof.

Remembering to work in a clockwise direction, we snap to the first two points along the bottom edge of the roof, rather than along the side. This bottom edge of the roof is aligned with the bottom of the screen, making it easy for us to orient ourselves to the new view, shown on the right of the example below.

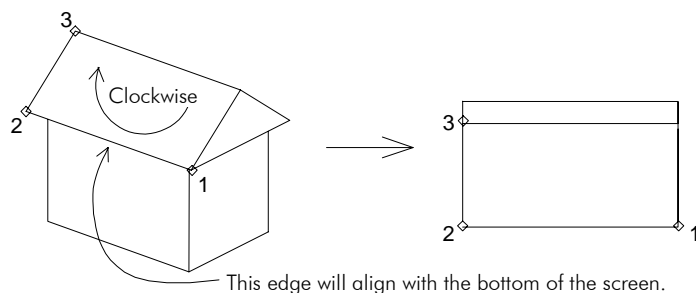


Figure 20.1: An example of effectively using PlneSnap to define a new zero plane

➔ To position the zero plane using PlneSnap:

1. Change to a parallel view that will gives you easy access to three points that lie on the plane that will be the new zero plane.
2. Click on DCAD 3D in the Edit menu in the Menu Window. The 3DEdit menu is displayed.
3. Click on 3DViews in the 3DEdit menu. The 3DViews menu is displayed.
4. Click on PlneSnap in the 3DViews menu. The PlneSnap menu is displayed, and the cursor changes to a 3D world-snapping cursor. You are prompted to “Select first point of snapping plane”. For details on the 3D world-snapping cursor, see “All About 3D Snapping” later in this chapter.
5. Enter the first point on the new zero plane and snap to a point on your model. You are prompted to “Select second point of snapping plane”.
6. Enter the second point on the new zero plane and snap to the point on your model. You are prompted to “Select third point of snapping plane”.
7. Enter the third point on the new zero plane and snap to the point on your model. The view is changed based on the plane you just defined. If your model moves off screen, click Reset in the PlneSnap menu. The view is changed back to the most recent plan view
8. Save this new view. Click on AddView in the PlneSnap menu, type a name for the view, and press **Enter**. The view is added to your go-to-view list. Keep in mind that view names can include up to eight characters but no spaces.

Changing 3D Settings

The 3D Settings menu has several toggles and other settings that affect the 3D menus. You can either click on Settings in the 3DEdit menu, or click on Options in the Tools pull-down menu and then click on 3D Settings in the submenu.

3D Settings Toggles

There are several toggles that affect drawing using 3D entities:

LyrSrch	Toggle on to scan all layers when you're selecting entities; this option works exactly like the one in 2D selection menus. See "More About Selection Menus" in "The Drawing Board" chapter for more information on Layer Search.
QckSrch	Toggle on to select curved 3D entities only by their center points; toggle off to select them by any displayed edge
LyrSnap	Toggle on to scan all layers when you're snapping to entities; toggle off to snap to entities on the current active layer only. See "More About 3D Snapping" later in this chapter for more information on quick snap.
QckSnap	Toggle on to snap only to straight edges or center points of 3D entities; toggle off to snap to these points as well as the endpoints of curved entities
DrawMrks	
MeshGrid	
MeshPnt	
GlobeDiv	
Dynamic	
SlabRefs	

Setting Primary and Secondary Divisions

Primary divisions are the number of curve divisions in the plane of the screen, also known as the sweep angle. *Secondary divisions* are those divisions that affect curves that bulge into or out of the plane of the screen; these are known as *rise or roll angles*. You can set the primary and secondary divisions for curved 3D entities using the 3D Settings menu.

➔ To set the number of primary divisions:

1. Click on Settings in the 3DEdit menu. The Settings menu is displayed.
2. Click on PrimDivs in the Settings menu. A value menu is displayed, and you are prompted to "Enter number of primary circle divisions".
3. Use the value menu or type a value, and press **[Enter]**. The primary divisions setting is changed, and you are returned to the Settings menu.

➔ To set the number of secondary divisions:

1. Click on Settings in the 3DEdit menu. The Settings menu is displayed.
2. Click on ScndDivs in the Settings menu. A value menu is displayed, and you are prompted to "Enter number of secondary circle divisions".
3. Use the value menu or type a value, and press **[Enter]**. The secondary divisions setting is changed, and you are returned to the Settings menu.

Setting Z-Elevations

You can change the Z-base and Z-height elevations in the 3D Settings menu, as well as two alternative Z-elevation settings.

➔ To change a Z-elevation:

1. Click on Settings in the 3DEdit menu. The Settings menu is displayed.
2. Click on the Z-elevation you want to change: Z-base, Z-height, or one of the two alternative Z-elevations, Z-User1 or Z-User2. A value menu is displayed, and you are prompted to enter a new value for the Z-elevation.
3. Use the value menu or type a value, and press **[Enter]**. The Z-elevation is changed, and you are returned to the Settings menu.

Drawing 3D Lines

3D lines are like wires or threads in space; they can connect any two points in 3D space. They are not constrained to lie in the XY plane; however, unlike 2D lines, 3D lines have no extrusion value.

When you draw 3D lines from Z-base to Z-height and select two points at exactly the same place on the screen, you create a line that is perpendicular to the screen. Such a line looks like a dot from the current view (a line viewed end-on).

Although this method works, it is usually conceptually easier to draw such lines flat across the screen in an appropriate elevation view.

You can specify the end points of 3D lines in the same way that you specify the end points of 2D lines. You can select points with the left mouse button, snap to points with the middle mouse button, or enter coordinates. You can also use the 3D cursor.

The first four options in the 3DEntity/3DLine menu are toggles that specify one of the four standard Z-elevations at which you can define entities. 3D lines can be drawn at Z-base, Z-height (Z-Hgt), or one of two alternative Z elevations (Z-User1 or Z-User2), which are set using the SetZUsr1 and SetZUsr2 options in the 3DLine menu, respectively.

➔ To set the height of the Z-User1 variable:

1. Choose SetZUsr1 from the 3DLine menu. A list of values is displayed.
2. Choose or type a height value and press **[Enter]**.

➔ To set the height of the Z-User2 variable:

1. Choose SetZUsr2 from the 3DLine menu. A list of values is displayed.
2. Choose or type a height value and press **[Enter]**.

You can also draw 3D lines that begin at Z-base (first point selected) and end at Z-height (second point selected) by toggling the Base/Hgt option on in the 3DLine menu. See “Z-Base and Z-Height” in the beginning of this chapter for more information on Z-elevations.

Further, you can toggle Chain on to create 3D lines that rubberband from the last point entered, just like 2D lines. If you toggle Chain off, you must select both end points of each 3D line you draw.

All about 3D Snapping

DataCAD 3D uses a different technique for object snapping than the two-dimensional DataCAD editor.

The most important difference between normal 3D snapping and its 2D counterpart is that normal 3D snapping attaches the cursor to the projection of the model on the screen, not to the model itself. A projection is a two-dimensional view of a three-dimensional object. If you want to attach the cursor to points on the model, you need to orient the zero plane (screen plane) so that it coincides with the part of the model you are snapping to.

Three-dimensional snapping is designed to work when the model is displayed in orthographic or any parallel view. See “The Zero Plane” earlier in this chapter for more information.

Zero Plane not aligned with the model.

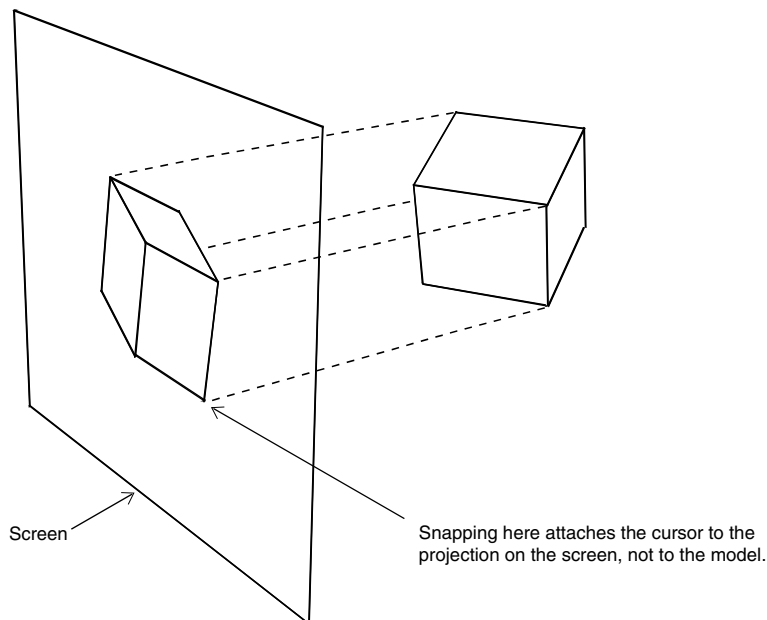


Figure 20.2: Normal snapping

Zero plane aligned with
the model.

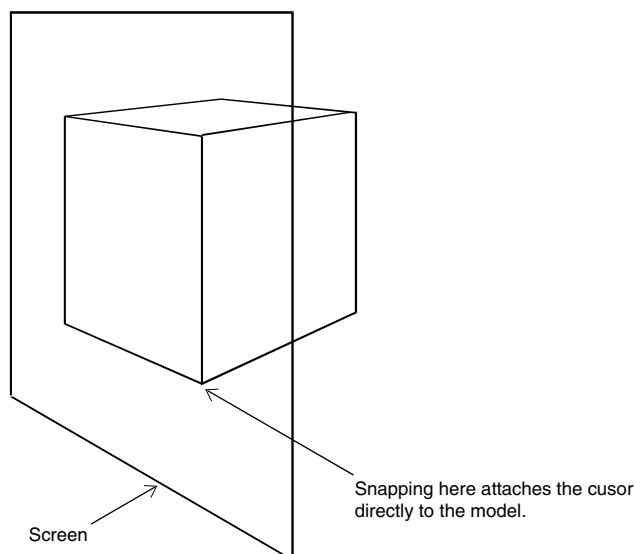


Figure 20.3: Zero plane alignment

In several menus in DataCAD 3D, you can use a 3D world-snapping cursor to snap to the end points you see in the model. This 3D world-snapping cursor snaps to the entities themselves rather than connecting to the projections of those entities. If the 3D cursor appears, then world snapping is on; if you press the middle button on the mouse, you snap to points on the model and not the projections of those points on the screen. World snapping works regardless of the location of the zero plane.

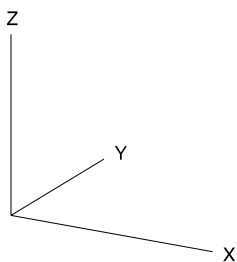


Figure 20.4: 3D world-snapping cursor

Zero plane not aligned
with the model

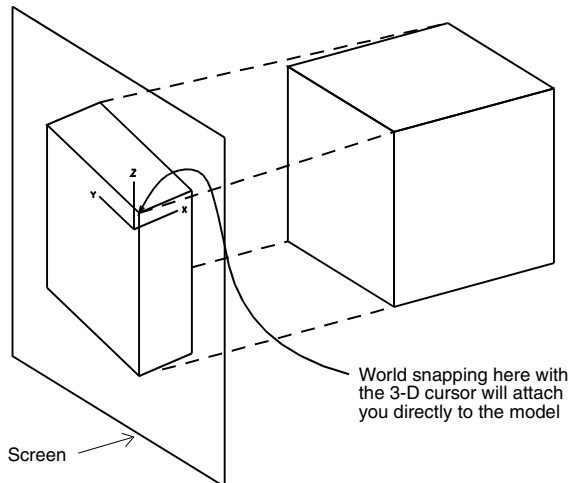


Figure 20.5: World snapping with 3D cursor

More About 3D Snapping

If you toggle Fast3D on in the Utility/ObjSnap menu or toggle QckSnap on in the 3DEdit/Settings menu, you can only snap to the edges of straight-edged objects and the center axis points of curved entities. If you toggle Fast3D or QckSnap off, you can snap to the end points of the circular divisions as well.

The following is a list of entities that you can snap to using Fast 3D and Quick Snap:

3D Lines	Snap to either end of a 3D line
Polygons	Snap to either end of any displayed edge (including the edges of voids)
Blocks	Snap to either end of any edge
Slabs	Snap to either end of any displayed edge (including the edges of voids)
3D Arc	Snap to the arc center marker; with QckSnap off, you can snap to the end points of the division segments that make up the arc itself
Horizontal or Vertical Cylinder	Snap to any of the three center axis markers; with QckSnap off, you can snap to the end points of the circular divisions of a cylinder
Cone	Snap to any of the three center axis markers; with QckSnap off, you can snap to the end points of the circular divisions of the cone
Truncated Cone	Snap to any of the three center axis markers; with QckSnap off, you can snap to the end points of the circular divisions of the truncated cone
Sphere	Snap to any of the three center axis markers; with QckSnap off, you can snap to any edge of the facets that make up the sphere
Torus	Snap to the center marker of the sweep or the center markers at the beginning and end of the roll; with QckSnap off, you can snap to any edge of the facets that make up the torus
Contour Curve or Mesh Surface	Snap to any control point; QckSnap has no effect on snapping to contours or mesh surfaces
Surface of Revolution	Snap to any control point; with QckSnap off, you can snap to any edge of the single-curvature facets which make up the surface of revolution, but you cannot snap to the edges of the double-curvature facets

Symbol	Snap to any displayed edge of any portion of the symbol according to the rules for each individual entity that makes up the symbol
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Using Markers

A marker is a 3D snapping point. When you choose Marker from the 3DEntity menu, the following options appear:

Z-Base	Places a marker at the current Z-base
Z-Hgt	Places a marker at the current Z-height
3DCursor	Snaps to any object with world snapping enabled; for more information on world snapping, see “3D Snapping” in the “3D Editing” chapter. See “All about 3D Snapping” earlier in this chapter for more information on world snapping.
3DViews	Accesses the 3DViews menu

Modeling with 3D Polygons, Slabs, and Blocks

In this chapter:

DataCAD's 3D polygons, slabs and blocks are closely related entities with a few important differences. A 3D polygon is a single entity with no thickness (unlike a 2D polygon, which is a group of lines drawn from Z-base to Z-height). Slabs, usually the most common entity in a model, are very similar to polygons, except that they have a thickness that you can specify. Blocks are simple solids with six rectangular or parallelogram-shaped faces. Because the top of a block doesn't have to lie directly over its bottom, blocks are particularly useful for modeling support members that lean to one side.

Another important distinction between 3D polygons and slabs is that slabs have a *reference face*, which is the face of the slab that orients the slab, controls the direction that you can pass a void through the slab, and is important when moving or adding vertices or voids to a slab. DataCAD automatically assigns one face of a slab to be the reference face, depending on what kind of slab you draw:

- Horizontal slabs** If you draw the slab from Z-base with the specified thickness (Base+Thk toggle), the reference face is at Z-base. If you draw the slab from Z-height with the specified thickness (Hgt+Thk toggle), the reference face is at Z-height. If you draw the slab from Z-base to Z-height (Base/Hgt toggle), the reference face is at Z-base.
- Vertical slabs** When you draw a slab from the top of the screen to the bottom, the reference face is on the left. When you draw a slab from the bottom of the screen to the top, the reference face is on the right.
- Inclined slabs** The reference face is always on the underside of the slab.

If you forget which face of a slab is the reference face, toggle SlabRefs on in the 3DEdit/Settings menu. Dots will be displayed on every vertex of each reference face, along with an additional dot on one of the opposite vertices.

- ⊕ Drawing horizontal and vertical 3D polygons
- ⊕ Drawing horizontal and vertical slabs
- ⊕ Using inclined polygons and slabs to draw roofs
- ⊕ Adding voids to 3D polygons and slabs
- ⊕ Editing 3D polygons and slabs
- ⊕ Drawing blocks

Drawing Rectangular 3D Polygons and Slabs

➔ To draw a rectangular 3D polygon:

1. Choose RectAngl from the Polygon menu.
2. Choose a Z elevation (Z-Base, Z-Hgt, Z-User1, or Z-User2) at which to draw the rectangle. To set the alternative Z elevations, choose SetZUsr1 to set the Z-User1 elevation or choose SetZUsr2 to set the Z-User2 elevation.
3. Select a point to start the rectangle.
4. Drag the cursor diagonally to size the rectangle and click. The rectangle is drawn. You can also create rectangular polygons with Horizntl, but using RectAngl is faster and easier.

➔ To draw rectangular slabs:

1. Choose RectAngl from the Slab menu.
2. Choose a Z elevation at which to draw the rectangular slab: from Z-base to a thickness (Base+Thk), from Z-height to a thickness (Hgt+Thk), or from Z-base to Z-height (Base/Hgt).
3. Determine if you are using Base+Thk or Hgt+Thk to draw slabs. Set the thickness value by choosing Thcknss from the RectAngl menu and then choosing or typing a value and pressing **Enter**. As with all Z measurements, a positive thickness extends out of the screen and a negative thickness extends into the screen.
4. Select a point to start the rectangle.
5. Drag the cursor diagonally to size the slab and click. The slab appears. You can also create rectangular slabs with Horizntl, but using RectAngl is faster and easier.

Drawing Horizontal 3D Polygons and Slabs

➔ To draw horizontal 3D polygons:

1. Choose Horizntl from the Polygon menu.
2. Choose the elevation at which you want to draw the polygon: Z-base, Z-height (Z-Hgt), or one of the two alternative Z elevations (Z-User1 or Z-User2). These alternative Z elevations are set using the SetZUsr1 and SetZUsr2 options in the menu, respectively. To set the height of an alternative Z elevation, choose SetZUsr1 or SetZUsr2 from the Polygon menu, and then choose or type a height value and press **Enter**. See “Z-Base and Z-Height” in the “Modeling Basics” chapter for details about Z-elevations.
3. Select the first point of the polygon. You can create polygons of up to 36 vertices this way.

4. Continue selecting points until you are ready to close the polygon. If you make a mistake in placing a vertex, use Backup to erase the last selected point. Use Backup to erase the last side of the polygon you drew. Backup is available as soon as you select the first point of a polygon and until you close the polygon or right-click to disconnect from the polygon you're drawing. To cancel the polygon before the polygon is completed and added to the drawing, choose Cancel. Use Cancel to erase the polygon you're currently drawing. Cancel is only available after you select the third point of the polygon and until you close the polygon or right-click to disconnect from the polygon you're drawing. You can also choose Exit, which is available until you select the third point of the polygon, when it becomes Close. Use Exit to stop drawing a polygon and erase what you've drawn to that point.

If you create a 36-vertex polygon, DataCAD closes the polygon for you. To circumvent this 36-vertex limitation, see "Editing 3D Polygons and Slabs" later in this chapter.

5. Connect the last point you selected with the first one and close the polygon by choosing Close or right-clicking.

➔ To draw horizontal slabs:

1. Choose Horizntl from the Slab menu.
2. Choose the elevation at which you want to draw the slab: Z-base extending to a thickness (Base+Thk), Z-height extending to a thickness (Hgt+Thk), or Z-base extending to Z-height (Base/Hgt). Backup is available as soon as you select the first point of a slab and until you close the slab or right-click to disconnect from the slab you're drawing.
3. Set a thickness for the slab by choosing Thicknss from the Horizntl menu. Then choose or type a value and press [Enter]. As with all Z measurements, a positive thickness extends out of the screen and a negative thickness extends into the screen. Set Thicknss when you are drawing slabs with Base+Thk or Hgt+Thk; otherwise, the Thicknss setting is ignored.
4. Select the first point or vertex of the slab. You can create slabs of up to 36 vertices. If you create a 36-vertex slab, DataCAD closes the slab for you. If you make a mistake in placing a vertex, use Backup to remove the last vertex you entered and then continue to place vertices. Use Exit to stop drawing a polygon and erase what you've drawn to that point. After you select the third point of the polygon, the Exit option changes to Close. See "Editing 3D Polygons and Slabs" later in this chapter for more information about simulating polygons or slabs with more than 36 sides.
5. Select a second corner point that, along with the first point, forms an edge for the base of the block.
6. Choose Cancel if you want to cancel the slab before it is completed and added to the drawing. Cancel is only available after you select the third point of the slab and until you close the slab or right-click to disconnect from the slab you're drawing.
7. Use Close if you want to connect the last point you selected with the first one and finish the slab.

Drawing Vertical 3D Polygons and Slabs

Vertical polygons and slabs are rectangular shapes drawn edge on. Vertical polygons are useful for quickly creating forms that show some kind of extruded quality, like gutters or irregular curtain walls, while vertical slabs are useful for quickly creating free-standing walls. Draw the edge of the vertical polygon or slab by selecting points on the screen; the height of the entity is determined by the separation of Z-base and Z-height (or the two alternative Z elevations).

➔ To draw a vertical 3D polygon:

1. Choose Vertical from the Polygon menu.
2. Choose to draw the polygon from Z-base to Z-height (Base/Hgt) or from Z alternative 1 to Z alternative 2 (Z1/Z2). These toggles set the height of the vertical polygon to run from one Z elevation to the other (from Z-base to Z-height or from Z1 to Z2). To set the height of the Z1 or Z2 elevation, choose Z-User1 or Z-User2 from the Polygon menu, and then choose or type a height value and press **[Enter]**.
3. Choose Chain to create vertical polygons in succession, with the first edge of each polygon connected to the last edge of the previous polygon. This is similar to the way 2D lines are drawn. When you toggle Chain off, DataCAD does not automatically connect lines, so you must select both points of each polygon edge you draw.
4. Select two points on the screen creating one edge of the vertical polygon. The polygon is drawn upright, into or out of the screen.

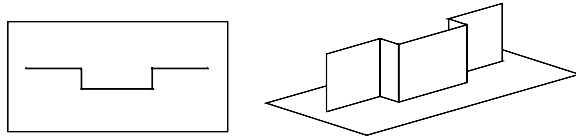


Figure 21.1: Vertical polygons

➔ To draw a vertical slab:

1. Choose Vertical from the Slab menu.
2. Choose to draw the slab from Z-base to Z-height (Base/Hgt) or from Z-alternative 1 to Z-alternative 2 (Z1/Z2). These toggles set the height of the vertical slab to run from one Z elevation to the other; they are mutually exclusive. To set the values of Z-alternative 1 and Z-alternative 2, choose Z-User1 or Z-User2 from the Vertical menu, and then choose a value from the list or type a value and press **[Enter]**.
3. Set the thickness of the slab by choosing Thicknss from the Vertical menu. Then choose or type a value and press **[Enter]**. Do not confuse this setting with the Thicknss setting in Horizontal and Rectangular slabs. Thickness for vertical slabs refers to the short dimension of the slab as seen across the screen, not the depth of the slab going into the screen.

4. Choose Left, Center, or Right to determine the side of the slab on which to attach the cursor. The terms Left, Center, and Right refer to the slab as drawn from the top of the screen toward the bottom. If you draw the slab from the bottom to the top with Left toggled on, your cursor appears attached to the right side of the slab. These toggles are mutually exclusive and can be activated at any time, even after you select the first point of the slab.

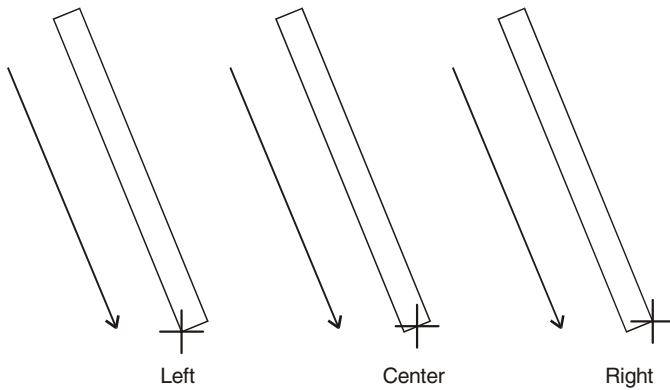


Figure 21.2: Cursor positions on vertical slabs

5. Select the first corner point of the vertical slab.
6. Move the cursor diagonally to the opposite point of the slab. The slab sizes dynamically with the cursor. Click to complete the slab. The slab is drawn upright, between the two elevations you selected in step 2.

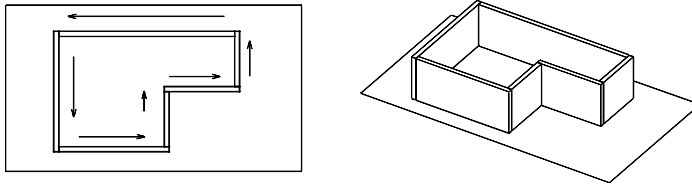


Figure 21.3: Vertical slabs

Use SlabRefs to display markers that indicate the reference face for a slab as well as the direction of extrusion. The reference face of the slab is determined by the points selected on the screen. The extrusion is determined by the relative Z-base and Z-height (or Z-User1 or Z-User2) settings you have entered.

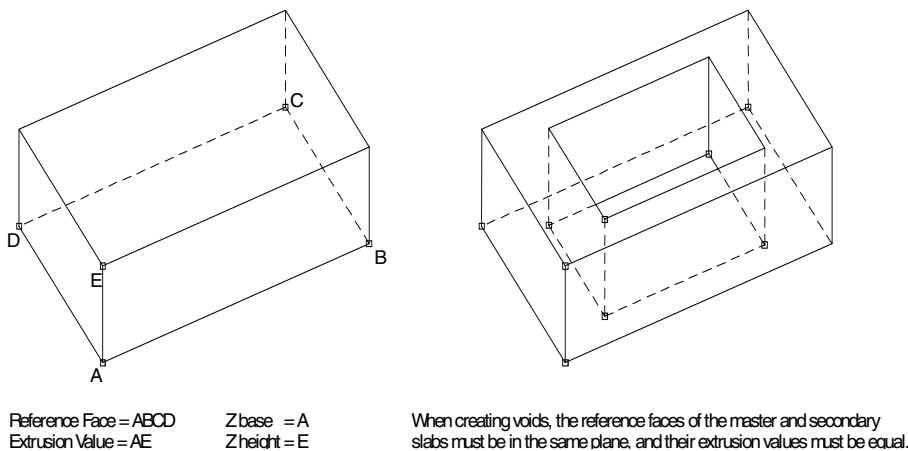


Figure 21.4: Reference faces

Drawing Inclined 3D Polygons and Slabs

Inclined polygons can be used to create roof elements. Roofs that you create with this menu do not have a thickness. To create roofs with thickness, use the Inclines menu in the 3DEntity/Slab menu, or use the RoofIt tool available from the Toolbox menu to create simple roofs automatically.

➔ To create roof elements using inclined polygons:

1. Use the 3DViews option from the Polygon menu to place the model in an orthographic view.
2. Set Z-Base to the height of the lowest point on the roof.
3. Set Z-Hgt to the height of the ridge line (peak) of the roof. DataCAD needs Z-base and Z-height information about the roofs that you are going to create—information that is easily calculated but not immediately at hand. Calculate the heights and breadths of roofs before using the Inclines menu.
4. Choose Inclines from the Polygon menu.
5. Continue with the sections below to draw the three- or four-edged inclined polygon you need.

➔ To create roof elements using inclined slabs:

1. Place the model into orthographic (plan) view.
2. Choose Inclines from the Slab menu.
3. Choose Thicknss to set the depth of the roof element; choose or type a value and press Enter.
4. Choose whether to add the thickness perpendicularly or vertically. Vertical adds the thickness of the slab to the roof element vertically, or perpendicular to the ground. This produces a plumb cut on the edge of the roof element. Perpend adds the thickness of the slab to the roof perpendicularly to the roof slope.
5. Continue with the instructions below to draw the three- or four-edged inclined slab you need.

➔ To draw gable, A-frame, and shed roofs:

1. Choose 4EdgPara from the Inclines menu.
2. Select the first corner point at the bottom edge of the slope.
3. Select the second corner point at the bottom edge of the slope.
4. Select the first corner point on the ridge of the roof. Enter this third point opposite the first point, as shown below. Locate this point by snapping or with a reference point shortcut and coordinate entry. The fourth corner point is automatically derived.

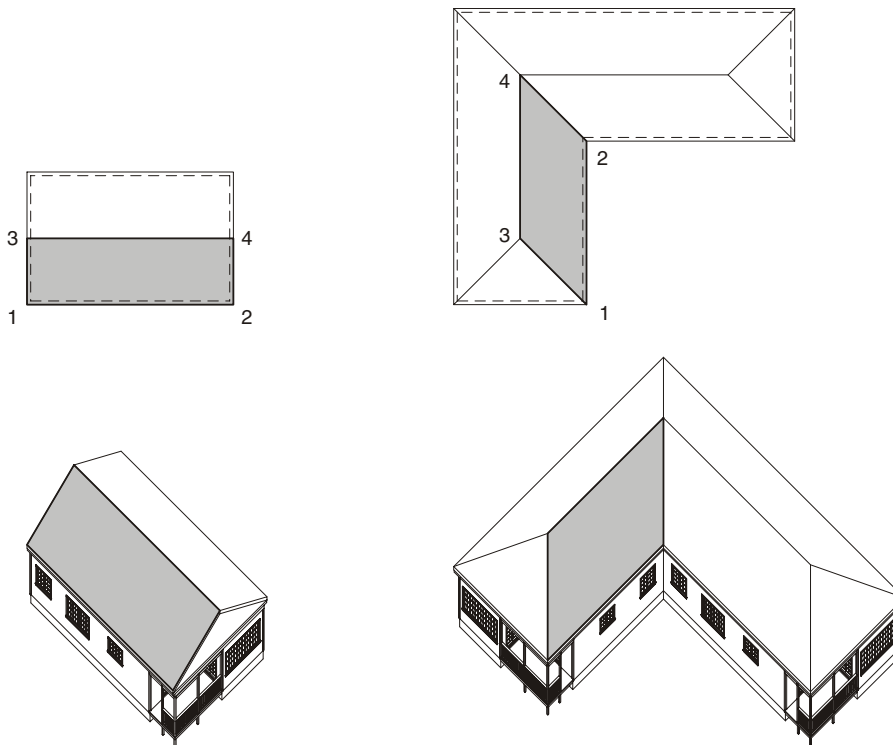


Figure 21.5: Four-edged parallelograms

➔ To draw hip and mansard roofs:

1. Choose 4EdgTrap from the Inclines menu.
2. Select the first corner point at the bottom edge of the slope.
3. Select the second corner point at the bottom edge of the slope.
4. Select the first corner point on the ridge of the roof. Enter this third point opposite the first point, as shown below. Locate this point by snapping or with a reference point shortcut and coordinate entry. The fourth corner point is automatically derived.

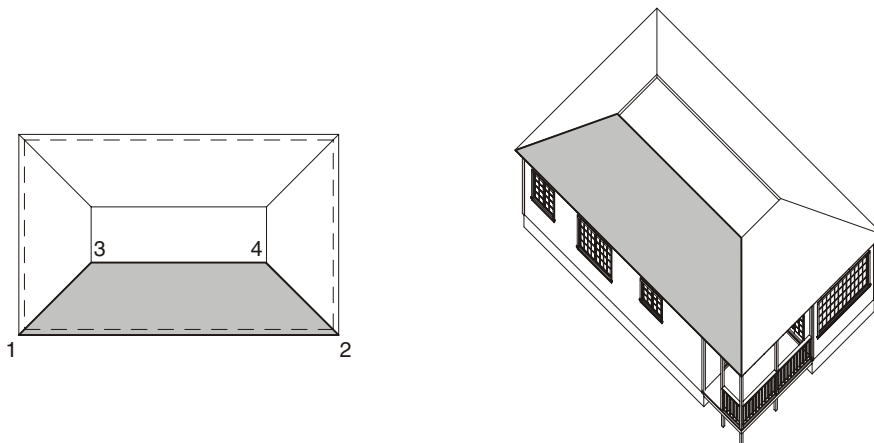


Figure 21.6: Four-edged trapezoids

When a portion of a roof has four edges, but the opposite edges are not parallel due to unusual intersections or valleys with other roofing elements, use the four-edged general inclined polygon or slab option.

➔ To draw dormers and other unusual roofs:

1. Choose 4EdgGen from the Inclines menu.
2. Select the first corner point at the bottom edge of the slope.
3. Select the second corner point at the bottom edge of the slope.
4. Select the first corner point on the ridge of the roof. Enter this third point opposite the first point, as shown below. Locate this point by snapping or with a reference point shortcut and coordinate entry.
5. Select the last corner point on the ridge of the roof.

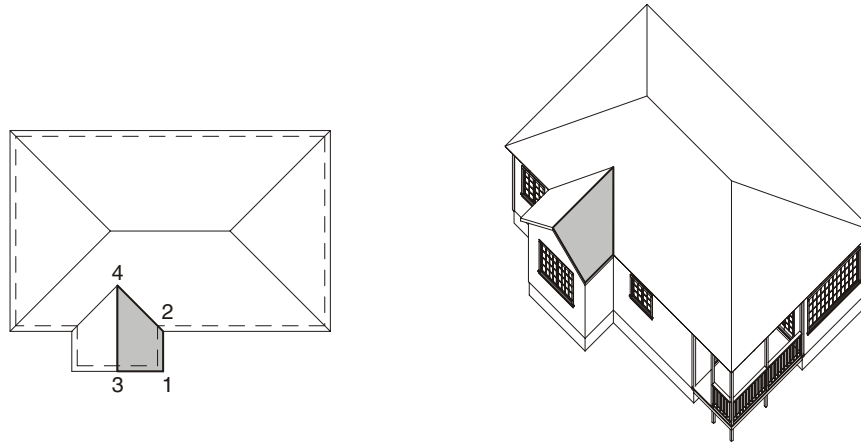


Figure 21.7: Four-edged general incline

➔ To draw the triangular pieces of hip roofs:

1. Choose 3EdgBot from the Inclines menu.
2. Select the first point at the bottom edge of the slope (at Z-base).
3. Select the second corner point at the bottom edge of the slope (also at Z-base).
4. Select the corner point on the ridge of the roof (at Z-height). Locate this point by snapping or with a reference point shortcut and coordinate entry.

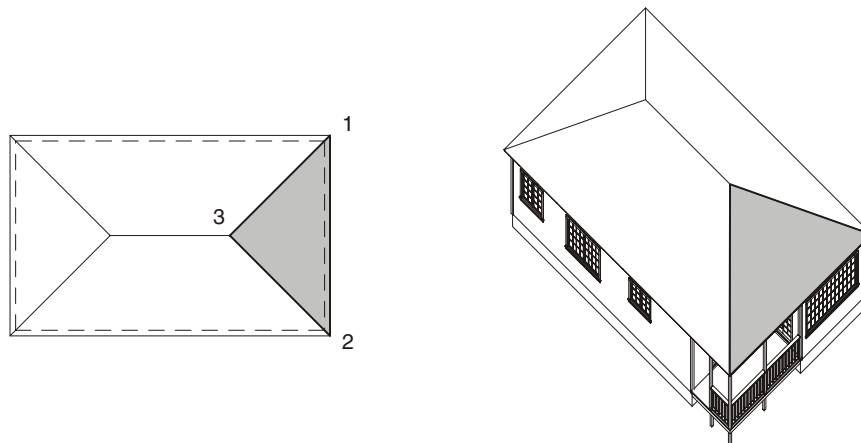


Figure 21.8: Three-edged bottom incline

➔ To draw simple dormers:

1. Choose 3EdgTop from the Inclines menu.
2. Select the first corner point on the ridge of the roof (at Z-height).
3. Select the second corner point on the ridge of the roof (also at Z-height).
Locate this point by snapping or with a reference point shortcut and coordinate entry.
4. Select the corner point at the bottom edge of the slope (at Z-base).

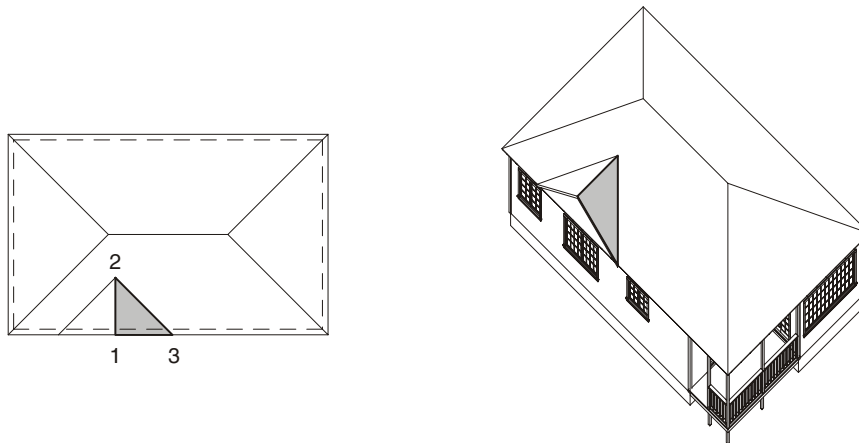


Figure 21.9: Three-edged top incline

The four-edged vertical incline is created edge on, very much like the vertical polygon or slab, except that the incline has an independent thickness associated with it that controls the depth of the beam.

➔ To draw rafters using vertical inclined polygons or slabs:

1. Choose 4EdgVert from the Inclines menu.
2. Choose Thicknss to set the depth of the polygon or slab (rafter). A list of values appears.
3. Choose or type a value and press .
4. Select the location for the lower edge of the polygon or slab, or the base of the incline, including the thickness of the polygon or slab.
5. Select the location for the upper edge of the polygon or slab, or the top of the incline. The polygon or slab ends at the Z-height, including the thickness of the polygon or slab.

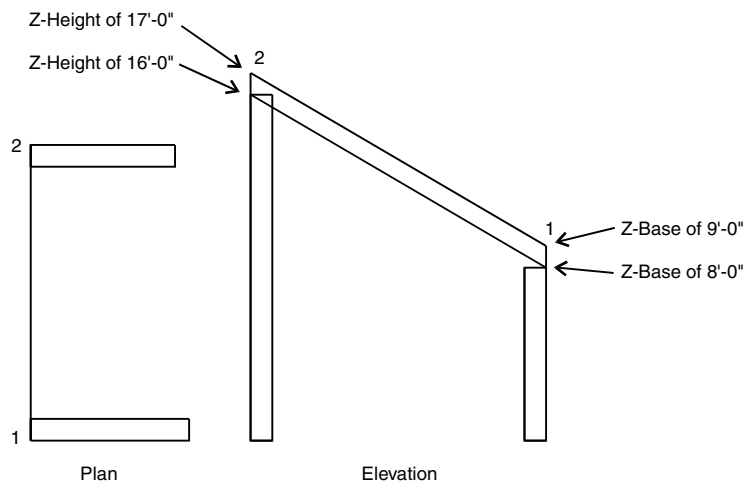


Figure 21.10: Four-edged vertical incline

➔ To create three-edged vertical, inclined polygons or slabs:

1. Choose 3EdgPoly from the Inclines menu.
2. Set the thickness of the slab using the Thicknss option if you're drawing a slab.
3. Snap to the first corner point on the incline. Use the 3D cursor to snap to the corners of roof elements already in place. You don't have to select the points in any particular order.
4. Snap to the second corner point on the incline.
5. Snap to the third corner point on the incline. The polygon or slab is drawn.

Adding Voids to Polygons and Slabs

The Voids menu in the Polygon and Slab menus has options for creating and deleting holes in polygons and slabs. You create a void by drawing a small polygon or slab (representing the void) inside a larger (primary) polygon or slab. The smaller polygon or slab is then converted into a void with the AddVoid option.

When creating slabs, the reference face of each void must coincide exactly with the reference face of the primary slab, and the voids must be exactly as thick as the primary slab.

➔ To add a void:

1. Draw a primary polygon or slab (the polygon or slab you will be adding a void to). Use PlneSnap, NewElev, or EditPlne (in the 3DViews menu) to bring the primary polygon or slab into the zero plane before adding voids. When you create both the primary and void polygons or slabs without changing the view, they automatically lie in the same plane.

2. Draw the void polygon or slab inside the primary polygon or slab in exactly the same plane, at Z-base (or whatever Z elevation you drew the primary polygon or slab at) equal to zero. You can draw more than one void polygon or slab at this time. Make sure the void polygon or slab lies completely within the primary polygon or slab.
3. Choose Voids from the Polygon or Slab menu.
4. Highlight the primary polygon or slab by selecting one of its edges.
5. Choose AddVoid.
6. Select the void polygon(s) or slab(s) you drew in step 2. The void polygons or slabs are highlighted as you select them. The primary polygon or slab now has voids in it.

When you add a void to a polygon or slab, you are left with one entity (a polygon or slab with a hole in it) where separate polygons or slabs existed before. If you erase or move the primary polygon or slab, the void moves with it, as you would expect of a single entity. The void polygons or slabs no longer have an identity apart from the primary polygon or slab to which they belong.

Use ConvVoid to convert voids back into the polygons or slabs from which they originated. You must convert a void when you want to edit the void by adding, deleting, or moving vertices using the Partial menu.

➔ To convert a void:

1. Select Voids from the Polygon or Slab menu.
2. Select a primary polygon or slab by indicating one of its edges. The primary polygon or slab and any voids are highlighted.
3. Choose ConvVoid from the Voids menu.
4. Select the voids that you want to convert back into polygons or slabs. Any converted voids now exist as individual, embedded polygons or slabs.

To add vertices to a void, convert the void back into a polygon or slab with ConVoid, add vertices with AddVertex from the Partial menu, then use AddVoid to turn the polygon or slab back into a void. You can also move a void or resize it using 3DEdit/Stretch.

Once you add a void to a primary polygon or slab, the void polygon or slab no longer exists as an independent entity; it becomes a hole in a larger object. Use ErasVoid to remove these voids from a primary polygon or slab.

➔ To erase a void:

1. Choose Voids from the Polygon or Slab menu.
2. Select the primary polygon or slab by indicating one of its edges. The polygon or slab and any voids are highlighted.
3. Choose ErasVoid from the Voids menu.
4. Select the voids that you want to delete from the polygon or slab. Selected voids disappear entirely, sealing the hole.

Editing 3D Polygons and Slabs

With the Partial menu options, you can edit - move, add, or delete vertices in - a polygon or slab. You can also toggle the edges of a polygon or slab to visible or invisible.

To maintain its integrity, edit the polygon or slab in the same plane in which it was created or at least in a plane parallel to the polygon or slab. Use PlneSnap, NewElev, or EditPlne from the 3DViews menu. See the chapter on “Viewing Your Model” for more information about the options for PlneSnap, NewElev, and EditPlne in the 3Dviews menu.

To simulate polygons or slabs with more than 36 sides, you can create a 36-sided polygon or slab and mark one of the edges as invisible. You can connect another polygon or slab with a shared invisible edge to the first invisible edge. By hiding the seams, you can build a patchwork of polygons or slabs that simulates a polygon or slab of an unlimited number of sides.

➔ To add vertices to a polygon or slab:

1. Choose AddVrtex from the Partial menu. Make sure that the current editing plane is parallel to the polygon or slab you want to edit.

Warning: Use caution with MovVrtex and AddVrtex. These commands can create physically impossible shapes. The most common problems include letting edges cross over one another and pulling vertices out of the plane of the polygon or slab, thus creating a twisted and ambiguous shape.

2. Select the edge of the polygon or slab to which you want to add a vertex. Remember that a polygon or slab can have no more than 36 vertices. Notice that this edge of the polygon or slab becomes a rubberband line with your cursor attached to the edge.
3. Position the new vertex.

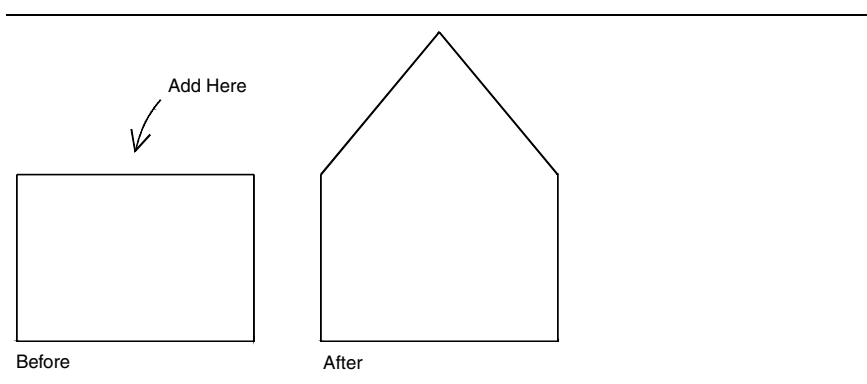


Figure 21.11: Adding a vertex

➔ To delete the vertices of a polygon or slab:

1. Choose DelVrtex from the Partial menu.
2. Select the vertex you want to delete. The vertex disappears and the two edges that led into the deleted vertex now become a single edge. Because a polygon or slab can have no less than three vertices, you cannot delete a vertex from a triangular-shaped polygon or slab.

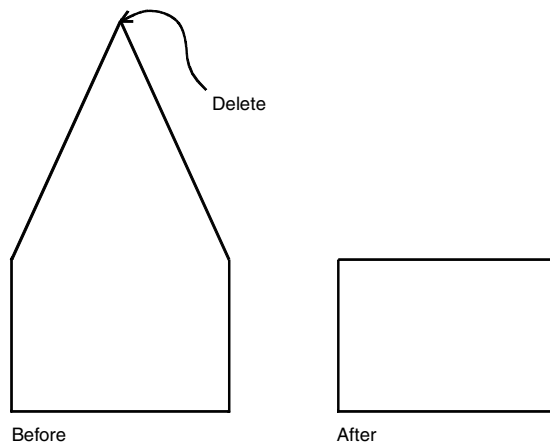


Figure 21.12: Deleting a vertex

➔ To reposition the vertex of a polygon or slab:

1. Choose MovVrtex from the Partial menu. Make sure that your current editing plane is parallel to the polygon or slab that you want to edit.
2. Select the vertex you want to move. Notice that this vertex is now attached to your cursor and the two edges of the polygon or slab that lead to this vertex become rubberband lines. You can also move a vertex with the Stretch option in the 3DEdit menu.
3. Select a new vertex position.

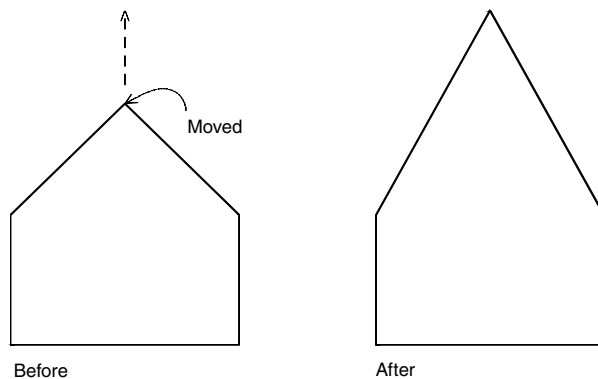


Figure 21.13: Moving a vertex

Use the MarkVis toggle to change the visibility of individual edges of a polygon or slab. Selecting a visible edge makes it invisible, while selecting an invisible edge makes it visible. Edges made invisible are not erased; they are suppressed from view, usually to hide the seams between adjacent polygons or slabs. Invisible edges remain invisible after hidden line removal, but the polygon or slab is processed as if they were visible; invisible edges can obscure objects that lie behind them.

➔ To make an edge visible or invisible:

1. Choose MarkVis from the Partial menu.
2. Select the edge of the polygon or slab you want invisible.

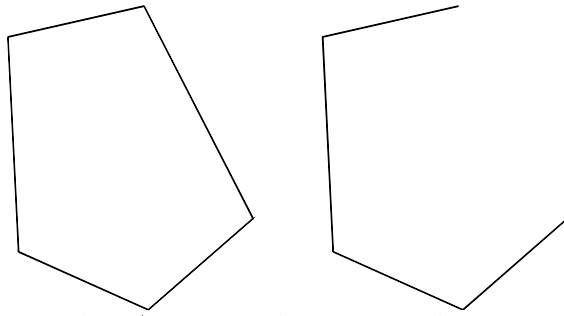


Figure 21.14: Using MarkVis on one edge

➔ To make all edges visible:

1. Choose MarkAll from the Partial menu.
2. Select the polygon or slab that you want visible. The edges of the selected polygon or slab become visible.

➔ To make all edges invisible:

1. Choose UnMrkAll from the Partial menu.
2. Select the polygon or slab that you want invisible. The edges of the selected polygon or slab become invisible.

Making a polygon or slab completely invisible with UnMrkAll has obvious problems. The best way to recover an invisible polygon or slab is to use Area or Fence from the selection menu with MarkAll in the region where you think the polygon or slab exists.

Drawing Blocks

A block is a simple solid with six rectangular or parallelogram-shaped faces. Because the top of a block is not constrained to lie directly over its bottom, blocks are particularly useful for modeling supporting members that lean to one side.

➔ To draw blocks:

1. Choose Block from the 3D Entity menu.
2. Select the shape of the base of the block: for a rectangular base, choose RectAngl; for a parallelogram base, choose Parallel.
3. Choose AutoHgt to automatically place the vertex at Z-height, centered over the center point of the base; otherwise, you must place the vertex manually.
4. Choose to place the reference face of the block with one of the following:
 - Base+Thk At Z-base, extending up through a distance equal to the thickness parameter set below
 - Hgt+Thk At Z-height, extending up through a distance equal to the thickness parameter set below
 - Base/Hgt At Z-base, extending up to Z-height

If you choose Base+Thk or Hgt+Thk, you can use Thicknss to set the thickness of the block. (Otherwise, the Thicknss setting is ignored.) As with all Z measurements, a positive thickness extends out of the screen, and a negative thickness extends into the screen. Choose Thicknss from the Blocks menu, and then choose or type a value and press **Enter**.

5. Continue with the sections below for either rectangular-based blocks or parallelogram-based blocks.

➔ To draw rectangular-based blocks:

1. Select the first corner of the block base.
2. Move the mouse diagonally, expanding the base of the block to the proper size. Select the second corner of the block base. If you selected AutoHgt, DataCAD places the top of the block directly over the bottom and the block is finished.
3. Turn AutoHgt off and select a location of the top of the block; the sides of the block are drawn, connecting the top and bottom faces.

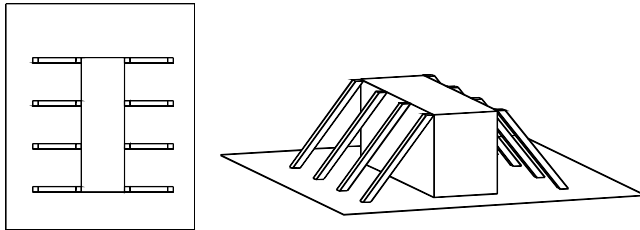


Figure 21.15: Rectangular-based blocks

➔ To draw parallelogram-based blocks:

1. Select a point to serve as one corner of the block.
2. Select a second corner that, along with the first point, forms an edge for the base of the block.
3. Select a third corner point to form the parallelogram base of the block. Notice how opposite sides of the base remain parallel but are not constrained to right angles. When you select AutoHgt, DataCAD places the top of the block directly over the bottom and the block is finished.
4. Turn AutoHgt off and select a location of the top of the block; the sides of the block lean to connect the top and bottom faces.

Drawing Single-Curved Entities

Unlike their 2D counterparts, all curved 3D entities in DataCAD are actually drawn using a number of short line segments to approximate true curves.

Some 3D entities, such as cylinders, 3D arcs, cones, and truncated cones, only curve in one direction, around the entity's center point or central axis. For these entities, you only need to set the number of primary divisions.

Entities that curve in more than one direction, such as domes and spheres, tori, surfaces of revolution, and mesh surfaces, have both primary and secondary division settings. See "Drawing Double-Curved Entities" for more information on drawing entities with both primary and secondary divisions.

In this chapter:

- ⊕ Drawing 3D arcs
- ⊕ Drawing horizontal cylinders
- ⊕ Drawing vertical cylinders
- ⊕ Drawing cones
- ⊕ Drawing truncated cones

Drawing 3D Arcs

3D arcs are essentially the same as their 2D counterparts, with the following exceptions:

- 3D arcs can exist at any orientation in space;
- 3D arcs can be specified as open or closed;
- 3D arcs have no extrusion value
- 3D arcs are approximations of arcs consisting of many short line segments called divisions.

The first eight options on the 3DArc menu are quite common and are seen wherever there is a curved 3D entity. Creating this 3D arc footprint is often the first step in creating a curved 3D entity. This footprint defines the sweep angles for the entity.

3D two- and three-point arcs and radius, diameter, and three-point circles are drawn exactly as their 2D counterparts. Use ClkWise, available only with 2PtArc, to create an arc in the clockwise direction rather than counterclockwise. Use Closed to close the arc and perform a hidden line removal. See “Drawing Curves” in the “Drawing Other Geometry” chapter for more information about arcs and circles.

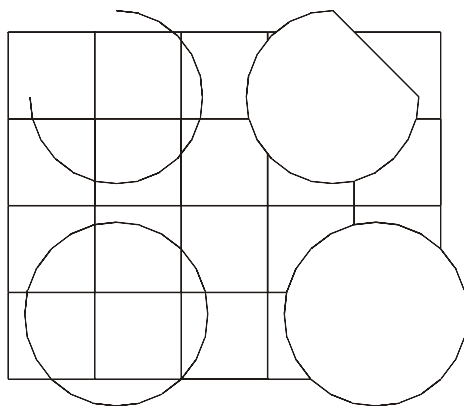


Figure 22.1: Open and closed arcs and circles

Use Tangent to draw arcs that are tangent to a specified line. This method is slightly different from the way tangent arcs are created in 2D.

➔ To draw a 3D tangent arc:

1. Choose Tangent from the 3DArc menu.
2. Select the first point of the tangent line. This is the line that the 3D arc draws to.
3. Select the beginning point of the arc.
4. Select the end point of the arc.

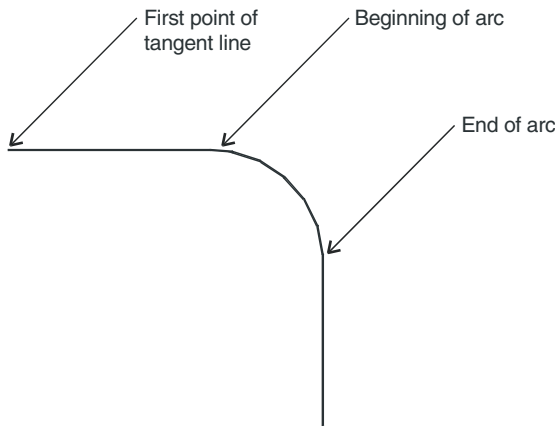


Figure 22.2: Drawing a tangent arc

A dog-leg arc is defined by the two tangent points of the arc and the point at which those two tangent lines cross (the knee point). You can use DogLeg to create an arc that must be tangent to existing lines.

➔ To draw a dog-leg arc:

1. Choose DogLeg from the 3DArc menu.
2. Select the first point of tangency, or the first point of the arc.
3. Select the knee point, the point where the two lines of tangency meet.
4. Select the second point of tangency, or the end of the arc.

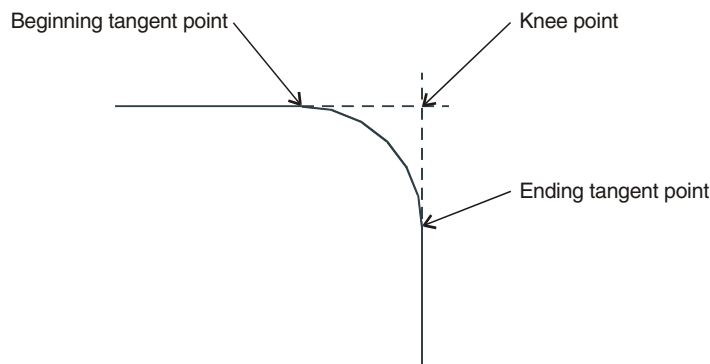


Figure 22.3: Drawing a dog-leg arc

You can enter the arc at the current elevation of Z-base or Z-height. Choose Z-base or Z-Hgt, respectively, and draw the arc.

➔ To create 3D arcs that curve out of the screen:

1. Choose Vertical from the 3DArc menu.
2. Select the center point of the arc.
3. Select a starting point for the arc. A list of values appears.
4. Choose or type a value and press **Enter**.
5. Type the angle of elevation for the first end point and press **Enter**.
6. Type the angle of elevation for the second end point and press **Enter**.

➔ To set the number of divisions for 3D arcs:

1. Choose Divisions from the 3DArc menu. A list of values appears.
2. Choose or type a division value and press **Enter**.

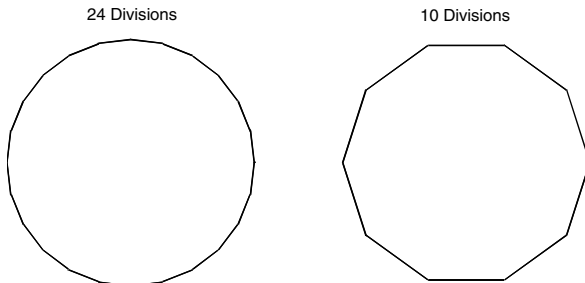


Figure 22.4: Divisions examples

Drawing Vertical Cylinders

Vertical cylinders are drawn as columns or posts standing on end. The diameter of the column is dynamically specified by drawing the footprint of the cylinder using one of the standard 3D arcs. The height of the column is controlled by setting Z-base, Z-height, and thickness.

➔ To draw a vertical cylinder:

1. Choose VertCyl from the 3DEntity menu.
2. Select one of these options: 2PtArc, 3PtArc, Tangent, DogLeg, RadCirc, DiaCirc, and 3PtCirc. These options are identical to those in the 3DArc menu. Choose one of these options to draw the footprint of the cylinder. A *footprint* is the shape of the base of the cylinder. To draw a full cylinder, for instance, choose RadCirc, DiaCirc, or 3PtCirc to draw a circular footprint.
3. Choose whether to draw the cylinder from Z-base to the thickness value (Base+Thk), from Z-height to the thickness value (Hgt+Thk), or from Z-base to Z-height (Base/Hgt).
4. Use Closed to create a closed cylinder for hidden line removals. The cylinder is processed as a solid element with Closed on.
5. Follow the prompts in drawing the footprint of the cylinder. For more information on using 2PtArc, 3PtArc, Tangent, DogLeg, RadCirc, DiaCirc, 3PtCirc and ClkWise, see “3D Arcs” earlier in this chapter.

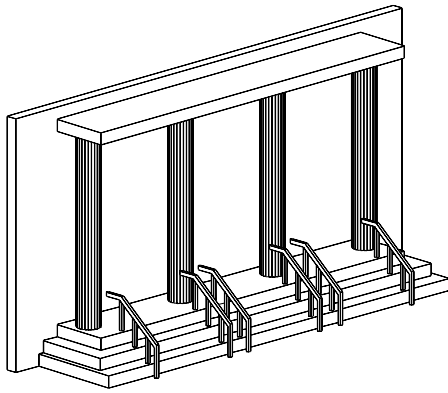


Figure 22.5: Vertical columns

Set Thickness when you draw cylinders by Base+Thk or Hgt+Thk; otherwise the Thickness value is ignored. As is true for all Z measurements, positive thickness extends out of the screen, while negative thickness extends into the screen.

➔ To set a thickness:

1. Choose Thickness from the VertCyl menu. A list of values appears.
2. Choose or type a value and press **Enter**.

➔ To set the number of divisions for the cylinders:

1. Choose Divisions from the VertCyl menu. A list of values appears. This setting is the same as PrimDivs in the 3D Settings menu.
2. Choose or type a value and press **Enter**.

The 3DViews option is a shortcut to the 3D Views menu. See the “Viewing Your Model” chapter for more information about the 3Dviews menu.

Drawing Horizontal Cylinders

Horizontal cylinders draw flat across the screen.

➔ To draw a horizontal cylinder:

1. Choose HorizCyl from the 3DEntity menu.
2. Select a radius for the cylinder.
3. Choose the extent of the sweep angles with StartAng and EndAngle, or select one of the four presets: FullCirc, TopHalf, BotHalf, or SideHalf.

For more control over the portion of the cylinder that appears, you can:

- reorient the view to draw a vertical cylinder and use the 3DArc menu to set the portion of the cylinder needed
 - stay with horizontal cylinders and explicitly specify sweep angles for the cylinder
4. Select two points to represent the end points of the cylinder.

To create a cylinder section that covers from the 10:00 position to the 2:00 position on a clock, decide which way you want to draw the cylinder across the screen. You can draw it from top to bottom or bottom to top. When drawing from top to bottom, remember that cylinders are drawn counterclockwise.

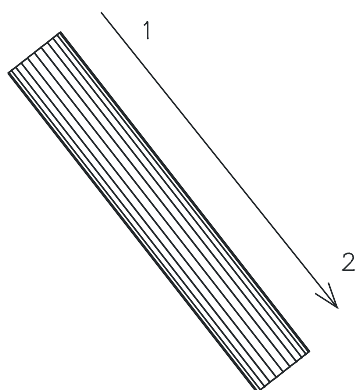


Figure 22.6: Horizontal cylinder

➔ To specify the beginning angle of the cylinder sweep angle:

1. Choose StartAng from the HorizCyl menu. A list of beginning sweep angles appears.
2. Choose or type a beginning sweep angle and press **Enter**.

➔ To specify the ending sweep angle of the cylinder sweep angle:

1. Choose EndAngle from the HorizCyl menu. A list of ending sweep angles appears.
2. Choose or type an ending sweep angle and press **Enter**.

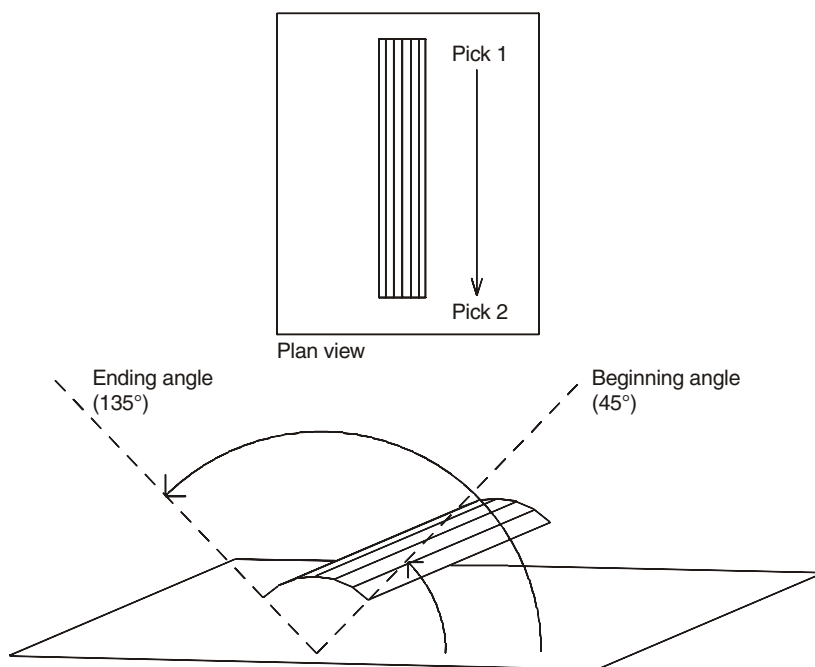


Figure 22.7: Computing sweep angles

- ➔ To control the radius of the cylinder:
 1. Choose Radius from the HorizCyl menu. A list of radius values appears.
 2. Choose or type a radius value and press **[Enter]**.
- ➔ To draw a full and complete cylinder:
 1. Choose FullCirc from the HorizCyl menu.
 2. Select the center point of one end of the cylinder.
 3. Select the center point of the other end of the cylinder.
- ➔ To draw the top half of a cylinder using a preset pair of sweep angles (much like a speed bump):
 1. Choose TopHalf from the HorizCyl menu.
 2. Select the center point of one end of the cylinder.
 3. Select the center point of the other end of the cylinder.
- ➔ To draw the bottom half of a cylinder using a preset pair of sweep angles (much like a trough):
 1. Choose BotHalf from the HorizCyl menu.
 2. Select the center point of one end of the cylinder.
 3. Select the center point of the other end of the cylinder.
- ➔ To draw half of a cylinder using a preset pair of sweep angles.
 1. Choose SideHalf from the HorizCyl menu.
 2. Select the center point of one end of the cylinder.
 3. Select the center point of the other end of the cylinder. When the two points used to define the cylinder are drawn from the top to the bottom of the screen, the resulting half-cylinder appears on the left.

Use Closed to create a closed cylinder for hidden line removals. The cylinder is processed as a solid element when Closed is on.

Use the Z-base toggle to set the centerline of the cylinder to lie at Z-base. If you draw a full circle cylinder, half of the cylinder lies above Z-base and half lies below. Similarly, the Z-Hgt toggle sets the centerline of the cylinder to lie at Z-height. If you draw a full circle cylinder, half of the cylinder lies above Z-height and half lies below.

- ➔ To set the number of divisions for the cylinders:
 1. Choose Divisons from the HorizCyl menu. A list of values appears. This setting is the same as PrimDivs in the 3DEdit/Settings menu.
 2. Choose or type a value for the number of cylinder divisions and press **[Enter]**.

3DViews accesses the 3DViews menu. See the “Viewing Your Model” chapter for more information about the 3Dviews menu.

Drawing Cones and Truncated Cones

Cones are entities that are not used often in architectural modeling, but they are sometimes seen in spires and pylons. You can create cones by drawing the base of the cone (at Z-base) using the standard 3D Arc menu and placing the vertex of the cone at Z-height. DataCAD automatically places the vertex, thus creating a right cylinder, or you can place the vertex yourself, creating a skewed cylinder.

➔ To draw a cone:

1. Choose Cone from the 3DEntity menu.
2. Choose AutoHgt to automatically place the vertex of the cone at Z-Hgt, centered over the center point of the base. Otherwise, you will have to manually place the vertex.
3. Use Divisions to set the number of divisions for the cone. (This setting is the same as the PrimDivs setting of the 3DEdit/Settings menu.) To set the divisions of a cone, choose Divisions from the Cone menu, and then choose or type a value for the number of divisions and press **[Enter]**.
4. Toggle Closed on to close the bottom of the cone, creating a solid base. This base won't be visible unless you perform a hidden line removal or shade the image.
5. Draw the footprint (sweep angle) of the cone. The base of the cone is always drawn at Z-base and the vertex is always drawn at Z-height. See "Drawing 3D Arcs" earlier in this chapter for more information about drawing the footprints of the cone or on using 2PtArc, 3PtArc, Tangent, DogLeg, RadCirc, DiaCirc, 3PtCirc, ClkWise and Closed.
6. Place the vertex of the cone if you toggled AutoHgt off. If you place the vertex to one side, you create a skewed cone.

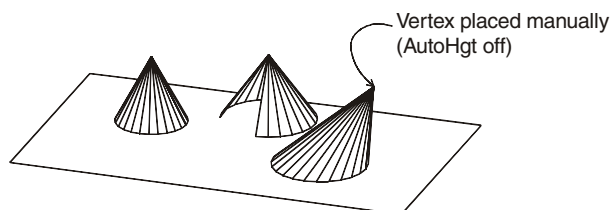


Figure 22.8: Cones

See "Drawing 3D Arcs" earlier in this chapter for more information about drawing the footprints of the cone or on using 2PtArc, 3PtArc, Tangent, DogLeg, RadCirc, DiaCirc, 3PtCirc, ClkWise and Closed.

Truncated cones have sheared off tops. Creating a truncated cone is similar to creating a regular cone, except you must also specify the diameter of the top opening of the truncated cone.

➔ To draw a truncated cone:

1. Choose TrunCone from the 3DEntity menu.
2. Set the Z-base and Z-height appropriately. As with regular cones, you draw truncated cones from Z-base to Z-height.

3. Toggle Closed on to close the bottom and top of the cone, creating a solid base and top. This solid base and top won't be visible unless you perform a hidden line removal or shade the image.
4. Choose the footprint (sweep angle) of the truncated cone by drawing one of the 3D arcs.
5. Determine whether AutoHgt is toggled on or off. If AutoHgt is toggled on, DataCAD prompts for the diameter of the top part of the truncated cone. If AutoHgt is toggled off, DataCAD prompts you for the center point of the top circle, and then the radius.

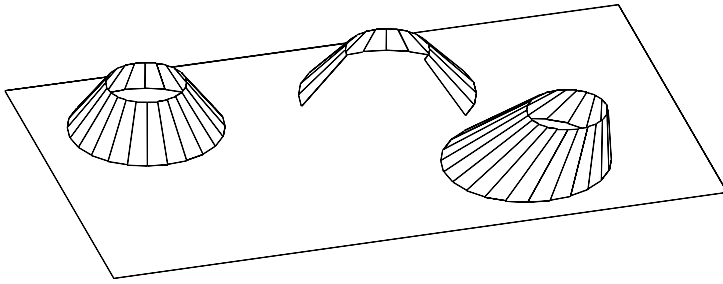


Figure 22.9: Truncated cones

The 3DViews option is a shortcut to the 3D Views menu. See the “Viewing Your Model” chapter for more information about the 3Dviews menu.

Drawing Double-Curved Entities

In this chapter:

Unlike their 2D counterparts, all curved 3D entities in DataCAD are actually drawn using a number of short line segments to approximate true curves.

Some 3D entities, such as cylinders, 3D arcs, cones, and truncated cones, only curve in one direction, around the entity's center point or central axis. See "Drawing Single-Curved Entities" for more information on drawing entities with only primary divisions.

Entities that curve in more than one direction, such as domes and spheres, tori, surfaces of revolution, and mesh surfaces, have both primary and secondary division settings. *Primary divisions* are the number of curve divisions in the plane of the screen, also known as the sweep angle. *Secondary divisions* are those divisions that affect curves that bulge into or out of the plane of the screen; these are known as rise or roll angles.

- ⊕ Drawing domes and spheres
- ⊕ Drawing tori
- ⊕ Drawing surfaces of revolution
- ⊕ Drawing mesh surfaces

Drawing Spheres and Domes

Spheres are any full sphere or section of a sphere. The menu combines features from both the vertical and horizontal cylinder menus as well as the 3D arcs menu. See “Drawing 3D Arcs,” “Drawing Vertical Cylinders,” and “Drawing Horizontal Cylinders” in the “Drawing Single-Curved Entities” chapter for more information.

➔ To draw a sphere:

1. Choose Sphere from the 3DEntity menu.
2. Choose a 3DArc option from the top part of the menu: 2PtArc, 3PtArc, Tangent, DogLeg, RadCirc, DiaCirc, 3PtCirc, or ClkWise to enter the footprint (sweep angles) of the sphere. See “Drawing 3D Arcs” in the “Drawing Single-Curved Entities” chapter for more information on using 2PtArc, 3PtArc, Tangent, DogLeg, RadCirc, DiaCirc, 3PtCirc, and ClkWise.
3. Set the rise angles with StartAng or EndAngle or use one of the presets: FullCirc, TopHalf, or BotHalf. This determines exactly how much of the sphere curves out of the plane of the screen.
4. Set the Z-base or Z-height properly to set the location of the sphere. The sphere is drawn with its center point at either Z-base or Z-height, depending on which is toggled on.
5. Draw the footprint with 3DArc options. DataCAD generates the dome from the footprint and the rise angles you supplied.

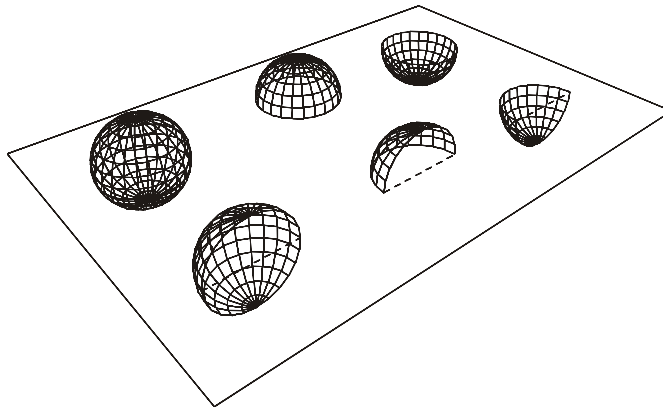


Figure 23.1: Spheres

For more control over the portion of the sphere that appears, you can specify rise angles. A *rise angle* describes the angle by which something tilts out of the plane of the screen. Something that lies exactly in the plane of the screen has a rise angle of 0°; something that extends straight out of the screen has a rise angle of 90°.

➔ To set the beginning rise angle for the sphere:

1. Choose StartAng from the Sphere menu. A list of beginning rise angles appears.
2. Choose or type a beginning rise angle and press .

➔ To set the ending rise angle for the sphere:

1. Choose EndAngle from the Sphere menu. A list of ending rise angles appears.
2. Choose or type an ending rise angle and press **[Enter]**. When setting the rise angles for the beginning and end of the sphere, remember that spheres are drawn counterclockwise.

Use FullCirc to create a sphere section encompassing both the north and south hemispheres. Use TopHalf to create a sphere section with just the northern hemisphere. Use BotHalf to create a sphere section with just the southern hemisphere. Use Z-Base to set the center of the sphere at the Z-base. Use Z-Hgt to set the center of the sphere at the Z-height.

➔ To set the primary (sweep) divisions of the dome:

1. Choose PrimDivs from the Sphere menu. A list of values appears.
2. Choose or type a value for the number of primary divisions and press **[Enter]**.

➔ To set the secondary (roll) divisions of the dome:

1. Choose StartAng from the HorizCyl menu. A list of values appears.
2. Choose or type a value for the number of secondary divisions and press **[Enter]**.

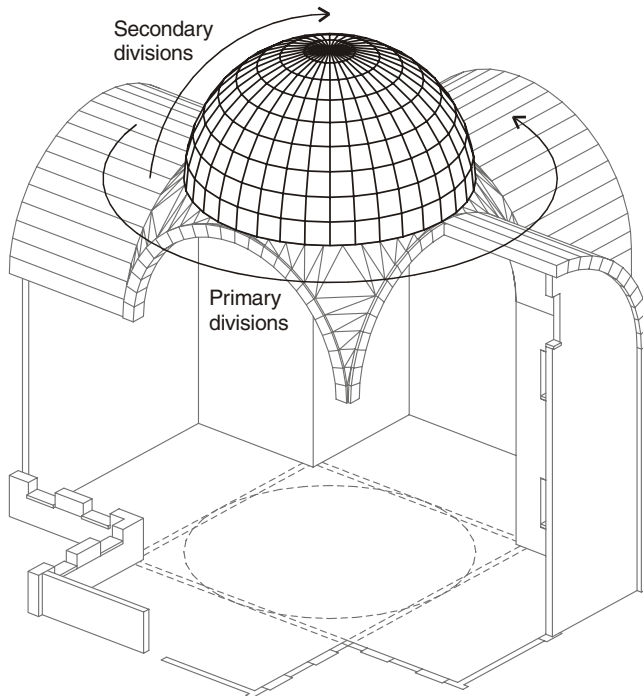


Figure 23.2: Primary and secondary divisions

Drawing Tori

A *torus* is a donut-shaped entity. Full tori are obviously very rare in architectural modeling, but sections of tori are fairly common. Typical applications include turns in railings, elbow connectors in pipes, and even children's playground slides.

➔ To draw a torus:

1. Choose Torus from the 3DEntity menu.
2. Choose a 3DArc option from the top of the menu: 2PtArc, 3PtArc, Tangent, DogLeg, RadCirc, DiaCirc, 3PtCirc, ClkWise, or Radius for creating the footprint (sweep angles) of the torus. See "Drawing 3D Arcs" in the "Drawing Single-Curved Entities" chapter for more information on using 2PtArc, 3PtArc, Tangent, DogLeg, RadCirc, DiaCirc, 3PtCirc, and ClkWise.
3. Set the roll angles for the tube part of the torus with StartAng or EndAngle or use one of the preset angles FullCirc, TopHalf, BotHalf, InHalf, or OutHalf.
4. Choose Z-base or Z-height. Center the torus at either the Z-base or the Z-height, respectively.
5. Draw the 3DArc footprint of the torus. The torus (or toroidal section) is calculated from the footprint and the roll angles.

For more control over how far the torus sweeps out of the plane of the sphere, you can specify roll angles. A *roll angle* describes the angle by which something tilts out of the plane of the screen. Something that lies exactly in the plane of the screen has a roll angle of 0°; something that comes straight out of the screen has a roll angle of 90°.

➔ To set the beginning roll angle for the torus:

1. Choose StartAng from the Torus menu. A list of beginning roll angle values appears.
2. Choose or type a beginning roll angle and press Enter.

➔ To set the ending roll angle for the torus:

1. Choose EndAngle from the Torus menu. A list of ending roll angle values appears.
2. Choose or type an ending roll angle and press Enter. When setting the roll angles, remember that tori are drawn counterclockwise as seen in a front elevation.

You can also use these preset options to set the most common roll angles:

FullCirc	Creates a full inner-tube torus section
TopHalf	Creates a torus section using only the top part of the torus roll
BotHalf	Creates a torus section using only the bottom part of the torus roll
InHalf	Creates a torus using only the inside part of the torus roll
OutHalf	Creates a torus using only the outside part of the torus roll

- ➔ To indicate the radius of the tube section of the torus (not the radius of the torus itself):
 1. Choose Radius from the Torus menu. A list of radius values appears.
 2. Choose or type a radius value and press **Enter**.
- ➔ To set the primary (planar) divisions of the torus:
 1. Choose PrimDivs from the Torus menu. A list of values appears.
 2. Choose or type a value for the number of primary divisions and press **Enter**.
- ➔ To set the secondary (roll) divisions of the torus:
 1. Choose ScndDivs from the Torus menu. A list of values appears.
 2. Choose or type a value for the number of secondary divisions and press **Enter**.

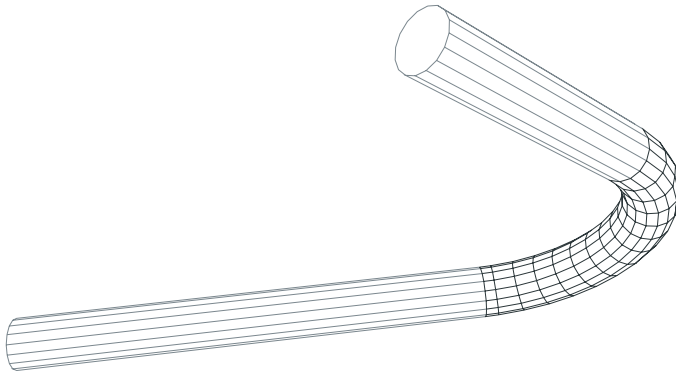


Figure 23.3: Torus connecting two cylinders

Drawing Surfaces of Revolution

Use RevSurf to create 3D solids with radial symmetry. Columns, lampshades, fountains, and canopies are all examples of objects with radial symmetry that you can model using RevSurf.

Surfaces of revolution are so named because of the way you construct them: a polyline profile revolved around an axis of revolution. As the profile revolves around a given axis, it “sweeps out” a surface in its wake. This surface is the surface of revolution. To create any surface of revolution, you must determine what profile and axis combination generates the surface you want.

Surfaces of revolution are single entities. To modify or edit a surface of revolution, you can explode it and stretch it, or enlarge it to create complex shapes. If you’re trying to model a shape that doesn’t have radial symmetry, but is close, try starting with a surface of revolution. Then through enlargements, explosions, and stretches, shape the solid into the form you need.

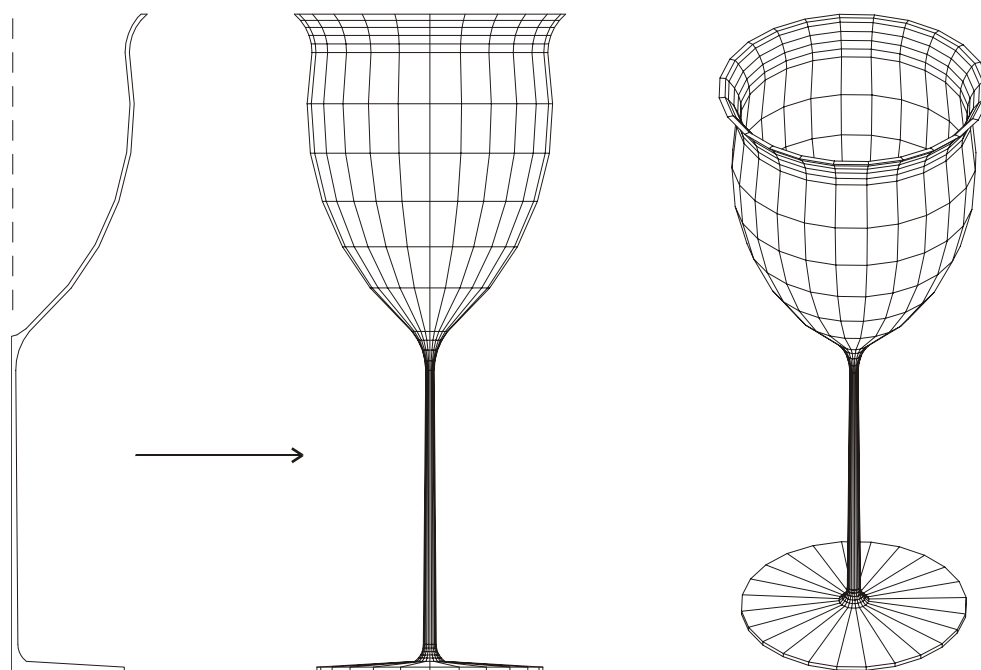


Figure 23.4: Surface of revolution

➔ To draw a surface of revolution:

1. Choose RevSurf from the 3DEntity menu.
2. Draw or select the profile to rotate. Create the cross section of the object as a polyline. If you already have a polyline, you can use Select to select the polyline. You can use Backup and Cancel to create polylines.
3. Choose Exit. The rise angle options are displayed.
4. Set the rise angles for the surface of revolution:

StartAng	Sets the beginning rise angle on the surface of revolution
EndAngle	Sets the ending rise angle on the surface of revolution
FullCirc	Sets the rise angle to rotate the profile 360° around, thus creating a full solid
TopHalf	Sets the rise angle to rotate the profile 180° out of the screen, creating just the top half of a full surface of revolution
BotHalf	Sets the rise angle to rotate the profile 180° into of the screen, creating just the bottom half of a full surface of revolution
SideHalf	Sets the rise angle to rotate the profile 180° into and out of the screen, on whatever side the contour appears, thus creating half of a full surface of revolution

5. Choose a surface of revolution:

Open	With open ends and sides; no surfaces are generated for these parts of the surface of revolution
ClosSid	With closed sides; <i>sides</i> are inside surfaces, the part of the surface of revolution you see if you stand on the axis of revolution and look outward toward the profile, or if you stand outside the surface and look inward toward the axis
ClosEnd	With closed ends; <i>ends</i> refer to the top and bottom, the part of the surface of revolution you see if you look along the axis of revolution

6. Choose PrimDivs to set the number of circular divisions along the roll of the surface of revolution (the surfaces which curve out of or into the plane of the screen, which rotate around the X or Y screen axes. Choose PrimDivs from the RevSurf menu, and then choose or type a value for the number circular divisions along the roll and press **Enter**.
7. Choose ScndDivs to set the number of circular divisions along the sweep of the surface of revolution (the surfaces that show curvature in the plane of the screen, which rotate around the Z screen axis). Choose ScndDivs from the RevSurf menu, and then choose or type a value for the number of circular divisions along the sweep and press **Enter**.
8. Choose Z-Base or Z-Hgt to create a surface of revolution whose axis of revolution is at the Z-base or Z-height, respectively.
9. Choose Exit when you are done with the polyline cross section.

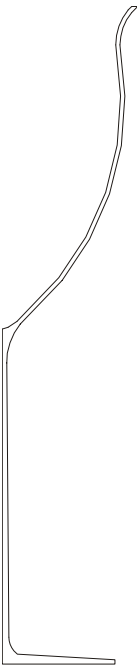


Figure 23.5: Polyline cross section

10. Select two points for the axis of revolution. The action here is to spin the polyline profile around the axis of revolution, sweeping out a solid in the process.

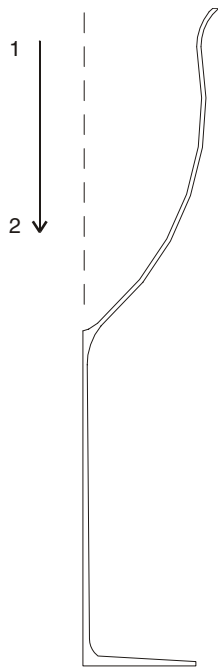


Figure 23.6: Selecting the axis of revolution

Drawing Mesh Surfaces

A *mesh surface* is a smooth patch that you can mold into a variety of useful shapes including free-form built structures, tents, and canopies. Mesh surfaces are the surface equivalents of Bezier curves.

Like contours, mesh surfaces are created through a series of control points that you specify. The greater the variation in the elevation of these control points, the more dramatic the curves in the surface.

There are always exactly 16 control points to specify for a mesh surface, and these 16 control points always form a 4x4 grid of points, although the spacing of these points may not be uniform.

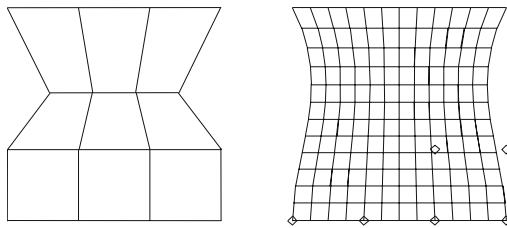


Figure 23.7: Mesh surfaces

There are three ways to create a mesh surface:

- Specify all 16 control points individually, as four rows, with four points per row. The first control point is numbered [1,1], meaning row 1, column 1 in the grid of points. The second control point is numbered [1,2], then [1,3] followed by [1,4], then [2,1], etc. You can change the Z-base or Z-height at any time during the entry of control points, thus creating a surface that curves into and out of the screen. The resulting surface is a Bezier-like curved sheet that curves into and out of the screen.

- Draw a rubberband box that forms the outer boundary of the mesh surface. The 16 control points are equally spaced in this box. The resulting surface is a flat sheet.
- Place four points in the drawing that represent the four rows of a rectangular sheet surface that go into the screen.

➔ To manually draw a mesh surface:

1. Choose MeshSurf from the 3DEntity menu.
2. Select the Z elevation of the first control point [1,1].
3. Select the first control point.
4. Repeat steps 2 and 3 until you enter all 16 control points.

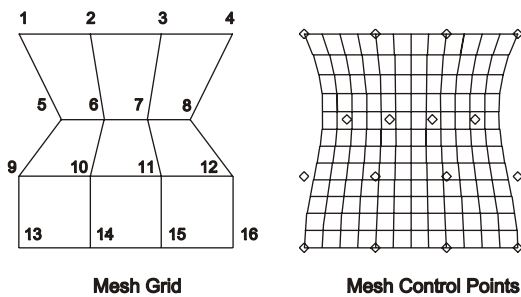


Figure 23.8: Mesh grid and its corresponding control points

➔ To create a rectangular mesh surface:

1. Choose RectSurf from the MeshSurf menu.
2. Select one corner of a rubberband box to contain the mesh surface.
3. Select the opposite corner of the rectangular region. The 16 control points are automatically placed within the boundary rectangle, all at the same elevation and an equal distance apart. To create a rolling surface, take an elevation view and use the stretch command to move the control points.

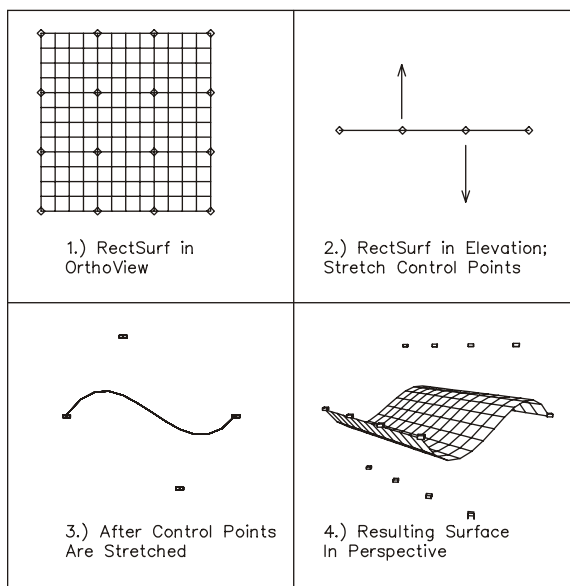


Figure 23.9: Rectangular mesh surface

➔ To create a vertical mesh surface from Z-base to Z-height:

1. Choose VertSurf from the MeshSurf menu.
2. Select four points across the screen. DataCAD generates a mesh surface that runs from Z-base to Z-height that fits the control points. For each control point you select on the screen, DataCAD generates four points: one at Z-base at that location, one at Z-height, and two more, equally spaced between these extremes.

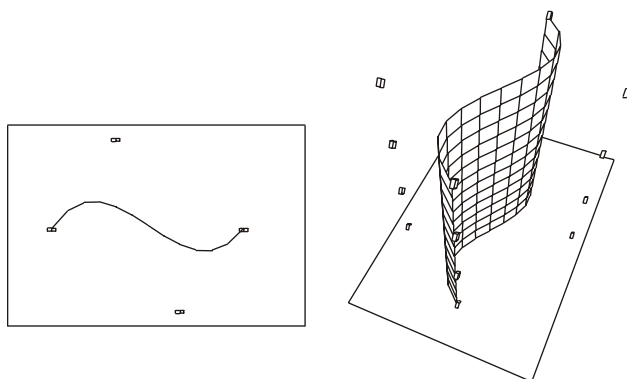


Figure 23.10: Vertical mesh surface

➔ To set the number of columns (divisions in the surface in direction 1):

1. Choose DivsDir1 from the MeshSurf menu. A list of surface division values appears.
2. Choose or type a value for the number of surface divisions in Direction 1 and press **Enter**.

➔ To set the number of rows (divisions in the surface in direction 2):

1. Choose DivsDir2 from the MeshSurf menu. A list of surface division values appears.
2. Choose or type a value for the number of surface division in Direction 2 and press **Enter**.

Use DispPts to display the control points for the surface. When you edit a mesh surface, toggle DispPts on. Toggle DispPts off for presentations.

Use DispGrid to connect the surface's control points with lines. Toggle DispGrid off for presentations.

As with Contours, you can specify an increment distance to add or subtract to the base elevation as you enter control points. SetIndx sets the index and AddIndx and SubIndx determine whether you add to or subtract from the current Z elevation value. See "Drawing Topographical Maps with Contours" earlier in the "Site Modeling" chapter for more information about contours.

➔ To specify and increment distance:

1. Choose SetIndx from the MeshSurf menu. A list of distance values appears.
2. Choose or type an increment distance and press **Enter**.

These are the four standard elevations at which you can enter a mesh surface control point. Use all four to create a mesh surface with varying Z elevations. Use AddIndx or SubIndx to change these values.

➔ To set the values for Z-User1 and Z-User2:

1. Choose one of these elevation options from the MeshSurf menu. A list of elevation values appears.
2. Choose or type an elevation value and press .

Site Modeling

DataCAD has a tool you can use for site modeling: the Drop Mesh tool. The Drop Mesh tool creates a triangulated regular network (TRN) of 3D polygons and is an averaged representation of data points.

Drop Mesh produces a "clean," rectilinear result, which may be desirable when accuracy is less of a concern than the visual representation.

Contours are curved 3D lines drawn through points that you enter. These are particularly useful for drawing topographical maps.

As with most tools in DataCAD, the Drop Mesh can be used to create not only site models but also any surface in your model. Contours can be used in other drawing situations as well.

In this chapter:

- ⊕ Using the drop mesh
- ⊕ Using contours to create topographical maps

Site Modeling with Drop Mesh

One additional option is displayed in the 3DEntity/Polygon menu. DropMesh is used to create a regular, triangulated, polygon mesh that lays over selected geometry or points, which can be created from a survey data file. This mesh can then be shaded or processed through a hidden line removal to provide a model of the building site.

A drop mesh is a group of triangulated polygons, not a single entity; thus, it can increase your drawing file size substantially. A mesh with 100 X divisions and 100 Y divisions will add 20,000 polygons to your drawing (X divisions x Y divisions x 2 polygons per grid division, or $100 \times 100 \times 2$), increasing your drawing file by 3MB. Additionally, since entities are not deleted from the drawing database even after you delete them from the drawing itself, your drawing file size can increase rapidly if you add a drop mesh, delete it, add another one, and so on. To remove unused entities from the drawing database, use Layer Utility in the Edit/Toolbox menu. It is recommended that you only use as many divisions as is necessary for a reasonably accurate model.

A drop mesh acts like a cloth dropped over the geometry. The way it drapes over the geometry can be adjusted with the Div-X, Div-Y, Stiffnss, Smooth, and SmthPass options. The mesh is made up of a grid of polygon divisions, with lines running in both the X and Y directions. These divisions are set with the Div-X and Div-Y options, and increasing these values adds flexibility to the mesh. Smooth adds points between the geometry to flatten the surface. SmthPass increases the number of smoothing passes. Stiffnss controls how closely the mesh coincides with the actual geometry; in other words, the lower the stiffness setting the more tightly it fits to the geometry.

➔ To drop a mesh:

1. Choose DropMesh from the 3DEntity/Polygon menu. Notice that the current settings are displayed in the Message Window.
2. Choose Div-X and Div-Y to set the number of X and Y divisions in the mesh grid.
3. Choose Stiffnss to set the stiffness of the mesh. Enter a number from 1 through 10, where 1 is the stiffest, and press **[Enter]**. The default setting is 5.
4. Choose Smooth to toggle on smoothing. This setting will have no effect on points created from an imported survey data file; it only works with vector geometry. Use of this option increases the time it takes DataCAD to generate the mesh.
5. Determine if Smooth is toggled on. If Smooth is toggled on, the SmthPass option becomes available on the DropMesh menu. Choose SmthPass to set the number of smoothing passes. Enter a number from 1 through 10, where 1 is the minimum number of passes. 1 is the default setting.
6. Toggle DropEdge on to model the site edges. When DropEdge is toggled on, EdgeHgt becomes available on the menu. Choose EdgeHgt to set the distance below the lowest, calculated point of the site model. The default edge height is 10 feet.

7. Toggle ShowDiag on if you want to display the diagonal polygon edges of the mesh.
8. Drop a mesh over either selected geometry or imported, survey points. To drop a mesh over existing geometry, use the selection menu at the top of the DropMesh menu to select geometry in your drawing. You can select all of the geometry or only a portion of it. When selection is complete, continue with step 10. You can choose Clear at any time to cancel your selections. To use survey data instead, skip to step 9. You can drop a mesh over any geometry in the drawing except geometric primitives, such as spheres, cones, or cylinders. To drop a mesh over geometric primitives, you must first explode the entities.
9. Make sure survey data is in a text file formatted in columns. You must use one of four data formats, including X-Y-Z, Y-X-Z, Angle-Distance-Height, and Distance-Angle-Height. To import one of these files, first choose ImprtScl from the DropMesh menu to set the scale for the data of the imported file. Then choose Import from the DropMesh menu, select a data format from the Files of Type drop-down box, and select a file to import and click Open. Points based on the survey data are displayed in the Drawing Window.
10. Choose Begin from the DropMesh menu to begin drawing the drop mesh. Press End to cancel the mesh at any time.

The following series of figures demonstrate the effect that the divisions, stiffness, and smoothing options have on a drop mesh.

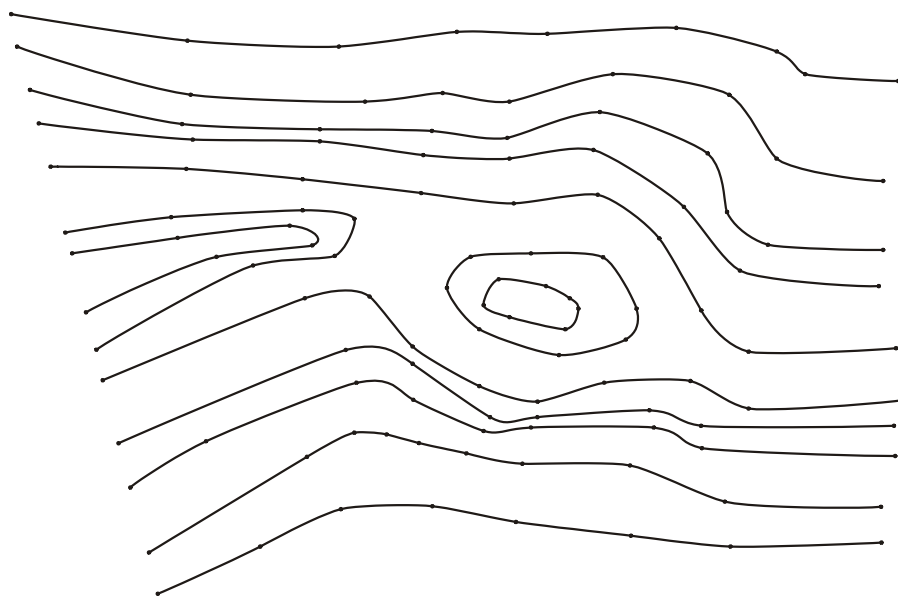


Figure 24.1: Site plan drawn using contours

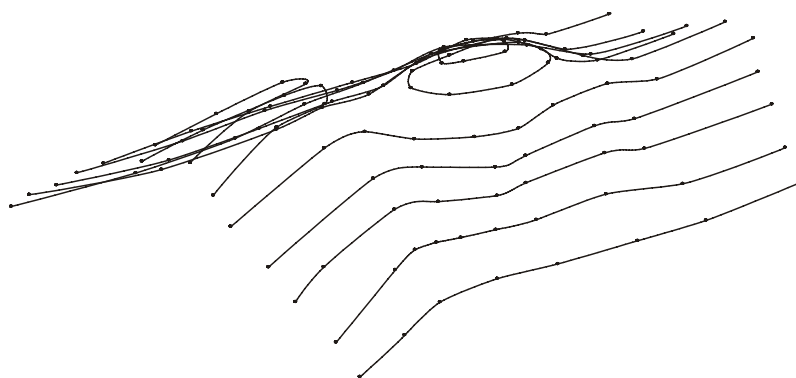


Figure 24.2: Isometric view of contour line drawing.

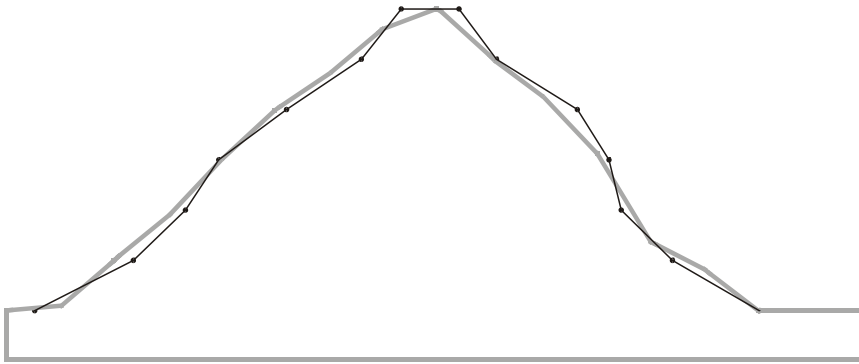


Figure 24.3A: Model using 16 x 30 divisions, a stiffness setting of 5, and Smooth toggled off. The isometric view to the right shows the results of these drop mesh settings. As you can see above, the mesh (represented by the gray line) does not always meet the points on the contour lines (represented by the darker line and dots). The result is a rougher approximation of the contour geometry.

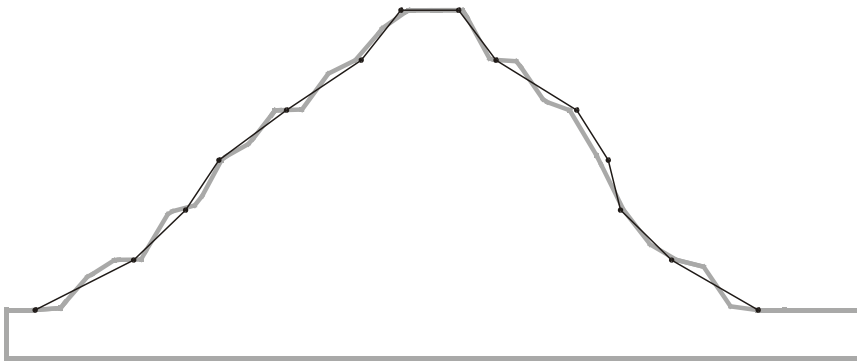


Figure 24.4A: Divisions are increased to 32 x 60, while the stiffness and smooth settings remain the same. A higher resolution mesh produces a stair-step effect with this stiffness setting.

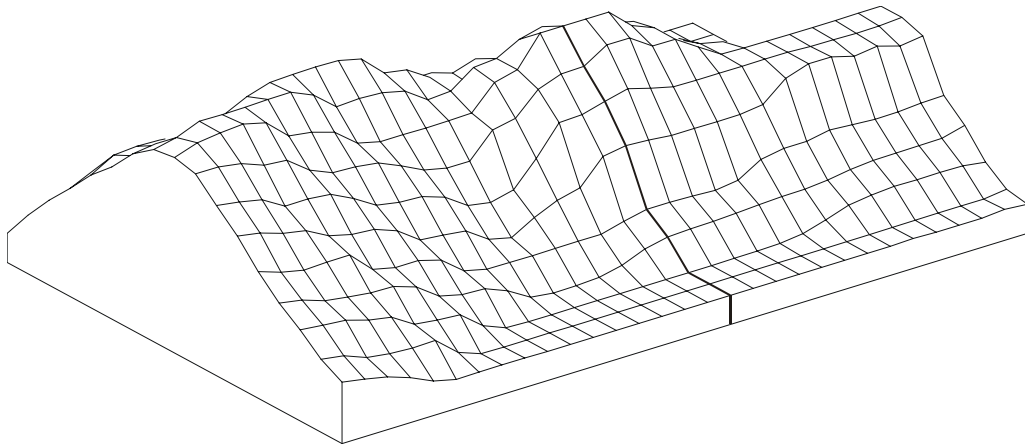


Figure 24.3B: Isometric view of model using 16 x 30 divisions with stiffness set to 5 and Smooth toggled off.

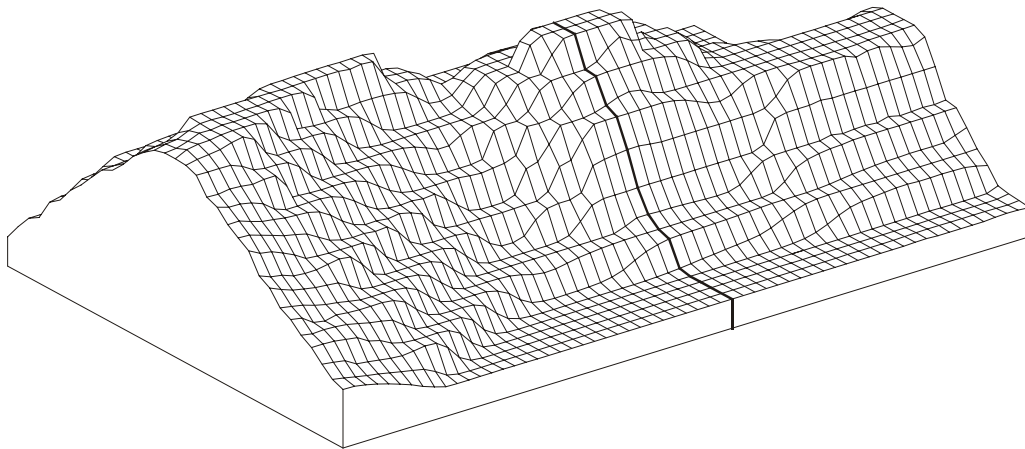


Figure 24.4B: Isometric view of model with divisions increased to 32 x 60.

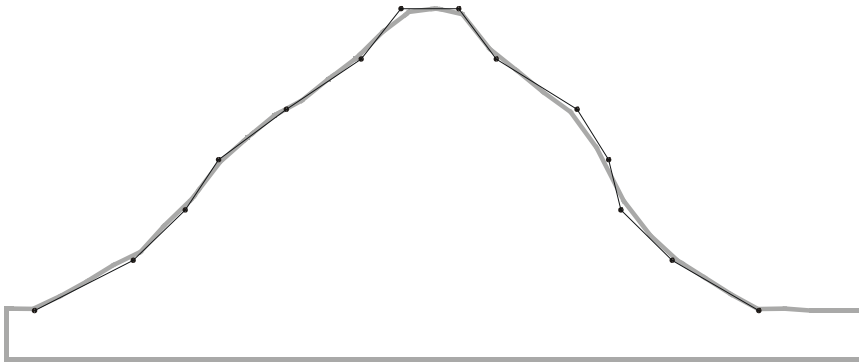


Figure 24.5A: Divisions remain at 32 x 60 with Smooth toggled off, but stiffness is increased to 9. The stiffness is adjusted to counter the stair-stepping effect, resulting in a closer relationship to the contour points.

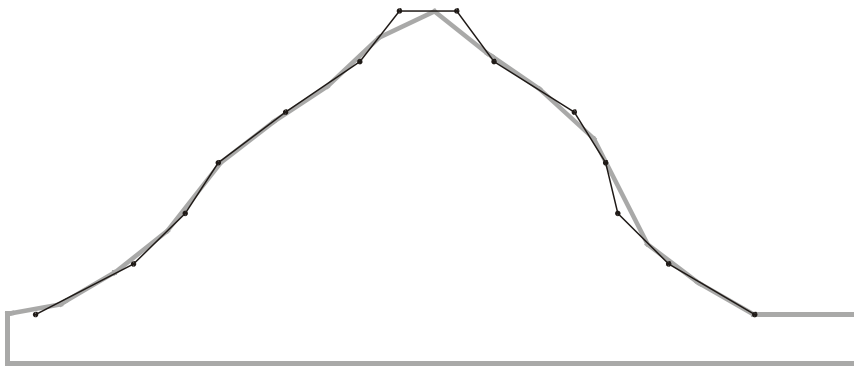


Figure 24.6A: Another way to smooth the drop mesh is to use the Smooth option. Here the divisions and stiffness have been reduced to their original values of 16 x 30 and 5, respectively. However, Smooth has been toggled on and SmthPass has been set to 5. The interpolation of the smoothing operation creates smoother transitions between contour points.

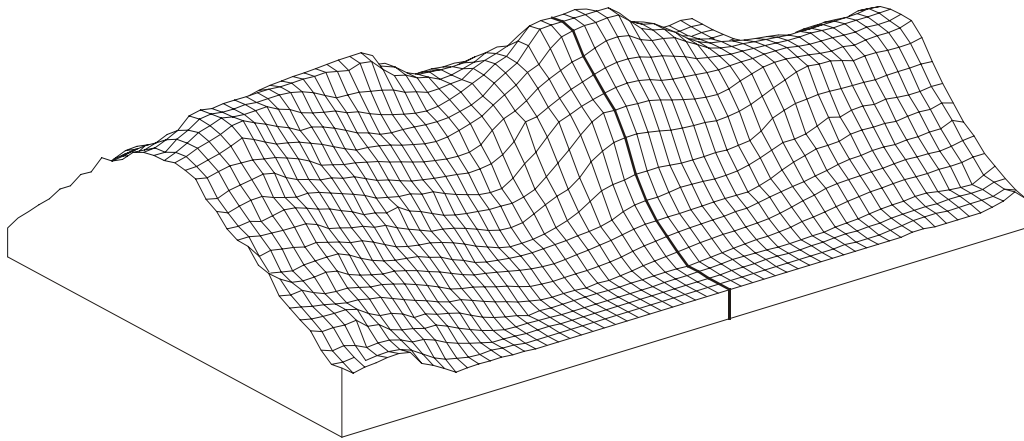


Figure 24.5B: Isometric view of model with stiffness increased to 9.

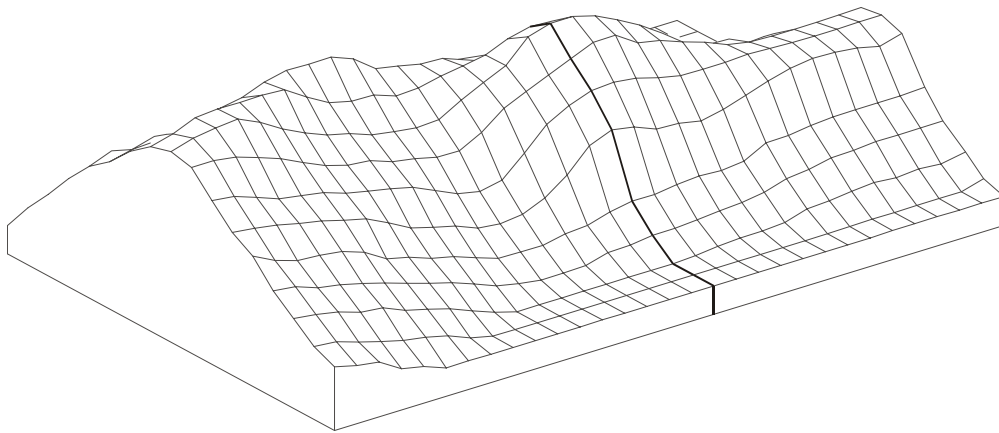


Figure 24.6B: Isometric view of model with Smooth toggled on and set to 5.

Drawing Topographical Maps with Contours

Contours are curved versions of 3D lines, and like 3D lines, they have no thickness. Use contours to create topographical maps.

Creating contours is similar to creating Bezier or B-spline curves. You place a number of control points that define the curve. DataCAD accepts up to 36 control points. Unlike Bezier or B-spline curves, the contour curve that DataCAD generates passes through the points that you draw.

➔ To draw a contour:

1. Choose Contour from the 3DEntity menu.
2. Choose the type of contour you want to draw: Natural, Cyclic, or Tangent. Natural creates a contour that begins at the first control point you select and ends at the last point. Cyclic creates a contour that connects the first and last control points with a smooth curve, creating a closed shape. Tangent creates a contour that connects two additional points to the control points, creating a tangency at the end of the contour.

3. Use the Stiffnss option to determine the curvature as the contour passes through each control point. A high stiffness forces the contour to remain flat as it passes through each point. To set a stiffness for the contour, choose Stiffnss from the Contour menu, and choose or type a stiffness value and press **Enter**.
4. Choose Divisons and enter a value to indicate the number of line segments you want to use in between each control point. The further apart your control points, the more divisions you need. If you set too many divisions, your display regeneration time slows.
5. Choose the elevation at which you want to draw this contour: Z-Base, Z-User1, Z-User2, or Z-Hgt. Z-Base, Z-User1, Z-User2, and Z-Hgt are the four standard elevations at which you can enter a contour control point. You can use AddIndx or SubIndx to alter these values. To set the Z elevations, Z-User1 and Z-User2, choose one of these elevation options from the Contour menu, and then choose or type an elevation value and press **Enter**.
6. Determine if you are working with tangent contours. Only with tangent contours, select a first tangent point, the point before the curve. This point determines the direction of the curve as it enters the first control point.
7. Select the control points. You can choose up to 36 control points for any one contour. You may select AddIndx or SubIndx to enter the next control point at a new elevation. Choose AddIndx to add the value of the Index setting to the current Z elevation value. Choose SubIndx to subtract the value of the Index setting from the current Z elevation value.
 - If you are working with tangent contours, specify the *ending* tangent point. This is the point *after* the curve. It determines the direction that the curve heads as it enters the last control point.

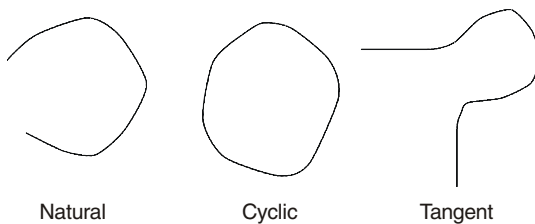


Figure 24.7: Types of contours

Use 3DCursor to attach a control point to a particular point on your model (vertex of a slab, center of a sphere, etc.), by snapping in world coordinates with the 3D cursor.

You can force the contour to lie in a plane by specifying that the control points have a fixed Z elevation. Use the Z-Base, Z-Hgt, Z-User1, or Z-User2 toggles to set the height.

Use SetIndx to help create a single contour that rises steadily in the Z direction. For example, to create a series of concentric contours set at 10' intervals, use SetIndx to set the index to 10'. After drawing the first contour at Z-base, select AddIndx and draw the second contour, which would be entered into the drawing 10' higher than the first, and so on.

➔ To draw a contour that steadily rises:

1. Choose SetIndx from the Contour menu. A list of index distance values appears.
2. Choose or type an index distance and press .

Editing 3D Entities

The 3DEdit menu looks similar to its 2D counterpart and, in most cases, works the same way. In fact, the Erase, Layers, and Identify options in the 3DEdit menu are actually shortcuts to those menus in the (2D) Edit menu. However, keep in mind that the 3DEdit menu options should only be used on 3D entities, unless otherwise noted. See the “Rules for Editing 3D Entities” section in this chapter for details.

Perhaps one of the most useful features of 3D entities is that you can edit them in any plan or parallel view. The 3DEdit menu works closely, in fact, with the 3DViews menu; all 3DEdit submenus have a 3DViews option, which takes you immediately to the 3DViews menu. In addition, to help you move quickly from menu to menu, the 3DEdit menu also includes shortcuts to the Edit menu and the 3DEntity menu.

In this chapter:

- ⊕ Rules for editing 3D entities
- ⊕ Moving, copying, rotating, stretching, mirroring, and enlarging 3D entities
- ⊕ Exploding 3D entities
- ⊕ Modifying 3D entity attributes

Rules for Editing 3D Entities

Keep in mind the following when editing 3D entities:

- **You can't erase edges.** The lines that make up a 3D entity represent the edges of a single entity; they are not an entity themselves. If you try to erase one edge of a slab, for instance, the entire slab is deleted.
- **Edit 2D entities with 2D editing options only.** The Edit menu contains the 2D editing tools you'll need to edit 2D entities. However, the Erase, Change, and Explode options in the 3DEdit menu can also be used on 2D entities.
- **Many 2D editing options can be used on 3D entities.** The Stretch option in the Edit menu, for instance, can be used on a horizontal slab. However, keep in mind that most 2D editing options work only in plan view. When you select such an editing option, DataCAD automatically changes to plan view.
- **Edit in plan or parallel views only.** If your model is not being displayed in plan or parallel view and you select a 3D editing option, DataCAD automatically changes to the last plan or parallel view.
- **Select entities in any view.** However, when using Entity or Group in 3D selection menus, you can select entities according to the following rules:

3D line	Anywhere on the line
Polygon	Any displayed edge (including the edges of voids)
Slab	Any displayed edge (including the edges of voids)
Block	Any displayed edge
3D arc	The arc's center marker
Cylinder	Any of the three center axis markers
Cone	Any of the three center axis markers
Truncated cone	Any of the three center axis markers
Sphere	The center marker
Torus	The center marker of the sweep or the center markers at the beginning and end of the roll
Contour	Any control point (you can't select the curve itself)
Mesh surface	Any displayed edge of the control grid or any of the displayed control points (you can't select the surface itself)
Surface of Rev.	Any control point
Symbol	The insertion point

With QckSnap toggled off (3DEdit/Settings menu), you can select any circular division of a 3D arc, cone, truncated cone, sphere, or torus; you can also select any edge of a single-curved facet of a surface of revolution but not an edge of a double-curved facet. With both QckSnap and Fast3D (Utility/ObjSnap menu) toggled off, you can select any displayed edge of a symbol, according to the rules for each entity in that symbol.

Moving 3D Entities

Use Move in the 3DEdit menu to move an entity to a different position on the screen. 3D Move works the same as the 2D Move option, with the exception of the following:

- When specifying the two points of the distance to move, the model can be in any parallel or orthographic view. DataCAD draws specific distances across the screen, but the distance and direction is taken with respect to the model in the current view. For example, if you view the model in an elevation view and specify a distance and angle to move, such as 10' at 90°, objects that you select move up, off the ground plane.
- The model displays in any view during the process of selecting entities to move. This makes the precise selection of individual entities or collections of entities easier, especially in cases where entities are stacked one upon another or when it is difficult to separate one entity from another.

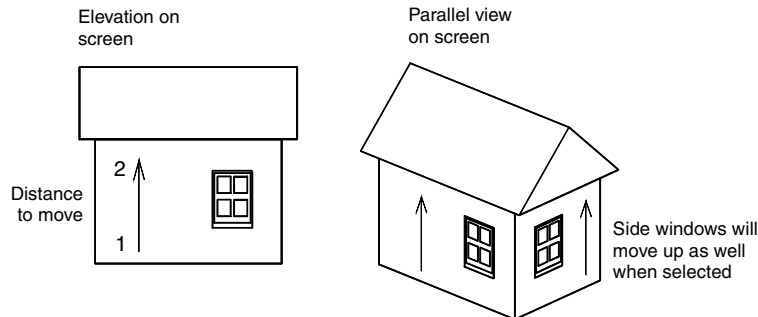


Figure 25.1: Using the 3D Move option

Choose 3DCursor from the Move menu to toggle on the world snapping cursor. The world cursor lets you snap directly to the objects in the model rather than to their projections on the screen. See “All About 3D Snapping” in the “Modeling Basics” chapter for more information on the world snapping cursor.

➔ To move an entity:

1. Choose Move from the 3DEdit menu.
2. Toggle AndCopy on to copy the entity at the same time you move it. If you toggle AndCopy off, the entities move to the new location but do not copy.
3. Select the first point of the distance to move.
4. Select the second point of the distance to move.
5. Choose Z-Dist to move the entity in the Z-direction relative to the plane on the screen. Positive distances indicate movement out of the screen; negative distances indicate movement into the screen. You can also use Z-Only to set the move distance to zero (0) in the X and Y directions. Then select the entities to move. Entities move relative to the screen Z-axis. DataCAD resets the Z-distance to zero each time you choose Move from the 3DEdit menu.
6. Choose a selection method and select the objects to move. Use the 3DViews menu to reorient your view to get the best vantage point for selecting objects. DataCAD moves the selected entities relative to the plane of the screen.
7. Choose Invert if you make a mistake and select the wrong entity to move; then, reselect the moved entity. The entity returns to its original position.

You can also use PrevDist to use the last distance and direction specified in an editing option. You can choose this option instead of specifying two points of the distance to move. Or use NewDist to enter a new move distance. This does not change the Z distance. NewDist only appears after you've entered a move distance.

➔ To move an entity to another layer in your drawing file:

1. Choose ToLayer from the Move menu.
2. Choose a layer from the list, or use Match or Filter to indicate the layer to move to. See the instructions below to use Match and Filter.
3. Choose a selection method and select the entity or entities to move.

Use Match to select a layer by selecting an entity on that layer. Set the active layer, move objects to another layer, or point DataCAD to a specific layer, all without having to remember layer names.

➔ To move entities to another layer using Match:

1. Select ToLayer from the Move menu. You are prompted to *Select layer to move to*.
2. Choose Match from the menu. You are prompted to *Select entity to match*. Use Match with any of the following Layers options: On/Off; SetActiv; ActvOnly; Lock; Name; EraseLyr; DelLayer; SaveLyr; ViewLyr, Select. It is also available when using ToLayer under Move and Copy or when using LyrSrch.
3. Select an entity to select the layer it is on. You are prompted to *Select entity to move*.
4. Select the entity you want to move to the selected layer. The entity is moved to the selected layer.

Use the Filter toggle to scan, or filter, for layers with similar names. When you specify a prefix or suffix that the filter searches for, a list of layers with that affix appears. To access a layer that you don't know the name of, use Filter to search for layers with a similar affix.

For example, if you specify layers by plumbing and electrical, you might have layers named 1FLPLUMB.LYR, 2FLPLUMB.LYR, 3FLPLUMB.LYR, and 1FLELECT.LYR, 2FLELECT.LYR, and 3FLELECT.LYR. To search for a plumbing layer, use ???plumb. You can search for first floor layers using a prefix of 1fl: 1fl* or 1fl????.

Use the asterisk (*) in place of any number of characters and the question mark (?) in place of each character you're unsure of.

The Filter toggle only appears when you are prompted to select from the list of layers. The filter disregards any characters following an asterisk. Therefore, if you entered *plumb to search for plumbing layers, all layer names would be displayed.

➔ To filter layers:

1. Choose Filter from the ToLayer menu to toggle Filter on.

2. Type the affix, including the wildcard character(s), for the layer you want to find and press **Enter**. A list of layer files with this affix appears.

DataCAD also allows you to scroll through different layer groups with similar affixes. If the group you initially searched for doesn't contain the layer you wanted, you can scroll through the remaining groups to find the appropriate layer.

FiltrBack Scrolls back to a group of similarly named layers.

FiltrFrwd Scrolls forward to a group of similarly named layers.

3. Choose the layers you want to use.

DataCAD remembers the last affix for which you filtered. The prefix name appears in the Message Window and the layer names appear in the Menu Window when you re-enter the Filter function.

Copying 3D Entities

The 3D Copy option works identically to the 2D Copy option: specify the distance to copy from any parallel or orthographic view, and then select the objects to copy in any view.

➔ To copy 3D entities:

1. Choose Copy from the 3DEdit menu.
2. Enter the first point of the distance to copy.
3. Enter the second point of the distance to copy.
4. Select the objects to copy using the Selection Menu. You can use the 3D Viewer to reorient your view and get the best vantage point for selecting the objects. DataCAD copies the selected entities relative to the plane of the screen.

Rectangular Arrays

Use RectArray to create three-dimensional rectangular arrays of 3D entities at any orientation in space. This option works like the 2D rectangular array option, except that you can copy objects in the X, Y, and Z directions simultaneously, each with its own spacing and number of repetitions.

➔ To create a rectangular array:

1. Choose Copy from the 3DEdit menu.
2. Choose RectArray from the Copy menu.
3. Set the Z distance to copy if you are creating an array of more than one element in the Z direction.
4. Select the first point of the distance to copy each element of the array.
5. Select the second point of the distance to copy each element of the array.
6. Type the number of repetitions in the array in the X direction, Y direction, and Z direction, pressing **Enter** after each value entered.

7. Choose a selection method and select the objects to copy in the rectangular array. You can use the 3DViews menu to reorient your view to get the best vantage point in selecting objects.

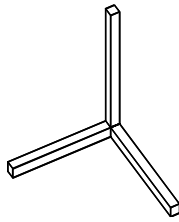


Figure 25.2: Original object to copy

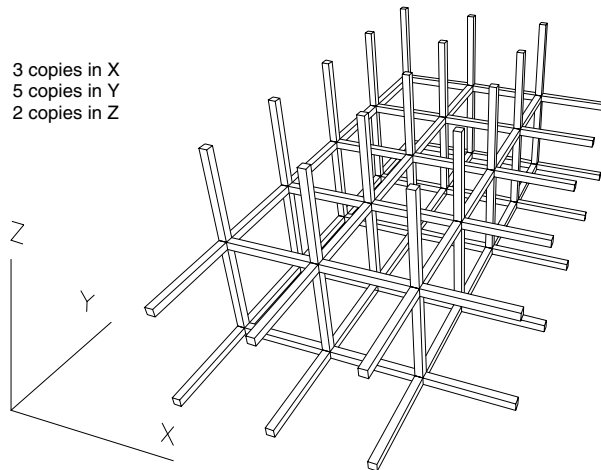


Figure 25.3: Resulting array

Choose 3DCursor to toggle on the world snapping cursor. The world cursor lets you snap directly to the objects in the model rather than to their projections on the screen. See “All About 3D Snapping” in the “Modeling Basics” chapter for more information.

Rotating 3D Entities

Like all 3D editing options, you can use the Rotate option in any parallel or orthographic view. Choose the center of rotation, and then select the objects to rotate. You can rotate the objects in any order.

➔ To rotate a 3D entity:

1. Choose Rotate from the 3DEdit menu.
2. Choose AndCopy to copy the entities at the same time you rotate them. To rotate entities without copying them, leave AndCopy toggled off.
3. Select a point that defines the center of rotation relative to the plane of the screen. If you want the Z-coordinate of the center of rotation to be something other than zero, use CenterZ to enter the Z-coordinate. See the instructions below to set the Z-coordinate. You can use NewCentr to re-enter the center of rotation relative to the current plane of the screen. This option does not change the Z-coordinate of the center of rotation. NewCentr only appears after you choose a center of rotation.

4. Choose which of the three axes (X, Y or Z) you wish to rotate around, relative to the screen. Use X-Axis to rotate selected entities about an axis parallel to the X axis of the screen and passing through the center of rotation. Use Y-Axis to rotate selected entities about an axis parallel to the Y axis of the screen and passing through the center of rotation. Use Z-Axis to rotate selected entities about an axis parallel to the Z axis of the screen and passing through the center of rotation.
 5. Choose XAngle, YAngle, or ZAngle to set the angle at which entities rotate around the X, Y, and Z axis, respectively, relative to the plane of the screen at the time the center of rotation is specified. The option you choose should match the axis that you selected in step 3. Use XAngle, YAngle, and ZAngle to Choose XAngle, YAngle, or ZAngle from the Rotate menu, and then choose or type a value and press **[Enter]**.
 6. Choose a selection method and select the objects that you want to rotate. You can use the 3D Viewer to reorient your view to find the best vantage point in selecting objects.
 7. Select Invert if you make a mistake and select the wrong entity to rotate. Then, reselect the entity. The entity returns to its original position.
- ➔ To set the Z-coordinate of the center of rotation relative to the plane of the screen:
1. Choose CenterZ from the menu. A list of values appears.
 2. Choose or type a value and press **[Enter]**. DataCAD resets CenterZ to zero each time you choose Rotate from the 3DEdit menu.

Mirroring 3D Entities

Use Mirror to mirror entities about a plane in space. You can view a model from any angle in space; the mirror plane is always perpendicular to the screen.

You define the mirroring plane the same way as with the 2D Mirror option.

- ➔ To mirror a 3D entity:
1. Choose Mirror from the 3DEdit menu.
 2. Choose AndCopy to copy entities at the same time you mirror them. To mirror entities without copying them, leave AndCopy toggled off.
 3. Select a point that defines one end point of a line in the mirroring plane.
 4. Select a point that defines the other end point of a line in the mirroring plane. You can use NewLine to re-enter the mirroring plane perpendicular to the current plane of the screen. NewLine only appears after you create a mirror line.
 5. Choose a selection method and select the objects that you want to mirror. You can use the 3D Viewer to reorient your view to find the best vantage point for selecting objects.

Stretching 3D Entities

3D Stretch lets you stretch three-dimensional entities similarly to 2D Stretch. As with other 3DEntity menu options, you can specify the distance, direction and entities to stretch from any parallel or orthographic (plan) view.

- Explode curved 3D entities into polygons or lines before stretching them. See “Exploding Entities” later in this chapter for instructions.
- Cylinders, 3D arcs, spheres, cones, truncated cones, and tori are non-stretchable entities. Use Enlarge to increase or decrease their size in one direction by an appropriate factor, or explode them into polygons and then stretch the polygons.
- 3D Stretch can only be used on 3D lines, polygons (and their voids, if any), blocks, slabs (and their voids, if any), contour curves, and mesh surfaces. You can use Stretch to edit polygons, slabs, and voids. You must define the distance to stretch in a plane parallel to the polygon or slab you want to stretch, or the resulting polygon or slab may become non-planar. To make sure that the plane of the screen is parallel to the slab, you can use PlneSnap, NewElev, or EditPlne.
- If the entity you select is contained completely within the limits of the selecting point, area, or fence, DataCAD moves the entity the distance you indicated to stretch.

➔ To stretch a 3D entity:

1. Choose Stretch from the 3DEdit menu. See the “Viewing Your Model” chapter for more information.
2. Select the first point of the distance to stretch.
3. Select the second point of the distance to stretch.
4. Choose Z-Dist to enter a distance to stretch portions of entities in the Z direction relative to the plane of the screen. Positive distances indicate movement out of the screen; negative distances indicate movement into the screen. You can use NewDist to re-enter the distance to stretch in the X and Y directions. NewDist does not change the distance to stretch in the Z direction.
5. Use PrevDist if you want to deal with the last X and Y distance and direction you specified in an editing option to stretch. You can use this option instead of specifying the distance to stretch in steps 2 and 3.
6. Choose a selection method and select the objects that you want to stretch. You can use the 3D Viewer to reorient your view to get the best vantage point for selecting objects.
7. Choose Invert if you make a mistake and select the wrong entity to stretch. Then, reselect the stretched entity. The entity returns to its original size. DataCAD resets the Z distance to zero each time you choose Stretch from the 3DEdit menu. Choose Z-Only to set the stretch distance in the X and Y directions to zero. Then select the entities to stretch.

Enlarging 3D Entities

Use Enlarge to enlarge or scale entities to different sizes in the X, Y, and Z directions. Enlarge works similarly to its 2D counterpart, except that you can set the center of enlargement in any orthographic or parallel view. You can also select entities you want to enlarge in any view or orientation once you enter the center of enlargement.

➔ To enlarge a 3D entity:

1. Choose Enlarge from the 3DEdit menu.
2. Choose AndCopy to copy entities at the same time you enlarge them. To enlarge entities without copying them, leave AndCopy toggled off. After you choose a center of rotation, you can use NewCentr to re-enter the center of enlargement. This option does not change the Z coordinate of the center of enlargement. To set the Z coordinate relative to the plane of the screen, choose CenterZ from the menu, and then choose or type a value and press **[Enter]**.
3. Select a point that defines the center of enlargement relative to the plane of the screen.
4. Use XEnlgmt, YEnlgmt, and ZEnlgmt to change the current value of the enlargement factor in the X, Y, and Z directions of the screen, respectively. The enlargement factor cannot be zero. Choose XEnlgmt from the menu, and then choose or type a value and press **[Enter]**. Use Reset to set all of the enlargement factors to 1.0. 2.0x sets all of the enlargement factors to 2.0. 0.5x sets all of the enlargement factors to 0.5.
5. Choose a selection method and select the objects that you want to enlarge. You can use the 3D Viewer to reorient your view to get the best vantage point for selecting objects.
6. Choose Invert if you make a mistake and select the wrong entity to enlarge. Then, reselect the entity. The entity returns to its original size. DataCAD resets the CenterZ to zero each time you choose Enlarge from the 3DEdit menu.

Exploding 3D Entities

Use Explode to separate 3D entities into their polygonal facets or into three-dimensional lines. After you explode an entity into lines or polygons, it is not possible to reverse the operation and recreate the original entity from these lines and polygons.

Explode can create a large amount of data in the drawing file. For example, take a single sphere having 20 divisions in the sweep direction (primary divisions) and 10 divisions in the rise direction (secondary divisions). After an explode operation, this single sphere produces some 200 polygons, each of which occupies approximately as much drawing storage space as did the original sphere. A mesh surface having 20 divisions in each direction would produce 800 polygons, which would collectively require 364 times as much drawing storage space as the original mesh surface.

- Explode separates curved entities (cylinders, cones, truncated cones, tori, spheres, mesh surfaces, and surfaces of revolution) into lines or polygons for subsequent editing operations (like stretch).
 - You can explode a text entity into 3D lines that you can then rotate in space. The lines then respond to a hidden line removal.
 - Explode determines the validity of exploding entities. For example, if you select a contour curve to explode into polygons, DataCAD ignores the selection and does not explode the curve because it has no polygonal components.
- ➔ To explode a 3D entity:
1. Choose Explode from the 3DEdit menu.
 2. Toggle ToLines on to explode entities into 3D lines, or toggle ToPgons on to explode entities into polygons.
 3. Choose a selection method and select the objects that you want to explode. You can use the 3DViews menu to reorient your view to get the best vantage point in selecting objects.

Changing 3D Entities

3D Change lets you change the divisions of circular entities and the stiffness of contours as well as colors, linetypes, line overshoots, and more.

- ➔ To change an attribute of a 3D entity:
1. Choose Change from the 3DEdit menu.
 2. Choose the option you want to change.
 3. Choose or enter a new value; follow the prompts in the Message Window.
 4. Choose Closed to indicate whether 3D arcs, cylinders, cones, and truncated cones are treated as closed solids or as open surfaces, then choose one of the following:

Yes	3D entities will be treated as closed shapes
No	3D entities will be treated as open shapes
 5. Choose a selection method and select the objects that you want to change. You can use the 3DViews menu to reorient your view to get the best vantage point for selecting objects.

Use PrimDivs to change the number of segment divisions of circular entities. PrimDivs affects 3D arcs, cylinders, cones, truncated cones, spheres, tori, surfaces of revolution, contour curves, and mesh surfaces. For spheres, surfaces of revolution, and tori, PrimDivs affects only the number of segment divisions in the sweep direction (in the plane of the screen). For mesh surfaces, it affects only the number of segment divisions in the first direction. For contour curves, it affects the number of segment divisions between each control point of the curve.

Use `ScndDivs` to change the number of segment divisions of certain circular entities. This toggle affects only spheres, tori, surfaces of revolution, and mesh surfaces. For spheres and surfaces of revolution, it affects the number of segment divisions in the rise direction. For tori, it affects the number of segment divisions in the roll direction. For mesh surfaces, it affects the number of segment divisions in the second direction.

Use `Stiffnss` to change the stiffness factor associated with contour curves. `Stiffnss` affects only contour curves. The stiffness can be any number except zero.

Viewing Your Model

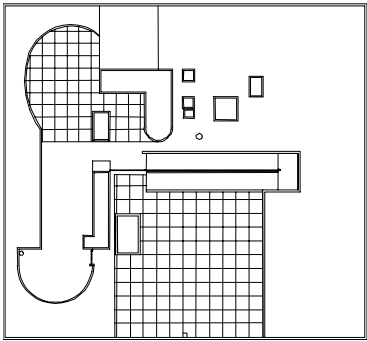
DataCAD gives you several ways to view your model to help you draw and create views for plotting or presentations. While there are many different kinds of views, only plan, parallel, perspective, and oblique are commonly used for architectural models, and DataCAD lets you customize each of these types of views. You must use a parallel or plan view for editing your drawing, while perspectives and obliques are most often used for visualization, rendering, and presentations.

Once you have a view defined, you can perform a hidden line removal of it or shade it. You can save any hidden-line, shaded, or standard 3D view and then link any entity in your model to that view to easily display it. You can even create walk-throughs of your model.

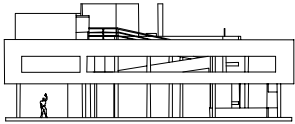
The o2c-based object viewer provides real-time, dynamic, shaded 3D views of DataCAD models. You can adjust many of the o2c settings and save them as high-quality rendered images.

In this chapter:

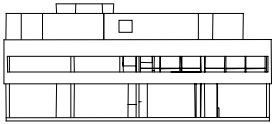
- ⊕ Creating and changing views
- ⊕ Shading your model
- ⊕ Performing hidden line removals on your model
- ⊕ Saving and linking to views
- ⊕ Creating walk-throughs
- ⊕ Using the o2c-based object viewer



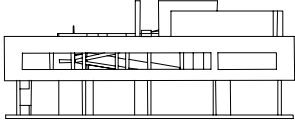
Ortho (Plan View)



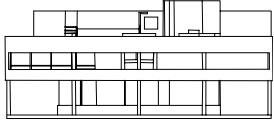
Front (Parallel View)



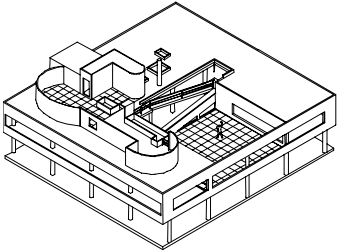
Left (Parallel View)



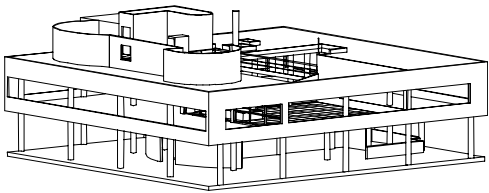
Back (Parallel View)



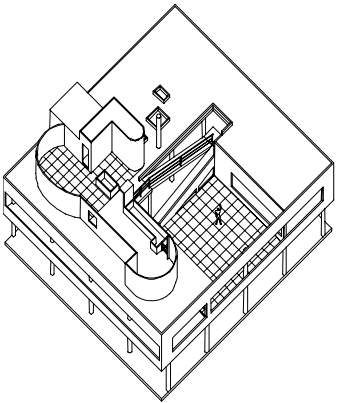
Right (Parallel View)



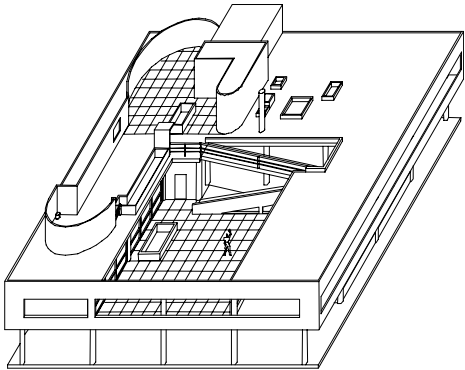
Parallel (Isometric View)



Parallel (Isometric View)



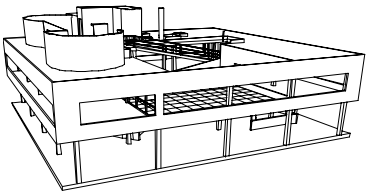
Plan Oblique (Axonometric View)



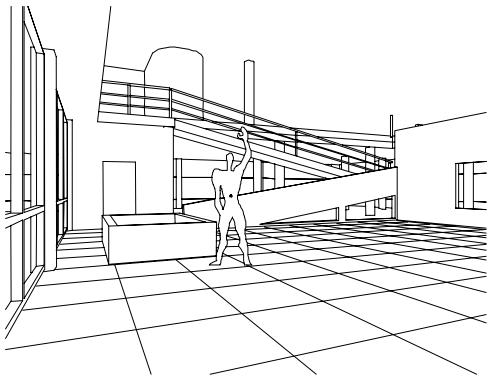
Elevation Oblique



Perspective (One point)



Perspective (Three point)



Perspective (Two point)

Figure 26.1: Examples of views you can create in DataCAD

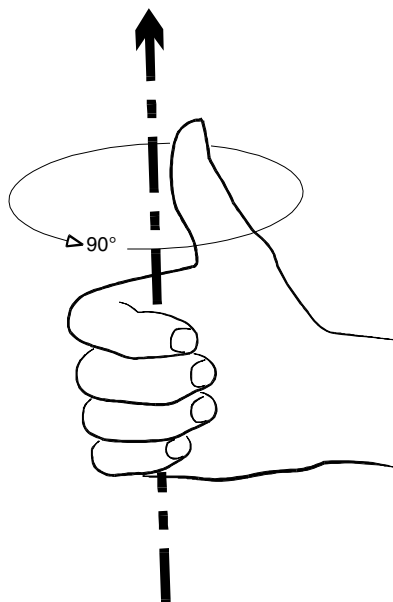
Creating Parallel Views

Parallel in the View pull-down menu restores the last parallel view you generated and displays the globe. Elevations and isometrics are parallel views but by no means the only ones. Because these two cases are used so frequently, they have their own menu selections. It may help, however, to remember that these are just different kinds of parallel views. You can, in fact, choose any eye point to create a parallel view. See “More About Changing the Eye Point for a View” later in this chapter for more information on the globe display and how it affects parallel views.

Elevations and isometric views can be quickly created in DataCAD in just a few steps. Isometric views can be created by simply choosing Isometric from the View pull-down menu in the Menu Bar or by choosing Isometric from the 3DViews menu. You can use the Viewer Toolbar to work with any parallel view. Choose Viewer Toolbar from the View pull-down menu. This toolbar will automatically change when you change your view to any parallel view or to orthographic (plan) view.

More About Rotation Angles in 3D Views

In 2D, positive rotation angles go counterclockwise; in 3D, the concept is a bit more complex. This rotation angle of -90° flips the model to the left. DataCAD uses the right-hand rule to indicate the direction in which positive angles run. To understand the right-hand rule, point the thumb of your right hand in the direction of the axis in question. Your fingers curl in the direction of positive rotation. The Y-axis goes up, so when you point your thumb up, your fingers curl toward you a counterclockwise direction. Negative angles, therefore, run in a clockwise direction.



POSITIVE AXIS OF ROTATION
Figure 26.2: Right-hand rule

More About Changing the View Center

When you create the first parallel view in a drawing, DataCAD calculates the view center by examining the extents of the model. DataCAD generates all parallel views with the assumption that the model is situated over a point called the view center, or the point in space at which the observer is looking. DataCAD places the view center in the middle of the extents, halfway between Z-base and Z-height. Ideally, this is somewhere near the center of the model.

The view center is not attached to the model in any way, so if the model moves, the view center does not change. Be careful when you use Globe. Sometimes models spin off the screen completely because the globe is always situated directly over view center. Reposition the view center to somewhere inside the model to avoid this problem.

The view center in a perspective view is always the point of intersection between the picture plane and the line of sight.

➔ To reposition the view center:

1. Choose Reset from the Controls menu. Reset places the drawing in the last orthographic view you generated. This helps you find a center of the model.
2. Choose ViewCntr; an Auto option becomes available.
3. Select a new center point in one of the following ways:
 - select a point in the center of the model
 - choose Auto to calculate a view center with the center point halfway between Z-base and Z-height

The new view center is set, and you return to the Controls menu.

Both the NewElev and EditPlne options change the view center implicitly. In both of these options, you select a point on the plane where $Z=0$. This point becomes the new view center for parallel views. This point also becomes the new grid origin for grid snapping. See “Creating Perspective Views” later in this chapter for more information on the view center in a perspective view.

More About Changing the Eye Point for a View

The globe display is used to reorient your eye point in a parallel or perspective view; it is automatically displayed on screen whenever you choose a parallel or perspective view. Selecting a point inside the globe display indicates the new observer position for viewing the model.

In perspective views, the globe changes the elevation of the observer’s eye point and the direction from which you observe the model (north, south, east, or west), but it does not change the view center, the distance to the object, or the cone of vision.

In a parallel view, the globe sets the observer’s eye point above or below the ground plane at which the observer stands. The center of the globe represents a point directly over view center and all points on the outer circle represent a point in space directly below the view center looking up.

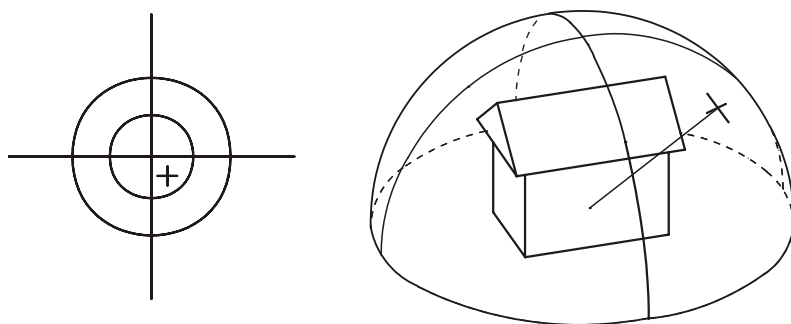


Figure 26.3: Globe perspective

To visualize how the globe operates, imagine that the model sits inside a large sphere. Half of the sphere is above the ground plane, and half of it is below, with the ground plane cutting the sphere at the equator. The equator of this sphere is represented as the inner circle on the globe. All points inside this circle, therefore, are the upper half of the sphere (above ground) and all points outside the inner circle are the lower half of the sphere (below ground).

While the Globe display is in the upper-left corner of the screen by default, you can use the Globe option in the 3DViews menu to reposition the globe on the screen. You can:

- Move the globe to a different location on the screen so that you can continue to work on a part of the model without interference
- Change the size of the globe; a larger globe gives you finer control over the rotation of a parallel or perspective view

➔ To move the Globe display:

1. Choose Globe from the 3DViews menu. The Globe menu is displayed:

Default	Returns the globe to its customary position in the upper-left corner of the Drawing Window
---------	--
2. Select a new center for the globe. Move your mouse to dynamically size the Globe display.
3. Select a point on the perimeter of the new globe. The globe moves to the selected position.

Use Controls to specify a vantage point for a parallel view. You can think of the Controls menu as a more accurate version of Globe and just another way of generating parallel views. You can use the Controls menu to position the view center inside the model for smoother operation of the globe.

When you choose Controls from the 3DViews menu, a toolbar appears at the top of the screen, along with the Controls menu in the Menu Window.

The cursor you see while you are in the Controls menu displays the current orientation of the X, Y, and Z world coordinate axes.

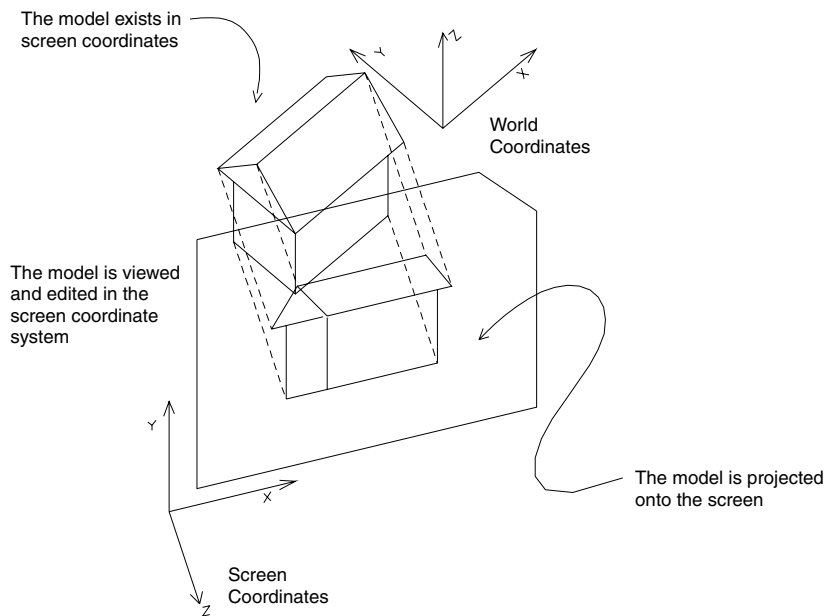


Figure 26.4: Screen coordinates vs. world coordinates

The Screen option rotates the view around the screen coordinate axes. Screen coordinates refer to the right-handed coordinate system that you worked with in 2D. The +X axis goes to the right, the +Y axis goes up, and the +Z axis comes out of the screen. This does not change.

The World option rotates the view around the world coordinate axes. World coordinates refer to the right-handed coordinate system associated with the model. Imagine that your model is in orthographic view, and you attach the screen coordinate axes to the model itself. What you have is a set of three axes known as world coordinate axes. These do not change with respect to the model but do change with respect to the screen.

When you choose any of the Rotate options, DataCAD rotates the view by a set number of degrees around the axis you indicate.

Reset changes the current view to the last orthographic view taken. Reset also brings the model back into view in case it moves off the screen. Use this option before using ViewCntr.

RotAngle controls the number of degrees that the object rotates under any Rotate+ or Rotate- options.

Four additional input modes are available in the Controls menu: relative and absolute cylindrical (distance, angle, Z coordinate) and relative and absolute spherical (distance, plan angle, rise angle). These modes provide additional variables to the Z component of coordinate entry, which may be helpful in specialized modeling applications like entering a 3D line that represents a solar angle.

➔ To set the rotation angle:

1. Choose RotAngle from the Controls menu. A list of rotation angles appears.
2. Choose or type a value and press **[Enter]**.

AddView adds the current parallel view to the list of saved views.

Creating an Isometric View

Isometric creates a true isometric view. Isometric views are a special parallel view where the X- and Y-axes of the model form a 30° angle from the horizontal. To make sure that the angle is precise, choose Isometric from the 3DViews menu in the Menu Window or choose Isometric from the View pull-down menu in the Menu Bar at the top of the screen.

Creating Elevations

Elevations are a special kind of parallel view. When you view an elevation in plan, the front of the model faces the bottom of the screen.

Use Front to view the front elevation of the model. Use Back to view the back elevation of the model. Use Right to view the right elevation of the model. Use Left to view the left elevation of the model.

In addition to the four preset elevations—Front, Back, Left, and Right—you can create non-orthogonal elevations. You can use New to generate views for rendering or to orient the model for a side view in which you can precisely place the zero plane, Z-base, and Z-height. If you choose a Front Elevation, you cannot guarantee where Z-base and Z-height are in relation to the new view.

➔ To create a new elevation:

1. Choose NewElev from the 3DViews/Elevation menu in the Menu Window, or choose Elevation/New from the View pull-down menu in the Menu Bar at the top of the DataCAD screen. The view changes into plan view so that you can specify the elevation line. CenterZ appears in the Menu Window.
2. Choose CenterZ to set the height of the view center. You must choose this option before you select the first point in the new elevation indicator. A list of values is displayed in the Menu Window.
3. Choose or type a value and press .
4. Select a point to lie on the new zero plane (such as the corner of a wall). This point also serves as the new view center and grid origin.

Move the cursor and it changes into a New Elevation Indicator. There are several parts to this cursor:

Z-base indicator

Z-height indicator

Zero plane (the plane at which $Z = 0$)

Direction of view

The Z-base and Z-height indicators may not be visible if their values appear off the screen or if their values are set to zero, in which case these indicators coincide with the zero plane.

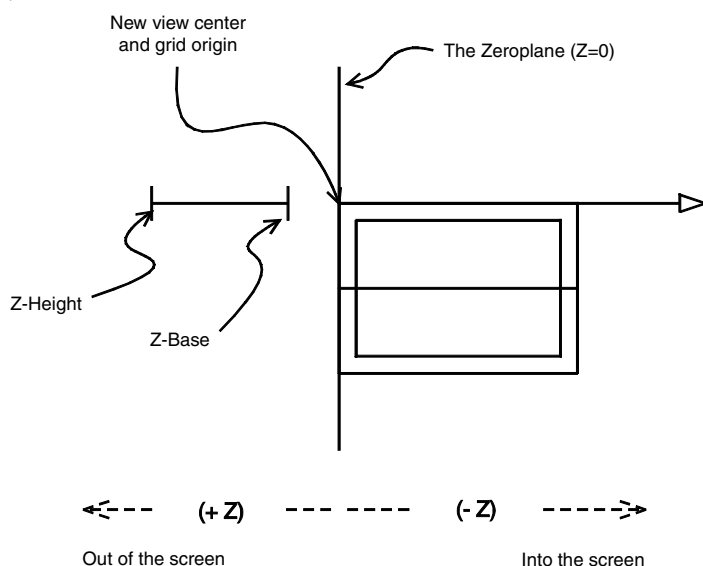


Figure 26.5: The new elevation indicator

5. Adjust Z-base and Z-height if necessary. The indicators change position to show you the new location of Z-base and Z-height. Remember that negative distances go in the direction of the view (into the screen).
6. Select a point in the direction of the view. The specified view appears on the screen.

Options for creating front, back, left, right, and new elevations are available from both the View/Elevation menu in the Menu Bar at the top of the DataCAD screen and the 3DViews/Elevation menu in the Menu Window.

➔ To add the current elevation to the list of saved views:

1. Choose AddView from the Elevation menu.
2. Type the name of the new view, up to eight characters, and press Enter. The view is added to the list of saved views.

More About the New Elevation Indicator Cursor

This cursor has an arrow indicating the direction of the new zero plane position and a tail representing the current Z-base and Z-height settings. (The end of the tail nearest the crosshair is Z-base and the end furthest from the crosshair is Z-height.) If Z-base is set to 0, the tail meets the crosshair; if Z-height is set to 0, the end of the tail won't be displayed.

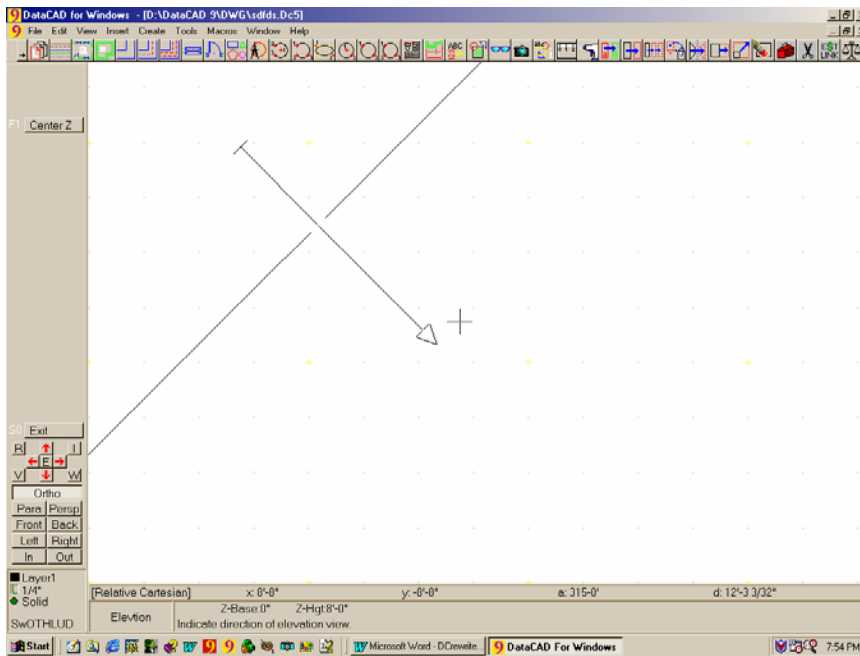


Figure 26.6: The New Elevation Indicator cursor with Z-base=0 and Z-height=8

Creating Perspective Views

DataCAD can generate one-, two-, and three-point perspectives. The differences between these perspective types lie only in the location of the eye of the observer (EyePntZ) and the point in the model at which the observer is focusing (CentPntZ).

You can use the Viewer Toolbar to work with your perspective view. Choose Viewer Toolbar from the View pull-down menu. This toolbar will automatically change when you change your view to any parallel view or to orthographic (plan) view.

For each perspective type, you must select an eye point for the observer and a view center to mark the point in the model where the observer is looking. When you select the first point the cone of vision is displayed. See “More About Changing the Eye Point for a View” earlier in this chapter for more information on the globe.

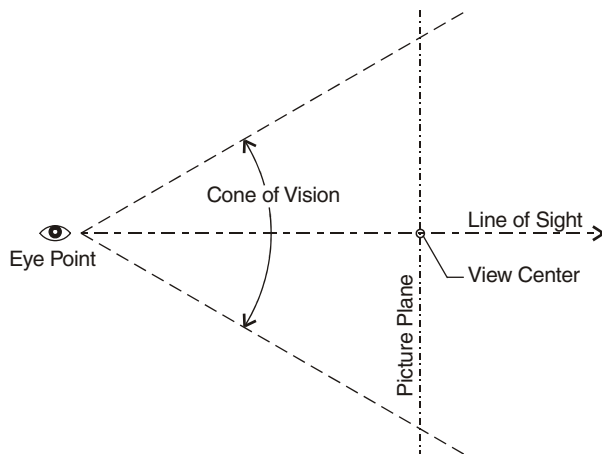


Figure 26.7: Cone of vision for perspective view

The line that cuts across the cone of vision is the picture plane. Things in front of this plane (on the same side as you, the observer) appear larger than normal; things behind the picture plane appear smaller.

The line that leads directly from the eye point to the view center is the line of sight. As with parallel views, always place the view center somewhere inside the model so that the globe is stable and predictable.

Establishing Perspective Settings

➔ To set the elevation of the observer's eye point:

1. Choose EyePntZ from the SetPersp menu. A list of perspective eye point values appears.
2. Choose or type a value and press **Enter**.

➔ To set the elevation of the center of the view:

1. Choose CentPntZ from the SetPersp menu. A list of values appears.
2. Choose or type a value and press **Enter**.

The Reset option in the SetPersp menu returns you to a perspective view using the current settings in this menu.

Use HthrClip to reduce the fisheye distortion that can appear in close-up perspectives. You can also increase the hither clip distance to partially cut your model shown in perspective view, and then create a shaded "sectional perspective" with the Quick Shader menu. The hither clip plane is displayed as a crosshair on the cone of vision.

➔ To use Hither Clip:

1. Choose HthrClip from the SetPersp menu. A list of values appears.
2. Choose or type a value and press **Enter**.

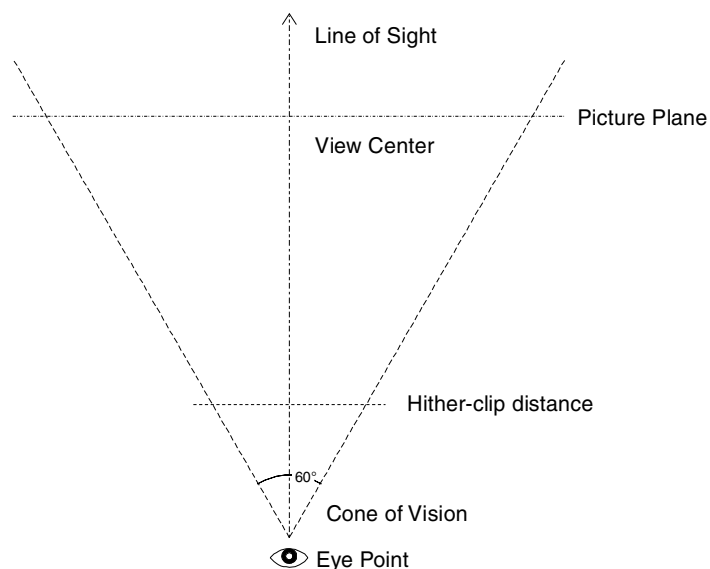


Figure 26.8: Cone of vision with hither clip

Toggle FixCone on in the SetPersp menu to fix the width of the cone of vision. The width of the field of vision in a fixed cone is set using either ConeAng or Camera.

- ➔ To set the width of the field of vision to an angle using ConeAng:
 1. Choose ConeAng from the SetPersp menu. A list of values appears.
 2. Choose or type a value between 1 and 179 degrees and press **Enter**.
- ➔ To set the width of the field of vision using Camera:
 1. Choose Camera from the SetPersp menu. A list of values appears.
 2. Choose or type a value and press **Enter**. Camera sets the width of the field of vision by expressing it as the length of a camera lens. The longer the lens, the narrower the cone of vision.

Creating a One-Point Perspective

Create one-point perspectives by choosing an eye point at the same elevation as the view center; the observer is looking straight ahead, neither up nor down. A one-point perspective must have the picture plane parallel to one of the frontal faces of the model.

- ➔ To create a one-point perspective:
 1. Choose SetPersp from the 3DViews menu. When you choose Prspect from the 3DViews menu for the first time in a drawing, the SetPersp menu is automatically displayed so you can set up the perspective view.
 2. Set EyePntZ to a reasonable value. When an observer is standing on the ground, 5' is a typical height for the observer's eye.
 3. Set CentPntZ to the same elevation you used in step 1.
 4. Select a point for the eye point.
 5. Select a point inside the model for the center point. Use ortho mode to make sure that the line of sight is perpendicular to one of the faces of the model.

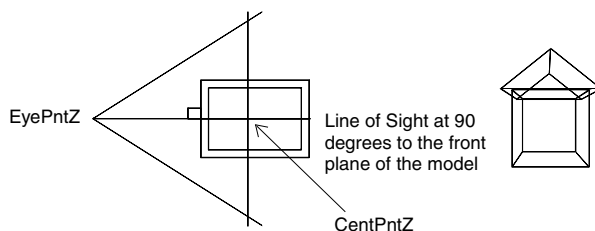


Figure 26.9: One-point perspective

Creating a Two-Point Perspective

Use two-point perspectives to relax one of the two constraints that you placed on one-point perspectives. You can either:

- Set the eye point and center point at different elevations and keep the line of sight perpendicular to one of the frontal faces of the model; or

- Keep the eye point and center point Z the same and set the line of sight at an angle other than 90°.

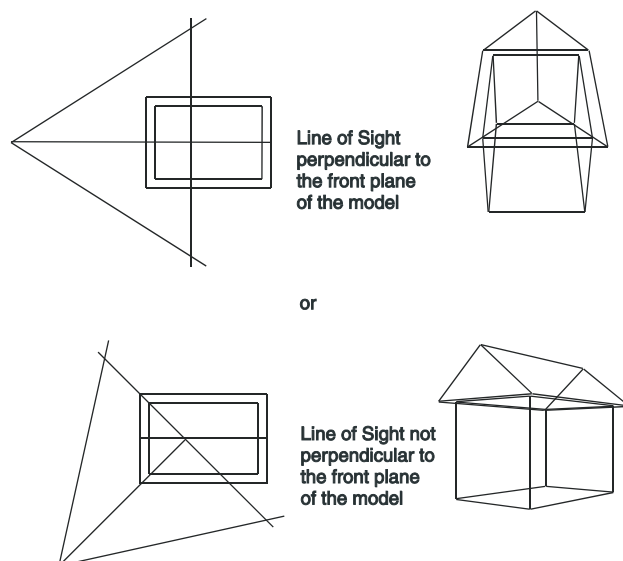


Figure 26.10: Two-point perspective

Creating a Three-Point Perspective

Three-point perspectives are generated when the eye point Z elevation is different from the center point Z elevation and the line of sight is at an angle other than 90°. With these constraints, set the eye point Z and the center point Z appropriately and choose these points in your model.

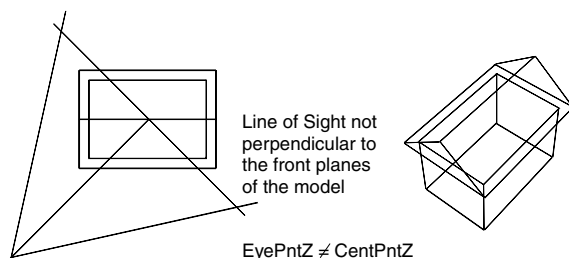


Figure 26.11: Three-point perspective

Perspective Navigation Controls

Hotkeys and MouseWheel controls automate perspective setup and navigation by using video game-style controls. For more detailed information about the navigation aids, see “Walking Through Your Model” later in this chapter.

Creating Oblique Views

DataCAD can create both plan obliques (architectural axonometrics) and elevation obliques. Choose PlanObl to create a plan oblique or choose ElevObl to create an elevation oblique. If this is the first time you’ve chosen ElevObl, follow the directions for selecting the NewElev option in the SetObliq menu. See “Creating Elevations” earlier in this chapter for more information on the NewElev option.

Establishing Oblique Settings

The SetObliq option, which is only available after you select ElevObl, defines which elevation of the model you want to use as your frontal elevation in constructing the oblique view. It works exactly the same way as the NewElev option in the 3DViews/Elevation menu.

OblqAng controls the angle of rotation before the plan oblique is taken. Angles are usually between 30° and 45°. For elevation obliques, this setting controls the angle at which the lines sweep away from the elevation.

Creating an Elevation Oblique View

➔ To create an elevation oblique view:

1. Select Elevation from the 3DViews menu.
2. Choose ElevObl from the Elevation menu.

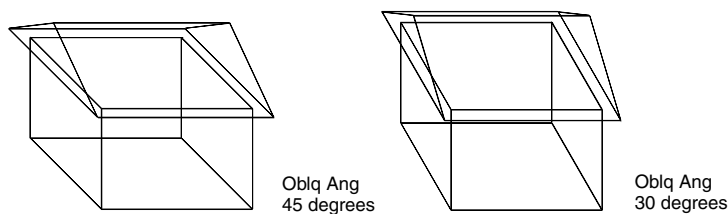


Figure 26.12: Elevation oblique

Creating a Plan Oblique View

➔ To create a plan oblique view:

1. Select Elevation from the 3DViews menu.
2. Choose PlanObl from the Elevation menu.

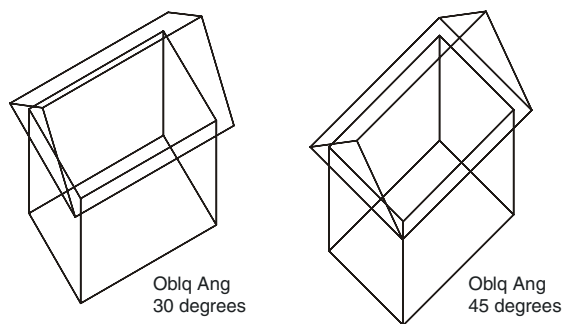


Figure 26.13: Plan oblique

Creating a Hidden Line Removal

Use Hide to perform hidden line removals in DataCAD. DataCAD's hidden line removal system is a two-pass system with an additional initialization step. The initialization step performs a few simple and quick operations: allocating memory, calculating the limits of your model for a specified view, and initializing the temporary hidden line removal database.

The hidden line removal process does not recognize text and associative dimensions; therefore, they do not appear after a Hide. To display these entities after a Hide, convert them into 3D lines with the 3D Edit, Explode command.

Pass one, the calculating pass of the hidden line removal system, scans through the entire model and temporarily separates it into a series of polygonal faces or free lines, as follows:

- DataCAD breaks entities like 2D lines (having height), polygons, slabs, blocks, cylinders, etc., into a series of polygons that the hidden line removal system can understand and interpret.
- DataCAD breaks entities like 3D lines, contour curves, 3D arcs, etc., into a series of free line segments. Free line segments are assumed to be infinitely thin.
- DataCAD also recognizes voids in polygons.

Pass two of the hidden line removal process is where most of the work takes place and, consequently, usually takes longer. Each line segment that is potentially a visible edge or a free line is compared with each polygonal face. DataCAD uses the tests to determine if the line segment lies completely in front of, completely behind, straddles, or pierces a given polygon.

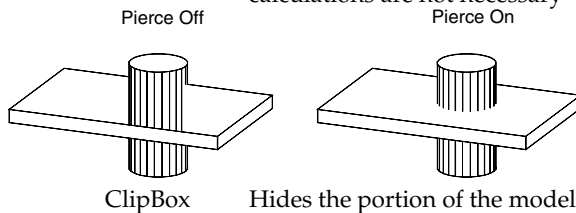
Depending on the results of these tests, the line is either eliminated from consideration, split into one or more pieces, or left intact to be tested later against the remaining polygons. If, after having been compared against all of the possible polygons, some portion of the line segment remains, DataCAD displays the segment and optionally adds it to a temporary database for the saved image.

DataCAD uses a number of optimization techniques for reducing the computing time of the display pass to a minimum.

➔ To perform a hidden line removal:

1. Choose Hide from the 3DEdit menu, or choose Hidden Line Removal from the View pull-down menu. Set the following options as necessary:

Begin	Starts the hidden line removal process and displays operation's progress in the Message Window; press [Delete] or [End] to cancel the hide at any time
SaveImg	Save the image produced by a hidden line removal
CropImg	Performs a hidden line removal only on the portion of the model that is visible on the screen
HLRpart	Divides the Drawing Window into smaller sub-areas for faster hides
Pierce	Performs piercing calculations during hide, causing slower hidden line removals; to speed up a hide, toggle Pierce off whenever piercing calculations are not necessary



ClipBox	Hides the portion of the model that is contained within the previously defined ClipCube; available only for parallel views of your model; to create a similar clipping effect for perspective hidden line images, use HthrClip in the 3DViews/SetPersp menu
All	Performs hidden line removals on entities on all layers toggled on
ActivLyr	Performs a hidden line removal only on the entities on the active layer
SelSet	Performs a hidden line removal only on the entities in the active selection set

- | | |
|---------|---|
| Options | Settings to perform different functions within a hidden line removal |
| 3DViews | Sets the view for the model prior to the hidden line removal; since the image created by the hidden line removal process contains only 2D lines, the image is best viewed in an orthographic view |
2. Choose 3DViews from the Hide menu and orient the model so that you can see the image.
 3. Choose Exit to return to the Hide menu.
 4. Choose Begin to start the hidden line removal process.

Tips on Maximizing Performance

- Use lines rather than polygons whenever possible. For example, if you want to draw some rectangles on the faces of a slab block to schematically represent windows, use 3D lines instead of polygons to outline the windows to increase performance. Since the rectangles are simply inscribed on the face of the slab, polygons are not needed; there is nothing to hide.
- Use three-dimensional entities like blocks, cylinders, and slabs instead of collections of 2D line and simple polygons.
- Keep models as simple as possible. As the size of the model increases, the time required to compute a given image increases at a faster rate. You may consider placing some of the detail for a given model on separate layers so that you can temporarily toggle some layers off for preliminary runs of the hidden line removal system. After you complete the modeling process, you can then toggle on all applicable layers and perform this more complicated hidden line removal analysis on the complete model.

Saving the Hide Image

If you toggle SaveImg on before you choose Begin, DataCAD saves the hidden line removal image for later use; if SaveImg is toggled off, DataCAD does not save the image and only displays it on the screen.

Once the hidden line removal process is complete, the SaveImg menu is displayed:

ActvLyr
NewLyr
LyrFile
Offset

➔ To add a retained image to the active layer:

1. Use **[Tab]** and **[Shift] + [Tab]** to scan through the available layers to select the layer to add the new image to. DataCAD saves the image to the active layer.
2. Choose ActvLyr after the hidden line removal process is complete. DataCAD adds the image to the active layer; you return to the Hide menu.

➔ To add the retained image to a new layer in the drawing:

1. Choose NewLyr at the completion of the hidden line removal process to create a new layer for the drawing.
2. Enter a name for the new layer.
3. Indicate whether you want the new layer on or off. If you plan to create a series of images, leave the layer off to return directly to the Hide menu and create a new image.

➔ To add the retained image to a layer file:

1. Choose LyrFile at the completion of the hidden line removal process.
2. Enter the name of the layer file you want to create. DataCAD lists the layer files in the current layer folder in a dialog box.

If you click Cancel in the dialog box without specifying a layer file, DataCAD deletes the image.

The image created by the hidden line removal process contains only 2D lines; you can use all of the normal DataCAD editing operations on the resulting image such as erasing, trimming, enlarging, moving, rotating, stretching, and hatching. Since the image contains 2D lines, an orthographic view is the best view.

Before you choose ActvLyr or NewLyr, you can offset the image in the XY-plane by some distance. This helps you to create a composition of views by arranging them on the screen apart from each other.

➔ To offset an image:

1. Choose Offset after completion of the hidden line removal process.
2. Select the first point of a distance to offset the image in the plane of the screen.
3. Select the second point of a distance to offset the image in the plane of the screen.
4. Choose either ActvLyr or NewLyr and proceed with the instructions listed above for adding the image to the drawing.

Choose Cancel to delete the retained image and return to the Hide menu. Remember that after the hidden line removal process, if you toggle SaveImg on, DataCAD saves the image before you select a destination for it. If you choose Cancel, DataCAD deletes the previously saved image from the drawing and makes this space available.

Increasing Hide Speed

Use HLRpart to partition the Drawing Window into smaller sections, allowing for faster hides. Hide treats each section separately, resulting in continuous lines being divided. It also allows you to see sections in their entirety as the hidden line removal progresses.

➔ To change Drawing Window partitions:

1. Choose HLRpart from the Hide menu.

2. Choose Seg-X.
3. Type a new value and press .
4. Choose Seg-Y.
5. Type a new value and press .

Hide Options

Use Options in the Hide menu to perform different functions within a hidden line removal.

Editing Hidden Line Removal Images

If you use HLRpart to partition your hidden line removal image, continuous lines will be broken wherever they cross a partition. While these breaks aren't visible, they can make editing that image time-consuming, if not difficult. Using the Join option allows you to more easily edit the hidden line removal image after it is complete. When you perform a hidden line removal with HLRpart on, toggle Join on in the Hide/Options menu to weld (reattach) lines on a model.

Deleting Double Lines

The DelDoubl option in the Hide/Options menu deletes any double lines in the model that were created as a result of performing a hide with Join and HLRpart on.

Drawing Hidden Lines

If the DrwHidn option in the Hide/Options menu is on before you choose Begin, DataCAD draws hidden lines using the line color, linetype, and line spacing you set by the HidnColr, HidnStyl, and HidnSpac options. If you toggle DrwHidn off, DataCAD does not draw the hidden lines.

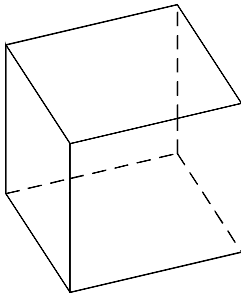


Figure 26.14: DrawHidn with a dashed linetype

Disregarding Back Faces

Use the BackFace option in the Hide/Options menu for a faster Hide performance. DataCAD calls all polygonal faces facing away from the viewer backfaces. With this toggle on, DataCAD ignores the backfaces for a hidden line removal. This decreases the amount of area to hide and the time it takes to complete the hidden line removal.

Hidden Line Colors

Use the HidnColr option in the Hide/Options menu to change the color used to draw the hidden lines. This color menu works exactly like its 2D counterpart. See “Color Menus” in “The Drawing Board” chapter for more information on the 2D options.

Hidden Linetypes

Use HidnStyl to change the linetype used to draw the hidden lines. When you toggle DrwHidn off, HidnStyl is ignored. When you toggle DrwHidn off, HidnColr is ignored.

➔ To change the hidden linetype:

1. Choose HidnStyl from the Options menu. A list of linetypes is displayed.
2. Select a new linetype for hidden lines.

Hidden Line Spacing

Use HidnSpac to change the line spacing used to draw hidden lines. When you toggle DrwHidn off, HidnSpac is ignored.

➔ To change the hidden line spacing:

1. Choose HidnSpac from the Options menu. A list of spacing values is displayed.
2. Choose or type a new spacing value and press **[Enter]**.

Closing 2D Arcs

Unlike 3D arcs and 3D cylinders, ordinary 2D arcs do not have an option for being treated as closed objects. If you toggle CloArc2 on in the Hide/Options menu, DataCAD treats 2D arcs as closed, which makes them opaque to the hidden line removal process. 2D arcs that have a Z-height work like cylinders with closed ends and sides. If you toggle CloArc2 off, during the hidden line removal process, DataCAD treats 2D arcs as ordinary curved lines (arcs with no Z-height) or open-ended cylinders (arcs having Z-height).

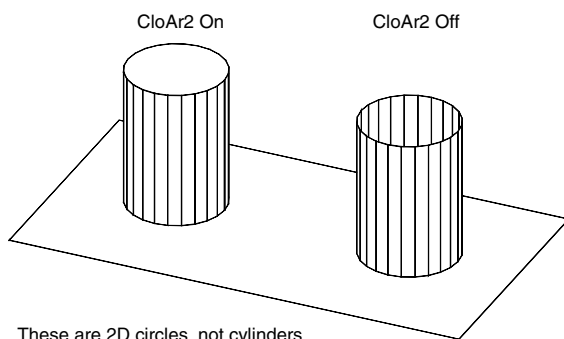


Figure 26.15: 2D circles

Closing 2D Ellipses

If you toggle CloEllp on in the Hide/Options menu, DataCAD treats 2D ellipses as closed, solid objects during the hidden line removal process. If you toggle CloEllp off, DataCAD treats ellipses as ordinary curved lines (no height) or curved surfaces (having height) with no top or bottom.

Displaying Outlines Only

Toggle NoEdge on in the Hide/Options menu to display the outline of tori and cylinders. NoEdge does not display any other divisions of these entities.

2D Arc Divisions

During the hidden line removal process, DataCAD uses the current value of the circle divisions setting. This Divisions option is duplicated on the Hide/Options menu for convenience in adjusting the current value of this variable. Choose Divisions from the Hide/Options menu and set it to an integer value from 4 to 36.

Automatic Hidden Line Removals

Use AutoHide to perform hidden line removal on multiple views, or batches. Choose AutoHide from the ViewMast menu, and choose from the following options:

Begin	Displays selected views on the screen in sequence; after the last view appears, the sequence begins again. To stop the sequence, press [End]
Select	Adds or removes views from the sequence; choose the view to add to the sequence, or choose the view to remove from the sequence
ClearAll	Clears all viewing selections and you can reset sequences
SelctAll	Selects all views and adds them to the sequence
LayrSet	Sets all layers as views
DrwHidn	Draws hidden lines in the hidden line removal images
ToScreen	Sends the hidden line removal images to the screen only
ToLayer	Sends the hidden line removal images to a layer only
ToLyrFil	Sends the hidden line removal images to a layer file only
LyrPath	Specifies the folder to contain the created layer files

Creating a Shaded Image

DataCAD's Shader creates a shaded image of your model by *temporarily* breaking it down into 3- and 4-sided polygons and then shading the surfaces according to light source and shader settings. You can define and place up to seven light sources in your model and use ambient light as well. These light sources are specific points that radiate light out in all directions and can be toggled on or off to create different lighting effects. Each surface of your model is illuminated independently of the others, so one surface will not cast a shadow on its neighbor. The only factor that affects the light on a surface is the distance of the light source from it. The Shader supports all 256 colors in the DataCAD palette as well as a rendering palette of 16.8 million colors.

➔ To create a shaded image of your model:

1. Go to the Edit menu in the Menu Window and click on DCAD 3D. The 3D Edit menu is displayed.
2. Click on Shader in the 3D Edit menu. The Shader menu is displayed.

3. Click on Settings in the Shader menu. The Shader Settings dialog box is displayed, allowing you to change light source settings, the background color or image of the shader window, how edges are shaded and more. The settings in the Shader Settings dialog box are also available individually in the Shader menu in the Menu Window. You can use either the dialog box or the menu to change settings; all changes will be reflected both in the dialog box and the menu options.

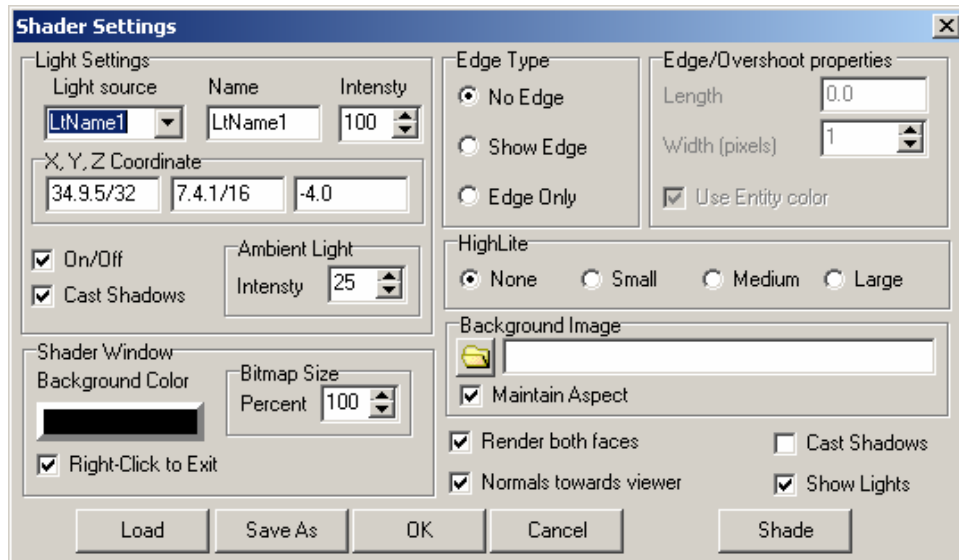


Figure 26.16: The Shader Settings dialog box

4. Change the Light Settings by choosing the light source whose settings you want to change from the Light Source list box. Then, follow the instructions below:
 - The light sources are named LtName1, LtName2, etc., by default. To rename the light source, click in the Name input box and type a new name. The name will be updated in the Light Source list box when you either click in the list box or click OK to save the shader settings.

You can also position a light source by simply clicking anywhere in the Drawing Area. Click on the light source name in the Menu Window, click On/Off in the menu if the light source has not yet been toggled on, and then choose Position from the menu. A 3D cursor is displayed in the Drawing Window; note the cursor's coordinates displayed in the Message Window. Click to place the light source. Please note that you cannot use the light source settings in the Shader Settings dialog box to position the light using this method.

- The light source's intensity is the percent of white used in the light, with percentage values ranging from 0 to 100. To set the intensity of the light source, use the up and down arrows to increase or decrease the percent value, respectively.
- To enter specific coordinates for a light source, click in the X, Y, Z Coordinate input boxes and type new coordinates. (The left input box is the X coordinate, the middle input box in the Y coordinate, and the right input box is the Z coordinate.)
- To toggle a light source on or off, click in the On/Off checkbox.

- To give the light source the ability to cast shadows, toggle Cast Shadows on. Note that shadows are not actually cast in the shaded image unless Cast Shadows in the bottom right corner of the dialog is checked.
 - The intensity of ambient light can be adjusted just as light source intensity was. To set the ambient light intensity, use the up and down arrows to increase or decrease the percent value, respectively.
5. Repeat the previous step for each light source you want to customize.
 6. Go to the Shader Window section. You can customize the background color of the Shader window, change the size of the shaded image, and change how you move from the shaded image back to your drawing:
 - To change the background color, click on the colored rectangle in the Shader Window section. Click on a new color in the Color Palette dialog box, and then click on OK.
 - Click on the Right-Click to Exit checkbox to check this option; this will allow you to close the shader window by simply right-clicking. Uncheck this option to use a pop-up menu to quickly save your image, move between images and your drawing, or import your shaded image into eZmeeting.
 - The shaded image is created as a bitmap (.BMP) file. You can adjust Bitmap Size by using the up and down arrows to increase or decrease the percent value, respectively, with 100% being the full size of the Drawing Area. You can go as high as 1,000%, depending on how much system memory you have available.
 10. Go to the Edge Type section. Click on No Edge to shade only the surfaces of your model, click on Show Edge to shade both the edges and surfaces, or click on Edge Only to shade only the edges (similar to a hidden line removal with lights). The default setting is No Edge.
 11. Add edge and overshoot lines to your shaded image.
 - To set the length of the overshoot, click in the Length input box and type a value. The default length is 0.0.
 - To set the width of the lines used for edges and overshoots, use the up and down arrows to enter a width in pixels. The default is 1.
 - To draw edge/overshoot lines in the same color as the entity's surface, check the Use Entity Color checkbox. This is checked by default. If you'd like to use a different color, uncheck the box, click on the color swatch to the right, and choose a new color. The Edge/Overshoot Properties section is only available when Show Edge is toggled on.
 12. Use the HighLight setting to produce a visible area of light around a light source, much like a lamp throws on a wall and ceiling. This setting is off by default, but you can set it to three intensities, with Large being the most intense.

13. Choose a background image for your shaded image by clicking on the folder icon in the Background Image section and choosing a .BMP or .JPG file. Leave Maintain Aspect checked to ensure the background image covers the entire Shader window. You cannot customize the size of the background image.
14. Go to the bottom right corner of the dialog box. Check Render Both Faces to render both sides of a polygon, instead of rendering only the side with the positive normal. When this option is unchecked, any negative normals of polygons that face you will not be rendered, unless “Normals towards viewer” is checked in the following step.
15. Check “Normals towards viewer” to set the positive normals of all polygons so that they face you. Uncheck this option to leave positive normals facing their original direction.
16. Cast shadows in your shaded image, according to the light sources you set for shadows in step 4, by checking the Cast Shadows box in the lower right corner of the dialog box. (Note that you must still complete steps 19 – 21 to actually draw the shadows.)
17. Make sure the Show Lights checkbox is checked on. This option displays the light sources as small symbols around your model, letting you see exactly where they are positioned. These symbols do not print; they are only displayed for visual reference.
18. Click OK to save your new settings and close the Shader Settings dialog box.
19. Cast shadows in your shaded image by clicking on Shadow in the Shader menu. Then, toggle AddShado on. You are prompted to “Select entity to receive shadow”.
20. Select the entity on which the shadow will fall, the polygon acting as the floor or ground plane for instance. Use the selection menu to select and highlight the entity. If you select the wrong entity, toggle AddShado off and select that entity again. Repeat this step to select the correct entity. An entity cannot cast a shadow and receive one (have a shadow fall on it) at the same time.
21. Right-click to exit the Shadow menu and return to the Shader menu.
22. Click Shade in the Shader menu. Your model is shaded and the bitmap image is displayed in the Drawing Area. You can now save the shaded image as a bitmap file or return to your drawing file. You can save your settings and then use them later without having to change all the Shader settings again. See “Saving and Using Shader Settings” later in this chapter.

More About Polygon Normals

A characteristic of a polygon surface is its orientation, or normal. Think of a polygon as a sheet of paper that only allows you to write on one side. This side is the positive normal, while the other side is the negative normal. In a visual sense, polygons are one-dimensional, as they may only be seen from one side in a rendered view.

To quickly shade your model without changing any Shader settings, click on Shade in the View pull-down menu. Your model is shaded.

You can't see polygon normals in DataCAD. While a normal is a condition of the coordinate points that describe a particular polygon, they really don't matter much in the DataCAD world. Whether you view, quick-shade or hide a polygon from either side, it will look and display exactly the same. The only way to know which way a normal faces is to remember how it was created. Did you define the polygon right to left counter clockwise, or left to right, clockwise? If you went counter-clockwise, then the normal faces towards you, or out of the screen. If you went clockwise, then it faces away from you, into the screen. This method of orientation is known as the right-hand rule. To further complicate things, if you mirror an entity, all of its normals will reverse.

Saving and Using Shader Settings

You can save your settings and then use them later without having to change all the Shader settings again.

➔ To save your shader settings:

1. Customize the options in the Shader Settings dialog box. See “Creating a Shaded Image” earlier in this chapter for more information on customizing shader settings.
2. Click on the Save As button in the Shader Settings dialog box. A dialog box is displayed, prompting you to “Save light information to”.
3. Change to the folder where you want to save your .LIT file, if necessary, and type a name for the file.
4. Click on Save. The dialog box closes and your .LIT file is saved.

➔ To use a .LIT file:

1. Click on Settings in the Shader menu. The Shader Settings dialog box is displayed.
2. Click on Load. A dialog box is displayed, prompting you to “Load light information from”.
3. Change to the folder where your .LIT file is stored if necessary. Then, click on the file.
4. Click on Open. The dialog box closes, and the light settings are loaded into the Shader Settings dialog box.

Saving Your Shaded Image

You can save your shaded image or send the bitmap to an eZmeeting conference.

➔ To save your shaded image:

1. Click on Save or Save As in the File menu, or right-click in the bitmap window to display a pop-up menu and click on Save Bitmap or Save Bitmap As. (Keep in mind that if you use the Save option, the bitmap is saved in the last folder you used.) The Right-Click to Exit option in the Shader Settings dialog box must be unchecked for the pop-up menu to display when you right-click in the Shader window.
2. Determine if you used the Save As option in the previous step. If so, type a name for the bitmap file, and click Save.

Using the OpenGL Shader

The OpenGL Shader lets you create preliminary architectural views of 3D models. Use the OpenGL Shader for shadow casting, custom edge color and thickness settings, custom overshoot settings, bitmap background image, and color mapping.

➔ To use the OpenGL Shader:

1. Select DCAD 3D from the Edit menu, choose Shader from the 3Dedit menu, and Settings from the Shader menu. The Shader Settings dialog box appears.

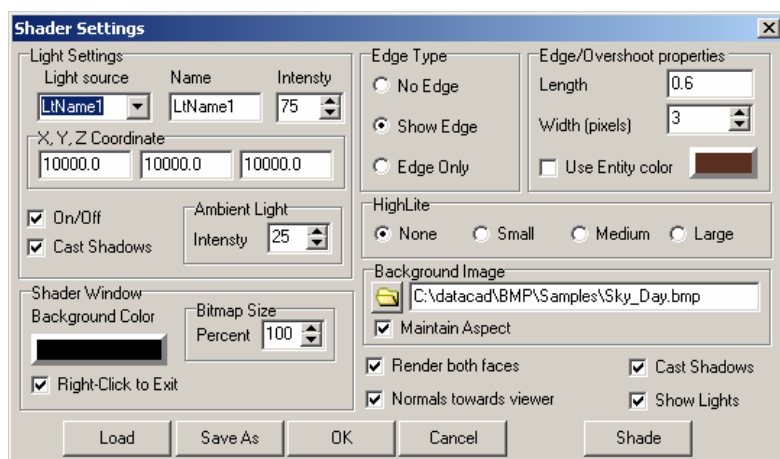


Figure 26.17: The Shader Settings dialog box

2. Make changes in the Shader Settings dialog box to meet your needs.
 - Enter a name for the light source.
 - Set the XYZ coordinates for the light. You can use 10000 to simulate sunlight.
 - Make sure that the On/Off box in the Light Source section of the dialog box contains a check.
 - Check the Cast Shadows box. This allows the light source to cast shadows.

- Put a check in the Cast Shadows box at the bottom right corner of the dialog box. This is a global setting that affects the OpenGL Shader.
- 3. Click OK to exit the Shader Settings dialog box and return to the Drawing Window.
- 4. Select 3DViews from the Shader menu. This sets up a perspective view of the model.
- 5. Select Prspect from the 3DViews menu to recall the last perspective view. Then, right-click to return to the Shader menu.
- 6. Select Shadow from the Shader menu. This determines which polygons will receive shadows.
- 7. Position the ground plane to receive shadows.
- 8. Select entities to receive shadows. Right-click when you are finished selecting entities to receive shadows.
- 9. Select Shade to render the model.
- 10. Use the arrow keys and (PageUp) and (PageDn) to adjust the perspective view for the next image. When you have the desired view, proceed.
- 11. Select Settings to adjust Shader options in the Shader Settings dialog box.
 - Turn on Show Edge in the Shader Settings dialog box.
 - Set the Edge Overshoot. For this example, we used 6 inches (0.6).
 - Set the rendered edge width.
 - Set a custom edge color.
- 12. Click on the Shade button to render the model.

Viewing Selected Entities in Your Model

Use ClipCube to restrict your view of the model so that only the entities inside a rectangular region are visible. If an entity is only partially inside the clip cube, DataCAD clips it at the point where it passes outside the cube. This reduces the visual clutter of a complex model as it develops. You cannot edit entities outside of the clip cube.

There are two clip cubes you can use: one for orthographic view and one for parallel views. These cubes are independent of one another. When you begin the clip cube operation from orthographic view, DataCAD uses the orthographic clip cube. When you use ClipCube while in a parallel view, the parallel clip cube is used.

The clip cube status is one of the attributes stored with the AddView function in the GoToView menu. Restoring a saved view also restores the clip cube that was in use when you saved the view.

The Arc Factor setting controls the smoothness of clipped arcs that use a solid linetype.

➔ To activate the clip cube:

1. Toggle ClipOn on.

➔ To create a new clip cube:

1. Choose New Cube from the ClipCube menu.
2. Set Z-Min and Z-Max so that they enclose the part of the model you want. First set the height of the floor of the clip cube by choosing Z-Min from the ClipCube menu, and then choosing or typing a value and pressing **Enter**.
3. Set the height of the ceiling of the clip cube by choosing Z-Max from the ClipCube menu, and then choosing or typing a value and pressing **Enter**.
4. Select the area with a rubberband box.
5. Right-click to exit, or select two new corners to reset the rubberband box.
6. Choose ClipOn to toggle the cube on.

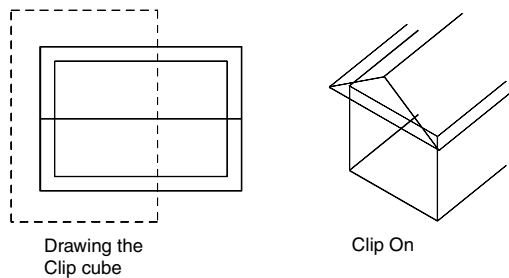


Figure 26.18: Clip cube

Refresh updates the screen after you activate the clip cube. When Refresh is off, press **Esc** to refresh the screen yourself to see the effects of the new clip cube.

You can insert an XREF into your drawing and define a clip cube within it. Either you can do this by using the Insert pull-down menu or by selecting XRefClip from the 3DViews menu.

➔ To define a clip cube in an XREF attached to your drawing with Reference File Management:

1. Open a new drawing or select an existing file.
2. Attach a reference file to it by selecting Reference File from the Insert pull-down menu.
3. Double-click on the name of the file you want to insert. Your cursor is now loaded with a boundary box that represents the extents of the reference file. Toggle on ByCenter in the InsX-REF menu. You are prompted to "Position XREF on drawing." Click the left mouse button to place the reference file by its center. The reference file appears in your Drawing Window.
4. Select Reference File Management from the Insert pull-down menu. Choose Clip Cubes from the extended menu. You are prompted to "Select XREF to modify ClipCube."
5. Click on the reference file you placed in your Drawing Window. Notice that the reference file's Z-base and Z-height appear in the Message Area.

6. Select Z-Min in the Identify menu, choose or type a value that is below the floor or Z-base of the model, and press (Enter). Similarly, select Z-Max to set the Z-height above the roof of the model.
 7. Click on NewCube. You are prompted to "Select the first corner of the clipping cube."
 8. Click where you want the clip cube to begin. Then, pull the rubberband out diagonally and click when it encloses the area you want to display in the clip cube.
 9. Select ClipOn. This toggles the clip cube on.
- ➔ To define a clip cube in an XREF attached to your drawing with XRefClip:
1. Open a new drawing or select an existing file.
 2. Attach a reference file to it by selecting Reference File from the Insert pull-down menu.
 3. Double-click on the name of the file you want to insert. Your cursor is now loaded with a boundary box that represents the extents of the reference file. Toggle on ByCenter in the InsX-REF menu. You are prompted to "Position XREF on drawing." Click the left mouse button to place the reference file by its center. The reference file appears in your Drawing Window.
 4. Make sure you are in the 3DViews menu. Select ClipCube from the 3DViews menu. The ClipCube menu appears.
 5. Select XRefClip from the ClipCube menu. You are prompted to "Select XREF to modify ClipCube."
 6. Click on the XRef you placed in your drawing. The Identify menu appears.
 7. Set Z-Min and Z-Max values so that they are respectively below and above the model's Z-base and Z-height.
 8. Click on NewCube. You are prompted to "Select the first corner of the clipping cube."
 9. Click where you want the clip cube to begin. Then, pull the rubberband out diagonally and click when it encloses the area you want to display in the clip cube.
 10. Select ClipOn. This toggles the clip cube on.

Saving a View

Use SaveImg in the 3DViews menu to create a snapshot of the current view. The snapshot consists of 2D lines and can be edited with normal 2D editing commands in orthographic view.

- ➔ To save an image:
1. Choose SaveImg from the 3DViews menu.

2. Choose an offset distance for the image, if necessary. After you select the first and second points of the image offset, the SaveImg menu is displayed:

ActivLyr	Save the image to the active layer
NewLyr	Save the image to an automatically created new active layer
LyrFile	Save the image to an automatically created layer file
Offset	Select the distance and direction to offset a saved image from the current geometry

3. Select the destination for the saved image.
4. Enter a name for the new destination layer (when you place the image on a new layer or in a layer file). DataCAD generates the image. The image that this procedure creates is made entirely of 2D lines with Z-base = 0 and Z-height = 0; therefore, DataCAD creates the image flat on the ground plane and in orthographic view.

GotoView

When you save a view using the GotoView menu in the 3DViews menu, DataCAD stores information about the current view including:

- viewing scale
- WindowIn location (what you have zoomed in on)
- view type (orthographic, parallel, perspective, or oblique)
- location and orientation of the zero plane
- location of the view center
- active and inactive layers
- size, location, and status (on or off) of the clip cube

Restoring these saved views actually restores all of these stored settings to their previous state. Please note, however, that restoring a view does not restore the values of Z-base and Z-height. The Layers On/Off setting may or may not be restored depending on whether LyrSet is on or off. The LyrSet option in the GotoView menu restores layers to their last saved condition (on, off, active). When LyrSet is off, DataCAD uses the current state of the layers but restores all the other parameters.

The top part of the GotoView menu displays the names of any saved views in the drawing. If there are more than ten views for this drawing, ScrlFwrd and ScrlBack appear on the menu to let you scroll through your views. Selecting a view from the menu restores that view to the Drawing Window.

➔ To add the current view to the list of saved views:

1. Choose Go-to-View from the View pull-down menu.
2. Choose AddView from the GotoView menu.
3. Type the name of the new view, up to eight characters, and press Enter. The view is added to the list of saved views.

- ➔ To add any changes you made to the view without creating a new view:
 1. Choose Go-to-View from the View pull-down menu.
 2. Choose Update from the GotoView menu.
 3. Choose the view that you want to update with the current settings. This update includes active or inactive layers, scale, and location of the zero plane.
- ➔ To delete a selected view from the menu:
 1. Choose Go-to-View from the View pull-down menu.
 2. Choose DelView from the GotoView menu.
 3. Choose the view that you want to delete. DataCAD deletes the view parameters only; the entities in the drawing are not affected by this operation.
- ➔ To move a view from one position in the menu to another:
 1. Choose Go-to-View from the View pull-down menu.
 2. Choose MoveView from the GotoView menu.
 3. Choose the view that you want to move from the list of saved views.
 4. Choose the view you want to appear after the view selected in step 2. The moved view appears before the view selected in step 3. This is useful for moving an often-used view into one of the first ten positions on the menu so you can access it quickly and easily.
- ➔ To rename a saved view:
 1. Choose Go-to-View from the View pull-down menu.
 2. Choose NameView from the GotoView menu.
 3. Choose the view that you want to rename.
 4. Type the new view name, up to eight characters, and press Enter. The view is renamed.
- ➔ To toggle between the previous and current views respectively:
 1. Use (Ctrl) + Left Arrow.
 2. Use (Ctrl) + Right Arrow.

Linking to a View

You can also link go-to-views to any entity in your drawing. For instance, you could link a door in your plan to a door schedule. Then simply pressing Alt and clicking on that door would display the go-to-view of your door schedule. You can also use the linking shortcut (press Alt and click on an entity) to select an entity and choose a view to link to. However, the following instructions offer greater flexibility when choosing entities to link to.

➔ To link an entity to a go-to-view:

1. Choose GotoView from the View pull-down menu. The GotoView menu is displayed in the Menu Window.
2. Choose HyprView from the GotoView menu. You are prompted to “Select the view to link to an entity.”
3. Choose a view from the list in the Menu Window. A selection menu is displayed in the Menu Window.
4. Choose a selection method. You can also toggle Multi on to select multiple entities.
5. Select the entity or entities to link to. The link to the go-to-view you chose in step 3 is established. Repeat steps 2 – 5 to continue linking entities to go-to-views. If no go-to-view is linked to the entity you select, you will be prompted to link one.
5. Restore the linked view by pressing and holding down **[Alt]** while selecting the entity with your left mouse button.

➔ To create hyperlinks between drawing entities and GotoViews, Internet URLs, and programs:

- Use (Ctrl) + Right-click while selecting an entity to access the hyperlink pop-up menu. You can add a link between the entity and a GotoView, a URL, or a file. In the extended menu, use Add to create a new link, Delete to eliminate one, or Edit to change an existing link.
- Use (Ctrl) + Right-click to recall a linked URL or file.
- Select Hyperlinks from the View pull-down menu to display existing hyperlinks in your drawing.

Walking Through Your Model

With the WalkThru option in the 3DViews menu, DataCAD simulates an observer walking through a perspective model. You can walk toward or away from the model, turn right or left, step up or down or to one side, or look up or down or to one side. Before using the options that move you around the model, you must set the distance or angle to control how far you will move with each option.

Before you can use WalkThru, you must set up a perspective view. After you’ve set up a perspective view, you can toggle on the Viewer Toolbar in the View pull-down menu to display a walk-through toolbar containing the same functions as the 3DViews/WalkThru menu.

Perspective Navigation Controls

Use hotkeys and MouseWheel controls to help you walk through your model. You can use these hotkeys in any perspective view.

(PageUp)	Walk backward
(PageDn)	Walk forward
MouseWheel	Walk forward/backward
(Up Arrow)	Walk forward
(Dn Arrow)	Walk backward

(Right Arrow)	Step right
(Left Arrow)	Step left
(Shift) + (Up Arrow)	Look up
(Shift) + (Dn Arrow)	Look down
(Shift) + (Right Arrow)	Turn right
(Shift) + (Left Arrow)	Turn left

The WalkThru, Options menu lets you customize the way you navigate.

➔ To customize walk-through options:

1. Go to the 3DViews menu and select WalkThru.
2. Select Options from the WalkThru menu.
3. Set the three toggle switches, keeping the following in mind:
 - If FixFocal is toggled on, you will always look at the same point in your model while navigating.
 - If FixFocal is toggled off, your focal point will move the same distance as your eye point while navigating.
 - If both Fix Focal and FixDist are toggled on, you will navigate spherically around a fixed focal point.
 - You cannot toggle off FixFocal and FixDist at the same time.
 - If PanView is toggled on, the walk-through controls are disabled. This means you can pan and zoom your perspective view as if it were a two-dimensional drawing.

Walking

WalkDist controls how far forward or backward the observer walks with WalkFwrd and WalkBack.

➔ To change the walking distance:

1. Choose WalkDist from the WalkThru menu. A list of values is displayed.
2. Choose or type a value and press Enter.

Now choose WalkFwrd or WalkBack from the WalkThru menu. WalkFwrd makes it look as if the observer has stepped toward the model by a distance equal to the walk distance by shifting both the eye point and the view center forward along the line of sight. WalkBack makes it look as if the observer has stepped away from the model by a distance equal to the walk distance by shifting both the eye point and the view center backwards along the line of sight.

Turning

TurnAngl controls how far the observer will turn with TurnLeft and TurnRight.

➔ To change the turning distance:

1. Choose TurnAngl from the WalkThru menu. A list of angles is displayed.
2. Choose or type a value and press Enter.

Choose TurnRight or TurnLeft from the WalkThru menu. TurnRight turns the observer's cone of vision to the right (clockwise) by an angle equal to the TurnAngl. TurnLeft turns the observer's cone of vision to the left (counterclockwise) by an angle equal to the TurnAngl.

Stepping

StepDist sets the step distance used with StepUp, StepDown, StepLeft and StepRight.

➔ To change the stepping distance:

1. Choose StepDist from the WalkThru menu. A list of values is displayed.
2. Choose or type a step distance and press .

StepUp moves both the observer's eye point and the view center upward, away from the ground plane. StepDist controls the distance that the observer's eye point is elevated. With StepDown, both the observer's eye point and the view center move downward, toward the ground plane. StepDist controls the distance that the observer's eye point is lowered.

StepLeft shifts the view to the right, as if the observer took a step to the left. Technically, this option causes both the observer's eye point and view center to move to the left by a distance equal to the StepDist. In this context, *left* means the observer's left, facing toward the view center. StepRight shifts the view to the left, as if the observer took a step to the right by causing both the observer's eye point and view center to move to the right by a distance equal to the StepDist. In this context, *right* means the observer's right, facing toward the view center.

Looking

LookAngl sets the angle that the line of sight changes using LookUp or LookDown.

➔ To set the rotation angle of the observer's line of sight:

1. Choose LookAngl from the WalkThru menu. A list of values is displayed.
2. Choose or type a value and press .

LookUp changes the view to simulate looking up, away from the ground plane. The eye point remains fixed while the view center moves up, away from the ground plane by an angle equal to the LookAngl.

LookDown changes the view to simulate looking down, toward the ground plane. The eye point remains fixed while the view center moves down, toward the ground plane by an angle equal to the LookAngl.

Saving a View

➔ To add the current perspective view to the list of saved views:

1. Choose AddView from the WalkThru menu.
2. Type the name of the new view, up to eight characters, and press . The view is added to the list of saved views.

Using the o2c-based Object Viewer

The Object Viewer lets you see your model and enhance the image by controlling the rendered colors and textures.

➔ To use the Object Viewer:

1. Select Object Viewer from the View pull-down menu. The o2c-based Object Viewer window appears.



Figure 26.19: The Object Viewer

2. Use the toolbar icons to control the view:
 - Sun – toggles sunlight on and off
 - Ambient lights 1, 2, and 3 – toggles ambient lighting on and off
 - Lamp – toggles interior lighting on and off
 - Sun/moon – switches between day and night
 - Parallel/perspective – switches between views
 - Clockwise rotation – hold down to move around the Z axis
 - Counterclockwise rotation – hold down to move around the Z axis
 - Decrease field of view
 - Increase field of view
 - Plan view
 - Elevation views – left, right, front, and back

- Isometric view
 - Oblique or axonometric view
3. Depress and hold the left mouse button while you move the cursor to dynamically adjust the view.
 4. Depress and hold the right mouse button while you draw the cursor toward you or away from you to zoom in and out. Alternatively, use the + and – buttons at the bottom right corner of the Object Viewer to zoom in (+) and out (-).
 5. Right-click to open the o2c pop-up menu. From here, you can further control the view or save it to a file.
 - Show all - makes the entire model fit in the screen. Or click on the “a” button at the junction of the scroll bars.

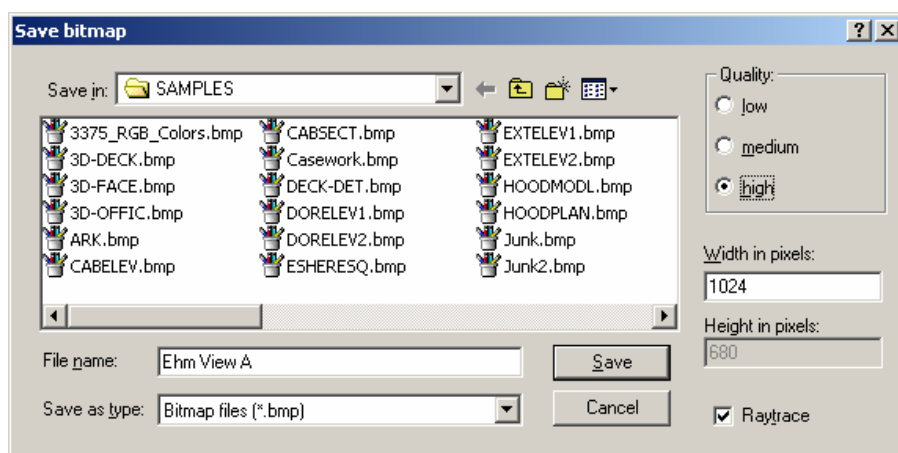


Figure 26.20: The Save bitmap dialog box

- Start raytrace - renders the model. Or click on the R button at the right edge of the horizontal scroll bar of the Object Viewer to raytrace the model.
- Save image - creates a permanent high-quality rendering. The Save bitmap dialog box lets you control the quality of the file while you save the view as a bitmap image. In the Quality section, high produces the best results, but it takes longer to process. Set the Width in pixels (try 320 for thumbnails, 640 for presentations, and 1024 for printing); DataCAD uses the width to automatically calculate the Height in pixels. Check Raytrace if you want shadows and reflections in your saved image; however, this takes a long time to process. Type the File name and select the appropriate folder (shown in the Save in box). Click on the Save button when you are finished and want to return to the Object Viewer.

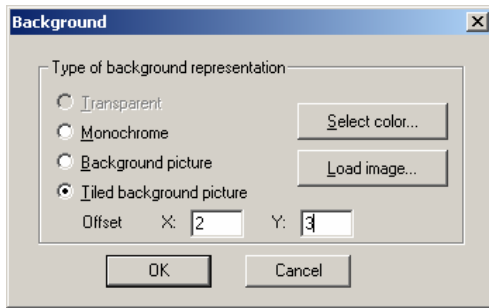


Figure 26.21: The Background dialog box

- Background – controls what is behind the model in the Object Viewer. Use Load image to access lists of available bitmap files; use Select color to choose from the Color dialog box. If you selected Load image, you can also check “Tiled background picture” to put many copies of the bitmap behind your model and further control the X and Y offset. If you clicked Select color, all options except Monochrome are unavailable (greyed).
6. Exit the Object Viewer by clicking on the X in the right corner of the title bar.

Color Mapping

Using the View/Shader color map dialog box, you can define materials for rendering and assign them to specific colors in the DataCAD palette. Mapping entity colors to material definitions controls rendered output. The material settings for a given drawing are stored in a .DMF (DataCAD Material File) file. You can save custom material files for use in other drawings. Also, you can save individual material definitions and load them one at a time. These definitions are compatible with other o2c-based programs such as DCvis, ArCon, and o2c Composer.

DataCAD lets you choose from a standard set of matte, glossy, and glass materials. By adjusting the various settings and experimenting a bit, you'll be able create your own materials to produce artistic, photo-realistic renderings of DataCAD models.

➔ To use the Viewer/Shader color map:

1. Access material settings for a given color by using (Ctrl)-right-click on an entity in the DataCAD Drawing Window. A small pop-up menu appears. For this example, just the STUCCO layer is active.
2. Select o2c Settings from the pop-up menu. This opens a dialog box that lets you precisely define the materials. The name of the active material file appears in the title of the Color Map dialog box.
3. Look at the Preview window to see the results of your choices in the rest of this dialog box.

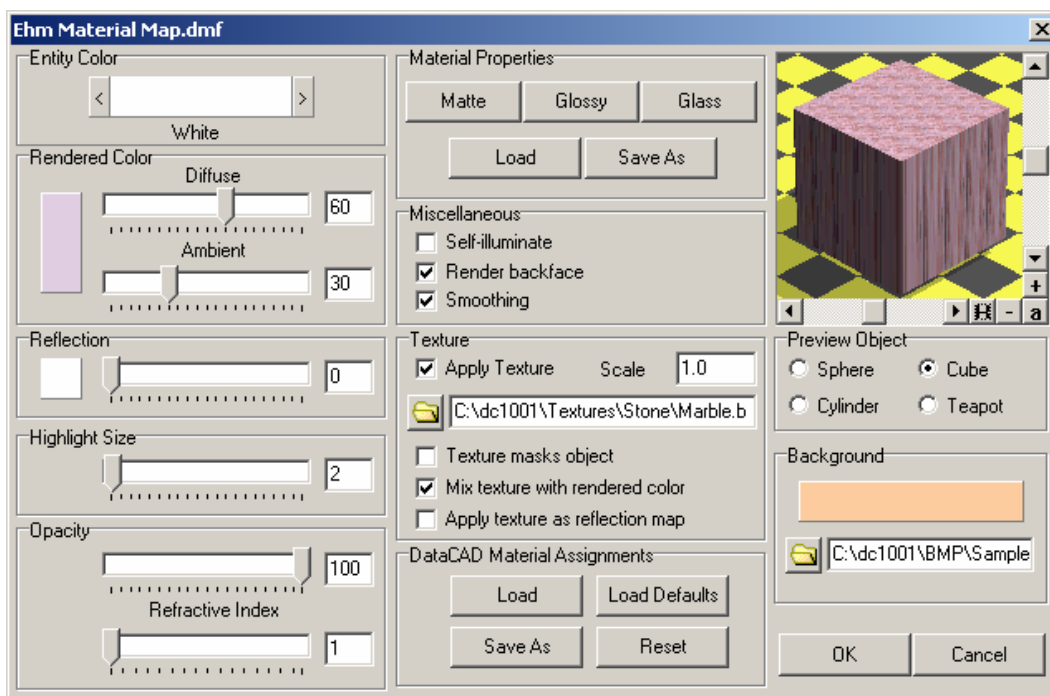


Figure 26.22: The color mapping decisions you make affect the Preview model.

4. Use the slider bars, input boxes, and check boxes in the various sections of the dialog box to define the material.
 - **Entity Color** – click on the rectangle to make your selection from the Color Palette. Alternatively, click on the arrows on the right and left sides of the color box to scroll through options in the palette. All entities that have the same color will also have the same material properties. The first 15 colors shown on the ColorPalette are most frequently used in DataCAD and are the most popular; however, you can define up to 255 unique materials per drawing by using custom colors (beyond yellow).
 - **Rendered Color** – click on the rectangle to make your selection from the Color Palette. Use the slider bars to further determine the proportion of diffuse and ambient lighting properties of materials. A high diffuse proportion will cause the light from defined light sources to reflect strongly into various nonspecific directions. Use low diffuse values to simulate light-absorbing materials such as velvet. If the ambient proportion is too high, the object will appear relatively pale (like plastic) because it will very strongly reflect the environment's light. Rendered Color is the material color for all entities that are the current Entity Color. For example, if you choose white in the Entity Color section, the Rendered Color you select for all white entities could be Dahlia Purple or any other color in the Color Palette, including a custom-mixed color that you define with the Color Picker dialog box.
 - **Reflection** - defines both the color and the proportional degree for reflections. In principle, the color defines the highlight on the objects. Higher reflection proportion values simulate glossy surfaces, such as chrome.

- **Highlight Size** – determines the size of reflected highlights from light sources. Adjustments to this value can only be seen in the o2c Player when using the raytrace option; they are not shown in the Preview window. You must use a relatively high reflective proportion to see the effect of changing this value. In principle, this value measures the intensity of a highlight as a function of the angle. The higher the value is, the smaller the highlight becomes. For matte-shining materials (such as unpolished gold or anodized aluminum), use small values around 3; for high gloss materials (such as polished gold or polished plastic), use large values.
- **Opacity** – the top slider bar determines the degree to which the material is opaque or transparent. A value of 50% means that when examining an object with this material, 50% of the color behind the object shows through it and 50% of the object's color is visible. The bottom slider bar affects how refractive (or hard) the transparent material is. To simulate glass, use a refractive index of 12; to simulate something like bubbles in water, use a refractive index smaller than 10. To see how your selections affect the material, you have to raytrace the object in the Preview window.
- **Material Properties** – mutually exclusive buttons apply preset Matte, Glossy, or Glass properties to the material. When you first work with the color map, you may rely on the presets; however, you will soon feel more confident about applying your own definitions via the slider bars and other options in this dialog box to produce photo-realistic materials. These presets affect the slider bars throughout the dialog box that keep the current rendered color, but change its reflective properties. Load lets you select an o2c color that will appear in the rectangle in the Rendered Color section; Save As lets you name and store your material definitions.

When simulating real-world elements using computer graphics, the surface properties of a material are based on various parameters. The level of realism depends partly on how the parameters are set. The o2c Player uses a standard computer material model known as PHIGS PLUS. This model simulates real-world material properties, but real-time renderers such as the o2c Player have limitations on how accurately they can be depicted in final rendered output.

The way an object (or surface) appears is affected by both the material properties and the lighting (including all sources which light the object). A rendered material is made up of a layering of different lighting properties (the diffuse portion simulates matte properties; the ambient portion reflects light in the environment; the reflective portion simulates polished material properties; and the opacity portion simulates transparent material properties).

By using bitmaps, you can also apply textures to enhance the material definition to simulate real-world surfaces such as carpet, tile, wallpaper, and brick. The preview window lets you see how a material will look before you apply it to your model.

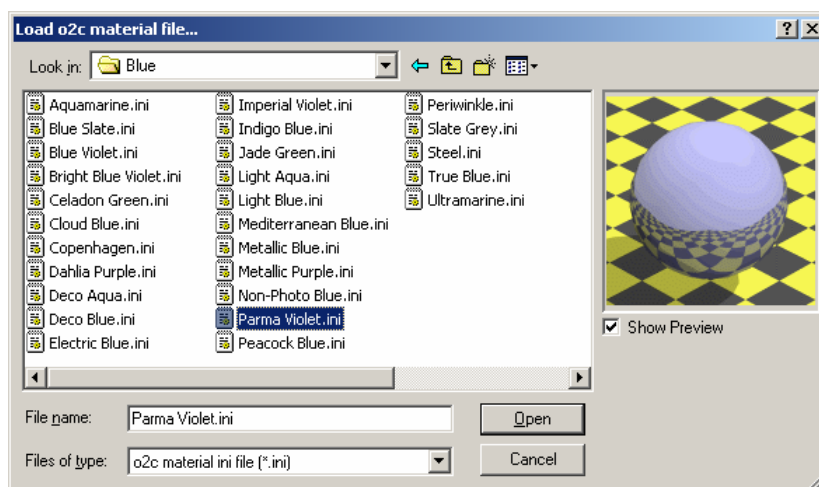


Figure 26.23: The Load o2c material file dialog box

- **Miscellaneous** – check the boxes that apply: self-illuminate, render backface, and allow smoothing. Select self-illuminate to see the material even without lights. This causes an object to simulate a light source, but it will not illuminate its surroundings. This material setting is ideal for neon and other low-level lighting that is normally used in dark settings. Render backface forces the o2c Player to render both sides of a surface. By default, render backface is on for all colors. Smoothing blends a material across adjacent, noncoplanar polygons (such as a mesh surface). Otherwise, each polygon will be rendered independently, revealing the creases between adjacent polygons. This option is especially useful for rendering reflective and transparent objects that are made up of many polygons (such as a sphere or a cylinder). Without this option, shiny objects will reflect a fragmented view of the surrounding environment.
- **Texture** – controls the interaction of textures and the defined material. Check Apply Texture to activate other options. Experiment until you get the results you want in the Preview window.

In some cases, you may want to mix the colors of the texture with the diffuse color. This makes it possible to tint (or lighten or darken) a texture without modifying the original bitmap.

If you check Apply texture as reflection map, textures appear to be reflected from, rather than mapped onto, a given surface. This simulates optimum mirror images, where the object's texture, not its environment, is reflected in the object.

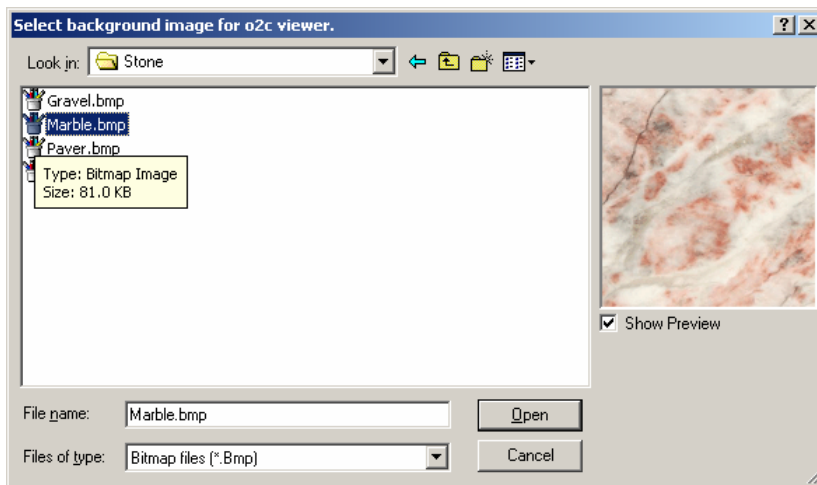


Figure 26.24: Select the background image for the o2c viewer.

- **DataCAD Material Assignments** – affect your settings, saved in a .DMF (DataCAD Material File). The Load button lets you access material assignments you created for other models. Save As lets you give the current definition a unique name. The current material file name is displayed in the title bar. In this example, we used the name Ehm Material Map that DataCAD saved as a .DMF file.
- **Preview Object** – lets you apply your material definitions to different shapes: sphere, cube, cylinder, or teapot. One object may give you a better idea of how the material definition will affect the entities in your drawing than another. For example, if you wanted a velvety material, you probably would steer away from displaying it on the teapot.
- **Background** – controls the color behind the image. This is a global setting, independent from the material file. When you click on the rectangle, the Color Palette appears so that you can select a color.

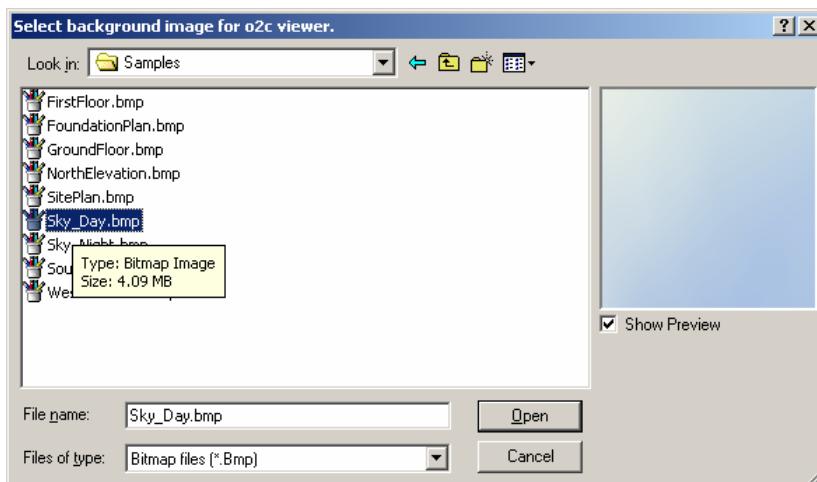


Figure 26.25: Select the background image for the o2c viewer.

- Preview window – lets you see your material assignments. This is a mini o2c object viewer. Click on “R” to raytrace the material. Some material settings can only be seen in a raytraced view. Although the default object is a sphere, you could change this to a cube, cylinder, or teapot to give you a better idea of how the material will look in your model.
5. Click on OK when you are satisfied with all the settings in the color mapping dialog box.
 6. Select Object Viewer from the View menu. The Object Viewer appears. The decisions that affect the STUCCO layer (see Figure 26.22) are displayed in the Object Viewer. If you are unhappy with the way the material looks, you can return to step 1 and go through the process again.

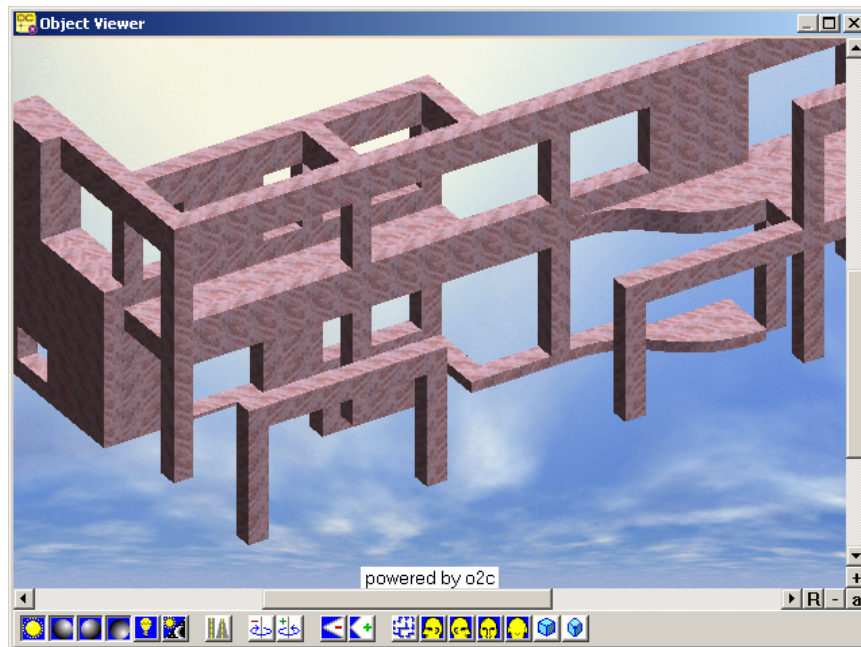


Figure 26.26: The results of color mapping decisions shown in Figure 26.22

Framing Your Model

27

FrameIt is a utility that helps you frame 3D floors, walls, and roofs. The RoofIt macro is used to create the outer portion of the 3D roof.

In this chapter:

- ⊕ Framing floors
- ⊕ Framing walls
- ⊕ Framing roofs
- ⊕ Saving and using framing settings

Framing Floors

➔ To frame a floor:

1. Choose Floors from the FrameIt menu.
2. Verify settings for Plate, Joist, and SubFloor. See “Sill Plate, Joist, and Sub-Floor Settings” below for more information.
3. Choose either Center or Sides to define the floor boundary from the center line of the vertical member or from one side of the floor to the other.
4. Define the floor to frame by defining a floor boundary or by choosing Select and selecting an existing floor boundary. To define the floor boundary, object snap to each point of the floor boundary, tracing the floor. If you make a mistake, use the Backup button to erase the last point you selected.
5. Choose Opening to define floor openings, such as chimneys or stairs. Select the first point of the floor opening, move the mouse to stretch the rubberband box around the opening, and select the second point of the rubberband box to define the floor opening.
6. Choose Beam to define beam style and location. Set the following options as necessary:

Single	Models single beams
Double	Models double beams
Custom	Customizes the number of entities that make up a beam
Color	Assigns a beam color. See “Color Menus” in “The Drawing Board” chapter for more information about how to choose a color.
Size	Sets a beam size (choose Change to change a standard lumber size in the menu)
Flush	Positions the top of the beam flush or equal to the top of the joist
Below	Positions the top of the beam equal to the bottom of the joist
Change	Changes the current framing member attributes (those attributes set when you selected or defined the floor boundary)
FileI/O	Accesses file options to save the current framing settings in a file, load saved settings, or delete or rename framing settings files

7. Make any changes before choosing Build. Choose Edit or Change to access DataCAD editing options, including Move, Copy, Rotate, Mirror, Stretch, Enlarge, Erase, and Layers, or to globally change any floor member attributes. If you choose Edit or Change, you must choose Build again for those changes to take effect.
8. Save floor settings to a file by choosing SaveFile from the FileI/O menu.
9. Choose Build and select the boundary on which to generate a floor system. Build generates a framing system based on the settings you defined above. See the “The Drawing Board” chapter for more information on using the selection menu to choose the boundary to frame.

Use the following toggles before you select the boundary to frame for added control over the Build process:

SchemLyr	Builds framing members on the schematic layer
ActvLyr	Builds framing members on the active layer

- | | |
|----------|--|
| FlrsOnly | Builds only floors with the current settings; otherwise, DataCAD builds all members; only available in the Floors/Build menu |
| WalsOnly | Builds only walls with the current settings; otherwise, DataCAD builds all members; only available in the Walls/Build menu |
| RfsOnly | Build only roofs with the current settings; otherwise, DataCAD builds all members; only available in the Roofs/Build menu |
10. Finish framing. When framing is complete, you have access to the following DataCAD options: 3DViews, Hide, and Layers. See the “Viewing Your Model” and “Before You Draw” chapters for more information on 3DViews, Hide, and Layers.
 11. Choose Exit to return to the Floors menu.

Sill Plate, Joist, and Subfloor Settings

Use Plate to set the following sill plate options:

DoPlate	Models sill plates
Single	Models single plates
Double	Models double plates
Custom	Customizes the number of entities that make up the plate
Color	Assigns a plate color. See “Color Menus” in “The Drawing Board” chapter for more information about how to choose a color.
Size	Sets a plate size; choose Change to change a standard lumber size in the menu

Use Joist to set the following joist options:

DoJoist	Models the joists
Size	Assigns a joist size, or defines a custom joist size; choose Change to change a standard lumber size in the menu
Spacing	Assigns joist spacing (measure spacing from the center line of one joist to the center line of the next joist)
Color	Assigns a joist color. See “Color Menus” in “The Drawing Board” chapter for more information about how to choose a color.
Angle	Changes the angle in which your joist member will be modeled within the schematic

Use SubFloor to set the following subfloor options:

DoSubflr	Models the subflooring
Single	Models single layer subfloors
Double	Models double layer subfloors
Custom	Customizes the number of entities that make up a subfloor
Color	Assigns a subfloor color. See “Color Menus” in “The Drawing Board” chapter for more information about how to choose a color.
Thicknss	Sets the subfloor thickness for each layer

Framing Walls

➔ To frame a wall:

1. Choose Walls from the FrameIt menu.
2. Verify the settings for WallHght, Plate, and Stud. See “Wall Height, Plate, and Stud Settings” later in this chapter for more information.
3. Choose either Center or Sides to define the wall boundary from the center line of the vertical member or from one side of the wall to the other.

4. Choose either Exterior or Interior. Use Exterior to create outside wall systems; DataCAD assumes this will be a closed boundary. Use Interior to create inside wall systems; DataCAD toggles Closed off and you are drawing a single or string-line wall that does not close itself.
5. Define the wall to frame by defining a wall boundary or by choosing Select and selecting an existing floor or wall boundary. If defining the wall by the boundary, object snap to the first point of the wall boundary to frame and continue tracing the boundary from that point. If you make a mistake, use the Backup button to erase the last point you selected.
6. Choose Opening to define wall openings. Choose either Window or Door (for Exterior) or PassThru or Door (for Interior). If you choose Window or PassThru, verify the settings for SillHgt, HeadHgt, and HeadStyl. For Door, define only HeadHgt and HeadStyl.

Window	Defines window openings in the wall
Door	Defines door openings in the wall
SillHgt	Defines the sill height; available only when Window is toggled on
HeadHgt	Defines the header height of the door
HeadStyl	Defines the header style:
Cripple	Member with smaller studs that run from the top of the header to the bottom of the lower top plate; maintains stud spacing
Boxed	Solid member that runs from the top of the header to the bottom of lower top plate
Change	Changes any opening attributes
FileI/O	Accesses file options to save the current framing settings in a file, load saved settings, or delete or rename framing settings files

7. Make any changes before choosing Build. Choose Edit or Change to access DataCAD editing options, including Move, Copy, Rotate, Mirror, Stretch, Enlarge, Erase, and Layers, or to globally change any wall member attributes. If you choose Edit or Change, you must choose Build again for those changes to take effect.
8. Save wall settings to a file by choosing SaveFile from the FileI/O menu.
9. Choose Build and select the boundary on which to generate a wall system. Build generates a framing system based on the settings you defined above. See “The Drawing Board” chapter for more information on using the selection menu to choose the boundary to frame.

Use the following toggles before you select the boundary to frame for added control over the Build process:

SchemLyr	Builds framing members on the schematic layer
ActvLyr	Builds framing members on the active layer
FlrsOnly	Builds only floors with the current settings; otherwise, DataCAD builds all members; only available in the Floors/Build menu
WalsOnly	Builds only walls with the current settings; otherwise, DataCAD builds all members; only available in the Walls/Build menu
RfsOnly	Build only roofs with the current settings; otherwise, DataCAD builds all members; only available in the Roofs/Build menu

10. Choose Exit when framing is complete to return to the Walls menu.

Wall Height, Plate, and Stud Settings

Use WallHgt to set wall height options. Choose a wall height from the list or type a wall height and press **[Enter]**.

Use Plate to set the following top and bottom plate options:

DoTop	Models top plates
Single	Models single top plates
Double	Models double top plates
Custom	Customizes the number of entities that make up a top plate
Color	Assigns a top plate color. See “Color Menus” in “The Drawing Board” chapter for more information about how to choose a color.
DoBottom	Models bottom plates
Single	Models single bottom plates
Double	Models double bottom plates
Custom	Customizes the number of entities that make up a bottom plate
Color	Assigns a bottom plate color. See “Color Menus” in “The Drawing Board” chapter for more information about how to choose a color.

Use Stud to set vertical framing member options:

DoStud	Models the vertical stud member
Size	Assigns a stud size or defines a custom stud size
Spacing	Assigns stud spacing (measure spacing from the center line of one stud to the center line of the next stud)
Color	Assigns a stud color. See “Color Menus” in “The Drawing Board” chapter for more information about how to choose a color.

Framing Roofs

Use Roof Framing to create three-dimensional roof framing systems. You can frame the following roof types: Shed, Gable, Hip, and Gambrel. See “3D Roofs” later in this chapter for more information on roof types

➔ To frame a roof:

1. Choose Roofs from the FrameIt menu.
2. Choose the roof type to frame: Shed, Gable, Hip, or Gambrel.
3. Choose Settings to set the options for roof settings, including pitch, purlin height, rafters, sheathing, and overhang. See “Settings” below in this section for more information.
4. Define the roof by defining a roof boundary or by choosing Select and selecting an existing wall or roof boundary. To define the roof with an existing boundary, choose that boundary and continue with step 6. To define the roof by a boundary, object snap to the first and second corners of the roof boundary to define the length of the roof parallel to the ridge. Make sure that you define the roof boundary at a Z-base set to the plate height of the wall.

The following options appear on the menu:

Closed	Models a closed roof boundary
Backup	Backs up one roof corner so you can enter a new position
Cancel	Returns you to the FrameIt menu without defining a roof perimeter
Close/Exit	Stops defining the roof perimeter

5. Choose the third corner of the roof perimeter to define the width of the roof perpendicular to the ridge if you define the roof by the boundary. The roof boundary is calculated and closed.
6. Choose Opening to define roof openings, such as chimneys. Select the first point of the roof opening, move the mouse to stretch the rubberband box around the opening, and select the second point of the rubberband box to define the roof opening.

7. Make any changes before choosing Build. Choose Edit or Change to access DataCAD editing options, including Move, Copy, Rotate, Mirror, Stretch, Enlarge, Erase, and Layers, or to globally change any roof framing member attributes. If you choose Edit or Change, you must choose Build again for those changes to take effect.
8. Save roof settings to a file by choosing SaveFile from the FileI/O menu.
9. Choose Build and select the boundary on which to generate a roof system. Build generates a framing system based on the settings you defined above. When you define a roof boundary, FrameIt bases the plate height on the current Z-Base setting. When you use Select to select an existing wall boundary, FrameIt uses the plate height of the wall. FrameIt always centers ridge boards when framing roofs
10. Choose Exit when framing is complete to return to the Roofs menu.

Settings

Use Pitch to set a pitch for Shed, Gable, and Hip roofs. Pitch is defined by rise and run; choose a pitch from the menu, or choose Custom, type a pitch, and press **[Enter]**. Use UpPitch and LwrPitch to set the pitch for the upper and lower sections, respectively, of a Gambrel roof.

Use MidRdgHt to set the height for purlins for Gambrel roofs. Choose a height from the menu, or choose Custom, type a height, and press **[Enter]**.

Use Rafters to set the following rafter options:

DoRafter	Models rafters; models collar ties every four feet
Size	Sets a rafter size from the list, or defines a custom rafter size
Spacing	Sets rafter spacing or defines custom rafter spacing (measure spacing from the center line of one rafter to the center line of the next rafter)
Color	Sets a ridge board and rafter color. See "Color Menus" in "The Drawing Board" chapter for more information about how to choose a color.
EaveSqr	Models fascia square to the angle of the rafter
EavePlmb	Models fascia parallel to the wall
BrdMouth	Assigns a Bird's Mouth dimension; choose a dimension from the menu, or choose Custom, type a dimension, and press [Enter]

Use Sheathng to set the following roof sheathing options:

DoSheath	Models roof sheathing
Single	Models single layer roof sheathing
Double	Models double layer roof sheathing
Custom	Models a custom number of roof sheathing layers
Color	Assigns a roof sheathing color. See "Color Menus" in "The Drawing Board" chapter for more information about how to choose a color.
Thicknss	Assigns the thickness of the roof sheathing material or defines a custom thickness

Use the following OverHang options to set the overhang distance. Measure OverHang from the outside face of the sheathing to the outside face of the rafter.

EaveOhng	Assigns an overhang distance at the eave
RakeOhng	Assigns an overhang distance on the gable end

RoofIt automates the construction of 3D roof models. You define the roof boundary and RoofIt automatically calculates and constructs a roof based on your settings.

The standard roof types available with RoofIt are Gable, Hip, Boston, Gambrel, Mansard, and Shed. You can also create your own customized roof shape. You can vary the pitch, overhang, and plate height settings, and specify the type of soffit and fascia you want to model. RoofIt also lets you join a new roof to an existing roof. You can create dormers or other complex roof shapes using this feature. A complete roof model is composed entirely of 3D polygons. Because of this, you can easily process the model with DataCAD's hidden line removal feature, Hide.

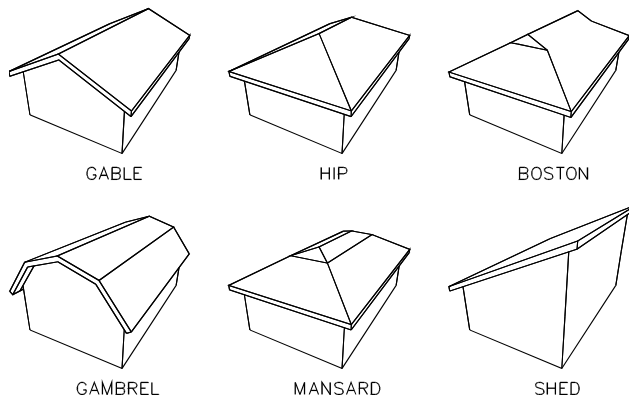


Figure 27.1: Standard roof types

➔ To create a roof model:

1. Choose RoofIt from the Toolbox menu. The following options are displayed:

RectRoof	Constructs an orthogonal (rectangular) roof with no variation in settings; RectRoof is the easiest way to model a rectangular roof. RectRoof and PolyRoof are mutually exclusive; only one option will be displayed in the menu at any given time. If RectRoof is displayed in the RoofIt menu, clicking on it will change it to PolyRoof, putting DataCAD in polygonal roof drawing mode; you can begin drawing polygonal roofs. To change back to rectangular roof drawing mode, click on the PolyRoof option to change it to RectRoof.
PolyRoof	Constructs a polygonal roof that is not orthogonal or a roof that has varied pitch, overhang, or plate height settings
Gable	Available roof type
Hip	Available roof type
Boston	Available roof type
Gambrel	Available roof type
Mansard	Available roof type
Shed	Available roof type
Join	Defines a Gable, Hip, or Shed roof that connects to an existing roof plane; the roof that you join onto may be any roof type you create using RoofIt
Settings	Sets roof options depending on the type of roof you're drawing. See "Setting Options" later in this section.
PlatHgt	Sets different plate height values for each roof line than what's currently set in Settings/PlatHgt during PolyRoof creation; when PlatHgt is on, RoofIt prompts you for the plate height of each roof; only changes with Gable and Gambrel roofs
Pitch	Sets different pitch values for each roof line than what's currently set in Settings/Pitch during PolyRoof creation; when Pitch is on, RoofIt prompts you for the pitch of each roof side; only changes with Gable and Gambrel roofs

- | | |
|----------|---|
| Overhng | Sets different overhang values for each roof line than what's currently set in Settings/Overhng during PolyRoof creation; when Overhng is on, RoofIt prompts you for the overhang of each roof side; only changes with Gable, Gambrel, and Shed roofs |
| Layers | Shortcut to the Utility/Layers menu without exiting RoofIt. See "Layers" in the "Before You Draw" chapter. |
| Hide | Shortcut to the 3DEdit/Hide menu without exiting Roof It. See the "Viewing Your Model" chapter for more information about Hide, 3DViews, and 3D Entity. |
| 3DViews | Shortcut to the 3DViews menu and look at your 3D model in different orientations without exiting RoofIt. See the "Viewing Your Model" chapter for more information about Hide, 3DViews, and 3D Entity. |
| 3DEntity | Shortcut to the 3DEntity menu without exiting RoofIt. See the "Viewing Your Model" chapter for more information about Hide, 3DViews, and 3D Entity. |
2. Choose RectRoof or PolyRoof to define a rectangular or polygonal roof. See "Defining a Roof Boundary" later in this chapter for more information.
 3. Choose one of the six standard roof types: Gable, Hip, Boston, Gambrel, Mansard, or Shed.
 4. Choose Join if you are defining a roof to join with an existing Gable, Hip, or Shed roof plane.
 5. Choose Settings to set roof options (if you don't need to vary any values, skip to step 6). Choose the option to set. These options vary depending on which roof type you selected in step 3. See "Setting Options" later in this chapter for more information
 6. Choose PlatHgt, Pitch, or Overhng from the RoofIt menu if you are creating a Gable, Gambrel, or Shed polygonal roof. This lets you change that setting for each roof edge you draw. Once you select the first point of the roof boundary, you can't move the control from one parameter to another. You can only change the value of one parameter as you define the roof boundary.
 7. Define a roof boundary by entering length and width values for a rectangular roof or by entering a series of four points for a polygonal roof. See "Defining the Roof Boundary" later in this chapter for more information.
 8. Choose the roof plane to join to when prompted if you are defining a roof to join.

Defining the Roof Boundary

The method you use to define the roof boundary depends on whether the RectRoof or PolyRoof option is displayed in the RoofIt menu. (The option that is displayed is the type of roof you'll draw.)

Rectangular Roofs

When you model a rectangular roof, you define the roof shape by selecting points (in a clockwise direction) that define its length and width. This results in an orthogonal roof shape.

➔ To define a rectangular roof boundary:

1. Choose PolyRoof if RectRoof is not displayed in the RoofIt menu. Then, toggle RectRoof on.
2. Select the first point of the roof boundary; this point is the first point of the roof length, which runs parallel to the ridge of the roof.
3. Select the second point of roof length.
4. Select the third point of the roof boundary to define the roof width. The roof width is the distance from one side of the building to the other; this is perpendicular to the roof length and roof ridge. On a Shed roof, this point locates the ridge (minus the overhang).

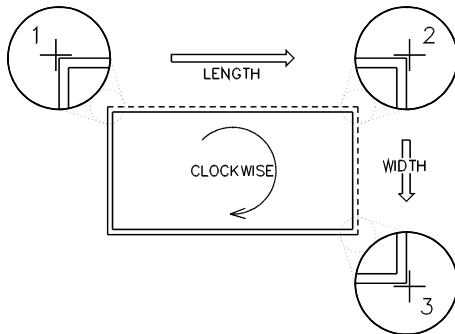


Figure 27.2: Defining a rectangular roof boundary

Polygonal Roofs

To model a non-orthogonal roof or to model a roof with varying parameters (PlatHgt, Pitch, or Overhng), use PolyRoof.

When you define a polygonal roof, you enter a series of four points. These points trace the edges of the roof boundary in a clockwise direction. The first and third edges of the boundary run parallel to the ridge; and the ridge bisects the second and fourth edges. Because the roof shape may be either orthogonal or non-orthogonal, the edges of the roof don't have to be parallel and perpendicular to the ridge.

➔ To define a polygonal roof boundary:

1. Choose RectRoof if PolyRoof is not displayed in the RoofIt menu. Then, toggle PolyRoof on.
2. Select the first point of the roof boundary. This is the first point of the roof length.
3. Enter the appropriate value if you're drawing a Gable, Gambrel, or Shed roof and you toggled PlatHgt, Pitch, or OverHng on in the RoofIt menu. RoofIt applies this value to the first edge of the roof.
4. Select the second point of the first roof edge.
5. Enter a value if you're drawing a Gable, Gambrel, or Shed roof and you toggled Overhng on in the RoofIt menu. RoofIt applies this value to the second edge of the roof.

6. Select the third point of the roof boundary. The second edge of the roof boundary is bisected by, but not necessarily perpendicular to, the ridge. The Shed roof is an exception; on a Shed roof, this point locates the ridge (minus the overhang).
7. Enter a value if you're drawing a Gable, Gambrel, or Shed roof and you toggled PlatHgt, Pitch, or OverHng on in the RoofIt menu. RoofIt applies this value to the third edge of the roof.
8. Select the fourth point of the roof; unless you toggled OverHng on in the RoofIt menu, the boundary closes and the roof model is constructed.
9. 9. Enter a value if you toggled Overhng on. RoofIt applies this value to the fourth edge of the roof. After you enter the value, the boundary closes and the roof model is constructed.

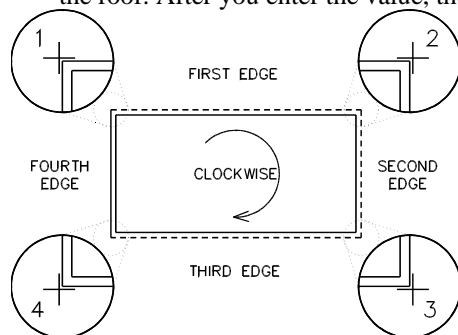


Figure 27.3: Defining an orthogonal PolyRoof boundary

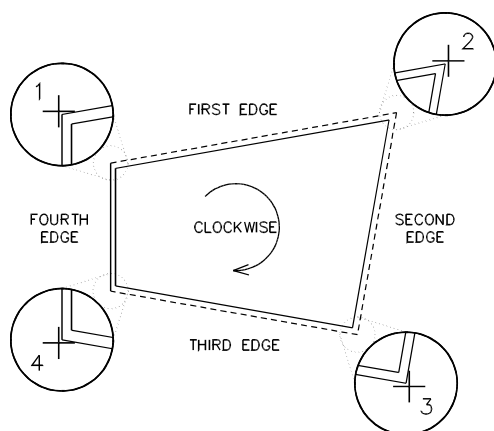


Figure 27.4: Defining a non-orthogonal PolyRoof boundary

Joining Roofs

Use Join to define a Gable, Hip, or Shed roof that connects to any one existing roof plane. The roof that you join onto may be any roof type created with RoofIt.

The following example shows how to use RectRoof to join a Hip roof; you can also use PolyRoof. Define the roof boundary in a clockwise manner regardless of which input mode you use: RectRoof or PolyRoof.

➔ To join two roofs:

1. Model the first roof using the steps outlined at the beginning of this section.

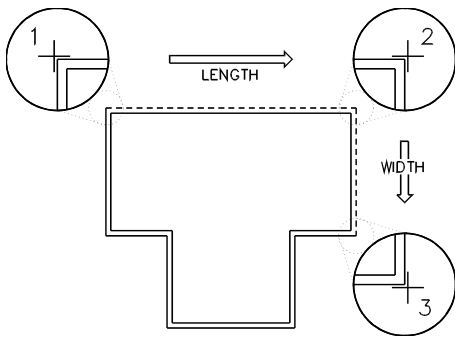


Figure 27.5: Defining the first boundary

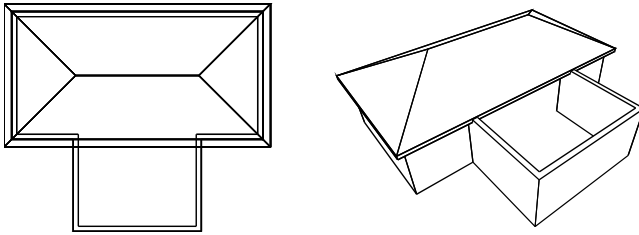


Figure 27.6: First boundary defined (plan view and perspective hidden-line image)

2. Toggle Join on in the RoofIt menu. This tells RoofIt that the next roof you define will join onto an existing roof.
3. Define the boundary of the second roof. RoofIt prompts you to select the roof plane to join onto. When defining the joining roof, if you selected any parameters to vary, RoofIt prompts you for values as outlined in the previous sections.

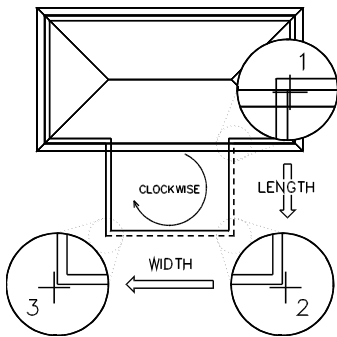


Figure 27.7: Join: Defining the second boundary

4. Select a point within the perimeter of the roof plane to join to. RoofIt highlights the roof plane and starts the join operation. The roof you defined extends to and clips at the selected roof plane.

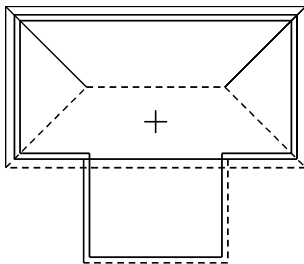


Figure 27.8: Selecting roof plane to join

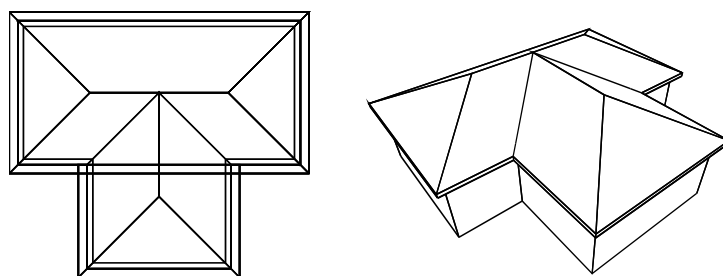


Figure 27.9: Result of join (plan view and perspective hidden-line image)

Settings Options

The options that appear in the RoofIt/Settings menu depend on the type of roof you're drawing. The following table illustrates which settings appear for each roof type:

	Gable	Hip	Boston	Gambrel	Mansard	Shed
PlatHgt	X	X	X	X	X	X
Pitch	X	X	X	X	X	X
MidRidge				X	X	
2ndPitch				X	X	
RoofThck	X	X	X	X	X	X
Overhng	X	X	X	X	X	X
GblOffst			X			
WallThck	X	X	X	X	X	X
Fascia	X	X	X	X	X	
EndWall	X			X		X

Settings options include:

PlatHgt	Sets the roof plate height; this value is an absolute Z-value and is not relative to the current Z-base and Z-height settings
Pitch	Sets the roof pitch in rise and run values (4:12, 5:12, 6:12, etc.); or enter a Custom pitch
MidRidge	Sets the purlin height on a Gambrel or Mansard roof; mid-ridge is a Z distance relative to the current PlatHgt setting
2ndPitch	Sets the second pitch on a Gambrel or a Mansard roof in rise and run values (4:12, 5:12, 6:12, etc.); or enter a Custom pitch
RoofThck	Sets the plate thickness of the roof
Overhng	Sets the roof overhang; measure overhang in the X, Y plane, from the roof definition boundary to the outermost point on the fascia
GblOffst	Sets the gable offset on a Boston roof; this distance is measured in the X, Y plane from the roof definition boundary to the exterior face of the gable end wall; only available when you choose to draw a Boston roof
WallThck	Sets the exterior wall thickness; controls the distance in the X, Y plane from the roof definition boundary to the intersection of the plate and the bottom plane of the roof
Fascia	Sets the width of the fascia on roofs with plumb fascia and horizontal soffits; does not affect the roof thickness (controlled by RoofThck); fascia width cannot exceed the plumb distance through the roof thickness; only available when SoffHorz and FaciaPlb are toggled on
SoffHorz/SoffIncl	Toggles between SoffHorz and SoffIncl to define the shape of the soffit; toggle SoffHorz on to draw the underside of the overhang (the soffit) horizontal, or parallel to the ground; toggle SoffIncl on to draw the soffit inclined, or parallel to the roof plane

FaciaPlb/FaciaSqr Toggles between FaciaPlb and FaciaSqr to define the shape of the fascia; toggle FaciaPlb on to draw the outside of the overhang (the fascia) plumb, or perpendicular to the ground; toggle FaciaSqr on to draw the fascia is square, or perpendicular to the roof plane

EndWall Toggles the construction of end walls as part of the roof model (end walls fill in the gaps between the plate and the underside of the roof); when you toggle EndWall on, RoofIt prompts you to enter the color of the end walls. EndWall only appears when you select a Gable, Boston, Gambrel, or Shed roof type. RoofIt determines the thickness of the end wall by using WallThck.

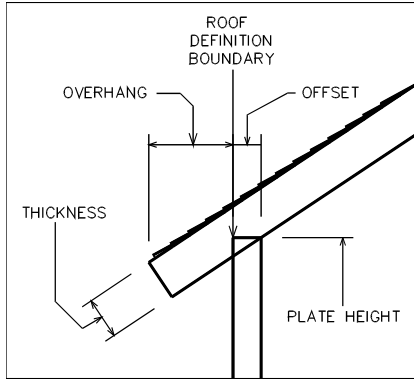


Figure 27.10: Roof parameters

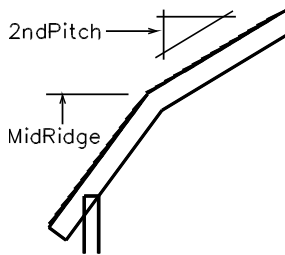


Figure 27.11: 2ndPitch and MidRidge settings

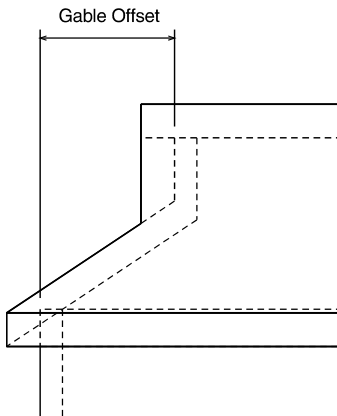


Figure 27.12: Gable offset

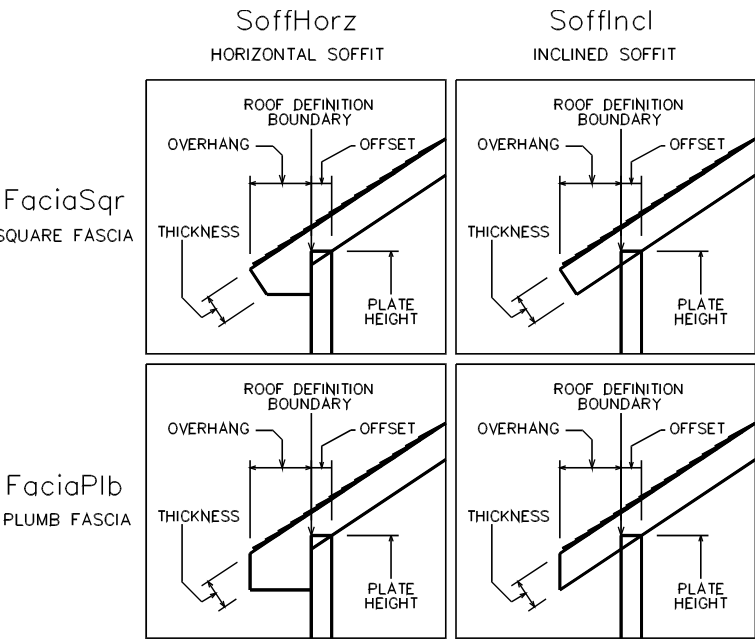


Figure 27.13: Types of soffit and fascia

Saving and Using Framing Settings

Use FileI/O to access file options:

- | | |
|----------|--|
| LoadFile | Loads a previously saved file of floor, wall, or roof framing settings |
| SaveFile | Saves the current framing settings in a file |
| DelFile | Deletes an existing file of framing settings |
| Rename | Renames an existing file of framing settings |

Modeling Windows and Doors

The AEC Modeler tool offers you simplified creation of 3D door and window modeling. You can model doors and windows in plan view or in elevation view.

In this chapter:

- ⊕ Modeling windows
- ⊕ Modeling doors

Modeling Windows

The Windows menu in AEC_MODL serves as the master level for accessing all three-dimensional window component groups. You can access window data, load and save previously designed windows, and select or enter window components and data. All windows open outward, away from you.

Cutting Walls

CutWall is a toggle used to automatically create voids in slabs or polygons for windows. It is available in the Windows menu when InPlan is toggled on. When CutWall is toggled on, a LyrSrch toggle also becomes available.

Window Files

The WndwFile option is a convenient way to expand on initial template designs, tailoring the product to your needs. The files then become additional resources for quick recall. The WndwFile menu options include:

LoadWind	Loads window settings from saved files; select or type a window filename and press [Enter] ; the window appears with all previously saved settings
SaveWind	Saves window settings to a file; type the filename and press [Enter]
DelWind	Deletes saved window files from the hard disk. To help prevent accidental deletions, the system prompts you for the name of the file you want to delete. Select the file you want to delete and press [Enter] . Choose Yes to delete the window file; otherwise choose No.
RenWind	Renames existing files; select a file to rename, type a new filename of up to eight characters long, and press [Enter]
CopyWind	Copies the contents of an existing window file to a new file; select the file to copy from and press [Enter] , then type the file to copy to and press [Enter]
Form	Displays a window data form. To save Form settings, press [Esc]

Window Forms

With the WndwForm option, you can view the settings for the current window. The design form offers a comprehensive view of the window parameters on one screen. These window parameters are divided into ten major groups.

Saving Windows as Symbols

Use the Template option to create and save windows as symbols, developing a library of window symbols that you can add quickly to new projects. See the “Symbols, Images, and Objects” chapter for more information on templates and symbols.

This Template option is a shortcut to the Template menu.

Window Height and Wall Thickness

You can set the head height and sill height of a window with the Head Hgt and Sill Hgt options, as well as use the Wall Thk option to change the wall thickness. The Head Hgt and Sill Hgt options only appear when In Plan is toggled on, while Wall Thk is only available when In Elev is toggled on. When you choose Head Hgt, sill Hgt, or Wall Thk, a list of values is displayed. Choose or type a new value and press **[Enter]**. The Head Hgt and Sill Hgt settings are relative to Z-base.

Window Types

The UnitType menu displays available window types. Hinged window types open outward, away from the designer. Sliding windows, when viewed from the interior design position, have a fixed sash on the right and a movable sash on the left.

Choose from the following window types:

Fixed	Windows with fixed panes of glass
Casement	Casement windows
Awning	Awning windows, hinged at the top
Hopper	Hopper windows, hinged at the bottom
DblHung	Double-hung windows
Sliding	Sliding windows
% open	Determines the amount that the window appears open in the model, with 0% being fully closed and 100% being fully open

Window Casings

The Casing option controls the casing or exterior window group. You can set any of the following window casing options:

AtHead	Models casing elements at the window header
Width	Sets the casing width at the window header, the window jamb, and the sill across the window plane; choose the Width option under AtHead, AtJamb, or AtSill, and then choose or type a width and press Enter
Thicknss	Sets the casing thickness at the window header, the window jamb, and below the sill; choose or type a thickness and press Enter
Extensn	Sets the extension length past the outside edge of the vertical casing components (jamb casing) and past the outside edge of the vertical jamb casing component to the sill. Choose or type an extension length and press Enter . For any extension, the measurement is always 0 when the outside edge of the vertical casing and the extension end are aligned. Positive numbers indicate an extension; negative numbers indicate a setback.
AtJamb	Models the casing element at the jamb (Width and Thicknss appear again below AtJamb)
AtSill	Models the casing elements below the sill. (Width, Thicknss and Extensn appear again below AtSill)

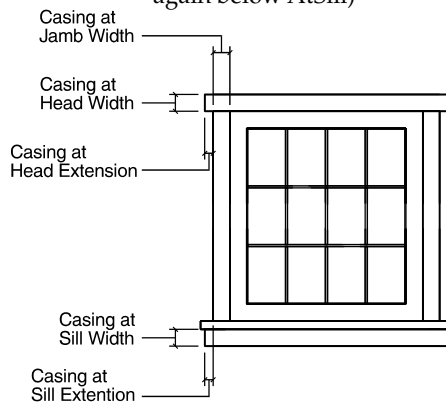


Figure 28.1: Window casing

Window Trim

Use Trim to control the trim or interior window component group. You can vary elements, such as the trim apron, according to aesthetic preference or design requirements:

AtHead	Models trim elements at the window header
--------	---

- Width** Sets the trim width at the window jamb, across the window jamb; choose or type a value and press **Enter**
- Thicknss** Sets the trim thickness at the window jamb and the window sill; choose or type a value and press **Enter**
- Extensn** Sets the trim extension length past the outside edge of the vertical jamb trim components (jamb casing) or past the outside edge of the vertical jamb casing component of the sill; choose or type a value and press **Enter**. For any extension, the measurement is always 0 when the outside edge of the vertical casing and the extension end are aligned. Positive numbers indicate an extension; negative numbers indicate a setback.
- AtJamb** Models trim elements at the jamb (Width and Thicknss appear again below AtJamb)
- AtSill** Models trim elements at the sill (Width, Thicknss and Extensn appear again below AtSill)

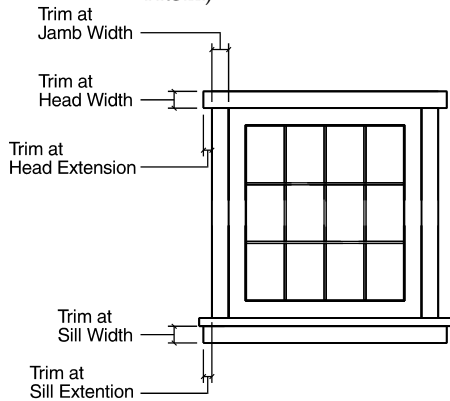


Figure 28.2: Window trim

Window Headers

The Head option lets you define the window header, or top of the frame. This area can become exceedingly detailed in a model rendering, with the inclusion of its structural and aesthetic elements in one master group. You can individually set the following components to provide a clearer view:

- Do Head** Models the header component
- HeadWdth** Sets the header width. Measure through the wall, from the inner surface of the wall, outward. This option appears when WalWdth is off. When you choose HeadWdth, a list of header width values appears. Choose or type a header width and press **Enter**.
- Head Thk** Sets the window header thickness. Measure from top, the wall opening, to bottom. When you choose Head Thk, a list of header thickness values appears. Choose or type a header thickness and press **Enter**.
- WalWdth** Calculates the header width equal to the thickness of the wall. When WalWdth is off, the width of the head is controlled by the header width setting.

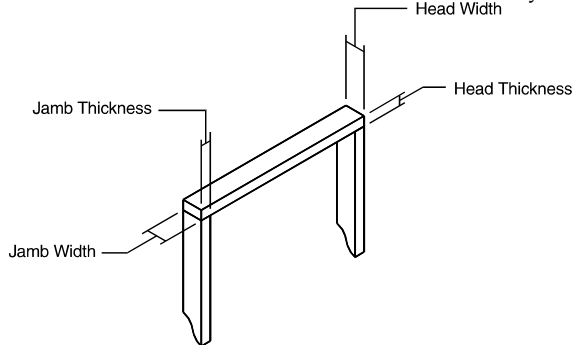


Figure 28.3: Window header

Window Jambs

Use the following Jamb options to define the window jamb:

Do Jamb	Models jamb components
JambWdth	Sets the jamb width. Measure through the wall from the inner surface of the wall, outward. This option appears when WalWdth is off. Choose JambWdth, and then choose or type a jamb width and press [Enter] .
Jamb Thk	Sets the jamb thickness. Measure from the void edge, or wall opening, inward toward the window center along the wall plane. Choose Jamb Thk, and then choose or type a jamb thickness and press [Enter] .
WalWdth	Sets the wall width. When this toggle is off, the width of the jamb is controlled by JambWdth.

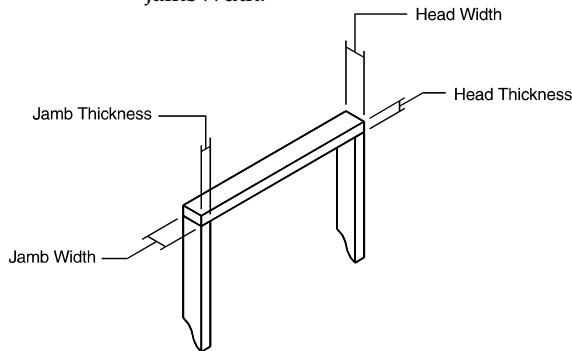


Figure 28.4: Window jamb

Window Sills

Sill lets you define the window sill. The sill is not broken down into stool (interior) and finish (exterior) sill components. Also, the sill horns are proportionate and do not carry different lengths on the interior and exterior extensions.

Do Sill	Models the sill
Thicknss	Sets the sill thickness from the bottom of the window opening, upward; choose Thicknss, and then choose or type a sill thickness and press (Enter)
In Extn	Sets the inside sill extension away from the plane of the wall toward the center of the room; choose In Extn, and then choose or type an inside sill thickness and press (Enter)
Out Extn	Sets the outside sill extension away from the plane of the wall outward; choose Out Extn, and then choose or type an outside sill extension and press (Enter)
SideExtn	Sets the inside and outside (sill horn) extensions from the sides of the window opening, outward, along the plane of the wall; choose SideExtn, and then choose or type a side extension and press [Enter]

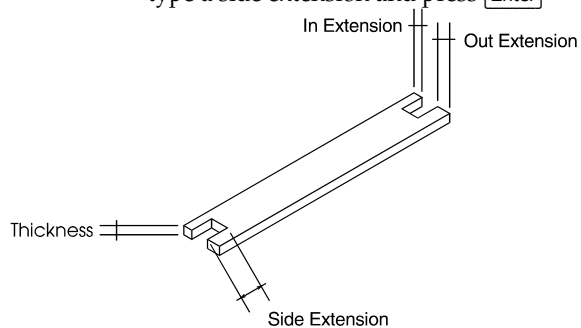


Figure 28.5: Window sill

Window Sashes

Sash defines the window sash assembly. The sash is the frame that holds the muntins and glass; without it, the muntins and glass would be suspended in 3D space. Single pane or double thermopane windows, while not having muntins, must have Centred toggled on to function within the confines of the wall plane.

Do Sash	Models the sash components; when Do Sash is off, muntins and glass are not added to the drawing even if DoMuntn and DoGlass are on
SashWdth	Sets the sash width for all components (top and bottom rails and stiles); choose SashWdth, and then choose or type a sash width and press <input type="button" value="Enter"/>
SashThk	Sets the sash thickness for all sash components; choose SashThk, and then choose or type a sash thickness and press <input type="button" value="Enter"/>
Centred	Centers a sash within the window frame. Centred and Offset are mutually exclusive; only one can be toggled on at any given time
Offset	Determines the sash offset placement from the inside of the window jamb. The distance is from the interior surface of the wall to the interior surface of the sash; choose Offset, and then choose or type a sash offset and press <input type="button" value="Enter"/>

Muntins

Use Muntins to define and place window pane dividers within the sash assemblies. You can also set the number of window panes and their placement.

DoMuntn	Models muntins, or window pane dividers
MuntWdth	Sets the muntin width along the plane of the wall; the width applies equally to vertical and horizontal muntins; choose MuntWdth, and then choose or type a muntin width and press <input type="button" value="Enter"/>
MuntThk	Sets the muntin thickness, which is measured through the window; choose MuntThk, and then choose or type a muntin thickness and press <input type="button" value="Enter"/>
PaneHorz	Sets the number of panes per window horizontally; choose PaneHorz, and then choose or type a value and press <input type="button" value="Enter"/> . PaneHorz and PaneVert are also available in the Glass menu. They are the same settings, so when you change either option in the Muntins menu, the new values are also displayed for the PaneHorz and PaneVert options in the Glass menu.
PaneVert	Sets the number of panes per window vertically; choose PaneVert, and then choose or type a value and press <input type="button" value="Enter"/>
Centred	Centers a muntin within the sash; Centred and Offset are mutually exclusive; only one can be toggled on at any given time
Offset	Sets the offset placement of the muntins from the interior surface of the sash to the interior surface of the muntins; choose Offset, and then choose or type a muntin offset and press <input type="button" value="Enter"/> ; Centred and Offset are mutually exclusive; only one can be toggled on at any given time
FullEdg	Sets a full-edge muntin at the window sash
HalfEdg	Sets a half-edge muntin at the window
No Edg	Sets a no-edge muntin at the window sash
Slabs	Constructs muntins as slabs; when Slabs is toggled off, muntins are constructed as 3D lines

Glass

Use Glass to define the number of glass panes horizontally and vertically. It also sets the glass thickness for the window panes in the model. Since glass is represented as a slab or solid object, it is often left out of a model so that a hidden line removal can be performed as if the glass were transparent.

DoGlass	Models window glass in the window panes
GlassThk	Sets the glass thickness; choose GlassThk, and then choose or type a glass thickness and press <input type="button" value="Enter"/>

PaneHorz	Sets the number of horizontal panes per window; choose PaneHorz, and then choose or type a value and press [Enter] . PaneHorz and PaneVert are also available in the Muntins menu. They are the same settings, so when you change either option in the Glass menu, the new values are also displayed for the PaneHorz and PaneVert options in the Muntins menu.
PaneVert	Sets the number of vertical panes per window; choose PaneVert, and then choose or type a value and press [Enter]
Centred	Centers a window pane glass in the muntin, through the window; Centred and Offset are mutually exclusive; only one can be toggled on at any given time
Offset	Sets the offset of the glass panes from the interior surface of the muntin to the interior surface of the glass; choose Offset, and then choose or type a glass pane offset and press [Enter] ; Centred and Offset are mutually exclusive; only one can be toggled on at any given time

Modeling Doors

The Doors menu in the AEC_Modl macro lets you access door data, load or save previously designed doors, and select or enter door components and data.

Cutting Walls

CutWall is a toggle used to automatically create voids in slabs or polygons for doors. It is available in the menu when InPlan is toggled on. When CutWall is toggled on, a LyrSrch toggle also becomes available.

Door File

Use the DoorFile option to load, save, delete, rename, or copy door files. Choose from the following DoorFile options:

LoadDoor	Loads door parameters from saved files; choose or type a door filename and press [Enter]
SaveDoor	Saves door parameters to a file; type a filename and press [Enter]
Del Door	Deletes saved door files from the hard disk. To help prevent accidental deletions, the system prompts for the filename requested. To delete a door file, select a filename and press [Enter] . Choose Yes to delete the file; otherwise choose No.
Ren Door	Renames existing files; select a file to rename, type a filename of up to eight characters long, and press [Enter]
CopyDoor	Copies the contents of an existing door file to a new file; type the name of the file to copy from and press [Enter] , then type the filename to copy to and press [Enter]
Form	Displays a door data form

All doors are created to open inward, toward you.

Door Forms

The DoorForm menu lets you view the settings for the door on which you are currently working. The design form offers a comprehensive view of the door parameters on one screen.

Saving 3D Doors as Symbols

Use the Template option to create and save doors as symbols, developing a library of door symbols that you can quickly insert into new projects. For more information, see “Templates and Symbols” in the “Drawing Elements” chapter.

Door Heights and Wall Thickness

You can set the head height and sill height of a door with the Head Hgt and Sill Hgt options, as well as use the Wall Thk option to change the wall thickness. The Head Hgt and Sill Hgt options are available only when In Plan is toggled on, while Wall Thk is only available when In Elev is toggled on. When you choose one of these options, a list of values is displayed in the Menu Window. Choose or type a new value and press **[Enter]**. The Head Hgt and Sill Hgt settings are relative to Z-base.

Door Types

UnitType displays available door types. Doors swing inward (toward you), so you should build designs from the interior of a structure.

Choose a door type from the following options:

- | | |
|---------|---|
| Single | Single door, hinged |
| Double | Double doors, hinged |
| Bifold | Bi-fold door |
| 2xBifld | Double bi-fold doors |
| Sliding | Sliding door |
| Pocket | Pocket doors |
| % open | Determines the percentage that a door appears open in the model, with 0% being fully closed and 100% being fully open |

Door Casings

Use Casing to control the casing or exterior door component group. You can set any of the following door casing options:

- | | |
|----------|---|
| At Head | Models casing elements in the door header model |
| Width | Sets the casing width at the door header, across the door plane, and at the door jamb; choose or type a value and press [Enter] |
| Thicknss | Sets the casing thickness at the door head and at the door jamb; choose or type a value and press [Enter] |
| Extensn | Sets the extension length past the outside edge of the vertical casing components (jamb casing). Choose or type a value and press [Enter] . For any extension, this measurement is always 0 when the outside edge of the vertical casing and the extension end are aligned. Positive numbers indicate an extension; negative numbers indicate a setback. |
| At Jamb | Models casing elements in the door jamb model (Width and Thicknss appear again after this option) |

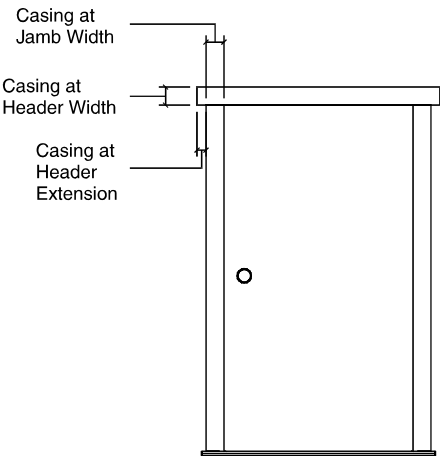


Figure 28.6: Door casing

Door Trim

Use Trim to control the trim, or interior door component group. You can vary such elements as the trim apron, according to aesthetic preference or design requirements.

At Head	Models trim elements at the door header model
Width	Sets the trim width at the door header and across the door jamb
Thicknss	Sets the trim thickness at the door header, through the door, and at the door jamb
Extensn	Sets the trim extension length past the outside edge of the vertical jamb trim components
At Jamb	Models trim elements in the door jamb model (Width and Thicknss appear again after AtJamb)

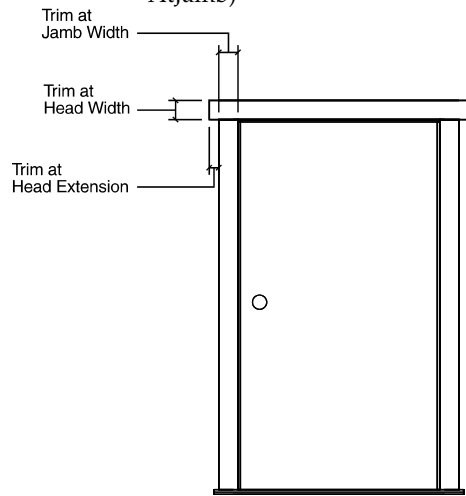


Figure 28.7: Door trim

Door Headers

Use Header to define the door header. This area can become exceedingly detailed in a model rendering, including its structural and aesthetic elements in one master group. With Header you can break down the various components to provide a clearer view.

Do Head	Models the door header
HeadWdth	Sets the door header width. Measure through the door, from the inner surface of the door outward. This option appears when WalWdth is off; choose HeadWdth, and then choose or type a header width and press Enter
HeadThk	Sets the door header thickness. Measure from top (wall opening) to bottom. When you choose HeadThk, a list of thickness values appears. Choose or type a header thickness and press Enter .
WalWdth	Calculates a header width equal to the wall thickness; when WalWdth is off, the width of the header is controlled by the header width (HeadWdth) setting

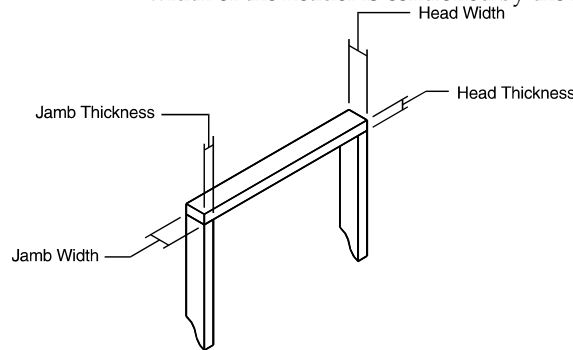


Figure 28.8: Door header

Door Jambs

The Jamb menu defines the door jamb, using the following options:

Do Jamb	Models jamb components
JambWdth	Sets the door jamb width (measure from the inner surface of the door, outward through the door); available only when WalWdth is off; choose JambWdth, and then choose or type a jamb width and press <input type="button" value="Enter"/>
Thicknss	Sets the door jamb thickness (measure from the void edge, or wall opening, inward toward the door center along the wall plane); choose Thicknss, and then choose or type a jamb thickness and press <input type="button" value="Enter"/>
WalWdth	Sets the jamb width equal to the wall width; when WalWdth is off, the width of the jamb is controlled by JambWdth; choose WalWdth, and then choose or type a jamb width and press <input type="button" value="Enter"/>

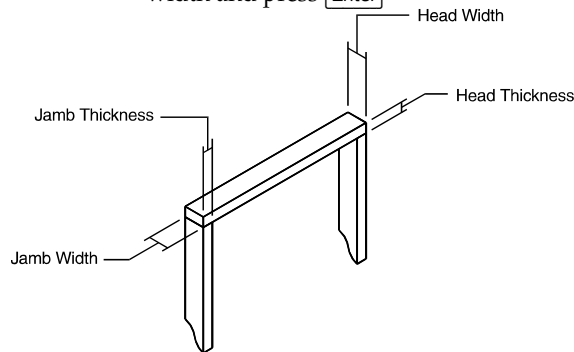


Figure 28.9: Door jamb

Door Sills

The Sill menu defines the door sill, using the following options:

Do Sill	Models the sill
Thicknss	Sets the sill thickness from the bottom of the door opening, upward at the sill middle; choose Thicknss, and then choose or type a sill thickness and press <input type="button" value="Enter"/>
EdgeThk	Sets the thickness of the tapered edge; choose EdgeThk, and then choose or type an edge thickness and press <input type="button" value="Enter"/>
InExtn	Sets the inside sill extension away from the plane of the door toward the center of the room; choose InExtn, and then choose or type an extension and press <input type="button" value="Enter"/>
Out Extn	Sets the exterior sill extension across the door plane from the exterior wall outward; choose Out Extn, and then choose or type an extension and press <input type="button" value="Enter"/>
SideExtn	Sets the exterior sill extension along the wall plane from the outside edge of the trim outward (also known as the "sill horns"); choose SideExtn, and then choose or type an extension and press <input type="button" value="Enter"/>

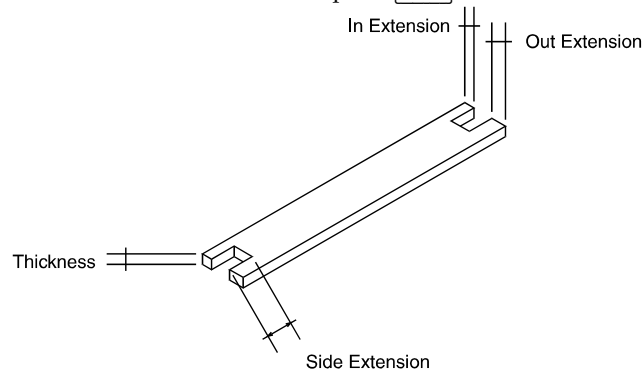


Figure 28.10: Sill thickness

Door Stops

Use Stop to place and size the door stops. The stop is drawn to the exterior of the door, whether the door is flush or offset. Door stops are not created for sliding or pocket doors, regardless of whether Do Stop is toggled on or not.

Do Stop	Models the door stop
StopWdth	Sets the stop width for all components (measure the stop width in the same direction as the jamb width and across the wall plane); choose StopWdth, and then choose or type a stop width and press <input type="button" value="Enter"/>
Stop Thk	Sets the stop thickness dimension (measure the stop thickness in the same direction as the jamb thickness and across the wall plane); choose Stop Thk, and then choose or type a stop thickness and press <input type="button" value="Enter"/>

Door Options

The Door menu sizes, positions, and hinges a door, using the following options:

Do Door	Models the door
Door Thk	Sets the door thickness; choose Door Thk, and then choose or type a door thickness and press <input type="button" value="Enter"/>
HingRht	Hinges doors on the right; determine the right side of the door by standing inside the structure and looking at the door
HingLft	Hinges doors on the left; determine the left side of the door by standing inside the structure and looking at the door
Flush	Sets the door flush with the outside wall
Offset	Sets the offset placement of the door from the interior wall surface across the wall plane; choose Offset, and then choose or type a value and press <input type="button" value="Enter"/>

Door Knobs

Use Knob to place, size, and identify the door knob; only round door knobs are available. When Do Door is off, you can't create a knob regardless of the Do Knob setting.

Do Knob	Models the knob
Diameter	Sets the diameter of the door knob; choose Diameter, and then choose or type a door knob diameter and press <input type="button" value="Enter"/>
Extnsion	Sets the knob extension out from the door; choose Extnsion, and then choose or type a knob extension and press <input type="button" value="Enter"/>
Knob Hgt	Sets the height of the knob from the bottom of the door; choose Knob Hgt, and then choose or type a knob height and press <input type="button" value="Enter"/>
Offset	Sets the knob offset from the door edge; choose Offset, and then choose or type a knob offset and press <input type="button" value="Enter"/>
Inside	Models a knob on the door interior
Outside	Models the knob on the door exterior; Outside and Inside can be toggled individually or together

Modeling Concrete Forms

With the Concrete tool, you can add six different types of preconstucted concrete forms to your model: columns, inverted tee, single tee, double tee, standard, and I-beam forms. These forms are actually slabs and can be edited the same way you'd change regular DataCAD slabs, using the editing options in the 3DEdit menu.

With the exception of columns, all forms are drawn based on a *defining line*, which is the line connecting the two points you enter that represent the two ends of the form. In plan view, this line can define the right, left, or center of the form and can lie at Z-base or Z-height. This line can also be defined in the Z direction. In addition, the form's top, bottom, or center can intersect the defining line in the Z direction. These options give you 18 possible positions for every form you draw.

In this chapter:

- ⊕ Drawing columns
- ⊕ Drawing inverted, single and double tee forms
- ⊕ Drawing standard forms
- ⊕ Drawing I-beam forms

Drawing Columns

Columns begin as rectangular boxes. You can chamfer the vertical edges of the column and add up to four seats to the column to support the girders and beams. The location of a column is defined by two points, at opposite corners of a box, indicating the overall dimensions of the column. The column extends vertically from the current value of Z-base to the current value of Z-height.

➔ To draw columns:

1. Choose Column from the Toolbox menu's Concrete submenu. The following options are displayed:

Width	Sets the column width
Depth	Sets the column height
Rotate	Enters columns that rotate about the entry point; you are prompted for the rotation angle of the column
LoLeft	Specifies the location of the column by its lower left corner
Center	Specifies the location of the column by its center point
UpRight	Specifies the location of the column by its upper right corner
TopSeat/ BotSeat BotSeat LftSeat / RhtSeat	Indicate the side of the column where the column seat is; you are prompted for the height of the column seat from the current value of Z-base (when a toggle is off, column seats are not added for that particular side)
SeatOnly	Adds column seats to an existing column or multiple sets of column seats to a single column in a multiple-story structure with monolithic concrete columns. When this toggle is on, you can add seats to a column, but you cannot add actual columns.
SeatWdth	Sets the distance from the face of the column to the outside of the column seat
SeatHght	Sets the distance from the top face of the column seat to the bottom edge of the column seat
Chamfer	Chamfers the edges of the column; otherwise, the column is angular in cross section (the size of the column seats automatically adjusts)
ChamSize	Sets the size of the column chamfers
2. Choose any options you want to set. When you choose Width, Depth, Rotate, TopSeat, BotSeat, LftSeat, RhtSeat, SeatWdth, and SeatHght a list of values appears.
3. Choose or type a value and press .
4. Select a point in the Drawing Window to place the column's location.

Drawing Inverted, Single and Double Tee Forms

Inverted tees, or girders, are typically used for supporting single or double tees across a long span such as a parking garage bay. Because they are in the shape of an inverted T, the beam portion of a single, double, or quad tee can be placed on a tee's shelf. You have a variety of options for changing the overall dimensions of the beam and for its placement in your model.

➔ To draw an inverted tee:

1. Choose InvertTee from the Concrete menu. The following options are displayed:

OvrlHgh	Sets the overall height of the inverted tee; choose OvrlHght, and then choose or type a value and press <input type="button" value="Enter"/>
---------	--

- | | |
|-----------|--|
| OvrIWidth | Changes the overall width of the inverted tee; choose OvrIWidth, and then choose or type a value and press <input type="button" value="Enter"/> |
| NtchDpth | Changes the depth of the notch in the inverted tee; choose NtchDpth, and then choose or type a value and press <input type="button" value="Enter"/> |
| BeamThck | Changes the thickness of the narrow portion of the inverted tee; choose BeamThck, and then choose or type a value and press <input type="button" value="Enter"/> |
| Left | Draws the form to the left of the defining line in the X-Y plane |
| Center | Draws the form centered about the defining line in the X-Y plane |
| Right | Draws the form to the right of the defining line in the X-Y plane |
| Top | Draws the form with its top aligned with the defining line |
| Center | Draws the form with its center aligned to the defining line |
| Bottom | Draws the form with its bottom aligned to the defining line |
| Z-Base | Sets the defining line at the Z-base |
| Z-Height | Sets the defining line at the Z-height |
| Single | Places a shelf on only one side of the inverted tee (to the left of the defining line) on the end of the first inverted tee that is entered |
| Double | Places shelves of identical size on both sides of the inverted tee |
2. Select the first end of the defining line.
 3. Select a second point that is the opposite end of the defining line.

Single tee concrete forms are used as combination beams and floor slabs. In applications where the primary function is creating floors, a double tee form is generally used. Industrial buildings and parking structures use double tees for upper floors that must support large floor loads. A double tee form is easier to position and place because of the stability provided by having two beam sections. Single tees are used in special cases, where spans are exceptionally long or floor loads are exceptionally high. In these cases, the weight and size of double tee forms can make them too large and heavy for practical fabrication, shipment, or placement.

➔ To draw a single tee:

1. Choose SngITee from the Concrete menu. The following options are displayed:

Width	Sets the overall width of the single tee form (the location of the beam flanges of the form are automatically adjusted); choose Width, and then choose or type a width and press <input type="button" value="Enter"/>
Depth	Sets the overall depth of the single tee form; choose Depth, and then choose or type a depth and press <input type="button" value="Enter"/>
WebThck	Sets the thickness of the floor slab portion of the single tee form; choose WebThck, and then choose or type a thickness and press <input type="button" value="Enter"/>
BeamThck	Sets the thickness of the two support beams for the single tee slab
Left	Draws the form to the left of the defining line in the X-Y plane
Center	Draws the form centered about the defining line in the X-Y plane
Right	Draws the form to the right of the defining line in the X-Y plane
Top	Draws the form with its top aligned to the defining line
Center	Draws the form with its center aligned to the defining line
Bottom	Draws the form with its bottom aligned to the defining line
Z-Base	Places the defining line at Z-base
Z-Hght	Places the defining line at Z-height
2. Set the size of cross-sectional dimensions of the single tee form.
3. Select the first end point of the defining line.
4. Select the second end point of the defining line.

➔ To draw a double tee:

1. Choose DblTee from the Concrete menu. The following options are displayed:

Width	Sets the overall width of the double tee form (the location of the beam flanges of the form are automatically adjusted); choose Width, and then choose or type a width and press Enter
Depth	Sets the overall depth of the double tee form; choose Depth, and then choose or type a depth and press Enter
WebThck	Sets the thickness of the floor slab portion of the double tee form; choose WebThck, and then choose or type a thickness and press Enter
BeamThck	Sets the thickness of the two support beams for the double tee slab; choose BeamThck, and then choose or type a thickness and press Enter
Left	Draws the form to the left of the defining line in the X-Y plane
Center	Draws the form centered about the defining line in the X-Y plane
Right	Draws the form to the right of the defining line in the X-Y plane
Top	Draws the form with its top aligned to the defining line
Center	Draws the form with its center aligned to the defining line
Bottom	Draws the form with its bottom aligned to the defining line
Z-Base	Places the defining line at Z-base
Z-Hght	Places the defining line at Z-height
2. Set the size of the cross-sectional dimensions of the double tee form.
3. Select the first end point of the defining line.
4. Select the second end point of the defining line.

Drawing Standard Beam Forms

Standard beams have a simple rectangular cross section, but they can also have chamfers along the four edges of the beam. You can vary the height and width of the beam. There are 18 possible ways to position the beam relative to a defining line that extends from the first point entered to the second point entered.

➔ To draw a standard beam:

1. Choose Std Beam from the Concrete menu. The following options are displayed:

Width	Sets the beam width
Depth	Sets the beam depth
Chamfer	Chamfers the beam on all edges; otherwise, the beam is rectangular in the cross section. When Chamfer is on, you must enter the chamfer size.
Left	Draws the form to the left of the defining line in the X-Y plane
Center	Draws the form centered about the defining line in the X-Y plane
Right	Draws the form to the right of the defining line in the X-Y plane
Top	Draws the form with its top aligned to the defining line
Center	Draws the form with its center aligned to the defining line
Bottom	Draws the form with its bottom aligned to the defining line
Z-Base	Sets the beam's defining line at the Z-base
Z-Hght	Sets the beam's defining line at the Z-height
2. Choose any options you want to set. When you choose Width, Depth, and Chamfer, a list of values appears. Choose or type a value and press **Enter**.
3. Select the first end of the defining line.
4. Select the second end of the defining line.

Drawing I-Beam Forms

I-beams are similar to standard beams, except that the beam is narrower at the middle of the cross section than it is at either end of the cross section. For a typical application, the stresses in the concrete are much higher at the top or bottom of the beam; therefore, an I-beam can be more economical to use than a standard rectangular beam.

➔ To draw an I-beam:

1. Choose I-Beam from the Concrete menu. The following options are displayed:

Width	Sets the overall horizontal measurement of the cross section of the I-beam
Depth	Sets the overall vertical measurement of the cross section of the I-beam
WebThick	Sets the thickness of the narrower portion of the beam independently of the overall width of the I-beam
FlngDpth	Sets the vertical dimensions of the I-beam flange independently of all other I-beam cross section dimensions
Left	Draws the form to the left of the defining line in the X-Y plane
Center	Draws the form centered about the defining line in the X-Y plane
Right	Draws the form to the right of the defining line in the X-Y plane
Top	Draws the form with its top aligned with the defining line
Center	Draws the form with its center aligned with the defining line
Bottom	Draws the form with its bottom aligned with the defining line
Z-Base	Sets the beam's defining line at the Z-base
Z-Height	Sets the beam's defining line at the Z-height
2. Choose any options you want to set. When you choose Width, Depth, WebThick, or FlngDpth, a list of values appears. Choose or type a value and press Enter.
3. Select the first end of the defining line.
4. Select the second end of the defining line.

Modeling Stairs

30

The 3D Stairs tool gives you everything you need to create any of seven predefined 3D stairs, including single-run, straight-run, double-back, L-shaped, open-well, curved, and spiral stairs.

For each stair type, you can customize settings for treads, risers, landings, stringers, handrails, balusters, and more. These settings can then be reviewed on screen and even saved to a stair (.STR) file that will allow you to quickly load customized settings for drawing additional stairs.

In this chapter:

- ✦ Drawing 3D stairs
- ✦ Customizing stair settings
- ✦ Saving and using stair settings

Drawing 3D Stairs

The 3DStairs menu provides everything you need to create a set of three-dimensional stairs. 3DStairs creates real stairs, not stacked slabs, making this representation of stairs more accurate than ever before. The 3DStairs macro has these main features:

- seven predefined stair types -- single-run, straight-run, double-back, L-shaped, open-well, curved, and spiral
- a stair calculator, which makes it easy for you to calculate the values of stair settings and then readjust the Settings menu values

With the stair forms and other settings you can customize the shape and placement of stair components. There are three ways to define the settings for the type of stairs you want to model:

- use the 3DStairs/Settings menu to select options and enter individual values for the various stair components
- use the Stair Form to display the settings on the screen for reference and/or customization
- save your stair settings to a file through the StrFile menu; when you want to model that specific stair type again, you can recall the file

Because of the interactive nature of stair components, changing one option can cause other options to change automatically. For example, when you close stringers, you can't have tread extensions.

The sections that follow describe how to create each stair type. You can only use the Stair Calculator for single-run stairs. See the "More About Drawing Single-Run Stairs" later in this chapter for more information.

➔ To draw 3D stairs:

1. Choose Toolbox from the Edit menu.
2. Choose 3DStairs from the Toolbox dialog box.
3. Choose a stair type.

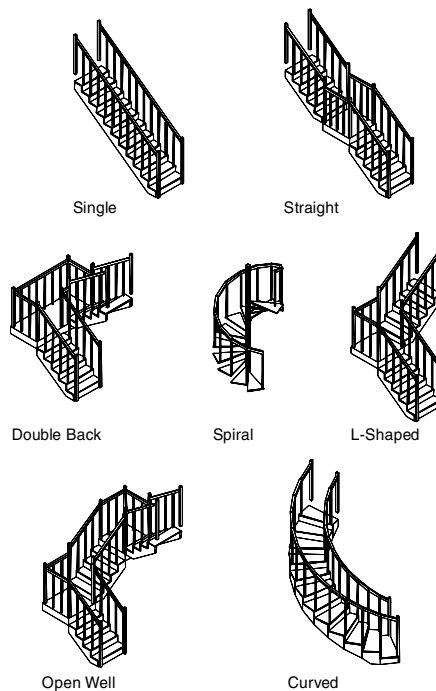


Figure 30.1: Stair types

4. Set the options for the stair style you want. You can use either the Settings menu, the Stair Form, or you can load a stair file. When you model single-run stairs, you can use the Stair Calculator to determine the settings. See the “More About Drawing Single-Run Stairs” later in this chapter for more information.
5. Choose Begin to place the stairs. The tool automatically changes to a plan view and a boundary box appears representing the run of stairs. When Ortho mode is on, the boundary box does not track with the cursor. Press ☐ to toggle Ortho mode off.
6. Drag the stairs by the lower outside corner of the first run to the new stair position. The boundary box rotates as you move your cursor.
7. Rotate the stairs to the angle you want. The box rotates as you move the cursor.
8. Click to place the stairs. The stair model appears in the selected location with the settings you chose.
9. Change the results by erasing the last group by pressing ☐ + ☐ and repeat steps 3 through 7.

More About Drawing Single-Run Stairs

The Stair Calculator (Calculat) is available only when you create single-run stairs. Use it to calculate one of six possible values: total rise, number of risers, riser height, total run, number of treads, or tread depth.

Use the Stair Calculator as you would use a desktop calculator; prompts are displayed in the Message Window at the bottom of the screen. Save the answers to automatically update the 3DStairs/Settings menu for single-run stairs.

➔ To use the stair calculator:

1. Choose **Calculate** from the **3DStairs** menu.
2. Select an option from the **Calculate** menu to calculate and modify:

TotalRise	The number of risers times the riser height; same value regardless of whether LandTop is on or off
# Risers	The total rise divided by the riser height, including the rise of the top landing; changing the number of risers automatically changes the number of treads and vice versa
RiserHgt	The vertical distance from the top of one tread to the top of the next tread; the total rise divided by the number of risers
TotalRun	The horizontal distance from the top of the run of stairs to the bottom of the stairs; the number of treads times the tread depth (the distance can include landing depth for the top and bottom landings when those toggles are turned on)
# Treads	The number of treads in the stair run; changing the number of treads automatically changes the number of risers and vice versa (you can set TreadDth from this menu to recalculate the number of treads)
TreadDth	The horizontal distance from the face of one riser to the face of the next riser; the total run divided by the number of treads (you can change # Treads in this menu)
LandBtm	When LandBtm is on, the total run value includes the bottom landing depth
LandTop	When LandTop is on, the total run value includes the top landing depth
Reset	Resets your calculator settings to the default values
Update	Overwrite the old settings in the Setting menu with new settings; saves your calculated values
Begin	Use Begin to start modeling stairs. Follow the steps in the “Using Stairs” section for information on using Begin .
Template	Shortcut to the Template menu; select a template from the menu, or type a template name, and press Enter
Hide	Perform hidden line removals; see “Hide” in the “3D Editing” chapter for more information. See the “Viewing Your Model” chapter for more information on templates, hidden line removals, and the 3DViews menu.
3DViews	Shortcut to the 3DViews menu; see the “3D Viewing” chapter for more information
3. Look at the submenu that is displayed after some **Calculate** menu options. If this occurs, choose an option to modify; if not, skip to step 5.
4. Choose a new value from the menu or type a different value. As you change the values of a setting, the results are automatically calculated and displayed. The stair calculator only accepts a whole number of treads or risers as a result. If the values you enter in a calculation result in a fractional solution, the stair calculator modifies the values.
5. Modify other options by repeating steps 3 and 4.
6. Choose **Update** to update the settings in the **3DStairs/Settings** menu.

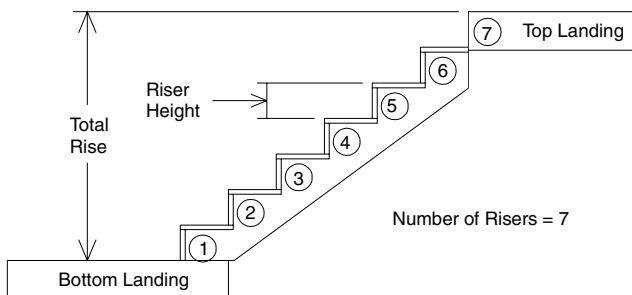


Figure 30.2: Total rise, riser height, and number of risers

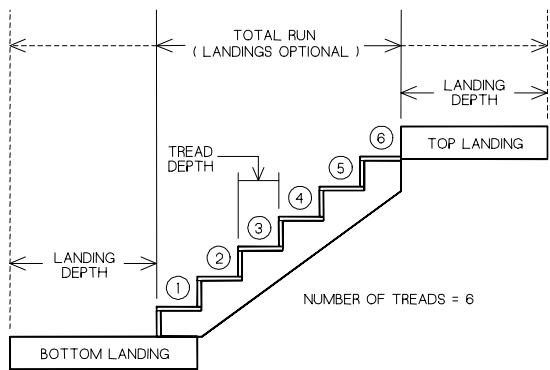


Figure 30.3: Total run, tread depth, and number of treads

For example, to calculate the number of risers through changing RiserHgt:

	Initial Settings	Modified Settings	Stair Calculator Results
TotlRise	12'0"	12'0"	12'3"
RiserHgt	6"	7"	7"
#Risers	24	20.57	21

Because #Risers is a fractional number, the Stair Calculator automatically adjusts the TotlRise so that the results are whole numbers. If you change TotlRise, Stair Calculator adjusts RiserHgt to find a whole number solution.

Customizing Stair Settings

Use the Settings menu to change the settings for 3DStairs. Different options appear depending on the stair type you selected from the 3DStairs menu.

The options in the top half of the menu are global settings that apply to all stair types. The bottom half of the menu are stair component submenus. Each of these submenus contains the various settings for that particular component.

When you choose any of the Settings options, a list of values appears. Choose a value or press **Enter** to accept the given value.

Base Elevation

Use BaseElev to change the base elevation of a flight of stairs. Base elevation is the height, or elevation, at which the flight of stairs begins.

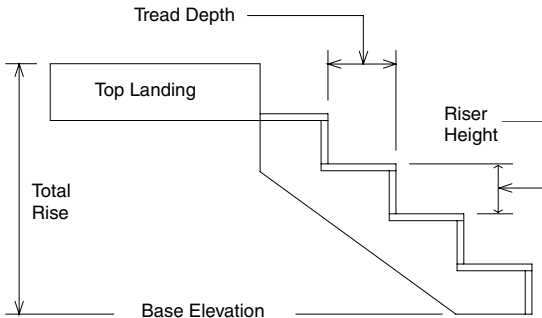


Figure 30.4: Base Elevation

Treads

Use `TrdLngh` to change the tread length. The tread length is the horizontal distance from one side of the tread to the other side. It does not include tread extension values. Any tread extensions are added to the tread length and only apply to stairs modeled with open stringers.

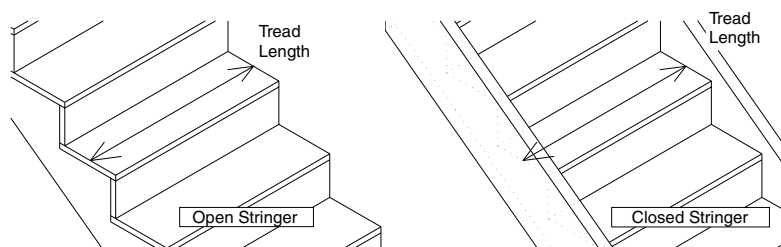


Figure 30.5: Tread length

Use `Trd/Run1` to change the number of treads in the first run of stairs. This applies to all stair types. The number of treads does not include the landing at the top of the run and is equal to one less than the number of risers.

Use `Trd/Run2` to change the number of treads in the second run of stairs. `Trd/Run2` is only available for straight, double-back, open-well, and L-shaped stairs.

Use `Trd/Run3` to change the number of treads in the third run of stairs. `Trd/Run3` is only available for open-well stairs.

Radius of Curved Stairs

Use `Radius` to change the inner radius of curved and spiral stairs. Measure `Radius` from the exact center point to the inside face of the stringer for curved stairs and the radius of the center pole for spiral stairs.

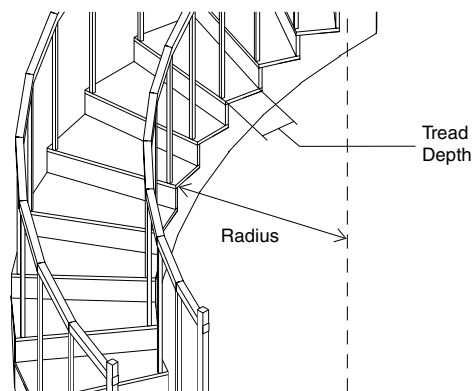


Figure 30.6: Radius on a curved flight of stairs

Clockwise

`Clkwise` applies to all stair types except single and straight. You can place the stairs clockwise or counterclockwise depending on your needs.

Chases

Use Chase to control the horizontal distance between the runs of a double-back flight of stairs. This setting effects the width of the intermediate landing and the tread length.

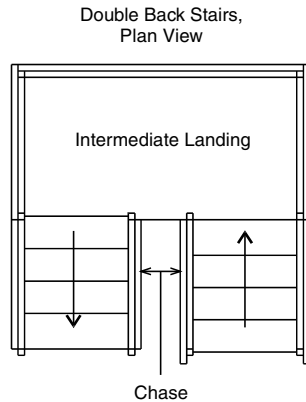


Figure 30.7: Chase

Landings

The top, bottom, and in some cases, intermediate landings all have depth and thickness settings. You can change the specific settings for each landing you create using the Landing menu in the 3DStairs/Settings menu. (Changing the settings on the top landing does not affect the settings of the bottom landing, and vice versa.)

Toggle DoLndTp on to model a landing at the top of the flight of stairs; toggle DoLndBt on to model a landing at the bottom of a flight of stairs.

The Depth option sets the landing depth. The *landing depth* is the horizontal distance from the front of the landing to the rear of the landing. When you model L-shaped or open-well stairs, you cannot set a landing depth on the intermediate landing. The landing depth equals the width of the run of stairs.

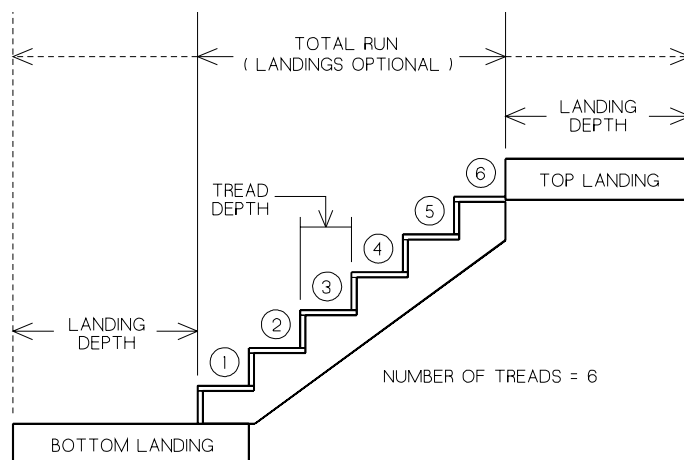


Figure 30.8: Landing depth

Thicknss shows the current thickness of a landing. Measure the landing thickness vertically from the underside to the top side. Thicknss sets the top, intermediate, and bottom landing thicknesses.

For stair types that have intermediate landings, IntrLand is always on. It is available for straight, double-back, open-well, or L-shaped stairs only. For rectangular or curved intermediate landings, use the Rectangl/Curved option; this is available for L-shaped, double-back, or open-well stairs only.

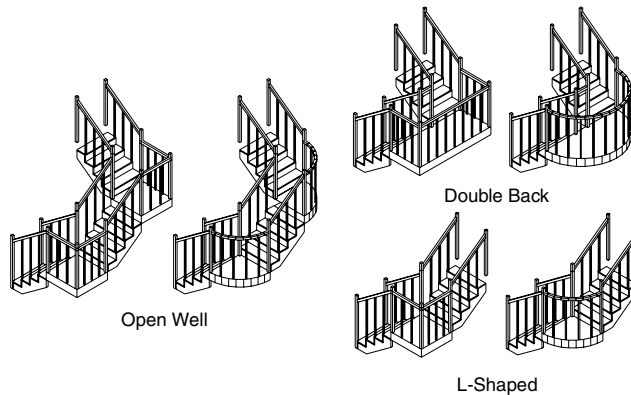


Figure 30.9: Rectangular / curved intermediate landings

Use Color to set the landing color. All landings in a single flight of stairs are modeled using the same color. See “Color Menus” in “The Drawing Board” chapter for more information about how to choose a color.

Risers

Use the Riser menu to set options that create risers and determine riser height, thickness, and color. Toggle DoRiser on to model risers on your flight of stairs.

RiserHgt displays the current riser height in the Message Window and lets you reset it. The riser height is the vertical distance from the top of one tread to the top of the next.

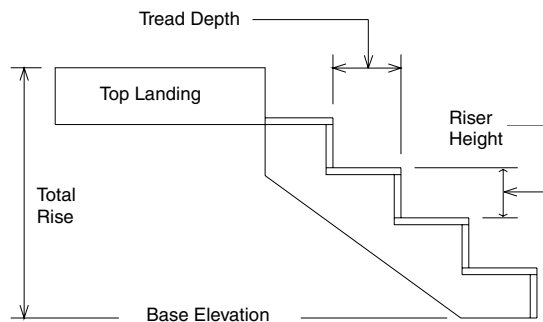


Figure 30.10: Riser height, total rise, and tread depth

The Thicknss option displays the current riser thickness in the Message Window and lets you reset it. The *riser thickness* is the horizontal distance from the front side of the riser to the backside of the riser when you choose Thicknss.

Use Color to set the landing color. All landings in a single flight of stairs are modeled using the same color. See “Color Menus” in “The Drawing Board” chapter for more information about how to choose a color.

Treads

Use Tread to set options that model treads and determine the depth, thickness, and nosing of the treads. Toggle DoTread on to automatically calculate and model treads on a flight of stairs.

Depth displays the current tread depth in the Message Window and lets you reset it. The tread depth is the horizontal distance from the front of one tread to the front of the next tread. When you select a spiral stair, TredDegs (tread degrees) is displayed in the menu in place of depth. Tread degrees displays the inclusive angle of the treads on spiral stairs in the Message Window and lets you reset it.

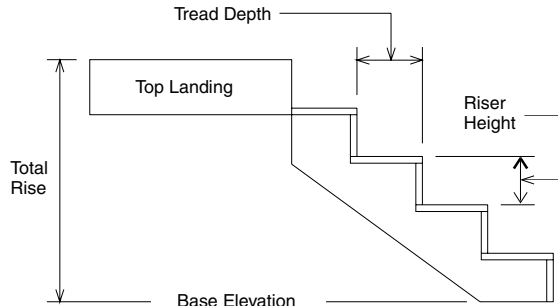


Figure 30.11: Tread depth

Thicknss displays the current tread thickness in the Message Window and lets you reset it. The tread thickness is the vertical distance from the top side of the tread to the bottom side of the tread.

LeftExtn and RightExtn display the current left and right extension, respectively, of the tread in the Message Window and lets you reset it. Left extension includes tread length, the distance from one stringer to the next. LeftExtn controls the distance that the tread extends beyond the outside of the stringer. LeftExtn is available only for stairs with open stringers.

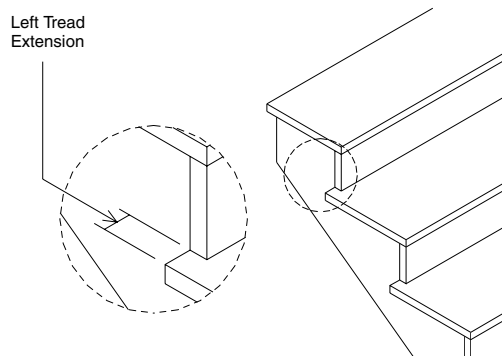


Figure 30.12: Tread extension

When Nosing is toggled on, you can set the nosing length and type for treads. 3DStairs prompts you for an overhang value when you toggle nosing on. Nosing controls the distance that the front of the tread extends beyond the front of the riser.

To set the nosing type to square or round, click on the RndNosng/SqrNosng toggle. This Nosing option changes between round and square just by clicking on it. If RndNosng is displayed in the Nosing menu, round nosing will be drawn on the stair treads; if SqrNosng is displayed in the Nosing menu, square nosing will be drawn on the stair treads.

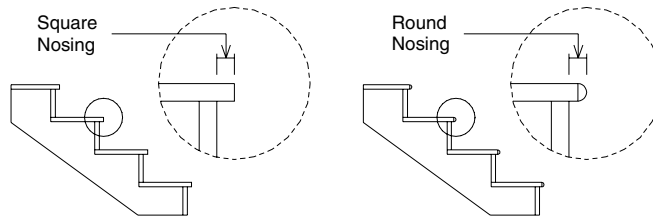


Figure 30.13: Nosing

Use Color to set the color for the treads. All treads in a flight of stairs are the same color. See “Color Menus” in “The Drawing Board” chapter for more information about how to choose a color.

Stringers

Stringer allows you to set options that create stringers and determine where and what kind you want to use. Toggle DoLeft on to model a stringer on the left side of the run of stairs, toggle DoCtr on to model a stringer in the center of a run of stairs, or toggle DoRight on to model a stringer on the right side of the stairs. When using DoCtr, the stringer is always modeled as an open stringer.

Use the ClosdStr/OpenStr toggle to set the stringer type to open or closed. When ClosdStr is displayed in the Stringer menu, stringers will be modeled as closed; when OpenStr is displayed in the Stringer menu, stringers will be modeled as open. Choose ClosdStr or OpenStr to toggle between the two options.

Width displays the current width of the stringer in the Message Window and lets you reset it. Measure the width from the top edge of the stringer to the bottom edge of the stringer.

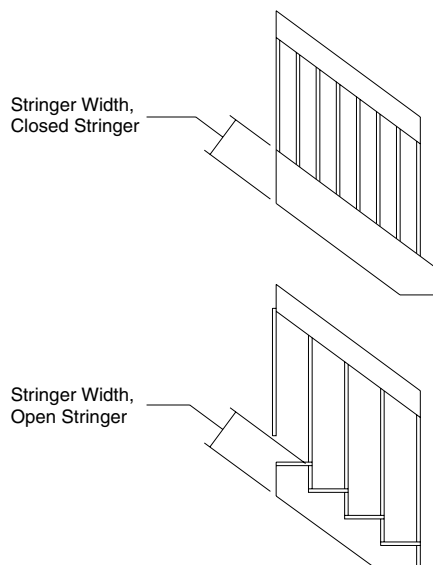


Figure 30.14: Stringer width (open or closed)

The stringer thickness is the horizontal distance from the inside of the stringer to the outside of the same stringer. Choose Thickness from the Stringer menu, and then choose or type a new thickness value and press **Enter**.

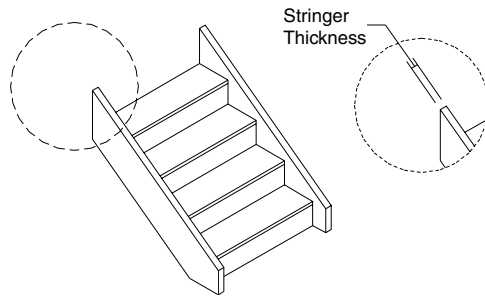


Figure 30.15: Stringer thickness

Use Color to set the color for the stringer. All stringers in a flight of stairs are modeled using the same color. See “Color Menus” in “The Drawing Board” chapter for more information about how to choose a color.

Handrails

Toggle DoLeft on to model a handrail on the left side of the flight of stairs. Toggle DoRight on to model a handrail on the right side of a flight of stairs. When either of these options are toggled off, 3DStairs does not model the handrails on that side of the stairs.

The Rectangl/Cylinder toggle lets you choose between modeling a cylindrical or rectangular handrail. When you select Cylinder, the Radius and Division options appear for you to enter settings. When you select Rectangl, the Width and Depth options appear for you to set.

The Radius/Width toggle displays the current radius or width of the handrail in the Message Window, depending on whether you selected cylindrical or rectangular handrails. When you model a cylindrical handrail, you can adjust the radius of the cylinder. When you model a rectangular handrail, you can adjust the width of the rectangle. Measure width from the top of the handrail to the bottom of the handrail.

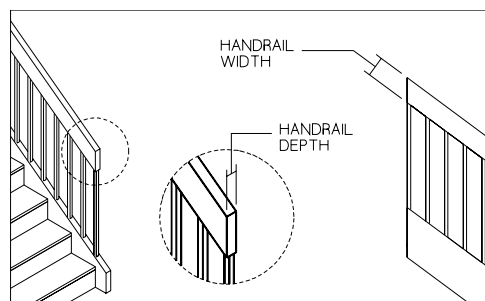


Figure 30.16: Handrail width and depth

The Depth/Division toggle displays the current depth or number of divisions in the handrail in the Message Window, depending on whether you select cylindrical or rectangular handrails. When you model a cylindrical handrail, you can adjust the division of the cylinder, or the number of sides in the cylinder. When you model a rectangular handrail, you can adjust the depth of the rectangle. Measure depth from one side of the handrail to the other side. You can use up to 36 divisions, but the more divisions you have, the longer it takes DataCAD to redraw.

Use Height to see or reset the current height of the handrail. Measure handrail height from the top surface of the handrail to the top of the tread.

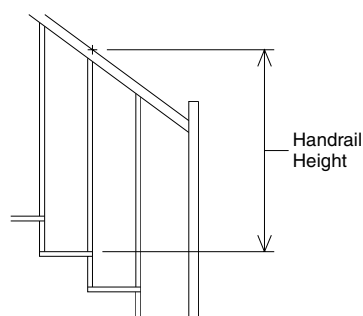


Figure 30. 17: Handrail height

Use Color to set the color for the handrails. All handrails in a flight of stairs are modeled using the same color. See “Color Menus” in “The Drawing Board” chapter for more information about how to choose a color.

Balusters

Use the Baluster menu in the 3DStairs/Settings menu to choose what kind of balusters, if any, to use for your stairs. Toggle DoBalus in the Baluster menu on to model balusters on your flight of stairs.

The Baluster menu has three options for baluster placement. OnSide models balusters on the sides of the stringers, OnTread models balusters on the treads, and OnStrng models balusters on the center of the stringer. OnStrng is available only when you model a closed stringer.

Depending on whether you choose OnSide, OnTread, or OnStrngr for baluster placement, either the Bal/Tred or Bal/Run1 option is displayed in the Baluster menu. The Bal/Tred option displays the number of balusters per tread in the Message Window and lets you change the value. Bal/Tred is only available when you use OnTread. The Bal/Run1 option displays the current number of balusters in the first run of the stairs in the Message Window and lets you change that value. Bal/Run1 is available only when you place the balusters on the side of the stringer (OnSide) or on the stringer (OnStrng).

The Bal/Run2 option displays the current number of balusters in the second run of the stairs in the Message Window and lets you change that value. Bal/Run2 is available only when you place the balusters on the side of the stringer (OnSide) or on the stringer (OnStrng) on straight, double-back, open-well, and L-shaped stairs.

The Bal/Run3 option displays the current number of balusters in the third run of the stairs in the Message Window and lets you change that value. Bal/Run3 is available only when you place the balusters on the side of the stringer (OnSide) or on the stringer (OnStrng) on open-well stairs.

Use Cylinder/Rectangl to choose between modeling a cylindrical or rectangular baluster. When you select Cylinder, the Radius and Division options appear. When you select Rectangl, the Width and Depth options appear.

The Radius/Width toggle displays the current radius or width of the baluster in the Message Window, depending on whether you selected cylindrical or rectangular balusters. When you model a cylindrical baluster, you can adjust the radius of the cylinder. When you model a rectangular baluster, you can adjust the width of the rectangle. Measure width from one side to the other of one baluster.

The Depth/Division toggle displays the current depth or number of divisions in the baluster in the Message Window, depending on whether you selected cylindrical or rectangular balusters. When you model a cylindrical baluster, you can adjust the division of the cylinder, or the number of sides in the cylinder. When you model a rectangular baluster, you can adjust the depth of the rectangle. Measure depth from the front to the back of the baluster. You can use up to 36 divisions, but the more divisions you have, the longer it takes DataCAD to redraw

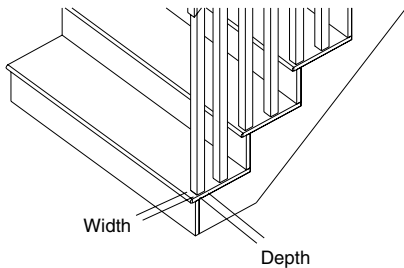


Figure 30.18: Baluster width and depth

Use Symbol to model balusters as symbols. When balusters are symbols, you can replace the balusters produced by 3DStairs with different, more detailed baluster symbols that you create.

➔ To use custom balusters in stairs:

1. Create your detailed baluster in the DataCAD drawing window and save it as a symbol. See “Creating and Editing Symbols” in the “Symbols, Images, and Objects” chapter for more information
2. Choose 3DStairs from the Toolbox dialog box.
3. Set the baluster options as necessary.
4. Toggle Symbol on to draw the balusters as symbols.
5. Create your run of stairs.
6. Replace the existing balusters with the symbol you created in step 1 using the Replace option in the Template menu. See “Replacing Symbols in Your Drawing” in the “Symbols, Images, and Objects” chapter for more information.

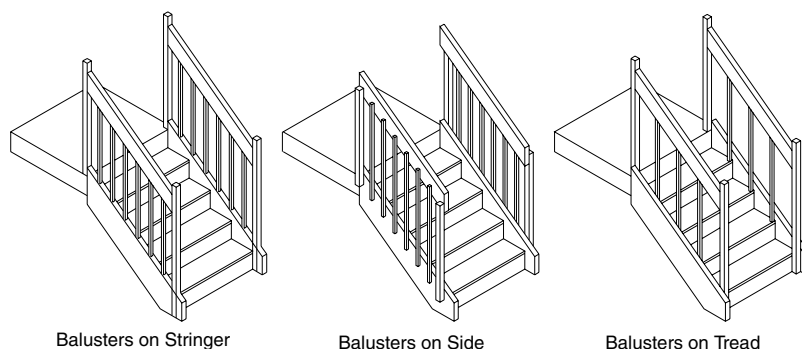


Figure 30.19: Baluster placement

VertOfst shows the current vertical offset of the baluster. Vertical is the distance that the bottom end of the balusters extends below the treads. Vertical offset appears only when you select OnSide.

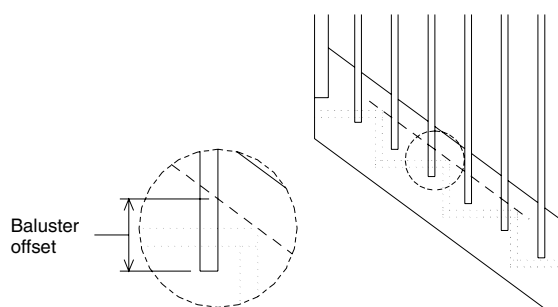


Figure 30.20: Baluster vertical offset

HorzOfst shows the current horizontal offset of the baluster. When you model balusters on the treads, measure the offset from the side edge of the tread to the middle of the baluster. HorzOfst only appears when you select OnTread.

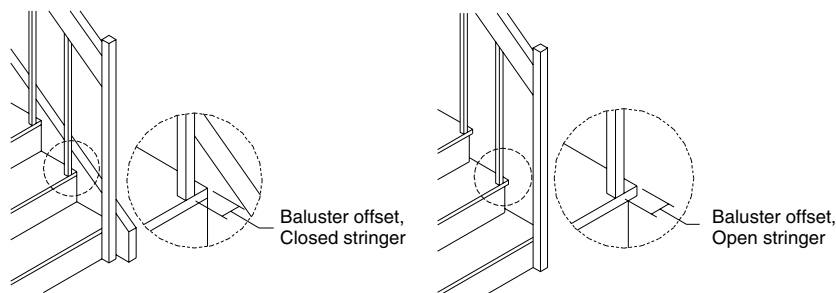


Figure 30.21: Baluster horizontal offset

Use Color to set the color for the balusters. All balusters in a flight of stairs are modeled using the same color. See “Color Menus” in “The Drawing Board” chapter for more information about how to choose a color.

Newel Posts

Use NewelPst to model newel posts on the stairs. Toggle DoNIPst on to model newel posts on a flight of stairs. When DoNIPst is toggled off, 3DStairs models balusters in place of newel posts.

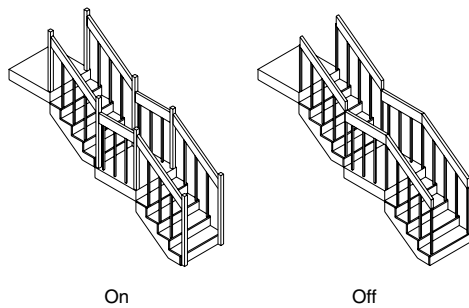


Figure 30.22: Newel posts on/off

Use Cylinder/Rectangl to choose between modeling a cylindrical or rectangular newel post. When you select Cylinder, the Radius and Division options are displayed. When you select Rectangl, the Width and Depth options are displayed.

The Radius/Width toggle displays the current radius or width of the newel post, depending on whether you selected cylindrical or rectangular newel posts, in the Message Window. When you model a cylindrical newel post, you can adjust the radius of the cylinder. When you model a rectangular newel post, you can adjust the width of the rectangle. Measure width from one edge, near a baluster, to the other edge, facing out, of the same newel post

The Depth/Division toggle displays the current depth or number of divisions in the newel post, depending on which of the previous options you selected, in the Message Window. When you model a cylindrical newel post, you can adjust the division of the cylinder, or the number of sides in the cylinder. When you model a rectangular newel post, you can adjust the depth of the rectangle. Measure depth from the edge of the newel post near the handrail to the open edge. You can use up to 36 divisions, but the more divisions you have, the longer it takes DataCAD to redraw.

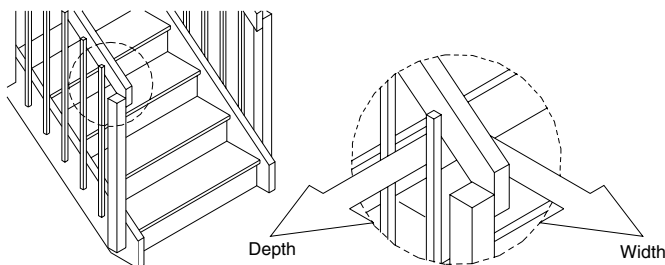


Figure 30.23: Newel post width and depth

Use Symbol to model newel posts as symbols. When newel posts are symbols, you can replace the newel posts produced by 3DStairs with custom newel post symbols that you create.

➔ To use custom newel posts in stairs:

1. Create your detailed newel post in the DataCAD drawing window and save it as a symbol. See “Creating and Editing Symbols” in the “Symbols, Images, and Objects” chapter for more information.
2. Choose 3DStairs from the Toolbox dialog box.
3. Set the newel post options as necessary.

4. Toggle Symbol on to draw the newel posts as symbols.
5. Create your run of stairs.
6. Replace the existing newel posts with the symbol you created in step 1 using the Replace option in the Template menu. See “Replacing Symbols in Your Drawing” in the “Symbols, Images, and Objects” chapter for more information.

Use Color to set the color for the newel posts. All newel posts in a flight of stairs are modeled using the same color. See “Color Menus” in “The Drawing Board” chapter for more information about how to choose a color.

Stair Form

➔ To display values and settings for all stair types:

1. Choose StrForm from the 3DStairs menu.
2. Press **[Tab]** to move forward or **[Shift] + [Tab]** to move backward through each line of the stair form.
3. Scroll through the two-page form by selecting ScrlFwrdd or ScrlBack from the menu.
4. Press **[Esc]** to save settings and exit the StrForm menu, or choose Exit to leave the StrForm menu without saving any changes. For settings with options, you can scroll through them using **[Spacebar]**.

Saving and Using Stair Settings

Use the StrFile option in the 3DStairs menu to save stair settings for future use, as well as load, save, delete, and rename stair setting files:

LoadStr	Loads a stair settings file; choose LoadStr from the StrFile menu and type the name of the file to load
SaveStr	Saves your current stair settings to a file; choose SaveStr from the StrFile menu and type the name of the file to save
DelStr	Deletes a stair settings file; choose DelStr from the StrFile menu and type the name of the file to delete
Rename	Renames an existing stair file; choose Rename from the StrFile menu, choose the file to rename, and type a new name
StrForm	Shortcut to the Stair Form menu. See “Stair Form” earlier in this chapter for more information on using this menu.

Appendix

This appendix contains additional information for advanced users as well as tables defining such things as DataCAD's standard linetypes, hatch patterns, and keyboard shortcuts.

In this chapter:

- ⊕ DWG/DXF translation tables
- ⊕ Creating custom toolbars
- ⊕ Creating linetypes
- ⊕ Standard hatch patterns
- ⊕ Creating keyboard shortcuts
- ⊕ Standard keyboard shortcuts
- ⊕ Standard fonts
- ⊕ Extended characters

DWG/DXF Translation Tables

Importing data from a DWG or DXF file to a DataCAD (.DC5) file is not always a one-to-one translation. There are a few instances where data is handled differently in DataCAD. For instance, entities in AutoCAD with a color-by-block attribute will be read into DataCAD as white.

Dimension arrowheads must also be changed in order to display in DataCAD. AutoCAD arrowheads, which are blocks (symbols), will be replaced *globally* with the arrowhead style set with the ArroStyl option of the Dmension menus. When exported back to a DWG/DXF file, all arrowheads will be set to the AutoCAD default. Dimension text and extension lines will remain the same color, but dimension lines will be changed to the same color as the extension lines.

Text, in some cases, is also treated differently in DataCAD. A single instance of vertical text in a DWG/DXF file will be translated to horizontal text. MText (AutoCAD’s multi-line text) is broken into single lines; it is no longer a single entity.

Third-party fonts may need to be scaled to display correctly. To adjust the scale and aspect of a font, open the CHR2SHX.DAT in the \CHR subdirectory of your DataCAD directory, locate the font in the list, and modify the values as necessary. Fonts can also be added to the list; remember to update the total number of fonts in the file (value located on first line of file). Be sure to make necessary adjustments *before* you import or export a file, so that the changes will appear in that file.

The Importing and Exporting sections that follow provide translation tables showing how entities are treated in “round-trip” scenarios, where a file would begin and end in AutoCAD or DataCAD.

Importing

<u>AUTOCAD</u>	<u>DATA CAD</u>	<u>AUTOCAD</u>
2D Polyline (parallel to X,Y plane)	Polyline	2D Polyline (rel.13 or 14) Polyline (rel.12)
2D Polyline (not parallel to X,Y plane)	Polyline Symbol‡	2D Polyline (rel.13 or 14) Polyline (rel.12)
2D Polyline (with width assigned to any vertices)	Polyline†	2D Polyline (rel.13 or 14) Polyline (rel.12)
3DFace	Polygon	3DFace

<u>AUTOCAD</u>	<u>DATA CAD</u>	<u>AUTOCAD</u>
3D Polyline (with varying Z values)	3D Line	Line
3D Polyline (planar--no varying Z values)	Polyline (parallel to X,Y plane) Polyline Symbol‡	2D Polyline
Arc (parallel to X,Y plane)	Arc	Arc
Arc (not parallel to X,Y plane; thickness= 0)	3D Arc	Arc
Arc (not parallel to X,Y plane; thickness ≠ 0)	Cylinder†	Arc
Attribute Definition	Text† (constant data is saved as attributes; variable data is lost)	Attribute Definition
Block⌘	Symbol	Block
Circle (parallel to X,Y plane)	Circle	Circle
Circle (not parallel to X,Y plane; thickness = 0)	3D Arc†	Circle
Circle (not parallel to X,Y plane; thickness ≠ 0)	Cylinder†	Circle
Dimensions	Associative Dimensions	Dimensions
Dimensions (angular, radius, or diameter dimensions and leaders)	Lines and text	Lines and text
Donut (parallel to X,Y plane)	Polyline†	Donut
Donut (not parallel to X,Y plane)	Polyline Symbol†‡	Donut
Ellipse (parallel to X,Y plane)	Ellipse	Ellipse (rel.13 or 14) Lines (rel.12)
Ellipse (not parallel to X,Y plane)	Ellipse Symbol‡	Ellipse (rel.13 or 14) Lines (rel.12)

<u>AUTOCAD</u>	<u>DATA CAD</u>	<u>AUTOCAD</u>
Hatch	Associative Hatch (if hatch boundary is LW polyline) Associative Hatch Symbol‡ (if hatch boundary is LW polyline but not parallel to X,Y plane) Lines (if hatch boundary includes any other boundary type besides LW polyline)	Lines
Leaders	Lines and Text	Lines and Text
Light-Weight Polyline	(See 2D Polyline above)	
Line (parallel to X,Y plane)	Line	Line
Line (not parallel to X,Y plane; thickness = 0)	3D Line	Line
Line (not parallel to X,Y plane; thickness ≠ 0)	Polygon†	Line (if coordinates unchanged) Line, with changed thickness (if polygon stretched along the axis of thickness) Polygon (if coordinates changed in directions other than axis of thickness)
MText	Text	Text
Mesh	Polygons	Polygons
Point (with no thickness)	Point£	Point
Point (with thickness)	3D Line† £	Point
Polyface Mesh	Polygons	Polygons
Spline	Point†	Spline (rel.13 or 14)
Solid	Polygons, triangulated	3D Faces
Text, single line or multi-line	Text	Text

† AutoCAD entity is imported into DataCAD as an entity with attributes. If the entity is edited in DataCAD, upon export it will return to its original (AutoCAD) entity type, incorporating the changes you made in DataCAD. Exceptions to this are exploding an entity, which permanently changes the entity type, or deleting an entity.

‡ AutoCAD entity is imported into DataCAD as a symbol. These symbols are like any other in DataCAD; they cannot be edited unless they are first exploded. If you explode an entity in 3D space, however, it will flatten to the X,Y plane. For instance, a Donut in a DWG file that is not parallel to the X,Y plane will be imported into DataCAD as a Polyline Symbol. If this symbol is not exploded, it will maintain its position when exported back to a DWG file. If the symbol is exploded, however, it will flatten to the X,Y plane, even when exported back to a DWG file.

£ Point shape is lost in translation.

⌘ A block comprised of entities on different layers will be imported into DataCAD as a symbol on the layer where the block was inserted; that is, all entities for that Block will be moved to the insertion layer. These entities will remain on this layer when exported back to a DXF or DWG file. Also, anonymous (unnamed) blocks in AutoCAD will be given a unique, numeric name when imported into DataCAD; this name will be saved with the block when exported.

There are a few AutoCAD entities that cannot be read, displayed, or otherwise saved during translation. These include: XREFs, viewports, ACIS data (Body), Rays (XLines), Ole Frames, Ole2 Frames, Tolerances, Images, Regions, Proxies, and named views.

Exporting

<u>DATA CAD</u>	<u>AUTOCAD</u>	<u>DATA CAD</u>
3D Arc	Arc	3D Arc
3D Line	Line	3D Line (if not parallel to X,Y plane) Line (if parallel to X,Y plane)
Arc	Arc	Arc
B-spline	Polyline	Polyline
Bezier	Polyline	Polyline
Circle	Circle	Circle
Cone	Polyface Mesh 3D Faces (if Decompose in Options dialog is on during export)	Polygons
Contour	Polyline	Polyline
Cylinder	Polyface Mesh 3D Faces (if Decompose in Options dialog is on during export)	Polygons
Dimensions (Associative)	Dimensions	Dimensions (Associative)
Ellipse	Ellipse (rel.13 or 14) Lines (rel.12)	Ellipse
Hatch (Associative)	Lines	Lines
Line	Line	Line
Mesh Surface	Polyface Mesh 3D Faces (if Decompose in Options dialog is on during export)	Polygons
Point	Point	Point
Polygon (with 3-4 vertices and no voids)	3D Face	Polygon
Polygon (with 5 or more vertices and/or voids)	Triangulated into multiple 3D Faces	Multiple 3-sided polygons
Polyline	Lightweight Polyline (rel.14) Polyline (<rel.14)	Polyline
Sphere	Polyface Mesh 3D Faces (if Decompose in Options dialog is on during export)	Polygons
Surface of Revolution	Polyface Mesh 3D Faces (if Decompose in Options dialog is on during export)	Polygons
Symbol	Block	Symbol
Text	Text	Text
Torus	Polyface Mesh 3D Faces (if Decompose in Options dialog is on during export)	Polygons
Truncated Cone	Polyface Mesh 3D Faces (if Decompose in Options dialog is on during export)	Polygons

Creating Custom Toolbars

Key (.KEY) files are the files that DataCAD uses to display and activate toolbars. You can create your own custom toolbar by creating a key file describing the bitmap files to use for the toolbar icons and the action DataCAD should take when each icon is selected.

The key file is a text file arranged in column groups, each group performing a different function. Once you are familiar with these group functions, you can easily edit the file. Each line of the key file is 185 columns long.

```
L|  Move      |move___m|1          |Move drawing objects |
L|  Copy      |copy___m|2          |Copy drawing objects |
M|  RoofIt    |roofit_m|RoofIt     |3D Roof Creation Tool|
C|  3dkey     |dcad3d_m|m_dcad3d   |3D Icon Toolbar      |
```

A description of each column group and its function follows:

Column(s)	Name	Function
1	Action Code	Contains an action code that identifies the function or program to access
3-4		Not currently used
6-13	Menu Level String	Contains an eight character description for the Message Window
15-22	Bitmap (.BMP) filename	Contains the bitmap (.BMP) filename
24-103	Action	Contains the action to perform, the menu or macro to access, or the sequence to perform
105-184	Message	Contains the message, or description of the function, that appears in the Message Window; acts as a sort of "tool tip"

Column numbers not listed represent pipe characters that separate the columns. You cannot adjust the position of the column separators or type in these columns.

Action Codes

The action codes, represented by L, P, M, and C in column one of the key file example above, identify the function or program to access.

Action Code	Action
!	Places comments or leave a note for yourself within the key file
A	Executes a macro key sequence
C	Loads a new key file for a new toolbar
K	Executes a DataCAD shortcut key
L	Accesses an internal DataCAD function, such as Move
M	Executes a DCAL macro or tool
T	Binds a template file to an icon

Menu Level String

The menu level strings, in columns 6-13, are eight-character descriptions of what each icon does or what it accesses. Any characters are valid and you can enter whatever you want. This description appears in the Message Window before the message. For example, the Move menu icon contains a menu level string that displays Move and a message line that displays Move drawing objects in the Message Window.

Bitmap Filenames

The bitmap (.BMP) filename column group, in columns 15-22, contains the name of the bitmap file, or icon, to associate with the specified DataCAD function. A list of currently used bitmaps in the DCADWIN.KEY and DCADWIN3.KEY files follows. Each bitmap filename matches a DataCAD function name.

Action

Each letter in the Action Code column represents an associated action in the Action column group, columns 24-103. For example, if the Action Code column reads L, the Action column group must contain the menu level string corresponding to the menu it accesses. The following are Action Codes, including necessary Action column group information:

Action Code Information Required in the Action Column Group

A	A key sequence, such as ^:^S6^F1^F1^F5, calls function keys, or keystrokes, in this order. This function allows you to create your own shortcuts to bind to icons. These sequences typically match a sequence from the DCAD.MCR file which contains only 26 user-definable sequences. See "DataCAD Keyboard Shortcuts" in this Appendix for information on creating keyboard shortcuts.
C	An eight-character key filename.
K	The key character that is used in keyboard shortcuts. This action binds a pre-existing sequence to an icon. For example, the shortcut for the Architect menu is [A]. Type a in the Action column, not A; these characters are case sensitive. You cannot use the [Alt], [Tab], or arrow keys in the key character column.
L	The internal menu function number listed in the following table.
M	The name of the tool to access: for example, RoofIt.
T	The path and the filename of the template file to bind to the icon. A comma is required between the path and filename with the file extension, as in the following example: <code>tpl\plumb,residenc.tpl</code> .

DataCAD Internal Menu Function Numbers

1 - 2dmove	31 - freehand	61 - selection sets	90 - rectangular
2 - 2dcopy	32 - linkents	62 - 3dviewer	polygon
3 - 2drotate	33 - fileio	63 - elevation	91 - rectangular slab
4 - 2dmirror	34 - directory	64 - view controls	92 - vertical polygon
5 - 2dstretch	35 - 2pt_arc	65 - editplane	93 - vertical slab
6 - 2denlarge	36 - 3pt_arc	66 - plnesnap	94 - inclined polygon
7 - identify	37 - centangarc	67 - setper	95 - inclined slab
8 - 2dchange	38 - centarc	68 - walkthru	96 - voids
9 - 2derase	39 - centchrd	69 - setobliq	97 - partial
10 - cleanup	40 - arctan	70 - 3dgotoview	98 - 3dmove
11 - architect	41 - ellipse	71 - saveimge	99 - 3dcopy
12 - 2dpolygons	42 - rad_circ	72 - clipcube	100 - 3drotate
13 - curves	43 - dia_circ	73 - hide	101 - 3denlarge
14 - text	44 - 3pt_circ	74 - 3dline	102 - 3derase
15 - linetype	45 - 2dmovedrag	75 - 3dsettings	103 - 3dstretch
16 - hatch	46 - 2drectarray	76 - blocks	104 - 3dchange

17 - 3dedit	47 - 2dcircarray	77 - vertcylnd	105 - 3dexplode
18 - windowin	48 - fillets	78 - horicylnd	106 - 3dedit
19 - to_scale	49 - chamfer	79 - cone	107 - 3dentity
20 - geometry	50 - 1ltrim	80 - truccone	108 - quick shader
21 - 2dgotoview	51 - 2ltrim	81 - sphere	109 - linear dimension
22 - grids	52 - weldline	82 - torus	110 - layer on/off
23 - layers	53 - tintsct	83 - meshsurf	111 - layer name
24 - template	54 - lintsct	84 - revsurf	112 - layer delete
25 - 2dsettings	55 - archdoor	85 - marker	113 - dxf read
26 - measure	56 - archwind	86 - 3dpolygon	114 - dxf write
27 - plotter	57 - cutwall	87 - slab	115 - pixelout
28 - dmension	58 - divide	88 - horizontal	116 - toolbox
29 - display	59 - intersect	polygon	117 - rstar2
30 - objectsnap	60 - tangents	89 - horizontal slab	

Message

Use the Message area column group, columns 105-184, for information about, or a definition of, the icon function. When you move the cursor over an icon, the Message Window displays this message

Creating a New Key File

1. Copy an existing key file from the \DATACAD\SUP\MENUPOF\ directory to a new filename. Store this file in the same directory as the other key files.
2. Edit this new file using existing icons and/or icons that you created. You can associate any bitmap (.BMP) file with any command or function, including functions that are already used in other key files.

Follow the steps in the "Editing a Key File" portion of "Editing Toolbars" later in this section. Remember you can change the bitmap (.BMP) filename in the bitmap (.BMP) filename column when creating a new key file. Changing the bitmap (.BMP) filename allows a different icon to appear in the toolbar. All bitmaps must be 24 x 24 pixels; 16-color bitmaps are recommended.

Incorrectly editing key files can cause system lockups. Make sure that column separators, or pipe characters, remain in their original positions.

Creating New Icons

1. Create a 16-color, square bitmap in bitmap (.BMP) format.
2. Save it with a filename of up to eight characters in the \DATACAD\SUP\MENUPOF\ directory.

Editing Toolbars

You can change any toolbar by editing its key file. You can also change the menu associated with any icon. Make a backup copy of the key file before you edit it.

Editing a Key File

Use the arrow keys to move within the key file. You cannot use Tab or place tabs anywhere in the key file. Incorrectly editing key files can cause system lockups. Please follow these steps carefully and make sure that column separators, or pipe characters, remain in their original positions.

1. Exit DataCAD and open a DOS text editor, such as Notepad. (This will ensure that all characters are displayed properly.)
2. Access the key file directory, \DATACAD\SUP\MENUPOF\, and open the key file you'd like to edit, S_DCAD2D.KEY, for example.
3. Move the cursor to the line where you want to change the menu/icon association.
4. Change any column group:

Action Code	Access a different program; for example, to access a tool instead of an internal DataCAD function, change an L to M.
Menu Level String	Type the eight-character description that you want to appear in the Message Window, a menu name for example.
.BMP filename	Leave the bitmap (.BMP) filename, which represents the icon, as it is. You are choosing a different menu to associate with the icon.
Action	Fill in the information requirements of the Action Code: the new internal function number (listed in the DataCAD Internal Menu Function Numbers table), path and program name, macro name, key sequence, key filename, or key character.
Message	Type the description of the function now associated with the icon.

5. Save the key file and exit the text editor.

All bitmaps must be 24 x 24 pixels. 16-color bitmaps are recommended.

Creating Linetypes

DataCAD allows you to use four standard linetypes and up to 175 user-defined linetypes. The definitions for linetypes provided with DataCAD are located at the end of this section. These definitions are in the DCADWIN.LIN file in directory \DATACAD\SUP\. Define new linetypes using any text editor. The lines consist of small segments linked together to form longer lines. Each segment is 100 "units" in length, and may be divided into as many as 20 definition components. One hundred units equals the line spacing length.

The first line of each definition must contain either an asterisk (*) or a greater than symbol (>), followed by a space and the title of the linetype. When an asterisk is used, DataCAD draws a short line, called a tail, to fill any remaining space not filled by the pattern. When a greater than symbol is used, DataCAD repeats the pattern as many times as possible to fill the specified space and then stretches the pattern slightly to fill any remaining space. The title must not exceed eight characters in length.

The second line specifies the factor by which one segment of the linetype must be multiplied to draw at the correct size. As 100 units equal the line spacing length to draw one segment or 100 units at a size of 8", a factor of 8/12 or .6667 is entered. Successive lines of the definition contain the origin, direction, and distance of each component in the line segment. Use one line for each component, with no more than 20 lines total, including the spacing value.

Each component is assigned a length based on the 100-unit dimension. Each component definition consists of two numbers. The first number specifies the line direction.

0	Right
90	Up
180	Left
270	Down



























The second number specifies the line length. A line length of 50 equals one half the distance of the total line segment. A minus sign (-) before the first number moves the pen clockwise. Pen down commands (visible lines) have a space between the two numbers. Pen up commands (invisible lines) use a space and a minus sign between the two numbers.

Definitions for linetypes provided in DataCAD are as follows:

>TelLine		>ElecLine		>Box	
4.0		4.0		4.0	
0	41	0	40	0	65.858
0	-6	0	-7	0	-10
71.56	-3.953	-71.56	-3.953	45	10
0	3.5	90	7.5	-45	10
-180	-1.675	0	3.5	225	10
-90	7.5	-90	-3.75	135	10
0	-1.675	-180	3.5	0	-24.142
71.56	-3.953	-90	-3.75		
0	-6	0	3.5		
0	41	71.56	-3.953	>PropLine	
		0	-7	8.0	
		0	40	0	75
>Plywood1		>Section		0	-5
0.0625		4.0		0	5
0	100	0	88	0	-5
270	-1	60	6	0	5
180	100	300	12		
270	-32.3333	60	6	>Insul	
0	100			0.1667	
270	-33.3334	>ShingleR		-118.0362	47.3592
180	100	1.0		-88.75	21.5060
270	-32.3333	5	100	-63.25	21.5060
0	100	275	4	-37.75	21.5060
270	-1	95	-4	-12.25	20.5060
180	100	185	-100	12.25	20.5060
90	-1	0	50	37.75	21.5060
45	45.7262	5	100	63.25	21.5060
135	47.1405	275	4	88.75	21.5060
45	45.7262	95	-4	118.0362	94.7184
270	-1	185	-100	88.75	21.5060
		0	50	63.25	21.5060
>Plywood2		>Hedge		37.75	21.5060
0.0625		1.0		12.25	20.5060
0	100	63.26	22.4	-12.25	20.5060
270	-1	0	10	-37.75	21.5060
180	100	289.59	39	-63.25	21.5060
270	-32.3333	21.48	18	-88.75	21.5060
0	100	101.19	17	-118.0362	47.3592
270	-33.3334				
180	100				

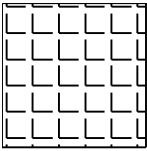
270	-1	14.02	13.7	>LapSidR	
0	100	277.08	26.9	0.5	
90	-1	0	10	98	12.5
135	45.7262	68.12	18	8	110
45	45.7262	51.20	21.3	278	12.5
90	-1	291.48	18	188	110
>CentrLin		>Shiplap		>GroundLn	
8.0		0.5		4.0	
0	85	90	5	0	35
0	-5	0	16	80	4
0	5	90	7.5	-80	4
0	-5	0	96	0	5
		270	5	80	2
		180	16	-80	2
		270	7.5	0	45
		180	96	80	3
>Brick				-80	3
0.6667				80	6
0	-2.605			-80	6
270	45.3125			0	13.924
0	28.125	>RigidIns			
90	45.3125	0.1667			
180	28.125	270	100		
0	-33.333	0	100	>ShingleL	
270	45.3125	-270	-25	1.0	
0	28.125	180	100	355	100
90	45.3125	-270	-25	85	4
180	28.125	0	100	265	-4
0	-33.333	-270	-25	175	-100
270	45.3125	180	100	0	50
0	28.125	-270	-25	355	100
90	45.3125	0	100	85	4
180	28.125	180	-75	265	-4
0	-2.604	270	100	175	-100
		0	-25	1	50
		-270	100		
		0	-25		
>Grass		270	100	>LapSidL	
0.5		0	-25	0.5	
0	9	-270	100	262	12.5
100	50			352	110
280	-50			82	12.5
0	17			172	110
90	30	>4 Block			
270	-30	0.6667			
0	-1	0	-2.3437		
85	40	270	45.3125	>12 Block	
265	-40	0	95.3125	0.6667	
0	16	90	45.3125	0	-2.3438
100	20	180	95.3125	270	145.3125
280	-20	0	-2.3437	0	95.3125
0	17			90	145.3125
75	30	>8 Block		180	95.3125
255	-30	0.6667		0	-2.3438
0	16	0	-2.3438		
100	20	270	95.3125		
280	-20	0	95.3125		
0	8	90	95.3125		
		180	95.3125		
		0	-2.3437		

Standard Linetypes

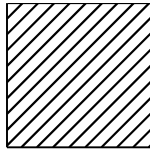
Name		Pattern	Line spacing values	Defined by:
Standard	Solid		N/A	Center
	Dotted		1 = 1'-0" on center dots	Center
	Dashed		1 = 1'-0" on center dashes	Center
	Dot-Dash		1 = 1'-0" on center dot to dot	Center
User-definable lines, 50 maximum	ElecLine		1 = 1'-0" on center E to E	Center
	TelLine		1 = 1'-0" on center T to T	Center
	Box		1 = 1'-0" on center box to box	Center
	PropLine		1 = 1'-0" on center double dashes	Center
	Insul		0.3 = 6" width insulation	Center
	Plywood1		0.0.3/4 = 3/4" width plywood	Left edge at line angle of 90°
	Plywood2		0.0.1/2 = 1/2" wide plywood	Left edge at line angle of 90°
	Hedge		1 = 4" wide line pattern	Center
	CentrLin		1 = 1'-0" dash to dash	Center
	Section		1 = 1'-0" on center breaks	Center
	ShingleR		0.8 = 4" to the weather (for line length divisible by set spacing)	Left edge at line angle of 270°
	LapSidR		0.4 = 4" to the weather (for line length divisible by set spacing)	Left edge at line angle of 270°
	ShipLap		0.4 = 4" to the weather (for line length divisible by set spacing)	Left edge at line angle of 270°
	Brick		0.8 = 4" nominal brick width	Left edge at line angle of 90°
	4Block		0.8 = 4" nominal brick width	Left edge at line angle of 90°
	8Block		0.8 = 8" nominal block width	Left edge at line angle of 90°
	12Block		0.8 = 12" nominal block width	Left edge at line angle of 90°
	RigidIns		0.2 = 2" thickness	Left edge at line angle of 90°
	Grass		0.5 = 3" high grass	Lower edge at line angle of 90°
	GroundLn		1 = 6" between pattern breaks	Lower edge at line angle of 90°
	ShingleL		0.8 = 4" to the weather (for line length divisible by set spacing)	Right edge at line angle of 270°
	LapSidL		0.4 = 4" to the weather (for line length divisible by set spacing)	Right edge at line angle of 270°

Standard Hatch Patterns

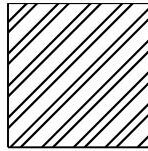
Hatch patterns included with DataCAD are pictured below.



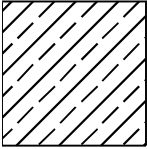
Angle



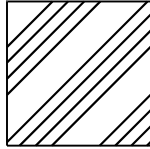
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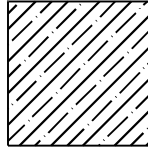
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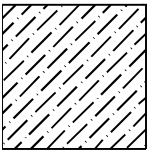
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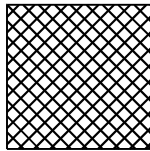
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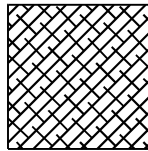
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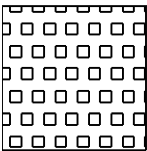
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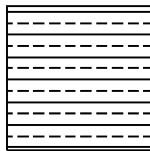
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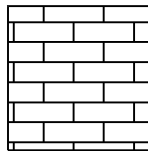
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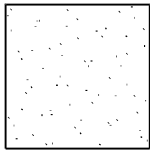
Box



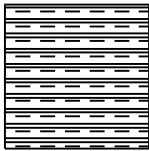
Brass



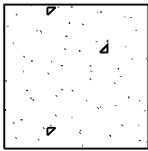
Brick



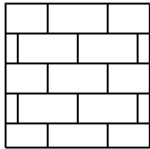
Cement



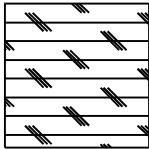
Clay



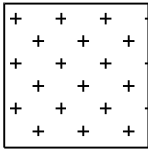
Concrete



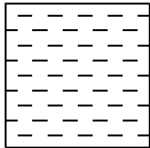
CMU



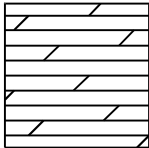
Cork



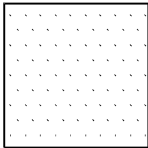
Cross



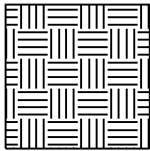
Dash



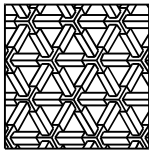
Dolmt



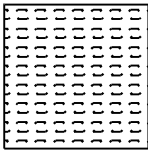
Dots



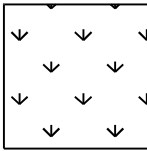
Earth



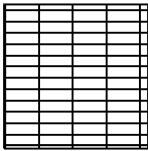
Escher



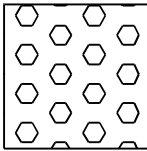
Flex



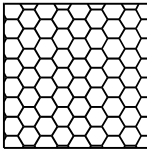
Grass



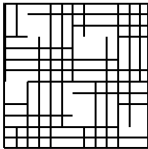
Grate



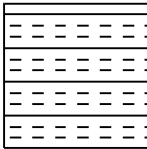
Hex



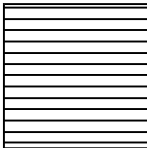
Honey



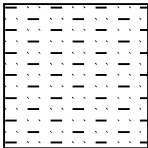
Hound



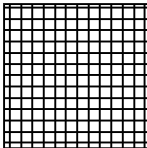
Insul



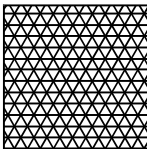
Line



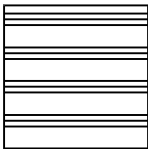
Mudst



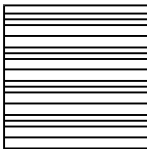
Net



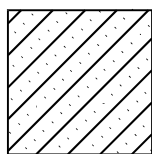
Net3



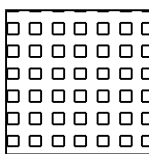
Plast



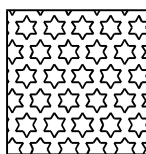
Plast1



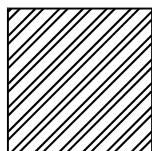
Sacncr



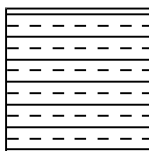
Square



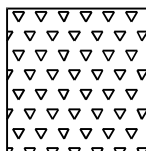
Stars



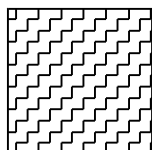
Steel



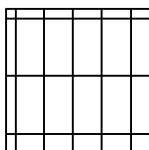
Trans



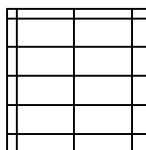
Triang



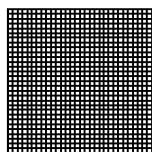
Zigzag



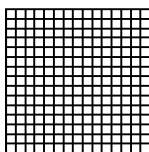
24x48Til



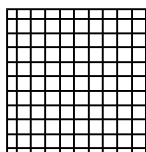
48x24Til



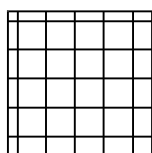
4x4Til



8x8Til



12x12Til



24x24Til

Creating Keyboard Shortcuts

You can use keyboard shortcuts for quick navigation or to perform functions. Shortcuts have also been called keyboard macros, keyboard interrupts, accelerator keys, hot keys and immediate mode commands. To set up your own shortcuts, see "Defining Shortcuts" below.

In this manual, all keyboard shortcuts are described exactly how they should be executed. If a shortcut requires a lowercase letter, you will be instructed to press only that letter. If a shortcut requires an uppercase letter, you will be instructed to press **[Shift]** and that letter. (If Caps Lock is on, you can just press the letter.) In some cases, both the uppercase and lowercase letter perform the same function.

In the following table, the Mnemonic column provides an easy way to remember some of the shortcuts. Some shortcuts, like Copy, are noted as being "2D/3D". In this case, when you use the shortcut from a 2D menu, you'll open the 2D Copy menu. Similarly, using the shortcut while in a 3D menu takes you to the 3D Copy menu.

Defining Shortcuts

To define your own shortcuts, edit the file DCADWIN.MCR in the \DATACAD\SUP\ folder using a text editor. You can only define shortcuts that use a letter in combination with **[Alt]**. You cannot define shortcuts that are accessed by a single letter or those that use a letter in combination with **[Shift]**.

You may have some older keyboard shortcuts that include **[Alt]** + 183 near the beginning of the line and end with **[Alt]** + 184. This was used to turn off menu display to increase the speed of shortcut execution in DataCAD for DOS. It is no longer needed in DataCAD for Windows, although these shortcuts will still function normally in DataCAD for Windows.

Define each shortcut key on a separate line. The following line defines a shortcut to Enlarge:

E;^f6

- The first character E, indicates the key that will start the shortcut. Pressing the character E will call the shortcut. This will also supercede the command that E currently does.
- The semicolon indicates that the command is in the Edit menu; a colon would indicate the Utility menu.
- The caret (^) separates each step through the menu structure.
- F6 indicates the function key that loads the Enlarge option on the Edit menu.
- If the function were on a menu under Enlarge, you would follow the F6 with an additional caret, followed by the appropriate function key on the next menu.

See "Using the Command Line Alias" in "The Drawing Board" chapter for more information on keyboard shortcuts.

Standard Keyboard Shortcuts

FUNCTION/MENU	MNEMONIC	KEY(S)
2LnTrim menu	Join	[Alt]+[J]
3D Edit menu		[J]
3D Entity menu		[J] or [Alt]+[U]
3D Views menu	Views	[Y] or [Shift]+[V] or [Alt]+[V]
Active layer, scroll backward to set		[Shift]+[Tab]
Active layer, scroll forward to set		[Tab]
Append selection set toggle	Append	[Shift]+[A]
Architect menu	architect	[A]
Cap wall ends toggle		[I]
Center screen to cursor location		[Home]
Change menu	Change	[Alt]+[C]
Colors, scroll backward	Kolors	[Shift]+[K]
Colors, scroll forward	kolors	[K]
Coordinate entry		[SpaceBar]
Copy to clipboard	Copy	[Ctrl] + [C]
Copy menu (2D/3D)	Copy	[C] or [Shift]+[C]
Cursor size toggle (big vs. regular)		[+]
Curves menu	Arcs	[Alt]+[A]
Cut to clipboard		[Ctrl] + [X]
Digitizer	digitIzer	[I]
Dimension menu	Dimension	[D] or [Shift]+[D]
Dimension, Linear menu	Dimension	[Alt]+[D]
Directory menu	directorY	[Alt]+[Y]
Display list, regenerate		[U] or [Shift]+[U]
Display associated saved view		[Shift]+[1] to [0]
Edit menu		[;] or [Ctrl]+[C]
EditSets menu	Selection sets	[Shift]+[S]
Elevation, back		[Ctrl]+[PgUp]
Elevation, front		[Alt]+[=]
Elevation, left		[F11]
Elevation, right		[F12]
Enlarge menu	Enlarge	[Alt]+[E]
Erase last entity		[.]
Erase last group		[<]
Erase menu (2D/3D)	Erase	[E] or [Shift]+[E]
Exit		(Ctrl)+(F4)
Fillets menu	Fillets	[Alt]+[F]

FreeHand menu	freeHand	Alt + H
Geometry menu	Geometry	Alt + G
GoToView menu	View	V
Grids menu	Grids	G or Shift + G
Hatch menu	Hatch	H or Shift + H
Help		Ctrl + F1
Hide menu		Shift + Y
Icon toolbar toggle		Ctrl + F11
Identify	Identify	Shift + I or Alt + I
Input mode:		Insert
1 : Relative Polar (distance, angle)		
2 : Absolute Polar (distance, angle)		
3 : Relative Cartesian (x, y coordinates)		
4 : Absolute Cartesian (x, y coordinates)		
Layer search toggle		'
Layers menu	Layers	L or Shift + L
Line overshoot amount		_
Line overshoot toggle		-
Line spacing, set		F
Linetype menu	Linetype	Alt + L
Linetypes, scroll backward		Shift + Q
Linetypes, scroll forward		Q
Line weight, decrement	BIG Weight to small	Shift + W
Line weight, increment	little Weight to big	W
Link Entities menu	linK	Alt + K
Measures menu		Alt + X
Mirror menu (2D)	Mirror	Alt + M
Move menu (2D/3D)	Move	M
Multi-view windows	multi-view Windows	Ctrl + W
New drawing, create	New	Ctrl + N or Alt + N
Object snap	sNap	N or Shift + N
Object Snap menu		Shift + X
Object Snap perpendicular toggle	Object snap	Alt + O
Online Help		Ctrl + F1
Open Files	Open	Ctrl + O
Ortho mode toggle	Orthogonal	O or Shift + O
Orthographic View	orthographic	Alt + O
Pan		(;) (¢) (£) (¥)
Parallel View, most recent		Alt + 9
Paste from clipboard		Ctrl + V

Perspective View, most recent		Alt + +
Plotter menu	Plotter	Alt + P
Polygons menu	Rectangle	Alt + R
Previous view, restore	Previous	P or Shift + P
Print	Print	Ctrl + P
Quit	Quit	Alt + Q
Redo		Ctrl + Y
Reference point entry		'
Restore last erased entity		.
Restore last erased group		>
Rotate menu (2D/3D)	Rotate	R or Shift + R
Save current drawing	Save File	Ctrl + S or Shift + F
Screen refresh		Esc
Screen refresh by layer, stop		End
Screen refresh, stop		Del
Select	sElect	Ctrl + E
Selection set, change active		Alt + 1 to 8
Snap grid spacing, set	spacing	S
Snap grid toggle		X
Snapping point entry		~
Stretch menu	Stretch	Alt + S
Symbol, insert by name		"
Tangents menu		B or Shift + B
Template menu	Template	T or Shift + T or Alt + B
Text menu	Text	Alt + T
Toolbox menu	Macros	Shift + M
ToScale menu	Zoom	Alt + Z
Undo		Ctrl + Z
Utility menu		:
Wall T intersection clean-up toggle		\
Walls toggle and set new wall thickness =		
Weld Line menu	Weld	Alt + W
Window In		/
World coordinate identify		?
Z-base or Z-height attribute entry	Z-base/Z-height	Z or Shift + Z
Zoom Extents		Ctrl + +
Zoom In		PgDn
Zoom Out		PgUp

Standard Fonts

DataCAD includes the following fonts; the first column lists the font name, while the second column provides examples of upper- and lowercase characters.

ARCDR2GP	ABCDEFG abcdefg 1234567890
ARCWY2FW	ABCDEFG abcdefg 1234567890
ARCWY2GP	ABCDEFG abcdefg 1234567890
ARCWY2HC	ABCDEFG abcdefg 1234567890
ARCWY2LC	ABCDEFG abcdefg 1234567890
ARCWY2TX	ABCDEFG abcdefg 1234567890
AVG_TP	ABCDEFG abcdefg 1234567890
BLOCK	ABCDEFG ABCDEF 1234567890
COMPLEX	ABCDEFG abcdefg 1234567890
HLV_BP	ABCDEFG abcdefg 1234567890
HLV_LP	ABCDEFG abcdefg 1234567890
HLV_MP	ABCDEFG abcdefg 1234567890
HLV_TM	ABCDEFG abcdefg 1234567890
HLV_TP	ABCDEFG abcdefg 1234567890
HLV_TPX	ABCDEFG abcdefg 1234567890
MCG_TP	ABCDEFG abcdefg 1234567890
ORIG	ABCDEFG abcdefg 1234567890
ORIG2	ABCDEFG abcdefg 1234567890
ROMAN	ABCDEFG abcdefg 1234567890
ROMAN2	ABCDEFG abcdefg 1234567890
ROMANC	ABCDEFG abcdefg 1234567890
ROMANS	ABCDEFG abcdefg 1234567890
SIMPLEX	ABCDEFG abcdefg 1234567890

Extended characters

Five DataCAD fonts feature extended characters. This allows you to include common drafting symbols, fractions, and exponents in your text:

ARCDR2GP	ArcDraft General Purpose
ARCWY2GP	ArcWyde General Purpose
ARCWY2HC	ArcWyde Heavy Chisel
ARCWY2LC	ArcWyde Light Chisel
HLV_TPX	Helvetica Thin Proportional with extended characters














The following chart shows the drafting symbols, exponents and fractions you can include in your text.

➔ To create drafting symbols and exponents:

- 1. Toggle NumLock on. The indicator light on your keyboard will be green if NumLock is on. If NumLock is not toggled on, you will not be able to create the drafting symbols and exponents.
- 2. Press **[Alt]** and hold it down while you type the corresponding four-digit number using the numeric keypad. The four-digit number always begins with zero. Do not use the number keys above the letters on your keyboard.

If you'd like to change your font but you've used extended characters in your text, make sure the new font also includes the extended characters. If you choose a font that does not include the extended characters, the content of your text may be changed.

To create fractions, use the Exponent/Numerator column of the chart to enter the numerator of your fraction; use the Denominator column to draw the denominator and the division sign. For example, to draw a fraction of 5/16 in your text, hold down **[Alt]** and type **0135** using the numeric keypad. When you release **[Alt]**, the 5 will appear. Depress **[Alt]** again and type **0153** on the numeric keypad. The fraction is completed when you release **[Alt]**.

Drafting Symbol			Exponent / Numerator		Denominator	
	Diameter	0140		0130		0150
	Square Feet	0141	1	0131		0151
	Property Line	0142	2	0132		0152
	Center Line	0143	3	0133		0153
	With	0144	4	0134		
	Degree	0145	5	0135		
	Plus-or-minus	0146	6	0136		
	Angle	0160	7	0137		
			8	0138		
			9	0139		

Glossary

3D face

The surface of a solid specified with 3D points.

3D mesh

A polygon mesh composed of vertices connected by lines.

3D polyline

Polylines with segments oriented in 3D space.

arc

A continuous section of a curved line.

area

Any part of a drawing within a rectangular window that is temporarily drawn during the selection process. All entities and groups enclosed completely within this rectangular window are included in the area. All entities and groups partially enclosed or not enclosed within the rectangular window are not included in the area.

attribute

A single value or characteristic associated with an entity.

bulge

A curved portion of a polyline.

center marks

Small marks displayed to define the exact center of arcs and circles. You can also use center marks with object snapping.

click

To position the cursor over a selection and press and release a mouse button (usually the left button). This command usually refers to selecting or deselecting an option.

choose

To select a command or option from the Menu Window.

command

An instruction from the user to the computer to perform a function. A command is usually selected from the Menu Window or the Menu Bar.

cursor

A graphic symbol displayed on the screen to show the location of the mouse.

default

Any setting or value that is preset by DataCAD.

dimension line

Also called the string line, the dimension line consists of a straight line parallel to the points to dimension. The dimension line also contains text showing distance information and termination marks, consisting of arrows or tick marks drawn at a 45° angle.

division

A graphical aid used in cylindrical balusters, newel posts, or hand rails to help you better visualize the cylinder.

doughnut (donut)

A circular object with hole in the center, composed of two concentric circles and a filled center.

Drawing Window

The area of the screen that is reserved for your drawing.

elevation

The height of an object. It defines the object's position in the direction of the Z-axis.

Also, an orthographic view that shows the front, back, right, or left sides of an object.

entity

A straight or curved geometric element defined by two endpoints. A circle, ellipse, or arc is an entity, as is a B-spline or Bezier curve. A line of text is also considered an entity in editing operations. Edit entities alone or when linked as a group.

extension line

Extension lines typically run from the object you dimension to the line where the dimension appears. Extension lines are used only for linear and angular dimensioning. You can control the length of the extension line. DataCAD typically draws one extension line at each end of a dimension line, but you can suppress the line.

extents

The outer-most edges of a drawing. To “display a drawing at its extents” means to adjust the view so that the drawing fills the Drawing Window as much as possible while still displaying it in its entirety; the outer-most edges of the drawing will fit just inside the frame of the Drawing Window. Clicking the E in the Navigation Pad does this automatically.

fence

Similar to an area but allowing more flexibility, a fence is any portion of a drawing within a closed polygon that you draw during the selection process. You can select using a concave or convex polygon having up to 36 sides.

fit

Specified fixed height of text that expands or contracts to the character width between designated points only.

function keys

Function keys, **[F1]** through **[F12]**, are located on the keyboard and are used to enter commands. In DataCAD, they are associated with the commands that appear in the Menu Window. You can select these commands by pressing the associated function key on the keyboard.

[S1] through **[S0]** provide access to ten additional commands, selected on the keyboard by holding down **[Shift]** while pressing the appropriate function key.

grids

Use display and snap grids to make cursor movement and positioning more accurate. The flexibility and control of grids are critical to computer-aided design.

group

A group is a set of entities linked together logically but not necessarily graphically. Entities that do not touch may be part of the same group, and entities that are contiguous may be part of different groups. A group may contain any number of entities - as many as hundreds or only one.

Entities are linked as a group in a number of ways. A group of entities drawn in succession without disconnecting the cursor are linked as one group. Similarly a series of lines of text entered at the same time are linked as one group. You can link any group of entities or groups using the LinkEnts command. Entities are also linked or associated as one group when they are created at the same time with the Copy command. When an existing entity is edited, with the Change or Move commands, for example, it retains the same group associations.

The grouping together of entities in DataCAD facilitates editing many related items without affecting other entities in the same portion of a drawing. Because they are also easily redefined, groups allow the designer to organize a drawing as a series of elements composed of logically related pieces or entities.

keyboard shortcuts

These key combinations perform an operation or select a setting in the middle of another command by interrupting the original command. Once the interrupt function is complete, DataCAD automatically returns to the original function.

layers

In computer-aided design layers are the electronic equivalent of overlaid pieces of tracing paper, each containing a small part of the drawing information. Each layer can also be saved and loaded for use in other drawing files.

limits

The maximum and minimum sizes indicated by a toleranced dimension.

line

The most fundamental drawing entity; defined by two endpoints.

Menu Bar

An area of the screen, located along the top of the DataCAD screen, that contains menus to give you quick access to many of DataCAD's most used functions.

Menu Window

An area of the screen that displays DataCAD's primary menu structure.



Message Window

The area of the screen reserved for DataCAD prompts and messages. The Message Window also displays information from the keyboard as it is entered.

Navigation Pad

An area of the screen below the Menu Window that displays buttons to pan around or zoom in on your drawing, view it at its extents, recalculate extents or access the 3DViews menu.

ortho mode

Used to draw entities at any angle. Press  to toggle Ortho mode off. To draw entities orthogonal to one another, press  again to toggle ortho mode back on. Modify this snap angle by setting the number of divisions under the SnapAng option of the Grids menu.

orthographic view

A parallel view whose lines of sight are parallel to each other and perpendicular to the plane of projection. Orthographic views include plan views and elevations.

pan

Move observer's view of a drawing by using the arrow keys to the right of the keyboard. Set the pan distance by using `ScrI Dist` in the Settings menu.

point

Use the mouse to point to objects on the screen. Move the mouse to move the cursor to the appropriate location on the screen.

Also, a point in space represented on the monitor as a dot, without width or height.

polyface

3D vertices connected in a specified order.

polyline

A connected sequence of lines and arc segments.

Projection Pad

An area of the screen below the Menu Window and Navigation Pad with buttons that change the view of your drawing.

prompt

A message that appears on the screen in the Message Window providing information or requesting a command or user action.

radius

The measurement used for a cylindrical baluster, newel post, or hand rail. The radius is the distance from the center point of the cylinder to the side of the cylinder.

raster printing

The rendering of a graphic as an image composed of dots rather than of vectors, or lines.

reference point

An arbitrary point in the Drawing Window from which the next point entered will be referenced.

right-click

To press and release the right mouse button.

right-hand rule

A rule that indicates the direction of angles. Point the thumb of your right hand in the direction of the axis in question. Your fingers curl in the direction of positive rotation. The Y axis goes up, so when you point your thumb up, your fingers curl toward you in the left to right direction. Negative angles, therefore, go from right to left.

screen refresh

Updates or redraws the screen from the drawing file in order to display the current status of your drawing. This is particularly useful to monitor your progress during editing operations.

scroll back

Use Scroll Back to view a previously-displayed portion of a list.

scroll forward

Use Scroll Forward to view more options when the list of available choices is too long to fit on a menu.

select

To highlight or otherwise mark an entity, icon, or text.

selection set

A user-defined set of entities and/or groups that contains any number of entities or groups. Selection sets provide increased efficiency during editing by reducing the time required to select entities, groups, or define areas or fences.

setting

An option that is set to a particular value. You can change settings at any time, and they remain in that state until you change them again.

solid

Solid filled figures, composed of quadrilateral (four-sided) or triangular (three-sided) polygons.

Status Area

The area of the screen reserved for status information such as the current layer, layer color, snapping on/off, etc.

string

A series of characters.

swap file

A temporary file created when you enter DataCAD. It is the working file that constrains your drawing during a work session. This working file is transferred to your drawing (.DC5) file when you exit DataCAD.

text

The alphabet, numerals, and special characters added to a drawing. It is created using fonts.

text leaders

Short, sometimes angled lines that are automatically drawn back to a point on the circumference of the object when a dimension distance is too short to contain dimension text.

thickness

The projection of a surface into 3D space.

toggle

An option switch that you can turn on or off. When a toggle is on, the button label is displayed in green; when a toggle is off, the button label is displayed in red.

tolerance

The total amount a specific dimension is allowed to vary, the difference between maximum and minimum limits.

Trace

A solid line with a specified width.

undo

An option that cancels the latest operation or command performed. It appears on the menu only after an action has taken place.

vector printing

Line-based drawings produced with commands that are either native to or added to a laser printer.


view center

The point in space at which the observer is looking. DataCAD places the view center in the middle of the extents, halfway between Z-base and Z-height.

void

An opening or hole created in a slab.

window in

DataCAD's "zoom" function. Displays objects at different scales for drawing tasks. To quickly focus on a portion of a drawing for detailed work, press  to expand that section.

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