OrCAD Layout™ for Windows®

Visual CADD™ Getting Started

*This book includes the **Visual CADD Getting Started** guide to illustrate how to use Visual CADD with OrCAD Layout for Windows. The **Visual CADD User's Guide** begins after the **Visual CADD Getting Started** guide.*

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Technical support	(503) 671-9400
Bulletin board system	(503) 671-9401
Administration	(503) 671-9500
Fax	(503) 671-9501
General email	info@orcad.com
Technical support email	techsupport@orcad.com
Web site	http://www.orcad.com

OrCAD®

9300 S.W. Nimbus Avenue
Beaverton, Oregon 97008 • USA

Contents

Welcome to Visual CADD

Whether you're new to CAD or an experienced CAD user, Visual CADD will help you become super-productive in your work.

If you're new to Visual CADD or a new CAD user, you'll find everything you need to know to get started right in this book.

Where to find help 6–7

Whether you're new to Visual CADD or already familiar with it, you'll be able to find the information you need quickly using these resources. Start with the lessons in this book for a Visual CADD primer.

Starting Visual CADD 8–11

After you install OrCAD Layout for Windows, review these pages to become familiar with the Visual CADD drawing window and tools.

Where to find help

This tutorial provides four lessons that you can use to learn the basics of Visual CADD. The lessons are based on the typical process of creating a drawing. In the short time it takes you to complete them, you'll see how quickly you can become an efficient Visual CADD user.

You can find additional information on using Visual CADD in several places. Use the table on the next page to find out what information is available to you, so you can quickly find answers to your questions.

Overview of the Tutorial lessons

Lesson...	What you learn...
Lesson 1	Preparing the drawing area. Drawing the board outline. Saving your work.
Lesson 2	Editing objects.
Lesson 3	Adding text and dimension labels.
Lesson 4	Printing—including basic printing and export options.

More Help for Learning Visual CADD

Look here...	For this information...
Visual CADD User Guide	The Visual CADD User Guide describes all procedures and settings in Visual CADD in detail. The task-oriented topics contain everything you need to do your work, and they provide numerous illustrations to help you understand CAD concepts.
Online Help	Help is always available at your fingertips. To access it, click Help Topics from the Help menu. Then, use either the Contents or the Index to find the information you need.
Dialog box Help	If a Visual CADD window contains a question-mark button  in the upper right corner, you can click it and then click any item in the dialog box to see a description of the item.

Starting Visual CADD

When you start Visual CADD, you will see the Visual CADD window and the drawing board, as shown here.

The Visual CADD drawing board contains a menu bar, speed bar, toolbar, and status bar. As you work, you can zoom in and out as far as you want, because the drawing area has no limits or page boundaries.

When the drawing board is visible, you can start a new drawing immediately. To work on an existing drawing, click Open from the File menu, and then find its folder. You can open and view up to 64 drawing files at one time, depending upon the memory available.

Once you have finished working on a drawing in Visual CADD and have saved it, you can close the drawing by clicking Close from the File menu. When you have completed a work session in Visual CADD, click Exit from the File menu.

Note: If you're an advanced user, you can use keyboard shortcuts to perform most Visual CADD commands. See the Visual CADD User Guide section of this book for a list of shortcuts.

A TOUR OF THE VISUAL CADD DRAWING BOARD

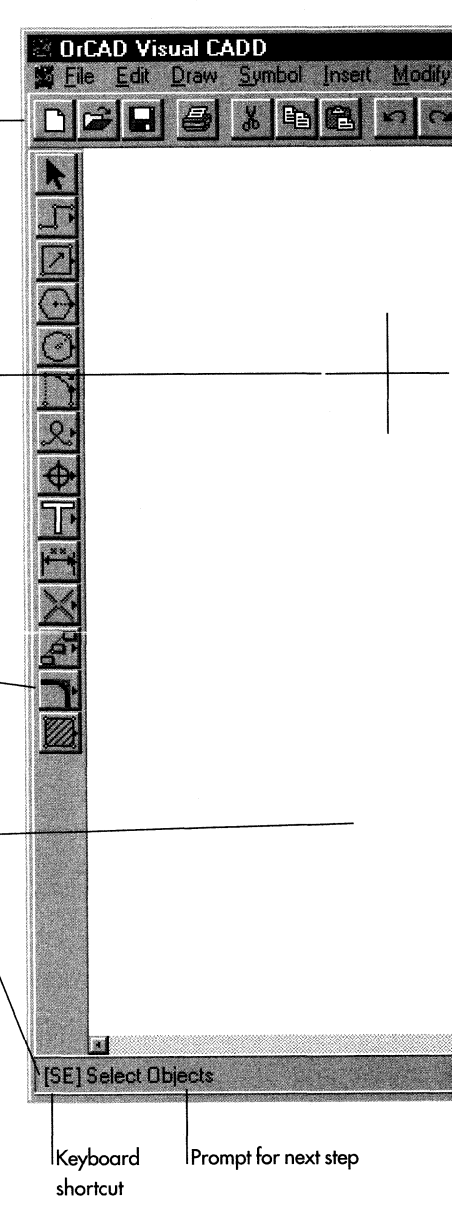
The speed bar displays different options depending on which tool and command you are using. Different speed bars appear when you choose certain commands from the menu bar or from the right mouse button menu, or when you use certain tool buttons.

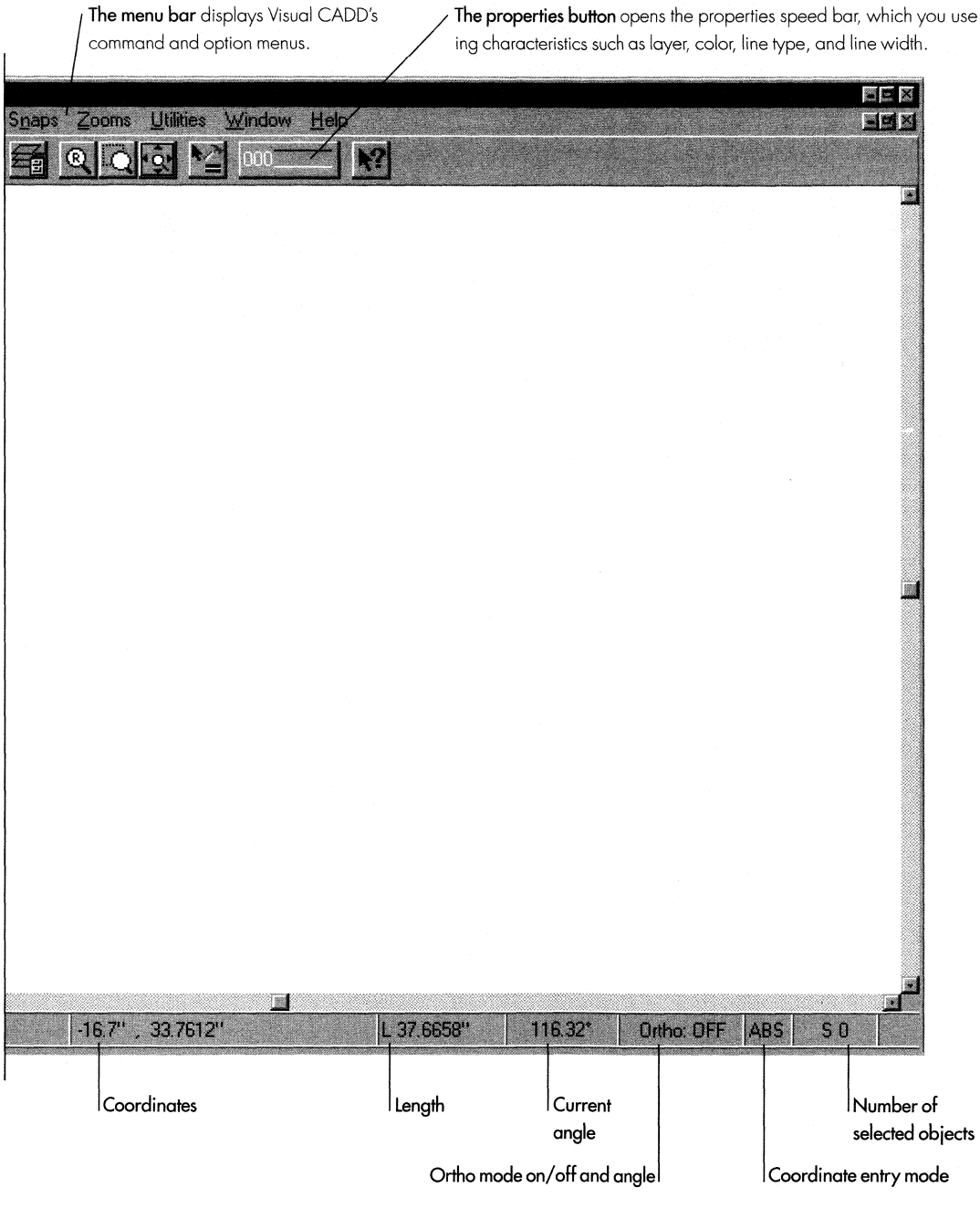
The cursor varies in appearance depending on the tool you use and the task you perform. Your mouse, digitizer puck, or keyboard instructions move the cursor.

The toolbar gives you quick access to all drawing tools and some important commands. For other tools related to the displayed tool, click any tool button with a right-arrow, and then click the tool you want; or click the right mouse button on any tool that has a right-arrow (▶), and then toggle through the related tools.

The drawing board has no limits or page boundaries, so you can zoom in and out as far as you want.

The status bar lists the keyboard shortcut for the current command or tool, prompts you with the next step in your task, gives you basic object information, and on high-resolution monitors, lists the ortho-mode (or constraint) status, the coordinate system, and the number of selected objects. Click the right mouse button on the corresponding area of the status bar to change the measurement system, to turn ortho mode on and off, and to change the coordinate system.




The menu bar displays Visual CADD's command and option menus.
The properties button opens the properties speed bar, which you use to set drawing characteristics such as layer, color, line type, and line width.
Snaps Zooms Utilities Window Help
-16.7" , 33.7612"
L 37.6658"
116.32°
Ortho: OFF
ABS
S 0
Coordinates
Length
Current angle
Ortho mode on/off and angle
Coordinate entry mode
Number of selected objects

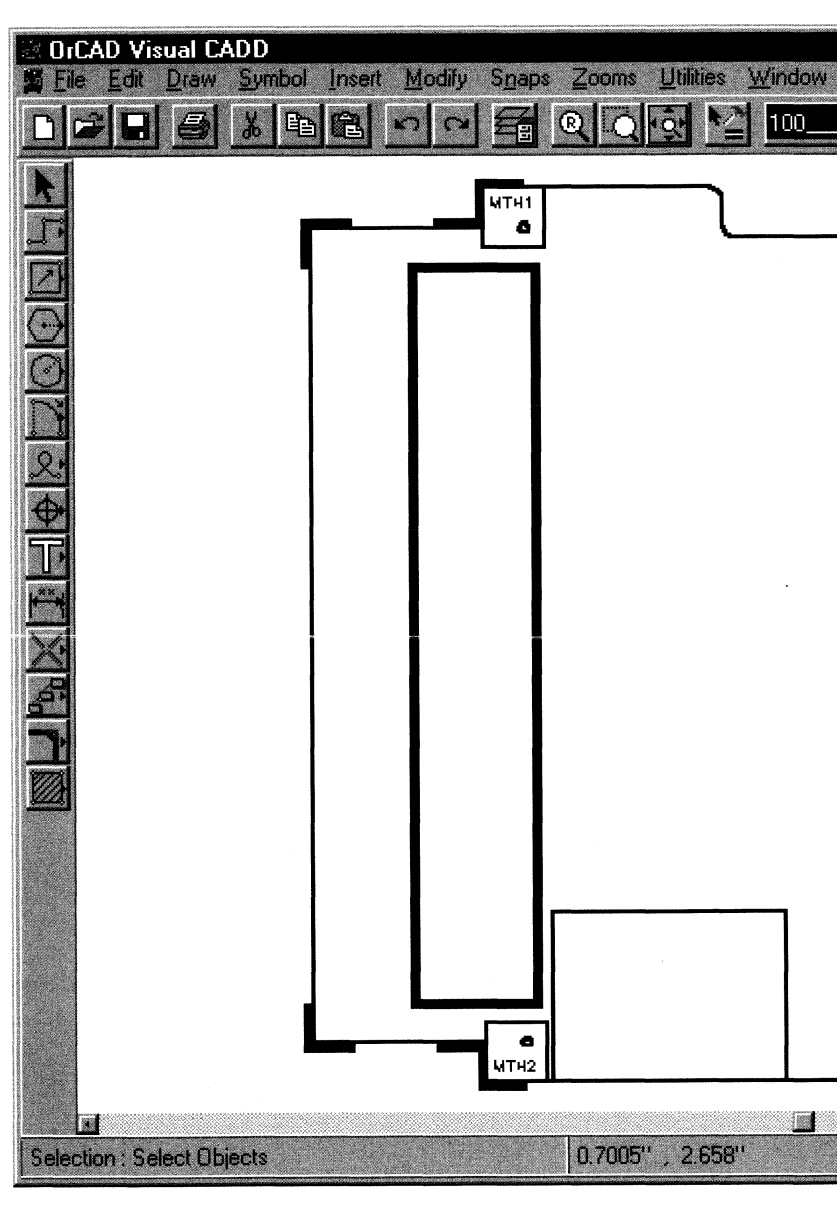
An overview of Visual CADD terms

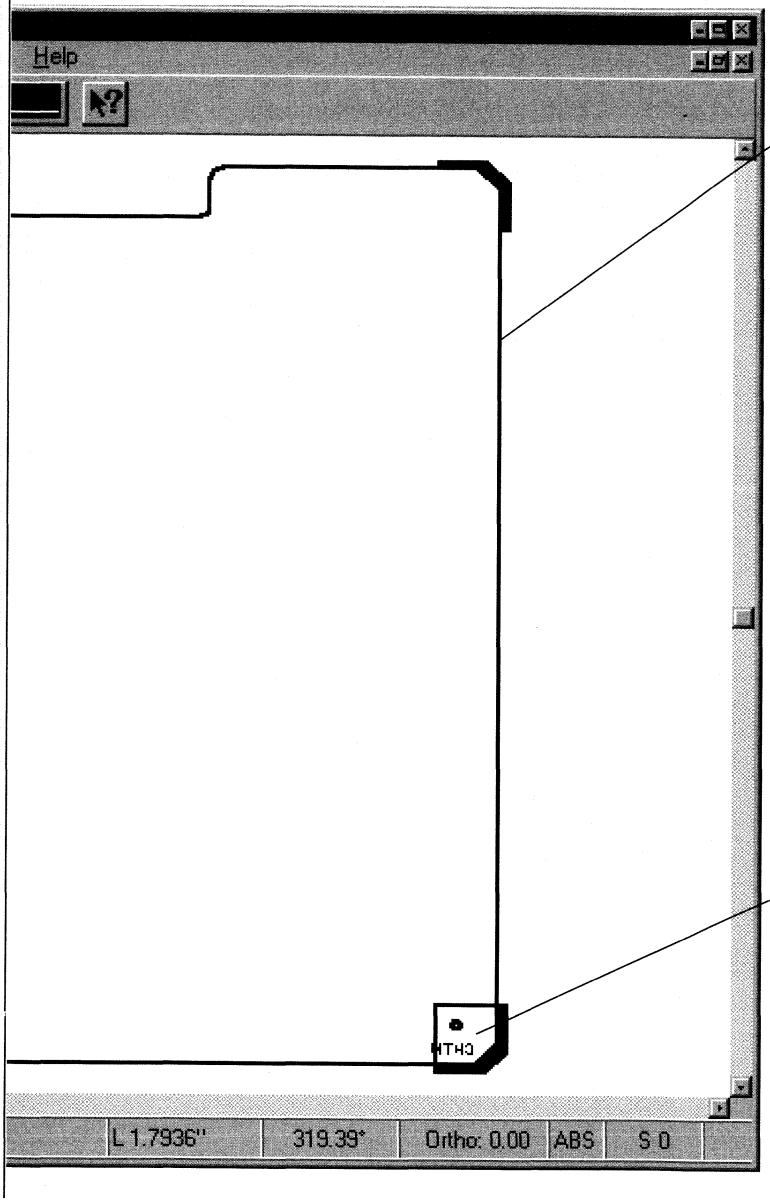
When you start working in Visual CADD, some terms might be unfamiliar, especially if you are new to computer-aided design (CAD). Use this illustration to become familiar with the basic terms used to describe the parts of your drawing.

For example, in Visual CADD, the lines you draw are called *entities*. As you draw, individual entities are combined to form *symbols*, such as an IC chip, mounting holes, or connection blocks.

Another difference between CAD and conventional drafting is that you do not lay down lines by drawing each stroke precisely. Instead, you place the points that define a drawing entity. Each point contains X and Y coordinates.

PARTS OF A VISUAL CADD DRAWING





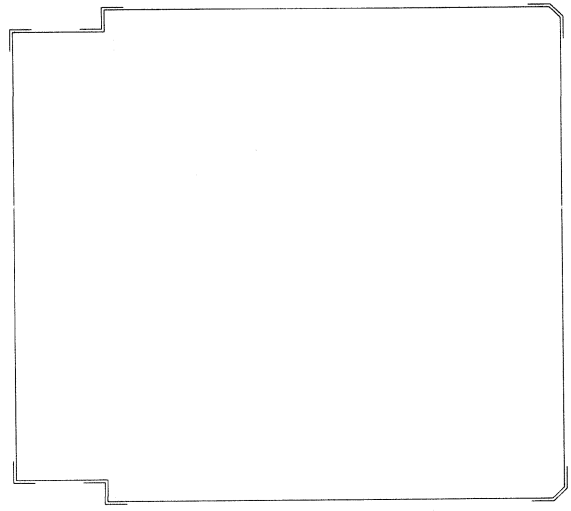
Entities are the building blocks of a Visual CADD drawing. Each arc, line, symbol, text block, and dimension is a separate entity.

Text resides within a block after you have typed it. You can resize, move, and edit each text block and change its properties separately.

Starting the drawing

Although Visual CADD is used in many different industries and applications, in these introductory lessons you will create a basic printed circuit board outline. The first lesson shows you how to size the drawing area, how to set up a unit of measurement to work in, and how to draw lines to create the board outline using exact dimensions.

When you finish this lesson, the board perimeter will be done, you will know how to use techniques essential to CAD drawing, and your drawing should look like the one shown here.



LESSON

1

Preparing to draw 14–15

The first task you undertake in any drawing is to set up the drawing area using the required dimensions and measurement unit. You also set up drawing defaults, such as line color, and linetype.

Creating the board outline 16–19

Here, you will learn how to create a board outline using basic drawing techniques and tools.

Adding crop marks 20–23

The crop marks you'll create here will show you how to make use of the Mirror command and how to use tracking. Tracking allows you to place points a certain distance from other points without drawing construction lines.

Preparing to draw

Before you begin drawing, you set up drawing properties in the Settings dialog box. This is where you define the measurement unit you want to work in, the line color, and the number of layers you want to work with

Working with layers

Layers help you organize your drawing. Creating layers is like drawing on transparent sheets; you can either view everything together, or view individual drawing components.

Creating the drawing view

As you work in these lessons, you will zoom in and out of various areas of the drawing to complete your work. In the beginning, however, you'll want to adjust the view of the drawing area so that the entire drawing will fit on the screen and the origin point (0,0) will be in the bottom left corner. You are always able to draw in unlimited space in Visual CADD.

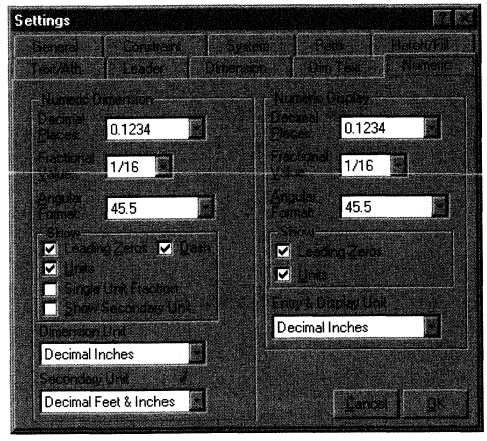
Visual CADD's default tool is the selection tool. For these lessons, make sure the selection tool is your default tool by clicking the Utilities menu, clicking Settings, clicking the System tab, and then verifying that Default Tool is set to Selection.

To start a new drawing

1. If Visual CADD is not open, select the Visual CADD option from the Tools menu in Layout.
2. To begin the lesson, click New from the File menu.

To set the measurement unit

1. On the Utilities menu, click Settings.
2. In the Settings dialog box, click the Numeric tab.
3. In the Numeric Display area, change the Decimal Places to **.1234**.
4. In the Entry & Display Unit list box, click Decimal Inches.
5. In the Numeric Dimensions area, change Decimal Places to **.1234**, and then click Decimal inches in the Dimension Unit list box.
6. Click OK to make the changes.



TIPS

Get help in any dialog box by clicking the help button and then clicking on the option about which you want to know more.

If you make a mistake, click the undo button on the main speed bar to undo the last editing or drawing action.

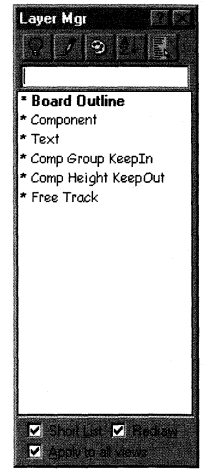
To set the line color

1. Click the Properties button on the main speed bar.
2. On the properties speed bar, change the Color to **9** (blue), and make sure all other fields are 0.
3. Click OK when you're finished.

To set up layers

1. On the Utilities menu, click Layer Manager.
2. Click **0**.
   A zero (0) appears highlighted in the edit box. The bold number or name indicates this is the current layer.
3. Click layer 0 a second time.
   An edit box appears, allowing you to enter the layer name.
4. Type **Board Outline** and then press Enter.
5. Click on the X in the upper right hand corner of the Layer Manager to close it.

   Note: Layer options are available from the right mouse menu during the placing of obstacles from the Insert pull-down menu.



To create the drawing window

1. Click the Snaps menu, and then click Manual Entry: Absolute.
   When you enter X,Y coordinates in Absolute entry mode, they are relative to the point of origin (0,0).
2. Click the Zooms menu, and then click Window.
3. Type **-1,-1**, and then press Enter.
4. Type **6,6** (do not put a space between the comma and the numbers).
5. Press Enter.

Your drawing window now represents a 6" x 6" Printed Circuit Board.

To turn on ortho mode

1. Click the Snaps menu, and then click Ortho Angle.
2. Type **0** in the Ortho Angle edit box, and make sure Ortho mode is checked.
3. Click OK to close the speed bar.

Creating the board outline

To create the board outline you will use these drawing techniques and tools, which can be applied to any line drawing:

- Selecting an obstacle from the Insert menu
- Drawing the board outline by using absolute and relative coordinates
- "Snapping" lines to other objects by using snap near point
- Drawing the board outline by using direct-distance entry and ortho mode

Selecting an obstacle from the Insert menu

All of the available obstacles that will be used for creating the PCB layout can be found on the Insert menu.

Drawing by using X,Y coordinates

To begin the outline, you will use Manual Entry: Absolute and then change to Manual Entry: Relative, which interprets X,Y coordinates as relative to the last point drawn rather than from the origin (0,0).

TIP

To undo a vertex, press the Backspace key. To cancel a command in progress, press Esc.

Using direct-distance entry

Instead of entering absolute or relative coordinates to draw the outline, Visual CADD provides *direct-distance entry*. Using direct-distance entry, you drag the cursor in the direction you want, type the distance of the line you want to create, and then press Enter.

Using ortho mode

In Visual CADD, you constrain lines to be horizontal, vertical, or other 90° relationships by using *ortho mode*.

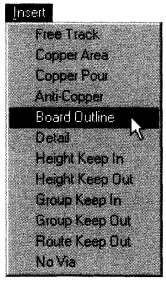
When ortho mode is on, Visual CADD constrains lines to 90° increments of the specified angle. This ensures square corners and true horizontal and vertical lines. Visual CADD uses polar coordinate notation, where 0° corresponds to a clock's 3:00 position and 90° to the 12:00 position.

Snapping lines for precision

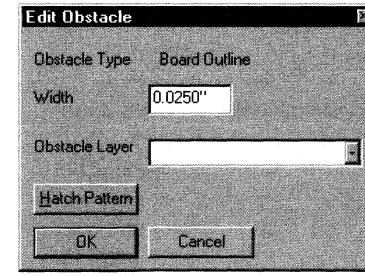
Using snaps in Visual CADD, you don't need to know the precise location of a point. Instead, you define the relationship of the point you are placing to an existing point.

Snap near point is the snap you will probably use most often because it snaps to the closest geometric point near your cursor. You can invoke it by typing NP, or by holding down Shift when you click the right mouse button.

To select an obstacle



1. Click the Insert menu, and then click Board Outline.
2. Move your cursor over the drawing area, click the right mouse button, and then click the Edit Obstacle option.
3. From the Obstacle Layer list select Board Outline.
4. In the Line Width box, type .025.
5. Click OK to make the changes.

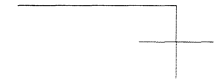


Setting the Grid spacing and display

1. Click the Snaps menu, and then click Grid/Grid Size.
2. On the speed bar, set the X value to .01. The Y value will update to match the X value automatically.
3. Click both Snap to Grid and Display Grid check boxes to turn them on.
4. Click OK to make the changes.

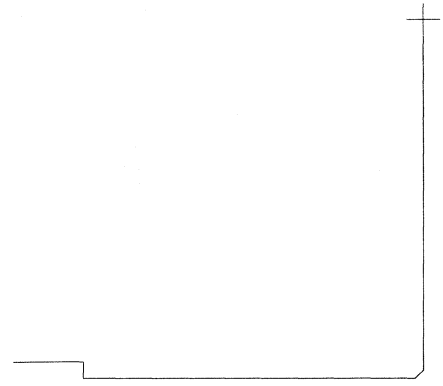
To create the board outline by using coordinates

1. To place the first point of the outline at the absolute coordinates of X = 0" and Y = 0", type **0,0**, and then press Enter.
2. Click the Snaps menu, and then click Manual Entry: Relative.
3. Type **0.85,0**, and then press Enter.
4. Type **0,-0.2**, and then press Enter.



To use direct-distance entry to continue the outline

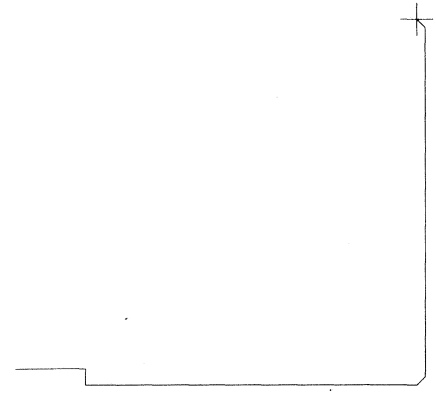
1. Drag the cursor to the right, type **4,** and then press Enter.
2. Type OR to use the keyboard shortcut to toggle Ortho Mode off.
3. Type **0.1,0.1**, and then press Enter.
4. Type OR again to toggle Ortho Mode back on. Drag the cursor up, type **4.3**, and then press Enter.
5. Toggle ortho mode off again by typing OR.
6. Type **-0.1,0.1** and then press Enter.
7. Click the Snaps menu, and then click Cursor Free.
8. Click the Snaps menu, and then click Ortho Mode.



To draw an angled line using ortho mode

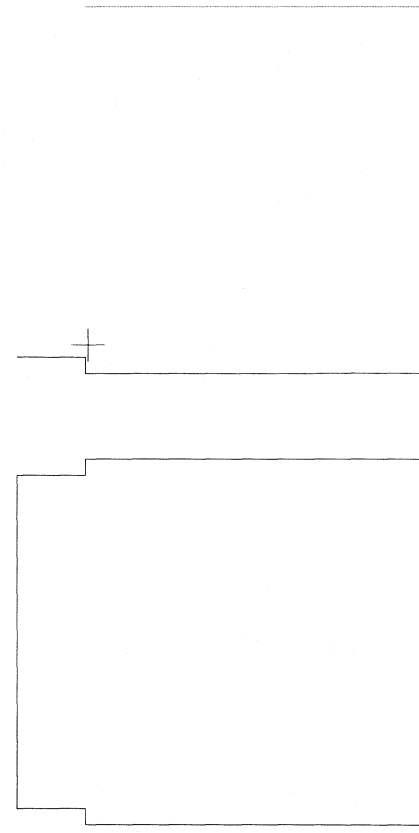
To draw the angled corner, you need to change the ortho angle to 45°. This setting constrains the cursor and the rubberband line to 45° rather than 0°.

1. Click the Snaps menu, and then click Ortho Angle.
2. On the ortho-angle speed bar, change Ortho Angle to **45**, make sure the Ortho Mode checkbox is checked, and then click OK.
3. Drag the cursor up and to the left, type **.1414**, and then press Enter.
4. Type **OA,** on the ortho-angle speed bar, change the Ortho Angle back to 0°, make sure the Ortho Mode checkbox is checked, and then click OK.



To draw a line by referencing other points in the drawing

1. Click the Snaps menu, and then click on Cursor Free.
2. Drag the cursor to the left showing the drag image moving horizontally to the left.
   Notice your line is still constrained, but the cursor is no longer attached to it.
3. Move your cursor down and to the left until you are next to the second point that you placed. With your cursor close to the line, type **NP** (snap near point).
4. Drag the cursor down, type **.2**, and then press Enter.
5. Drag the cursor to the left to start the constrained line moving in the appropriate direction, and then move the cursor down next to the original starting point.
6. Hold down the Shift key and click the right mouse button on the original start point (this is the same as typing NP, as you did in step 3).
7. Click the right mouse button, and then click Pen Up to complete the board outline.



To save your work

1. From the File menu, click Save As.
2. In the File Name box, type **Outline**, and then click Save.

Adding crop marks

To place the crop marks at a specific distance from the board outline, you use *tracking*. Tracking allows you to place points a certain distance or angle from other points without drawing construction lines. To ensure that you start tracking exactly where the points intersect, you will use *snap near point* again to snap to the closest geometric point near your cursor.

To set the properties for the first crop mark

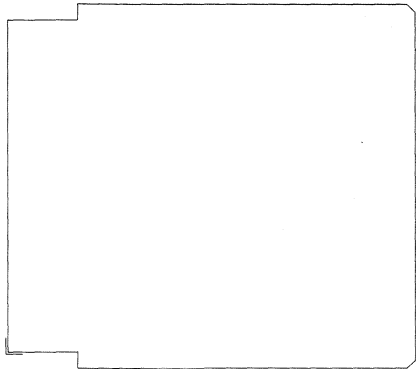
1. Click the Insert menu, and then click Free Track.
2. Click the right mouse button, and then click Edit Obstacle.
3. In the width box, type **.05**, and then press Enter.
4. Click Board Outline in the Obstacle Layer list box, and then click OK.

To draw the crop mark

1. Click the Snaps menu, and then click Track.
2. When you are prompted for the starting point, move the cursor to the lower left corner of the board outline.
3. Type **NP** (snap near point) to start tracking exactly where the two lines intersect.
   Until you end Tracking, rubber-banding lines will appear, indicating your current position.
4. Drag the cursor to the left, type **.025**, and then press Enter.
5. Drag the cursor up, type **.175**, and then press Enter
6. Click the right mouse button, and then click Track End.
7. Drag the cursor down, type **.2**, and then press Enter.
8. Drag the cursor to the right, type **.2**, and then press Enter.
9. Click the right mouse button, and then click Pen Up to release the Free Track tool.

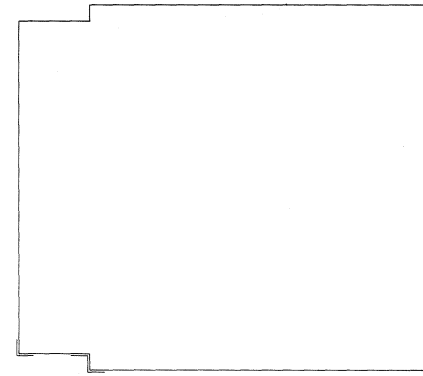
To zoom in on the lower left corner of the board outline

1. Click the Zooms menu, and then click Window.
2. Type **-.5,.5**, and then press Enter.
3. Type **1.5,-.75**, and then press Enter.
   This will change the display so that you have a closer look at the lower left corner of the board outline.



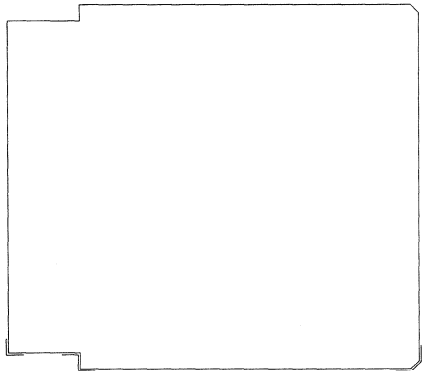
To draw the next crop mark

1. Click the Insert menu, and then click Free Track.
2. Click the right mouse button, and then click Edit Obstacle.
3. Type **.05** in the Width box, click Board Outline for the Obstacle Layer, and then click OK to make the changes.
4. Click the Snaps menu, and then click Track.
5. When you are prompted for the starting point, move the cursor to the upper right corner of the board outline in the current view.
6. Type **NP** (snap near point) to start tracking exactly where the two lines meet.
7. Drag the cursor to the left, type **.225**, and then press Enter.
8. Drag the cursor down, type **.025**, and then press Enter.
9. Click the left mouse button, and then click Track End.
10. Drag the cursor to the right, type **.2**, and then press Enter.
11. Drag the cursor down, type **.2**, and then press Enter.
12. Drag the cursor to the right, type **.2**, and then press Enter.
13. Click the right mouse button, and then click Pen Up to release the Free Track tool.



To draw the third crop mark

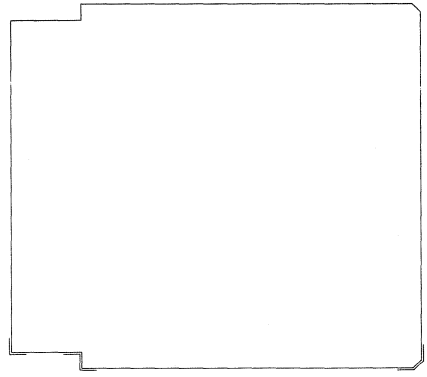
1. Use the scroll bar at the bottom of the screen to move to the right until you see the lower right corner of the board.
2. Click the Insert menu, and then click Free Track. Set the properties just as you have done in the last 2 steps. (Width: **.05**, Layer: Board Outline).
3. Click the right mouse button, and then click Track.
4. Move the cursor over the lowest of the two points in the corner, and then type **NP**.
5. Drag the cursor to the left, type **.1896**, and then press Enter.
6. Drag the cursor down, type **.025**, and then press Enter.
7. Click the right mouse button, and then click End Track.
8. Drag the cursor to the right, type **.2,** and press Enter.
9. Type **OA**, then type **45**, and then press Enter.
   This sets the Ortho Angle to 45°.
10. Drag the cursor up and to the left, type **.162**, and then press Enter.
11. Type **OA**, then **0**, and then press Enter.
12. Drag the cursor up, type **.2**, and press Enter.
13. Click the right mouse button, and then click Pen Up.
14. Type **ZA** to display all of the work you have done so far.
15. Type **DS** to save.



Drawing the final crop marks

All three of the remaining crop marks can be drawn in one step using the Mirror command.

1. Click the Edit menu, and then click Select: Object.
2. Move the cursor over one of the three crop marks and click the left mouse button.
   Note: If at any time you select an entity that you did not want, simply click on it again to deselect it.
3. Hold down the CTRL key and click the left mouse button on each of the remaining crop marks to select all three.
4. Click the Modify menu, and then click Mirror.
5. Click the Snaps menu, and then click Midpoint.
6. Move the cursor over the right edge of the board outline, and then click the left mouse button.
7. Drag the cursor to the left.
   You will see an outline image at the top of the board that represents the placement of the mirrored crop marks.

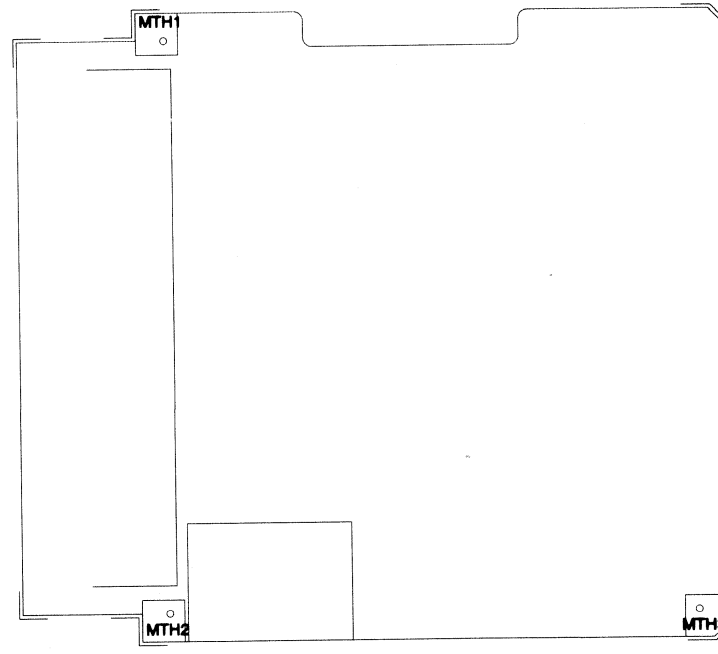


Editing the drawing

Once your basic framework is created, you'll use the Visual CADD tools to add entities to your drawing and to modify existing entities. In this lesson, you will learn how to:

- Add internal obstacles
- Clean up lines and entities
- Place and edit text
- Create new objects from existing objects
- Use the fillet tool to round corners

Your drawing should look similar to this one when you finish the lesson.



LESSON

2

Adding internal obstacles 26–27

Here, you will add KeepIn and KeepOut areas. You will also create and place mounting holes.

Placing and editing text 28–29

You will label the mounting pads.

Creating new objects from existing objects 30–32

You will add a notch to the top of the board outline. You will use the match tool to create the inside of the notch.

Finishing the notch 33

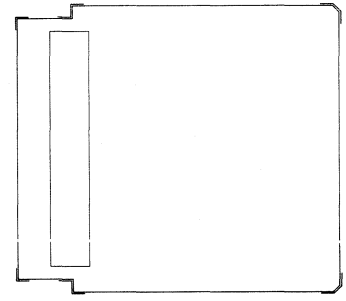
You finish the notch by using the Multiple Trim command. Here, you learn how to trim to a path of objects. The Fillet command is used to round the corners of the notch.

Adding internal obstacles

Once you have finished the outline of the board, you will need to define internal obstacles to create restricted areas or boundaries. Here you will add a Height KeepOut area and a Group KeepIn area. You will also define the boundary of a Copper Area.

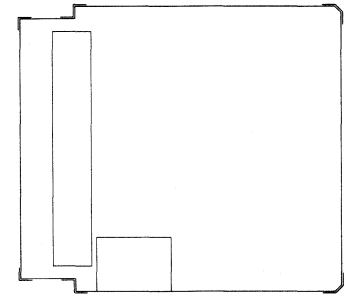
To place the Height KeepOut

1. Click the Insert menu, and then click Height KeepOut.
2. Click the right mouse button, and then click Edit Obstacle.
3. Set the width to **.05**, and then click OK to make the changes.
4. Click the Snaps menu, and then click Manual Entry: Absolute.
5. To place the first point of the obstacle, type **.5,.2**, and then press Enter.
6. Click the Snaps menu, and then click Manual Entry: Relative.
7. Drag the cursor to the right, type **.6**, and then press Enter.
8. Drag the cursor up, type **3.7**, and then press Enter.
9. Drag the cursor to the left, type **.6**, and then press Enter.
10. Drag the cursor down, type **3.7**, and then press Enter.
11. Click the right mouse button, and then click Pen Up.



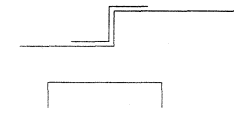
To place the Group KeepIn

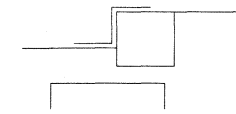
1. Click the Insert menu, and then click Group KeepIn.
2. Click the right mouse button, and then click Edit Obstacle.
3. Set the width to **.015**, and then click OK to make the changes.
4. Click the Snaps menu, and then click Manual Entry: Absolute.
5. To place the first point of the obstacle, type **1.175,-.2**, and press Enter.
6. Click the Snaps menu, and then click Manual Entry: Relative.
7. Drag the cursor to the right, type **1.15**, and then press Enter.
8. Drag the cursor up, type **.85**, and then press Enter.
9. Drag the cursor to the left, type **1.15**, and then press Enter.
10. Drag the cursor down, type **.85**, and then press Enter.
11. Click the right mouse button, and then click Pen Up.



Creating a mounting pad

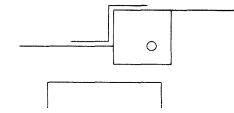
1. Type **MO** to set the manual entry mode to Absolute.
2. Click the Zooms menu, and then click Window.
3. To place the first corner, type **.5,4.5**, and then press Enter.
4. To place the second corner, type **1.4,3.75**, and then press Enter.
5. Click the Insert menu, and then click Group KeepOut.
6. Click the right mouse button, and then click Edit Obstacle.
7. Set the width value to **.015** and then click Plated Mounting Holes from the layer list.
8. Click OK to make the changes.
9. Hold down the Shift key, and then click the right mouse button near the upper left corner of the board outline.
10. Drag the cursor to the right, type **.3**, and then press Enter.
11. Drag the cursor down, type **.3**, and then press Enter.
12. Drag the cursor to the left, type **.3**, and then press Enter.
13. Drag the cursor up, type **.3**, and then press Enter.
14. Click the right mouse button, and then click Pen Up.





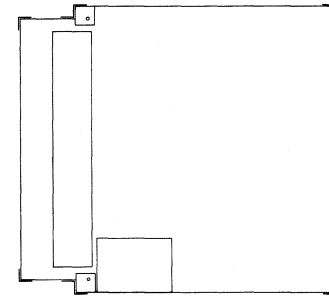
Creating a mounting hole

1. Click the Draw menu, and then click Circle: 2 point.
2. Click the Snaps menu, and then click Track.
3. Hold down the **Shift** key, and then click the right mouse button near the lower right corner of the mounting pad.
4. Drag the mouse up, type **.1**, and then press Enter.
5. Drag the mouse to the left, type **.1**, and then press Enter.
6. Click the right mouse button, and then click Track End.
7. Drag the cursor to the right, type **.025**, and then press Enter.



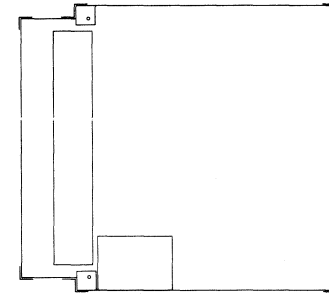
Creating the next mounting hole

1. Click the mounting pad, hold down the CTRL key, and then click the mounting hole so that both the hole and the pad are selected.
2. Type **ZA** to zoom out to the extents of your drawing.
3. Click the Modify menu, and then click Mirror.
4. Click the Snaps menu, and then click Midpoint.
5. Click the right side of the board outline.
6. Drag the cursor to the left to create a horizontal axis for the entities to be mirrored about.
7. Click to finish the mirror operation.



Creating the last mounting hole

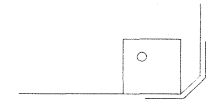
1. Select both the new mounting pad and mounting hole by clicking one, and then pressing the CTRL key while clicking the other.
2. Click the Modify menu, and then click Mirror.
3. Click the Snaps menu, and then click Midpoint.
4. Click the bottom edge of the board outline.
5. Drag the cursor up to create a vertical axis for mirroring.
6. Click to finish the mirror operation.

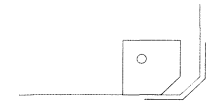


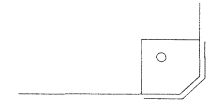
Chamfering the last mounting pad

The third mounting pad needs to fit into the existing corner. You will use the chamfer command to create the corner angle.

1. Click the Zooms menu, and then click Window.
2. Type **4,.25**, and then press Enter.
3. Type **5, -.3**, and then press Enter.
4. Click the Modify menu, and then click Chamfer.
5. Click the right mouse button, and then click Chamfer Dist.
6. In the speedbar at the top of the screen, enter **.1** for the Dist. 1 value.
7. Move the cursor over the Dist. 2 edit box and the value will update to match the first value.
8. Click OK to make the changes.
9. Click the bottom edge of the mounting pad.
10. Click the right edge of the mounting pad.
11. Click the Edit menu, and then click Select: Adjoining.
12. Click any where on the outside edge of the mounting pad.
13. Hold down the CTRL key and click the mounting hole.
14. Click the Modify menu, and then click Move.
15. Click anywhere in the drawing area, and then drag the cursor to the right.
16. Type **.1**, and then press Enter.





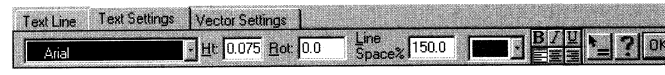


Placing and editing text

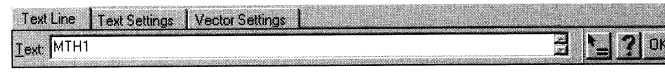
After creating the mounting pads, you will need to label them. For this you will use the Text Tool to place the first label and then use the Array Copy tool to place labels on the remaining pads. Finally, you will use the Text Editor to edit the labels.

Labeling the first pad

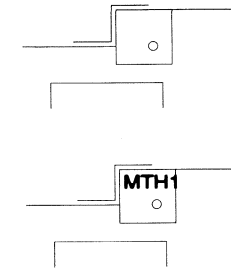
1. Click the Zooms menu, and then click Window.
2. Type **.5,4.5**, and then press Enter.
3. Type **1.4,3.75**, and then press Enter.
4. On the Toolbar, click the Text Line button.



5. Click the right mouse button, and then click Text Settings.
6. Select Arial from the font listbox on the Text Settings speed bar at the top of the screen.
7. Click the Ht edit box and type **.075**.
8. Click OK to make the changes.



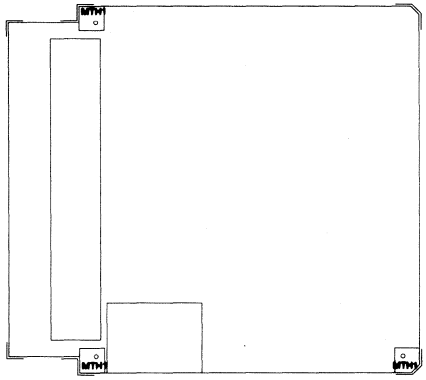
9. Type **MTH1** in the edit window on the speed bar.
10. Move the cursor to the top left of the mounting pad, and then place the text so that it is above the mounting hole. If you don't like where it is, click the text again to pick it up, and then move it to a new location. Once you have located it where you want it, click the OK button on the speed bar at the top of the screen.



Using Array Copy to place the remaining labels

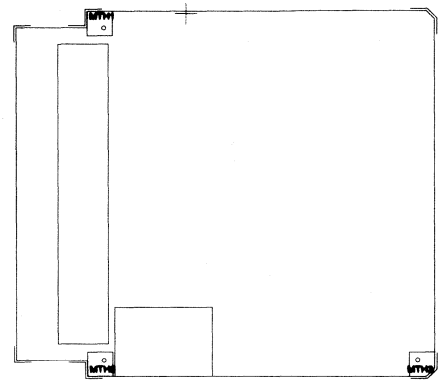
1. Click the label text to select it.
2. Click the Modify menu, and then click Copy: Array.
3. On the speed bar at the top of the screen, set the number of copies to **1** and the number of rows to **2**.

4. Click OK to make the changes.
5. Click near the text label.
6. Type **ZA** to set your current view to the entire board.
7. Drag the cursor down, type **4.35**, and then press Enter.
8. Move the cursor up toward the original label. Drag the cursor to the right, type **3.75**, and then press Enter.
9. Select the extra label in the upper right corner of the board by clicking the text.
10. Press the DELETE key on your keyboard to erase the extra label.



Editing the labels

1. Select the label in the lower left corner of the board by clicking the text.
2. Click the right mouse button, and then click Text Line.
3. Change the text string in the edit box at the top of the screen to read **MTH2**, and then click OK to keep the changes.
4. Repeat steps 1 through 3 for the label in the lower right corner of the board. Change the text string to read **MTH3**.



Creating new objects from existing objects

In Visual CADD, it's easy to create objects from existing ones using the Match Tool and Match Entity commands.

Here, you'll create a notch in the top edge of the board outline. You will use the Match Entity command to set all of your drawing properties to that of the existing board outline.

TIP

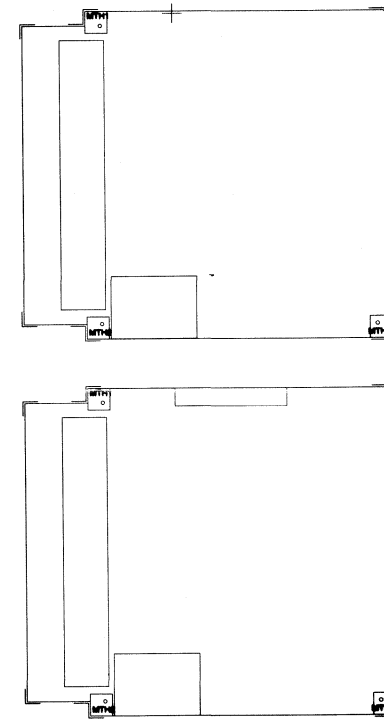
Select more than one object by holding down CTRL as you click each object.

To match the existing board outline

1. Click the Utilities menu, and then click Match Entity.
2. Click the top edge of the board outline.

To draw the notch

1. Click the Draw menu, and then click Line: Continuous.
2. Click the Snaps menu, and then click Object.
3. Click the top of the board outline, about one-third of the way in from the left.
4. Drag the cursor down, type **.25**, and then press Enter.
5. Drag the cursor to the right, type **1.5**, and then press Enter.
6. Drag the cursor up, type **.25**, and then press Enter.



Finishing the notch

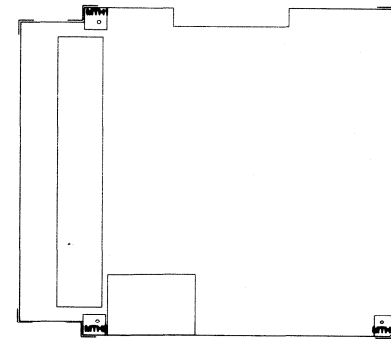
To finish the notch, you'll perform two operations.

First, to open the board outline to include the notch, you'll trim the existing board outline to a path. A path is a group of objects joined by their end points.

Next, you'll round the corners of the notch using the fillet tool. The fillet tool is used to curve the intersection of two lines. When you fillet the intersection of two lines, an arc joins them.

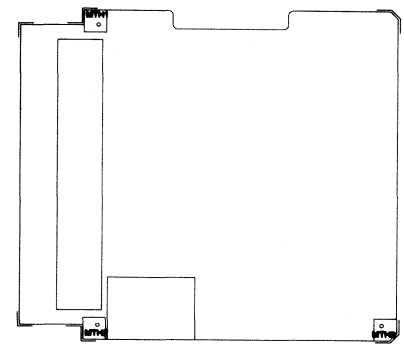
To open the notch

1. Click the Edit menu, and then click Select: Adjoining.
2. Select the top of the board outline by clicking the outline.
3. Click the Modify menu, and then click Trim: Multiple.
4. Click the inside edge of the notch to define it as the trim boundary. Move the mouse around to see a preview of possible trims.
5. Move the cursor toward the center of the board, so that the preview shows the notched area open, and then click to complete the trim operation.



Rounding the corners of the notch

1. Click the fillet tool.
2. Click the right mouse button, and then click Fillet Radius.
3. On the fillet-radius speed bar, make sure the Preview Fillet box is checked, change the Fillet Radius to **.062"**, and then click OK.
4. Click near the intersection of the upper left corner of the notch, and then drag the cursor until the correct fillet appears.
5. Click to accept the fillet. Repeat for the other three corners by clicking near their intersections.
6. Press Esc twice to end filleting.
7. Click the Zooms menu, and then click All.
8. Save your work by clicking the save icon on the speed bar.



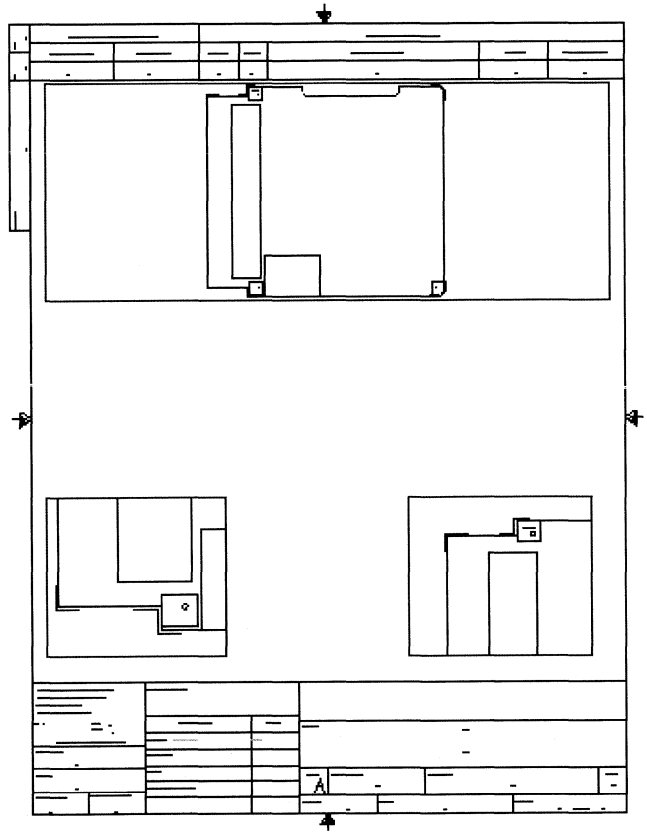
TIP

The fillet tool is persistent; that is, it repeats until you press Esc or select another tool or command.

Printing

In this lesson, you will print your drawing both to fit any size paper and to a specific scale. You'll use Visual CADD's reference frame entities to print your drawing within a prepared title block at three different scales.

When you print your drawing using reference frames, your output will look similar to this illustration.



Printing your drawing 36–37

Visual CADD can automatically scale your drawing to fit the paper in your output device. This is particularly practical for creating drafts of your project. For more technical drawings, you can define a specific scale at which to print.

Printing to scale by using reference frame entities 38–39

Reference frame entities give you the ability to put several drawings or parts of drawings on the same printout. Each drawing or drawing detail can be scaled separately. And because each one is a *reference* to the original drawing or detail, when you update the original, the combined drawing is updated as well.

Printing your drawing

The easiest and most common way to print your drawing on any output device is to use the Fit to Page command, which scales your drawing to the paper you have selected.

You may prefer to set a specific scale for your drawing, particularly for technical drawings such as blueprints.

When you print to scale, Visual CADD uses the measurement unit you specified in the drawing. However, you can use any measurement unit in the Print dialog box, and Visual CADD will recognize it.

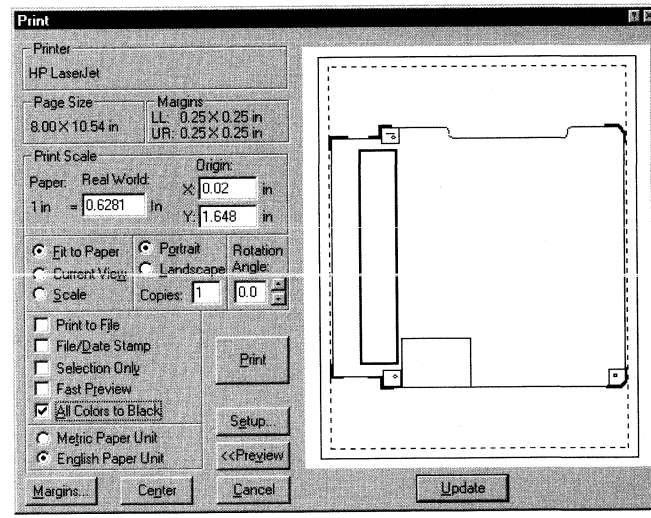
Setting up your printer

Visual CADD uses the default printer and paper size that you have set up in Windows. For this lesson, you will create output on 8.5" x 11" paper. If your default printer is not set up to print at this size, you need to change it before starting the following exercises. Also check that your printer is properly connected and functioning. Your page size may be slightly different than that shown here.

Note: The page size is the paper size minus the margins. For example, 8.5" x 11" paper with .25" margins results in 8" x 10.5" page size.

To print to fit the page

1. Click the Print icon on the main speed bar.
2. In the Print dialog box, make sure that the page size reflects that your default paper size is 8.5 x 11, and you are using portrait orientation.
3. Click the Preview button.
4. Click Fit to Paper, and then click Update.
5. Click Print to send the drawing to the printer.

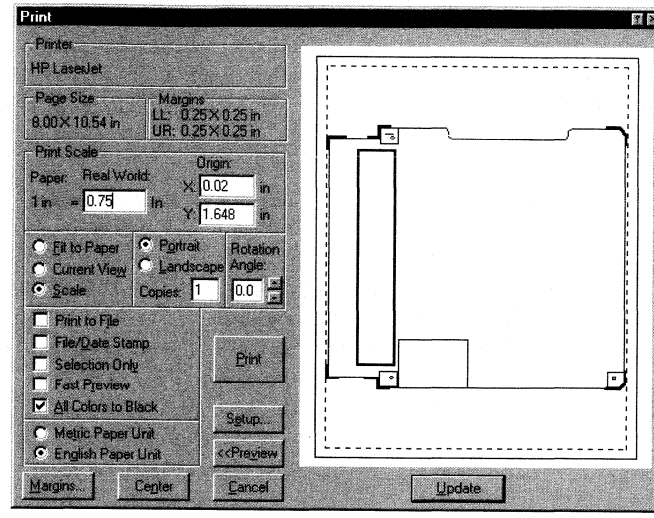


TIP

When you have color in your drawing, but you are printing to a black-and-white output device, you do not need to change any settings. In the Print dialog box, just make sure that the option All Colors to Black is checked.

To print to scale

1. Click the Print icon on the main speed bar.
2. Click the Preview button.
3. Change the Real World value to **.75**, and then click Center Drawing.
   Visual CADD uses the measurement unit you specified in the drawing, so here it will print at a scale of 1" = ¾".
   The Center button centers the drawing on the paper and updates the print preview to match the new scale.
4. Click Print to send the drawing to the printer.



Printing to scale by using reference frame entities

When you need to output a drawing at two or more different scales, you can use reference frame entities to make this an easier task.

Reference frame entities create a reference, or link, to any other drawing on your hard disk.

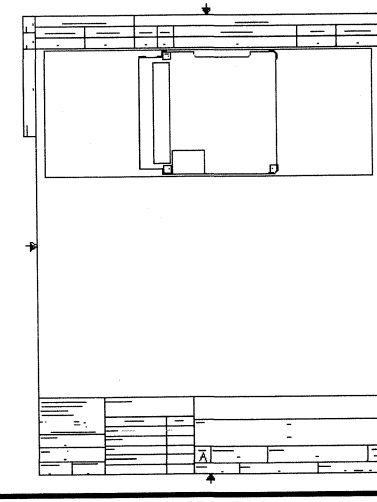
For example, you can create a file that has a title block drawn to the size of your paper, and then reference another drawing to print at a different scale on the same page. The entire drawing can then be printed at a scale of 1:1.

When you save this file (let's name it Printout.vcd), the title block and the drawing files are linked together. If you change one of the drawing files, it will be automatically updated in the file Printout.vcd as well.

Note: Reference frame entities have their own zoom commands, which are available only by using the right mouse button to click the reference frame.

To use a reference frame

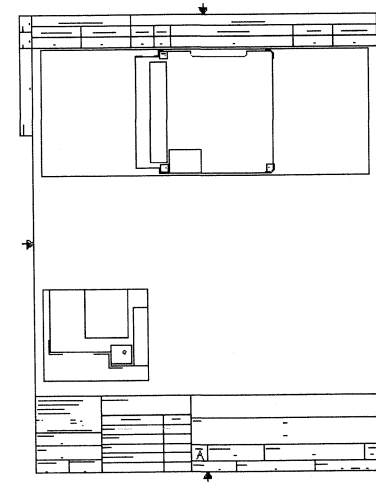
1. Click Open from the File menu.
2. Open the file Titleblock.vcd in the Visual CADD\Tutorial folder. The Title Block is 7.5"x10" designed for use on an 8.5"x11" piece of paper.
3. On the Draw menu, click Reference Frame, and then click Create Reference Frame.
4. For the first point of the reference frame, type **.75,7.0**, and then press Enter.
5. Type **7,5** and then press Enter for the second point.
6. On the reference frame speed bar, click the MDI tab.
7. Click Outline.vcd from the drop-down box, and then click OK.
8. Click the Reference frame to select it, and then click the right mouse button and click RF Zoom All.



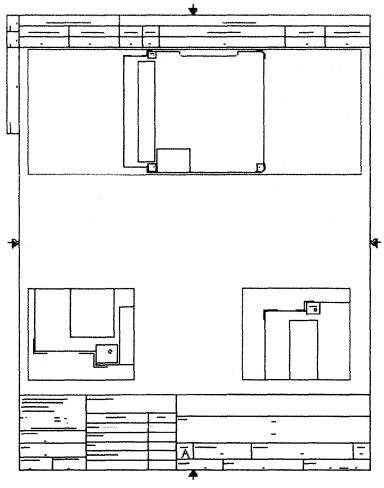
To insert a detail

1. On the Draw menu, click Create Reference Frame.
2. For the first point of the frame, type **.75,2.5**, and then press Enter. For the second point, type **3,4.5**, and then press Enter.

3. Click the MDI tab from the speed bar at the top of the screen.
4. Click Outline.vcd from the drop down box, and then click OK.
5. Use the right mouse button to click the frame you just placed, and then click RF Zoom Window.
6. Click to draw a window around the lower left corner of the board.



To insert a detail to scale



1. On the Draw menu, click Create Reference Frame.
2. For the first point of the frame, type **5.25, 2.5**, and then press Enter. For the second point, type **7.5, 4.5**, and then press Enter.
3. Click the MDI tab from the speed bar at the top of the screen. Click Outline.vcd from the drop-down box, and then click OK.
4. Use the right mouse button to click the frame you just placed, and then click RF Zoom Scale.
5. On the Zoom scale speed bar, type in **1"**, and then click OK to set the zoom scale to 1" = 1" (an inch on the drawing is equal to an inch when printed).
6. Click the right mouse button, and then click RF Pan.
7. Click in the upper left corner of the reference frame to define a new center of the screen.
8. Press the spacebar to repeat the pan command, and then click in the upper left corner of the reference frame.
   The top of the mounting hole is in the middle of the reference frame.

To print to scale with reference frame entities

Since the reference frame entities are already scaled, you just need to print your drawing at a scale of 1:1.

1. Click the Print icon, and then click Preview.
2. Change the Print Scale to **1**.
3. Click Center Drawing.
4. Click the Print button to send the drawing to the printer.

Congratulations!

You have successfully completed the OrCAD Layout for Windows *Visual CADD Getting Started* lessons and have mastered the Visual CADD basics. As you work in Visual CADD, you can get additional help from the *Visual CADD User Guide* and from online Help.

Visual CADD offers two kinds of online Help.

- While working in Visual CADD, you can click the Help menu, and then click Help Topics to find the help topics that interest you.
- In any Visual CADD dialog box or window that has a help button  you can click the help button and then click any item in the dialog box to find out more about it.

You are now ready to become a super-productive Visual CADD user!

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Visual CADD™ User's Guide

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Technical support	(503) 671-9400
Bulletin board system	(503) 671-9401
Administration	(503) 671-9500
Fax	(503) 671-9501
General email	info@orcad.com
Technical support email	techsupport@orcad.com
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Beaverton, Oregon 97008 • USA

Credits

Writing: Mark Ashley, J. Gustave Elliott
Illustration: Amy Carroll Smith, William Schaefer, Keith Smith
Designs by Visual CADD users:
Plat, page 28: Sarasota Computer Associates, Inc.
High-speed clipper, page 54: Claude Hilico
Clothing pattern, page 108: Lisa A. Shanley
Yacht, page 136: Dickes Yacht Design
Single-family residence, page 152: Linda Joy Weinstein, AIA
Editing: Nancy Raiken
Production: Mark Ashley
Book Design: Allen Matsumoto
Cover Design : Tim Girvin Design
Special thanks: Kristy Collins, Bob Dewing, Craig Fischer, Brent Gilbert, Kelly Kimball, Cathy McBride, Tim Murphy, Sam Taylor, Micah Vivion

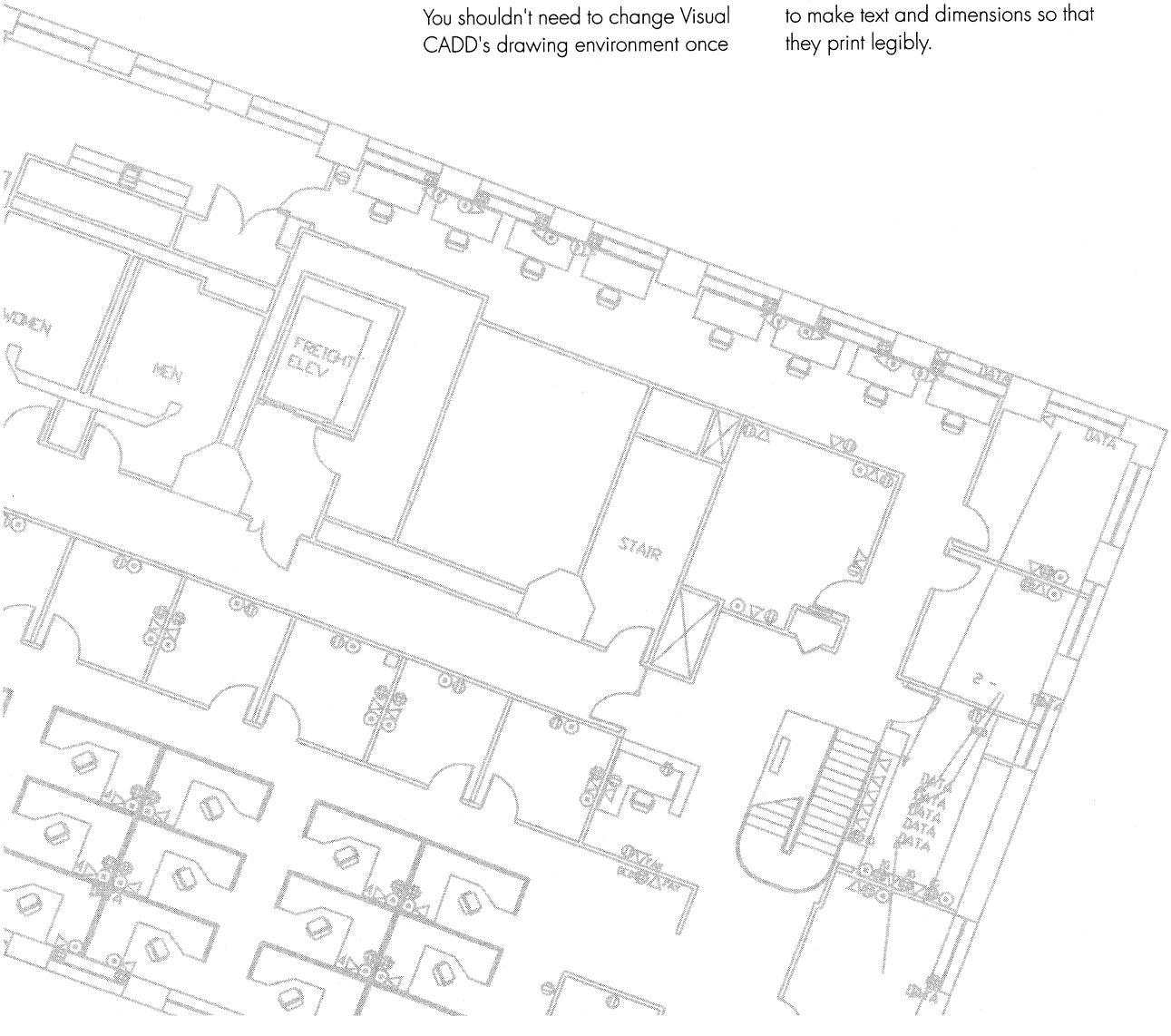
Contents

Setting up a drawing

In conventional drafting, you usually draw objects at a scale different than their actual size because your paper dictates the size of your drawings.

Visual CADD's drawing board is limitless, so you draw with the actual dimensions of the objects you're drawing. You decide on the paper size at the end of the drawing process.

You shouldn't need to change Visual CADD's drawing environment once you have set it up. Whenever you start a drawing, however, you need to decide on the measurement system you'll use, the kinds of information your drawing will contain (so that you can organize it into layers), and what size to make text and dimensions so that they print legibly.



CHAPTER

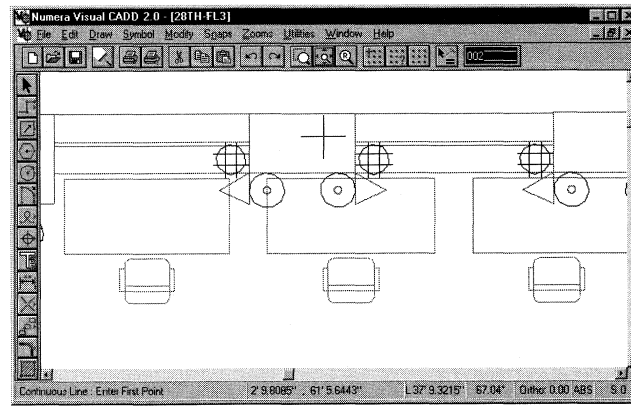
1

Understanding Visual CADD

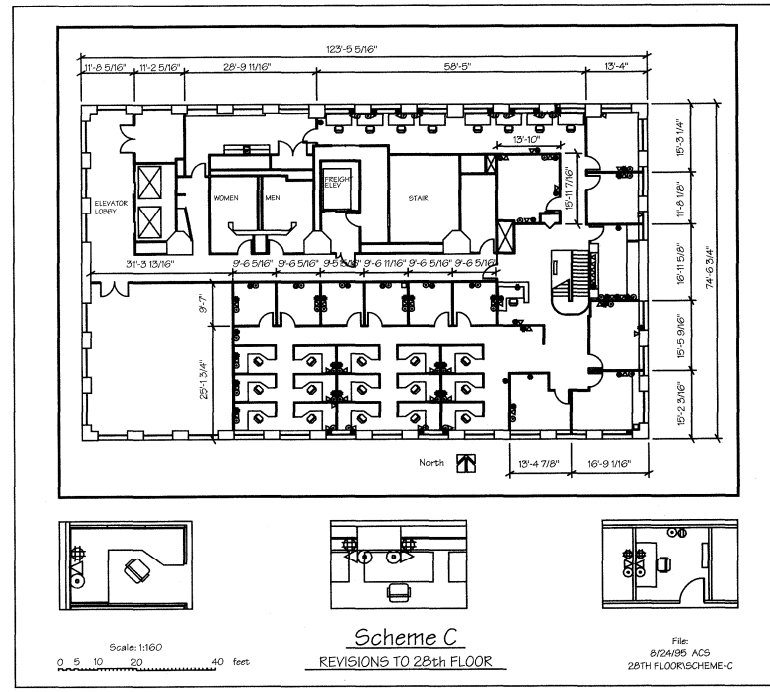
Computer-aided design (CAD) creates a database of coordinates in two dimensions that closely mimics three-dimensional reality.

In conventional drafting you draw each line precisely. In Visual CADD you place only the points necessary to define corners, centerpoints, vertices, foci, and control points of drawing entities. Visual CADD creates the objects from the coordinate information of the points you place. You can place points quickly and precisely by typing coordinates on your keyboard, and you can change the coordinate system as you work to key the points you're placing off of the origin, a basepoint, or the last point entered, depending on which is most efficient.

CREATING A DRAWING IN VISUAL CADD



Draw full-scale reality.



Develop a layout for communicating the appropriate information at a legible scale, and then send the final layout to the printer or plotter.

FOR MORE INFORMATION

Planning your drawing

Before you begin to draw, you'll want to set up Visual CADD to help you work most productively. You can choose the default tool, decide which drawing elements you want to appear on the screen, change the size of the cursor, and change the colors of the drawing environment.

Setting up a measurement system

In Visual CADD, you can enter numeric values using feet and inches, decimal feet, or meters at any time. Dimensions can contain up to two different measurement units.

Setting up layers

Traditional drafting keys separate drawings to a particular point on each sheet of paper. In Visual CADD you usually key all parts of your drawing to a single origin point (0,0), basepoint, or survey monument, enabling you to easily draw all parts of a drawing in the same file rather than on separate sheets. When drawing a building project, for example, you can draw the different floors and building systems, as well as sections and elevations, on different layers, and then display only the layers you want to work with or print.

Setting a scale

Once you've completed a drawing in Visual CADD, you can print it on paper or film, or save it in a print file. Whatever your final output, your drawing should be at a scale that legibly conveys the information you want to communicate. For example, you can set the scale to fit your drawing to the paper on which you're printing, planning the size of text and dimensions so that they are neither too small to read nor disproportionately large. You can add details to your drawing, each with a different scale if necessary, by borrowing portions of other (referenced) drawings using *reference frame entities*.

FOR MORE INFORMATION

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Using layers to simplify a complex drawing 20
Drawing an object 28–53
Printing your drawing 168
Adding reference frame entities to your drawing 172

Starting, saving, and closing a drawing

When you start Visual CADD, a new untitled drawing opens. Once Visual CADD is running, you can open another new drawing file or an existing one. You can open and view up to 64 drawing files at a time, depending upon the memory available on your computer.

Save a Visual CADD drawing before closing it as you do in any Windows application, or save it with another name to create a new drawing while preserving the original drawing file.

If you are sharing drawing files with other Visual CADD users on a network, you can lock Visual CADD drawing files so that only one person can make changes to them at a time, enabling you to control which drawing is the most recent version.

To...	Do this...
Start a new drawing	Click the File menu, and then click New.
Open an existing drawing	Click the File menu, click Open, find the filename, and then click Open.
Save a drawing	Click the File menu, and then click Save. If you are saving the drawing for the first time, type a name, and then click Save.
Save a drawing under another name	Click the File menu, click Save As, type a new name, and then click Save.
Lock a drawing for version control	Click the Utilities menu, click Settings, click the System tab, and then make sure Use File Locking is checked.
Close a drawing	Click the File menu, and then click Close. Visual CADD prompts you to save the drawing, if you haven't done so since the most recent edit.
End a Visual CADD work session	Click the File menu, and then click Exit. Visual CADD will prompt you to save the drawing, if you haven't done so since the most recent edit.

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Using Visual CADD to match your working style

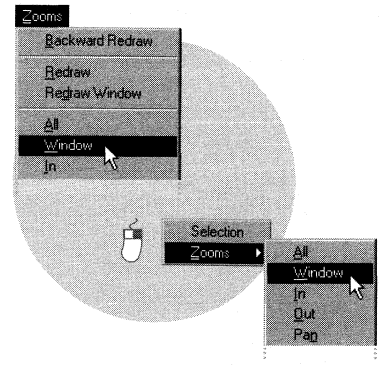
You can perform almost any task in Visual CADD in at least three different ways. If you are new to Visual CADD, you probably will use a mouse or digitizer puck to choose commands and tools, and to change many options on the speed bars.

As you become more familiar with Visual CADD, you might find the keyboard shortcuts faster for the commands you use regularly. You also can click the right mouse button to display a pop-up menu of commands related to the object you have selected.

Whether you choose a command from a menu, a tool from the toolbar, or enter a keyboard shortcut, the status bar displays it and provides a prompt for the next step of the task.

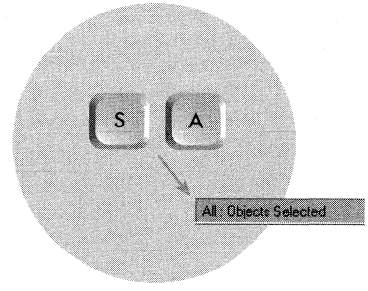
You can assign scripts to the function keys and other keys as well as mouse and digitizer buttons to run commands. You can customize the menus and toolbars with the commands, options, and tools you use most often, as well as create new ones.

WAYS OF WORKING IN VISUAL CADD

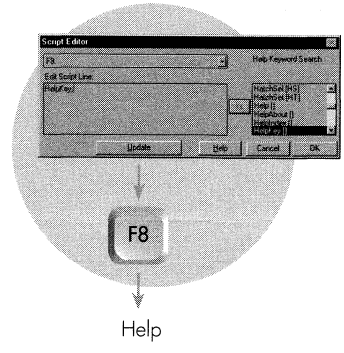


Use a mouse or digitizer puck to choose a command or tool. Or click the right mouse button to use context-sensitive commands.

Use a mouse or digitizer puck to click buttons and select options on the context-sensitive speed bars.



Enter keyboard shortcuts for commands, tools, or options. You don't need to press ENTER after the keyboard shortcut.



Assign scripts to keys and mouse and digitizer buttons to automate command sequences.

FOR MORE INFORMATION

Changing the coordinate system

In Visual CADD, construction points, which have a location, a distance, and a direction, define each entity that you draw. For instance, to draw a 2-point circle, you specify the circle's center and the point at a certain distance from the center (its radius).

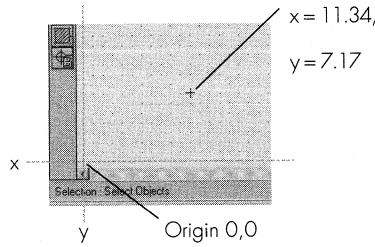
You place a point most accurately by manually entering its coordinates in the status bar, and then pressing ENTER. You can use one of three coordinate systems and switch between them as needed:

- Absolute coordinates
- Basepoint coordinates
- Relative coordinates

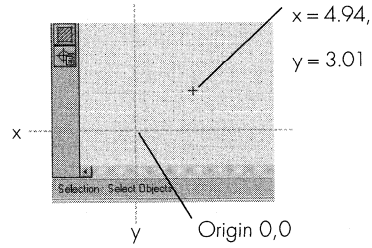
TIP

Change coordinate-system options quickly on high-resolution monitors by clicking the coordinate system section of the status bar until you see the option you want.

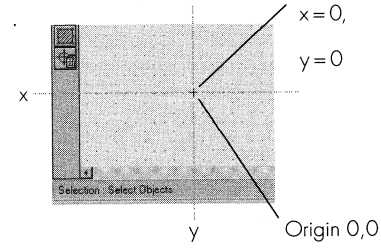
OVERVIEW OF THE THREE COORDINATE SYSTEMS



Absolute coordinates All distances are measured along the X and Y axes from a fixed origin, with coordinates 0,0 at the lower left corner of a new drawing file.

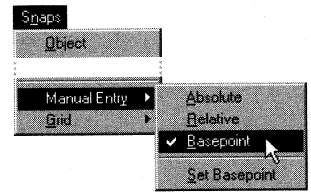


Basepoint coordinates Distances are measured along the X and Y axes from a temporary origin, set with the Set Basepoint command on the Manual Entry submenu of the Snaps menu. For example, when drawing a house far from the true origin, you might want to set a corner of the house as a basepoint.

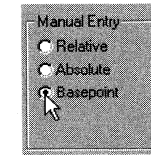


Relative coordinates Distances are measured along the X and Y axes from the last point entered. For example, when breaking a line a specific distance from its end, you might want to set the coordinate system to Relative to locate the break.

To change the coordinate system



Click the Snaps menu, click Manual Entry, and then click an option.



OR

Click the Utilities menu, click Settings, click the Constraint tab, select an option in the Manual Entry section, and then click OK.

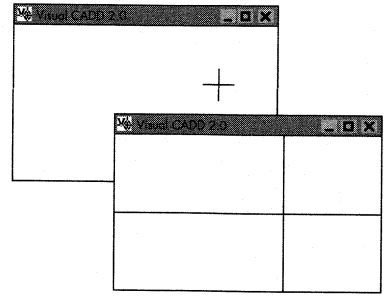
FOR MORE INFORMATION

Setting Visual CADD's defaults

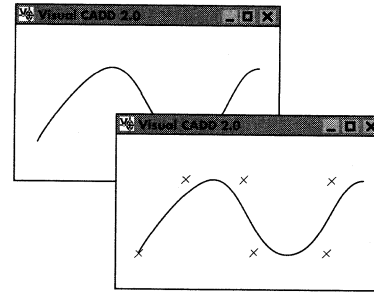
A new Visual CADD drawing opens with built-in settings, or defaults, that control the drawing's coordinate system, units of measure, tools, and drawing board elements. You can modify many of these to suit your work.

Initially, you might want to set only the appearance and behavior of Visual CADD's basic features. For example, you can choose a default tool, change the size and color of the cursor, change the color of the drawing background, and display or hide various drawing elements. You can do that on the Systems and General tabs of the Settings dialog box. As you gain more experience working with Visual CADD, you can customize it further.

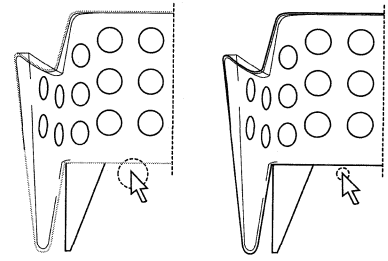
VISUAL CADD DEFAULTS



By default, the cursor size is set at one screen inch. You can set it to any dimension, including full-screen size.



By default, line types, line widths, fills, hatches, and points are all displayed. You can also set construction and handle points to be displayed. The more items you display, however, the longer it takes to redraw the displayed objects.



By default, the cursor must be within ¼" of an object when you click to select it. You can change the search tolerance to a larger or smaller distance.

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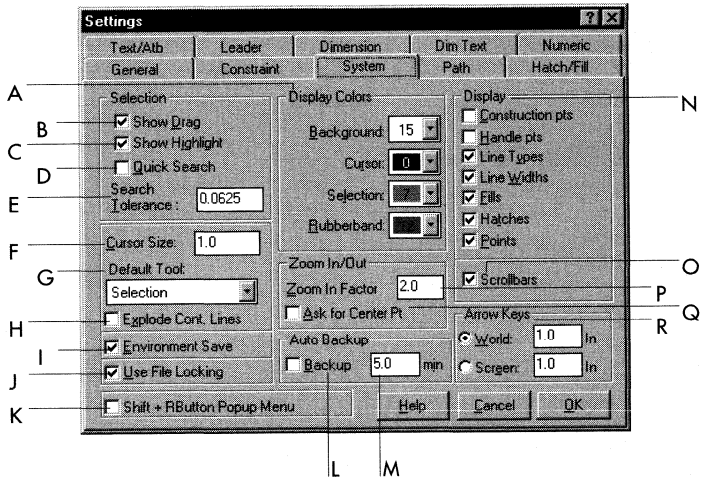
Customizing Visual CADD's settings 150

SYSTEM SETTINGS



Opens when you click the Utilities menu, click Settings, and then click the System tab.

A Sets the colors of the drawing environment without affecting output colors.

B Sets selected objects to visibly drag across the screen when you move or copy them.

C Sets selected objects that are checked under the Display heading to highlight with the selection color.

D Sets Visual CADD to select the earliest-drawn object that falls within the cursor's search tolerance.

E Sets the maximum distance, in inches, the cursor can be from an object for Visual CADD to snap to or select it.

F Sets the size, in inches, of the cursor's crosshairs. For a cursor that spans the screen, type **0**.

G Sets the active tool in new drawings and the tool to which Visual CADD reverts after completing most operations.

H Sets Visual CADD to draw continuous lines as a series of single lines rather than as a single object.

I Saves settings upon exiting Visual CADD for use in future work sessions.

J When checked, subsequently opened drawings cannot be modified by another Visual CADD user on your network until the drawing is saved or closed. Other users can only open, view, and copy the drawing.

K Changes activation of context-sensitive menus, enabling you to program your middle and right mouse buttons to perform other functions.

L Saves named drawings that have changed since the last save. Auto Backup saves drawings with a .vbk extension.

M Specifies the number of minutes between automatic backups. Backups occur only when no dialog boxes are open and no tools are active.

N When checked, sets these items to display, print, and plot. When unchecked, points, fills, and hatches do not display. Line types display, print, and plot as solid lines. Line widths display, print, and plot as line width 0 (1 pixel wide).

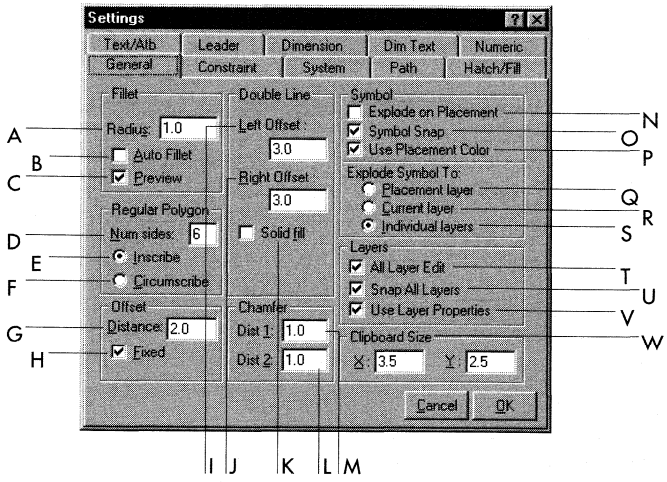
O When checked, turns on scrollbars.

P Sets the multiplier of the Zoom In command for changing view magnification. The multiplier for the Zoom Out command is the reciprocal of this number.

Q Displays a prompt during zoom operations for you to select a point on the screen to be the center of the zoomed view.

R Sets the distance that each arrow key advances the cursor from the location of the last click. If World is checked, the distance on the screen varies according to the zoom value. If Screen is checked, the distance remains constant no matter the zoom value. If Screen is checked and the grid is displayed, each arrow key advances the cursor to the next grid point regardless of the increment entered here.

GENERAL SETTINGS



Opens when you click the Utilities menu, click Settings, and then click the General tab.

A Sets the radius of fillets.

B Sets corners to fillet automatically as you draw with the double-line tool.

C Sets the possible fillet options to display as you move the cursor around an intersection or corner during a fillet operation.

D Sets the number of sides drawn with the center-polygon and side-polygon tools.

E Sets the center-polygon and side-polygon tools to place the second point as a vertex of the polygon, inscribed in a two-point circle.

F Sets the center-polygon and side-polygon tools to place the second point as the midpoint of a side of the polygon, circumscribed around a two-point circle.

G Sets the distance from the original object to the offset copy.

H Locks the offset copy to the distance set.

I Sets the offset distance of the double line to the left of the cursor.

J Sets the offset distance of the double line to the right of the cursor.

K Fills the space between double lines with a solid color.

L Sets the distance from the corner or intersection to the start of the chamfer on the second line selected.

M Sets the distance from the corner or intersection to the start of the chamfer on the first line selected.

N Sets Visual CADD to place symbols as the individual entities that compose them.

O When checked, enables snaps to target any point within a symbol. When unchecked, enables snaps to target only symbol handle points.

P When checked, each symbol placed uses the current color setting. When unchecked, each symbol placed uses the color setting assigned to each entity within the symbol.

Q Sets the entities of an exploded symbol to be located on the same layer as the symbol.

R Sets the entities of exploded symbols to be located on the layer that is current when you place each symbol.

S Sets the entities of exploded symbols to be located on the layers where the entities were located when they were saved as a symbol.

T When checked, enables objects on all visible layers to be edited. When unchecked, enables objects on the current layer only to be edited.

U Enables Visual CADD to snap to objects on all layers, rather than just the current layer.

V When checked, objects drawn on each layer take on the properties assigned to that layer.

W Sets the size of objects copied from Visual CADD to the Windows clipboard.

Entering numbers

In Visual CADD, you enter numbers to define coordinates, distances, angles, and sizes.

At any time, you can enter coordinates and distances in another measurement system by entering the abbreviation for the system after the number. If your current unit of measure is feet and inches, for example, you can enter meters by typing **m** after the number.

On the Numeric tab of the Settings dialog box, you can set the unit of measurement you use to enter and display numbers in the status bar and dimension strings.

Mathematical expressions

You can enter mathematical expressions to have Visual CADD calculate a value for you. Visual CADD evaluates mathematical expressions in the following order:

1. Numbers or subexpressions enclosed in parentheses
2. Trigonometric or exponential functions
3. Multiplication
4. Addition or subtraction

Note: Operations of equal priority are carried out from left to right.

Measurement abbreviations

Abbreviation	Units
"	Inches
'	Feet
' "	Feet and inches
mm	Millimeter
cm	Centimeter
m	Meter

Formats for entering numbers

Format	Distinction
3'6"	No space between the foot mark and inches value
3' 6"	Space between the foot mark and inches value
3'6	Inch mark omitted
3' 6 1/2"	Mixed feet/inches/fraction
3'6.5"	Mixed feet/decimal inches
3.5'	Decimal feet
3 1/2'	Fractional feet
3.5m	No space between number and abbreviation
3.5 m	Space between number and abbreviation

Note: Visual CADD interprets a hyphen separating feet and inches as a mathematical expression (for example, entering 5'-4" results in a measurement of 4' 8"). Similarly, Visual CADD interprets fractions not separated from inches by a space as a mathematical expression.

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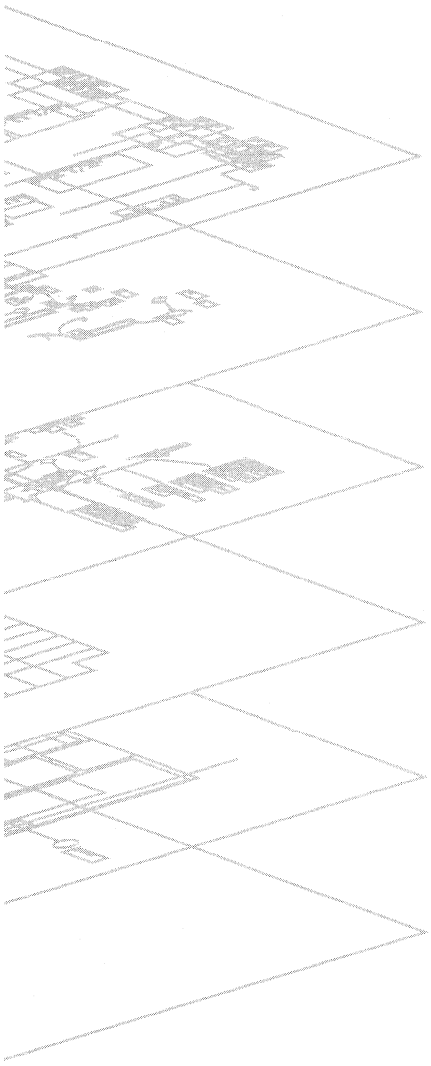
Operators for mathematical expressions

Operator	Function
+	Adds preceding value to following value
–	Subtracts following value from preceding value
*	Multiplies preceding and following values
/	Divides preceding value by following value
$SIN(A)	Sine of A
$COS(A)	Cosine of A
$TAN(A)	Tangent of A
$ATN(A)	Arc or inverse tangent of A
$LOG(A)	Log (base 10) of A
$NLN(A)	Natural log (base E) of A
(A)$EXP(B)	Exponent (A to the B power)
$SQR(A)	Square of A (A to the 2nd power)
$SQT(A)	Square root of A
$ABS(A)	Absolute value of A

Mathematical expressions

Format	Result
2*3	6
2*3-1	5
6*3-1/4	17.75
6*(3-1/4)	16.5
23' 6 1/2" +9' 4 1/2"*2	42.29166667' or 42' 3 ½"

Organizing your drawing



On complex projects, conventional drafting benefits from the use of overlay drafting systems. Different building systems can be separated easily by combining the overlay sheets, say, for the building's structure and then printing them. To prepare a cohesive set of drawings, complex projects usually require a large team of drafters, which must use the same drawing conventions—from line weights to symbols and abbreviations.

The more complex your drawing, the more you benefit by using Visual CADD's organizing tools: layers and styles. Layers and styles speed the editing, redrawing, and printing of even the most complex drawings.

Create portions of a drawing on different layers to separate the drawing into logical groups. Use layers to display or hide information and to print groups of information separately. Use styles to maintain a consistent format for different types of objects. Styles enable you to change drawing settings quickly and to maintain drawing conventions.

When you develop a scheme for setting up layers and styles before beginning a drawing, you build a foundation for organized, efficient work.

CHAPTER

2

Setting up layers 20–25

Layers organize a drawing into different portions, as though it were on transparent overlays. Using layers, you can view, hide, select, move, and print groups of entities, and assign properties to them based on the layer you draw them on.

Using styles 26–27

Styles help maintain consistency of format by enabling an entire project team to easily apply the same settings to all drawings. You can create styles based on drawing, dimension, text, hatch, symbol, selection filter, and file exchange settings, as well as Visual CADD's drawing environment.

Using layers to simplify a complex drawing

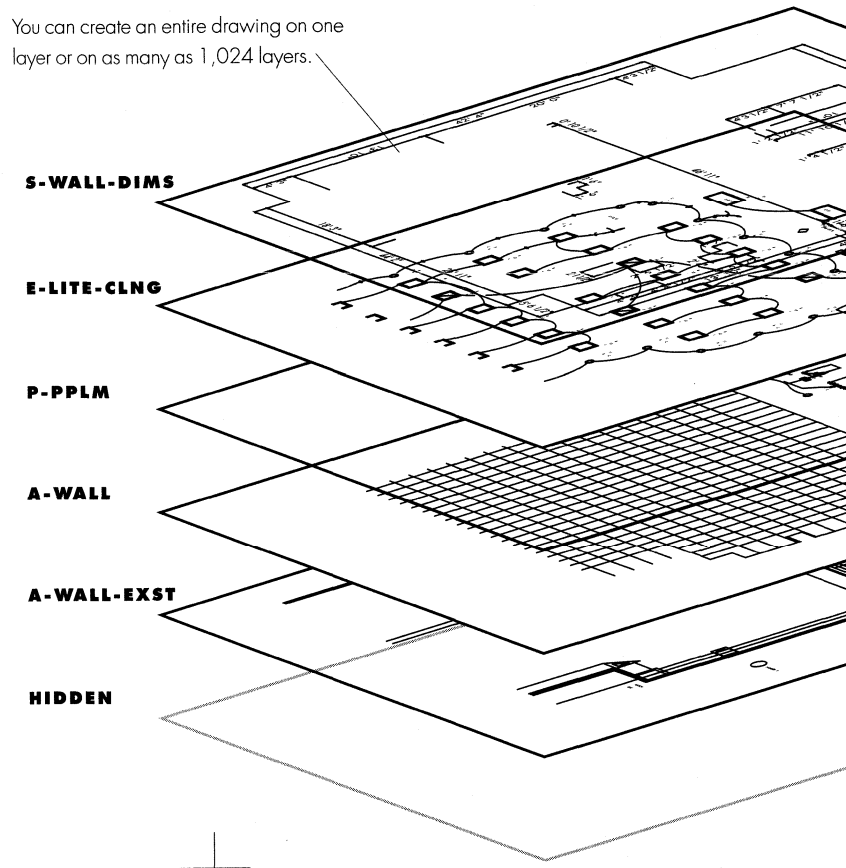
As with conventional drafting's transparent overlays, you can use layers to organize your drawing into distinct categories that you can manage easily as your drawing increases in complexity. For example, you might want to organize the floor plans for a renovation into the following layers: existing walls, new structure, existing doors, new doors, electrical system, plumbing, mechanical system, and so on. You could then work with each group in turn, viewing, drawing, editing, and printing it with any combination of other layers.

Most layer functions are available from Layer Manager. You can work with the default set of numbered layers and name them. You can also save a layer or set of layers as a new drawing to use a portion of the original drawing as the basis for a new project. In addition, in the Settings dialog box you can set all dimensions and text to reside on any layer, no matter which layer is set as the current drawing layer.

You can draw an object on any layer and with any properties you choose, or you can designate any layer to apply certain properties automatically when you draw objects on that layer.

LAYERS IN A DRAWING

Using layers, you can easily view, select, move, and print groups of objects, and quickly assign properties to them based on the layer you draw them on.



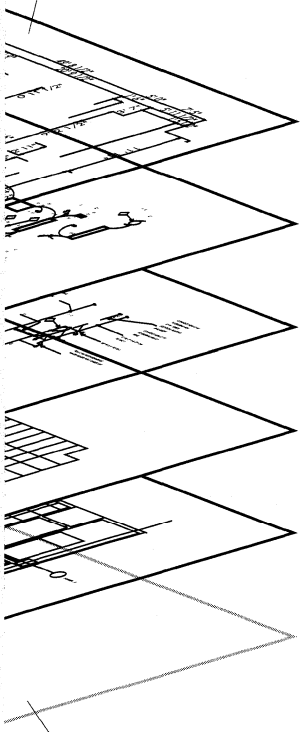
FOR MORE INFORMATION

TIPS

Shorten the time in which your drawing redraws and prints by hiding the layers on which you're not working.

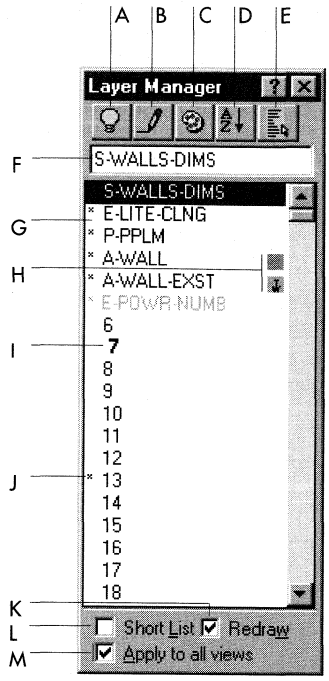
For quick access to basic layer functions without using Layer Manager, click the Properties button on the speed bar.

Although you can move objects from one layer to another, objects on the top layer are not actually in front of objects on other layers—they won't hide or be hidden by objects on other layers.



You can hide any or all layers except the current drawing layer. Hidden layers won't print.

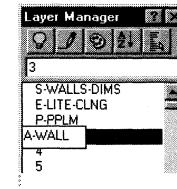
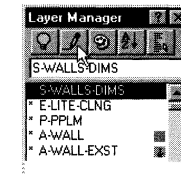
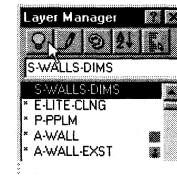
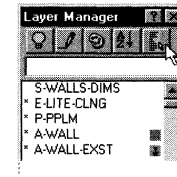
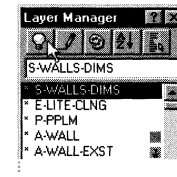
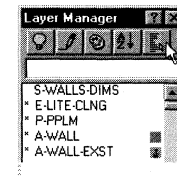
LAYER MANAGER OPTIONS



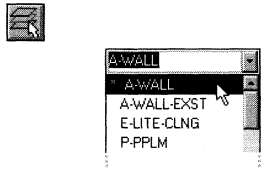
Opens when you click the Utilities menu and then click Layer Manager.

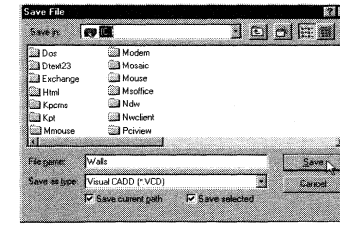
- A Displays or hides all selected layers.
- B Makes the selected layer the active drawing layer.
- C Opens the Layer properties speed bar, on which you can assign properties to the selected layer(s).
- D Sorts the list of layers alphabetically by name, or restores the list of layers to its original numerical order.
- E Selects all layers listed.
- F Text box for entering the layer(s) you want to select. A hyphen separating two numbers designates a range of layers. A comma separates noncontiguous layers.
- G Lists all available layers. Hidden layers are dimmed.
- H Indicates a layer that has been assigned layer properties. If so, the assigned color is shown.
- I Designates the current drawing layer (in boldface).
- J Indicates a layer containing at least one object.
- K When checked, redraws objects on newly hidden or displayed layers as soon as you hide or display them. When unchecked, objects redraw after you close Layer Manager.
- L When checked, lists only layers containing an object and named layers.
- M When checked, Layer Manager settings apply to all windows. When unchecked, Layer Manager settings apply only to the active window.

Note: Double-clicking the Layer Manager title bar rolls it up and down.

To...	Do this in Layer Manager...	
Name a layer	Select a layer, press ENTER, type a name, and then press ENTER.	
Select a layer on which to draw	Select a layer, and then click the Current button, or double-click the layer name.	
Display a layer	Select a hidden layer, and then click the Display button.	
Display all layers	Click the Select All button, and then click the Display button.	
Hide a layer	Select a displayed layer, and then click the Hide button.	
Hide all but the current layer	Click the Select All button, and then click the Hide button.	

To save a layer as a new drawing



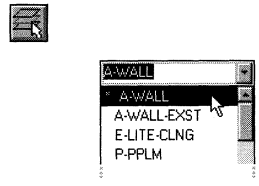


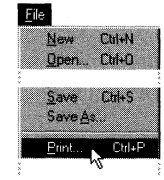
1 Type S1, click the Select Layer button, select a layer from the drop-down list on the speed bar, and then click OK.

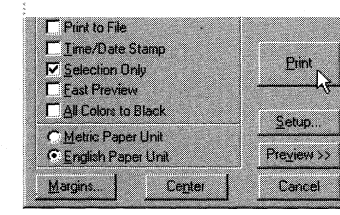
2 Click the File menu, and then click Save As.

3 Click the Save Selected checkbox, enter a new filename, and then click Save.

To print all objects on a layer





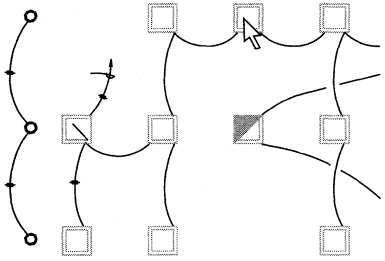


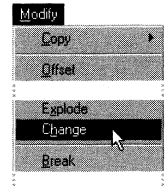
1 Type S1, click the Select Layer button, select a layer from the drop-down list on the speed bar, and click OK.

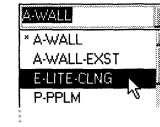
2 Click the File menu, and then click Print.

3 In the Print dialog box, click Selection Only, and then click Print.

To move an object or objects to another layer





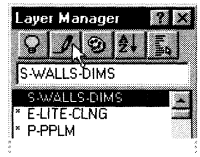


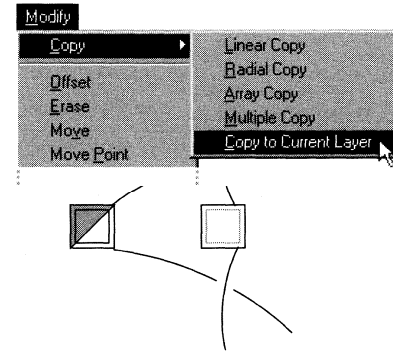
1 Select an object or objects.

2 Click the Modify menu, and then click Change.

3 Select a layer from the Layers list, and then click OK.

To copy an object to another layer





1 In Layer Manager, click the layer to which you want to copy the object, and then click the Current Layer button.

2 Select the object you want to copy, click the Modify menu, click Copy, and then click Copy to Current Layer.

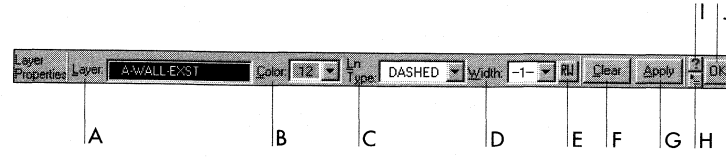
Assigning properties to a layer

You can assign properties to a layer so that all objects you subsequently draw on that layer have those properties in common. Layer properties can speed your work; to change the properties of the object you want to draw, you need only change the current layer.

When you use layer properties, any objects you draw take on the color, line type, and line width properties assigned to the layer they are drawn on. If you assign, for example, only a color to a layer, objects you draw on the layer take on the layer's assigned color property and Visual CADD's current line type and line width.

After you click the Utilities menu, click Settings, click the System tab, and then check Use Layer properties, layer properties apply to any objects you draw on layers to which properties are assigned. If you later uncheck the Use Layer Properties option, each object takes on the properties that were current at the time it was drawn.

LAYER PROPERTIES SPEED BAR



Opens when you click the Layer Properties button in Layer Manager.

- A Designates the layer (or first in a series of layers) for which you want to set properties.
- B Assigns the color property to the designated layer(s).
- C Assigns the line type to the designated layer(s).
- D Assigns the line width to the designated layer(s).
- E Toggles line width options between predefined and real-world line widths.
- F Removes all layer properties assigned to the designated layers.
- G Applies the settings to the designated layers without closing the layer properties speed bar.
- H When clicked, sets the layer properties to match those of the next object you click.
- I Opens online Help.
- J Applies the selected properties options to the designated layers and closes the layer properties speed bar.

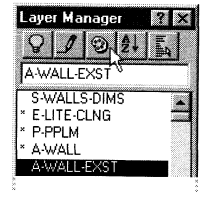
TIPS

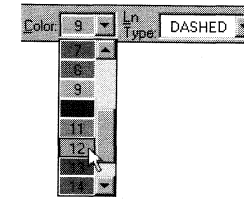
To assign layer properties to objects drawn on a layer before properties were assigned to it, select the objects, click the Modify menu, click Change, and then set the objects' properties to "LP."

If you want to change properties of an object that has been assigned layer properties, click the Modify menu, click Change, and then select different options from the properties drop-down lists.

FOR MORE INFORMATION

To assign properties to a layer





1 In Layer Manager, select a layer (or hold down Shift and select a series of layers), and then click the Properties button.

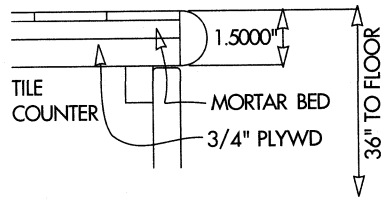
2 On the Layer Properties speed bar, select options from the drop-down lists to set the layer properties, and then click Apply.

Using styles to maintain standards

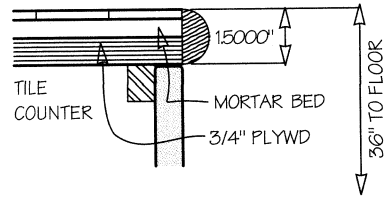
A style is a collection of settings you define, name, and save. When you save a group of settings as a style and then apply the style to another drawing, you change all of the drawing's settings in one step, saving time and maintaining consistency.

In addition to creating styles for the properties of objects, you can also create styles that save Visual CADD's drawing environment, selection filters, and file-exchange settings.

WHY USE STYLES?



ORIGINAL



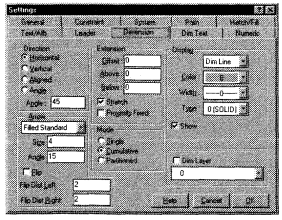
NEW STYLE LOADED

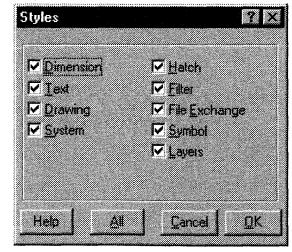
Use styles to maintain project or company drawing standards. Using the same styles, all members of a project team can create drawings that use the same line widths, colors, hatches and fills, text and dimensioning conventions, and layering system.

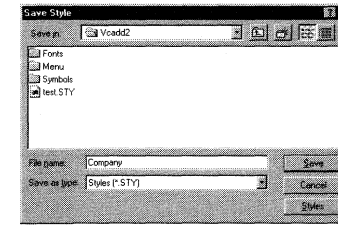
FOR MORE INFORMATION

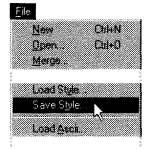
Setting Visual CADD's defaults 13
Working with layers 22
Setting line properties 32
Selecting objects by properties and type 60
AutoCAD and Generic CADD translation settings 162

To save a style







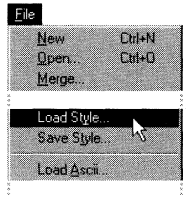


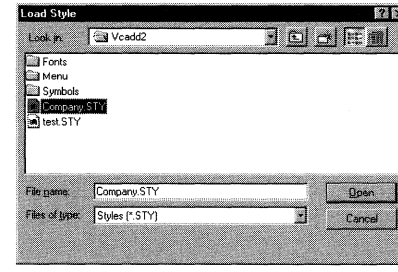
1 Specify the settings to save, click the File menu, and then click Save Style.

2 Click Styles, check the style categories that you want included in the style file or click All to select all categories, and then click OK.

3 Enter the name of the style file, and then click Save.

To load a style





1 Click the File menu, and then click Load Style.

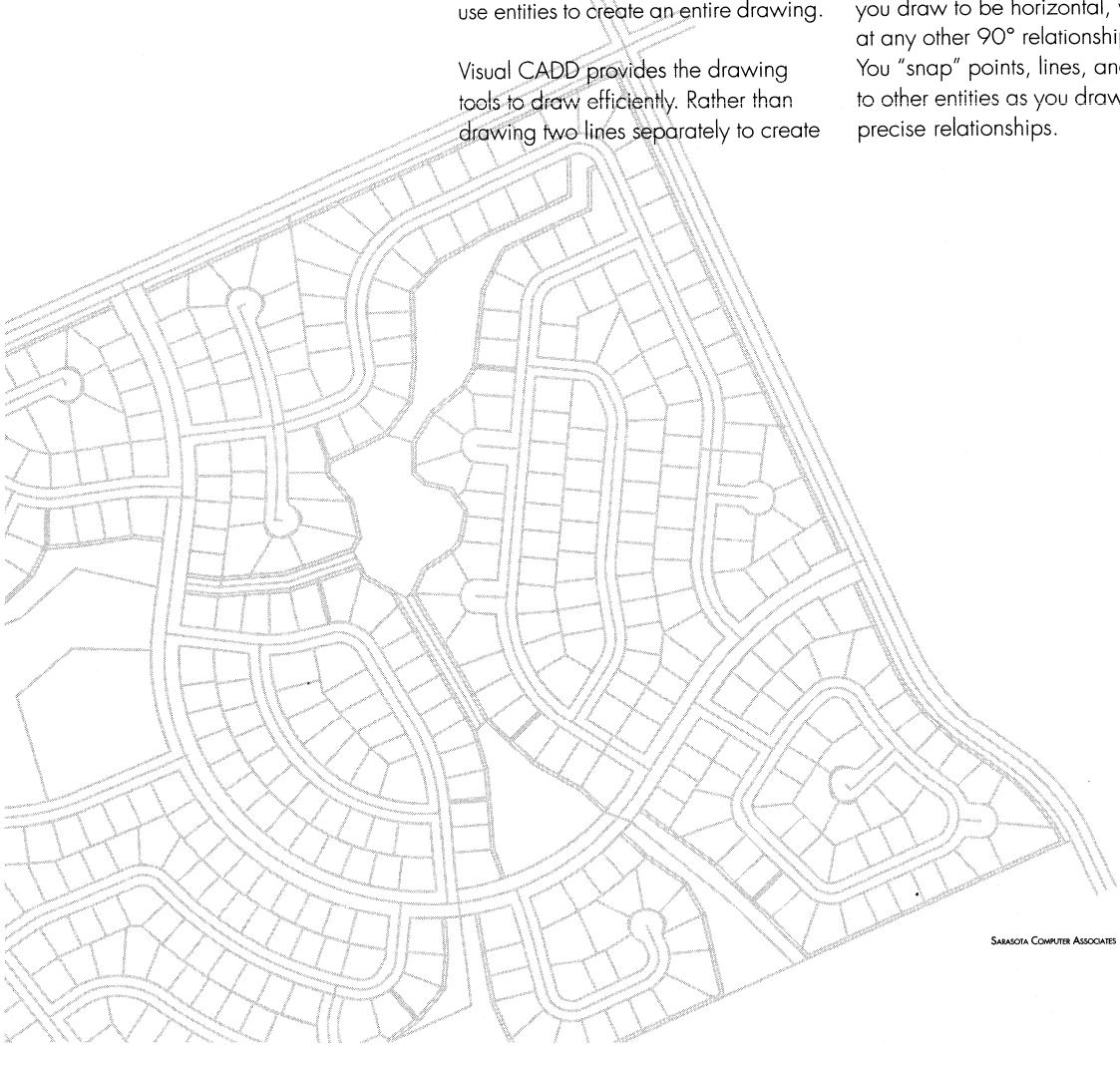
2 Select a style file, and then click Open.

Drawing an object

In conventional drafting, you lay down lines by drawing each stroke exactly how you want the finished line to look. In CAD, you draw major lines (for example, the perimeter of a building), and then delete parts that are unneeded, for example, to make openings for windows and doors.

In CAD, you draw lines and objects by placing the points that define a drawing entity, such as a line, a circle, and a polygon. Each point contains X and Y coordinates. You can duplicate and re-use entities to create an entire drawing.

Visual CADD provides the drawing tools to draw efficiently. Rather than drawing two lines separately to create a wall, for example, you use the double-line tool to lay down two lines at a specified distance from each other. In place of T-squares and triangles, you use ortho mode to constrain the lines you draw to be horizontal, vertical, or at any other 90° relationship to a grid. You "snap" points, lines, and objects to other entities as you draw to create precise relationships.



Sarasota Computer Associates

CHAPTER

3

Introduction to the drawing tools

To draw objects in Visual CADD, choose a drawing tool, and then place the points that define the object by either clicking its position or keying in the points' coordinates.

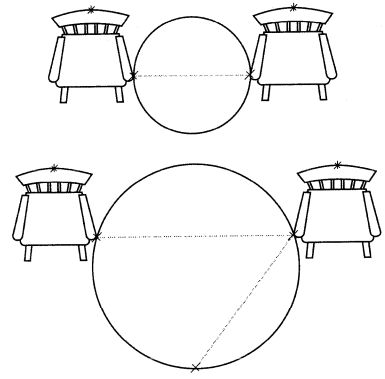
Some tools create a series of adjoining line segments or curves. Others create a closed object, such as a polygon. Most tools create objects that maintain their shape when you move or modify them. You can also choose to "explode" such an object—to select its component line segments and move or modify each individually.

The continuous tools

Four tools—continuous line, double line, continuous Bézier, and spline curve—are continuous; that is, they continue to be active until you either double-click, press Esc, or click the right mouse button, and then click Pen Up or Close Contour.

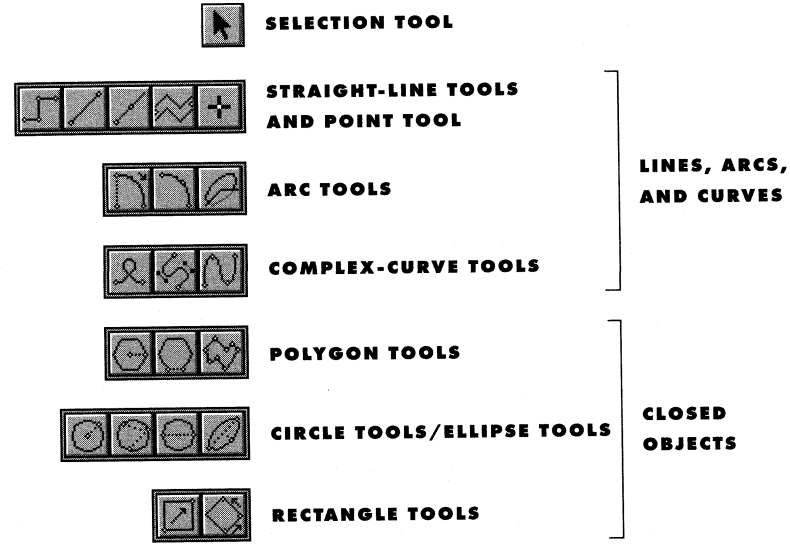
For more information on the individual tools, see pages 32–43.

CHOOSING THE APPROPRIATE TOOL FOR THE TASK



In many cases, several tools are available for defining the object you want to draw. For example, if you want to draw a circle between and adjoining two objects, you can choose the diameter-circle tool, which places the full diameter of the circle between the two objects; or, you can choose the three-point-circle tool to place a point on one object, another on the second object, and the third point elsewhere to define the circle's size.

THE DRAWING TOOLS

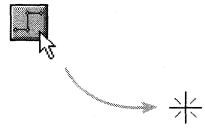


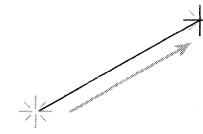
TIPS

Find related tools by clicking a tool button with the right mouse button until you see the tool you want.

Reselect a tool immediately after using it by pressing the Spacebar.

To draw by placing points

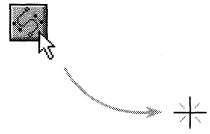


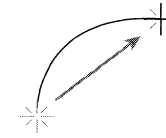


1 Select a drawing tool, and then click at the point you want to start, or type X and Y coordinates and then press ENTER to locate the first point.

2 Click where you want to locate additional points, or enter their X and Y coordinates.

To draw by using direct-distance entry

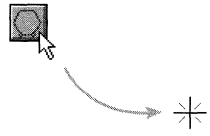


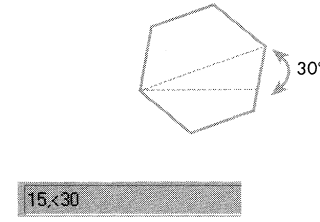


1 Select a drawing tool, and then click the mouse, or type X and Y coordinates and press ENTER to locate the first point.

2 Move the mouse in the general direction of the second point, type a distance, and then press ENTER.

To draw by entering polar coordinates





1 Select a drawing tool, and then click the mouse, or type X and Y coordinates and press ENTER to locate the first point.

2 Type a distance and an angle, separated by a comma and the symbol for angle, and then press ENTER.

Setting line properties

For any line, you can define four properties.

Line type Choose from 42 line types, in addition to those you modify or create yourself.

Line width 16 predefined line widths are available, as well as any real-world line width you enter.

Layer Layers help you organize your drawing but do not change its appearance unless you hide a layer. You can assign entities with varying properties to the same layer, or you can assign properties to a layer so that all objects you draw on that layer have the same properties.

Color You can assign any one of 256 colors to an entity depending on your monitor and video card. Note that color prints correctly only on a plotter equipped with the designated color ink, using pen mapping. You can also choose to print all colors as black.

You can preset the properties for a drawing so that they apply to all subsequently drawn objects, and you can change them at any time. Until you change them, the properties of the last-drawn entity apply to the next entity you draw.

FOR MORE INFORMATION

Using layers to simplify a complex drawing 20
Assigning properties to a layer 25
Introduction to the drawing tools 30
Changing the properties of objects 81
Creating new line types 148
Using direct plotting 176

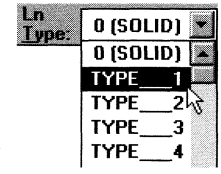
LINE TYPES

THE LINE WIDTHS

Number	Inches	Metric	On-screen	Sample
0	.0078	.2	1 pixel	
1	.0156	.4	1 pixel	
2	.0234	.6	2 pixels	
3	.0312	.8	3 pixels	
4	.0390	1.0	4 pixels	
5	.0468	1.2	5 pixels	
6	.0546	1.4	6 pixels	
7	.0624	1.6	7 pixels	
8	.0702	1.8	8 pixels	
9	.0780	2.0	9 pixels	
10	.0858	2.2	10 pixels	
11	.0936	2.4	11 pixels	
12	.1014	2.6	12 pixels	
13	.1092	2.8	13 pixels	
14	.1170	3.0	14 pixels	
15	.1248	3.2	15 pixels	

Real-world line widths In addition to Visual CADD's 16 predefined line widths, you can set a line to any width by clicking the properties button on the main speed bar, clicking the RW button, and then typing a line width in the Line Width text box. Real-world line widths print and display at the width you enter, regardless of the scale.

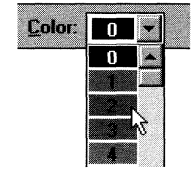
To set the properties of a line before drawing



1 Click a drawing tool, and then click the properties button on the speed bar.

2 Click options from the drop-down lists.

OR



1 Click a tool, and then enter the two-letter command for the individual property you want to set.

2 Click an option from the drop-down list.

Drawing with the straight-line tools

Visual CADD offers four tools for drawing straight lines:

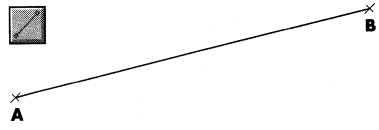
- Single-line tool
- Midline tool
- Continuous-line tool
- Double-line tool

TIPS

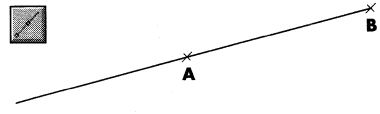
When you complete an object drawn with a continuous tool, such as the continuous-line tool, click the right mouse button, and then click Pen Up.

Delete the last point placed when drawing a continuous line or a double line by clicking the right mouse button, and then click Undo Vertex.

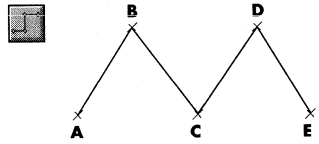
WHEN TO USE THE LINE TOOLS



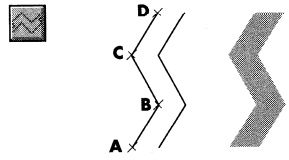
The single-line tool creates a line between two endpoints **A** and **B**.



The midline tool places the midpoint **A** and one endpoint **B**. Use this tool when you need to draw the endpoints of a single line segment equidistant from a specific point.



The continuous-line tool places the endpoints **A**, **B**, **C**, **D**, and **E** of adjacent line segments. Use this tool to draw a continuous, segmented line. When you select a segment of a continuous line, you select the entire line, unless you set Visual CADD to explode continuous lines on placement.

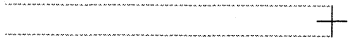


The double-line tool, a continuous tool, places the endpoints **A**, **B**, **C**, and **D** of the adjacent line segments of two parallel lines. Use this tool to draw walls and roads. You can set the *offset* or size of the space between the lines, and you can add a solid fill to the space between the lines.

FOR MORE INFORMATION

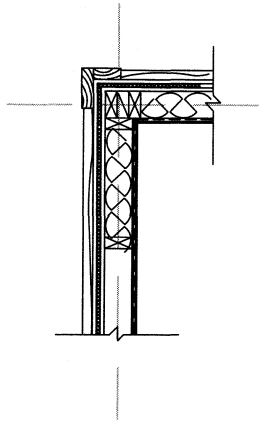
SETTING THE OFFSET OF DOUBLE LINES

Line offset: Left = 0 Right = 5



Line offset: Left = 5 Right = 5

The line offset specifies the distance from the cursor to each line drawn with the double-line tool. An offset of 0 locates the line on the cursor. When left and right offset values are equal, the cursor lies equidistant between the two lines. Left and right offsets are defined relative to the cursor's forward movement.



Using the line offset Change offset values when drawing different materials. For example, when drawing the different materials in a wall, you might set both left and right offset with equal values, align the center line of one wall with a structural gridline, change the right offset value to 0 and the left offset value to the thickness of the drywall, and then draw the finish material at its precise thickness.

To change the offset of a double line and apply a solid fill

1 Click the double-line tool.

2 Click the right mouse button, and then click DB Line Settings.

3 On the speed bar, enter values in the Left and Right Line Offset boxes, click the Solid Fill box to check it, and then click OK.

The rectangle tools

Visual CADD offers two tools for drawing rectangles:

- 2-point-rectangle tool
- 3-point-rectangle tool

Each rectangle tool draws a single entity that also is a closed, continuous line.

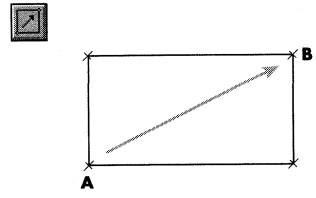
TIPS

When you know the length and angle of the first side of a rectangle, draw it using polar-coordinate entry set to relative mode, with ortho mode off.

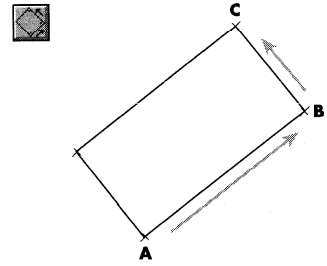
For an efficient method of creating a two-point rectangle, use manual entry set to relative mode, with X and Y coordinates representing the rectangle's width and height, respectively.

Place the second and third points of a three-point rectangle most efficiently using direct-distance entry or a snap command.

USING THE RECTANGLE TOOLS



The 2-point-rectangle tool places two points to establish the rectangle's opposite corners **A** and **B**. Use this tool to create horizontal and vertical rectangles.



The 3-point-rectangle tool places the two endpoints of the rectangle's sides **A** and **B**, establishing the length and angle, and then places the third point **C** to establish the rectangle's height. Use this tool to create rectangles at arbitrary angles and orientations.

FOR MORE INFORMATION

The polygon tools

Visual CADD offers three tools for drawing polygons:

- center-polygon tool
- side-polygon tool
- irregular-polygon tool

Each polygon tool draws a single entity that also is a closed, continuous line.

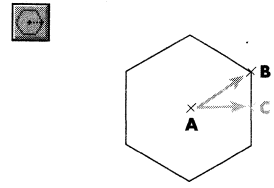
Both the center- and side-polygon tools can draw polygons with up to 99 sides. Set the number of sides after selecting the tool by clicking the right mouse button, and then clicking Number of Sides.

The center-polygon tool draws polygons either inscribed or circumscribed on a circle. To set an inscribed polygon, click the center-polygon tool, click the right mouse button, and then click Inscribed. To set a circumscribed polygon, click the center-polygon tool, click the right mouse button, and then make sure Inscribed is unchecked.

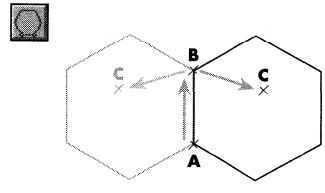
TIP

Draw a square with the center or side polygon tool by clicking the right mouse button, clicking Number of Sides, and then clicking 4.

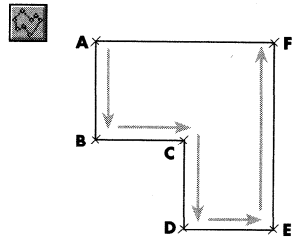
USING THE POLYGON TOOLS



The center-polygon tool places the center point **A** and, if inscribed, one vertex point on the polygon's perimeter **B**, or if not inscribed, one midpoint of a side of the polygon **C**. Use this tool to snap a vertex or midpoint of a side to an existing point.



The side-polygon tool places two adjacent vertices on the polygon's perimeter **A** and **B**, defining one of its sides, and then establishes the polygon's orientation **C**. Use this tool to snap the side of a polygon to an existing line.



The irregular-polygon tool, a continuous tool, places points that create an irregular shape with three or more sides of any length or angle. Use this tool to draw an irregular, closed object.

FOR MORE INFORMATION

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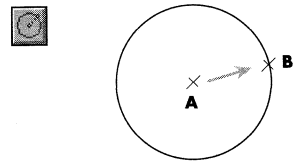
The circle and ellipse tools

Visual CADD offers four tools for drawing circles and ellipses:

- 2-point-circle tool
- 3-point-circle tool
- diameter-circle tool
- ellipse tool

Each of these tools draws a single entity that also is a closed object.

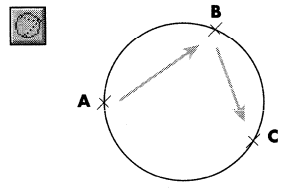
USING THE CIRCLE TOOLS



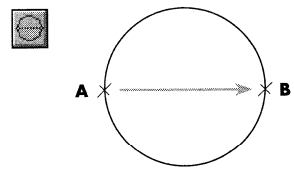
The 2-point circle tool places the centerpoint **A** and one point on the circle's circumference **B**. Use this tool when you know the location of the circle's centerpoint and any point on its circumference.

TIP

Have Visual CADD calculate the distance for you when placing a circle's second point with the 2-point or diameter circle tools by entering a mathematical expression for the radius (diameter/2) or the diameter (radius*2) in a direct-distance entry.



The 3-point-circle tool places three points on the circle's circumference **A**, **B**, and **C**. Use this tool when you don't know the center or radius, but you know the location of two or three points through which the circle's circumference must pass.



The diameter-circle tool places points opposite one another on the circle's circumference **A** and **B**. Use this tool when you need a circle to exactly fill the space between two objects.

FOR MORE INFORMATION

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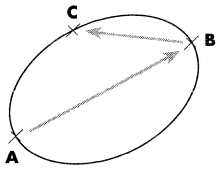
USING THE ELLIPSE TOOL



The ellipse tool places the endpoints **A** and **B** of the ellipse's major axis, and then a point at the end of the ellipse's minor axis **C**.

The arc and complex-curve tools

Visual CADD offers three tools for drawing arcs:

- radius-arc tool
- 3-point-arc tool
- elliptical-arc tool

Three additional tools draw complex curves:

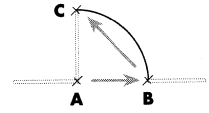
- single-Bézier-curve tool
- continuous-Bézier-curve tool
- spline-curve tool

TIPS

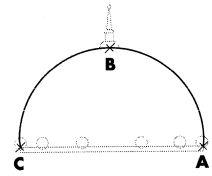
Delete the last point placed when drawing a continuous Bézier curve or spline curve by clicking the right mouse button and then clicking Undo Vertex.

When you complete an object drawn with a continuous tool, such as the continuous-line tool, click the right mouse button, and then click Pen Up.

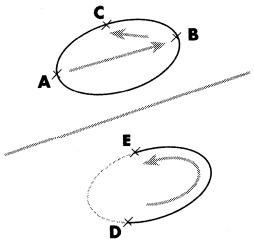
USING THE ARC TOOLS



The radius-arc tool places the centerpoint **A** of the circle describing the arc, the arc's starting point **B**, and then the arc's endpoint **C**. Use this tool to draw an arc with a precise centerpoint or radius, such as a door swing.



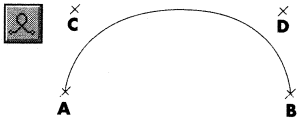
The 3-point-arc tool places the arc's starting point **A**, an intermediate point **B**, and the arc's endpoint **C**. Use this tool for shaping an arc as you draw it.



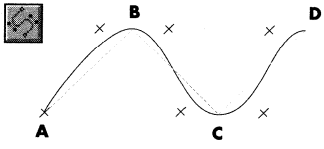
The elliptical-arc tool first establishes the ellipse that describes the arc, placing the ellipse's first and second endpoints **A** and **B** on the major axis of an ellipse, and a point **C** at the end of the ellipse's minor axis. It then places the arc's starting point **D** and finally its endpoint **E**. Use this tool to draw an arc that follows the contours of an ellipse.

FOR MORE INFORMATION

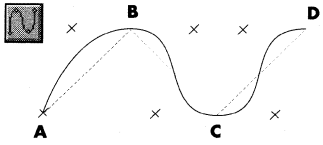
USING THE COMPLEX CURVE TOOLS



The single-Bézier-curve tool places the endpoints **A** and **B** of the curve, and then the control points **C** and **D** that define the slope of the curve. Use this tool to draw a single Bézier curve.



The continuous-Bézier-curve tool places the starting point **A** and subsequent tangent points **B**, **C**, and **D** of adjoining Bézier curves. When you select any portion of a continuous Bézier curve, you select the entire line. Use this tool for drawing steep curves such as highway acceleration curves.



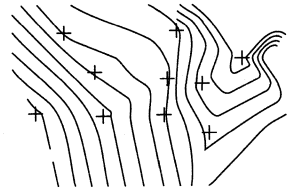
The spline-curve tool, a continuous tool, places the starting point **A** and subsequent tangent points **B**, **C**, and **D** of adjoining spline curves. When you select any portion of a spline curve, you select the entire line. Use this tool for drawing the broad curves common to aerodynamics, automotive, and marine design.

Point basics

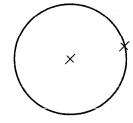
In Visual CADD, you place points to draw an entity. An entity's first and last points are its endpoints. On open objects, endpoints don't touch. On closed objects, the endpoints touch.

You turn the display of each point type on and off individually on the System tab of the Settings dialog box. To open the Settings dialog box, click the Utilities menu, and then click Settings.

COMPARING THE FOUR TYPES OF POINTS

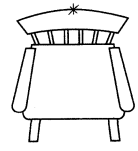


Standard points are the points you place with the point tool, marking positions that are not part of any objects. You can snap to standard points while drawing objects. Unlike the other points, standard points are entities to which you can assign properties.



Construction points are the points you place to draw objects or to move when editing most objects. The endpoints of lines and arcs and the centerpoints of circles, arcs, and polygons are construction points.

Control points, a special kind of construction point, are the points you place after placing the endpoints of Bézier and spline curves. Defining a tangent to and lying off of the curve, control points alter the curve's size and shape.



Handle points are the points you drag to locate entities. They also are the points you snap to on symbols, text blocks, and dimensions.

FOR MORE INFORMATION

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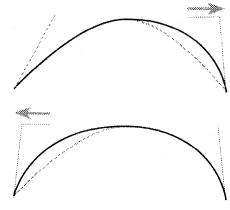
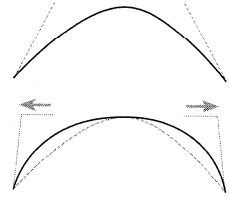
Using control points to change a curve's shape

Moving, extending, or retracting control points alters the curve of the adjacent line segments. With construction points checked on the System tab of the Settings dialog box, you can see the control points on a selected Bézier or spline curve.

HOW CONTROL POINTS SHAPE A CURVE



A control point controls the portion of the curve on the same side of the construction point.

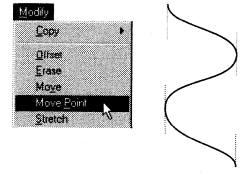


As you drag a control point farther from its corresponding construction point, the curve becomes broader.

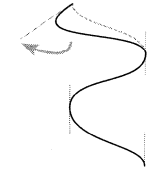
TIP

To turn off the display of control points so that you don't accidentally snap to or alter the shape of a curve, type **DV**, click the Zooms menu, and then click Redraw. Type **DV** again to display control points.

To alter a complex curve by moving a control point



1 Click the Modify menu, click Move Point, and then click the construction point on the curve you want to reshape.



2 Drag the control point to its new position.

FOR MORE INFORMATION

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Drawing efficiently

Visual CADD replaces parallel rules, T-squares, triangles, and other conventional drafting equipment with ortho mode, snaps, and grids.

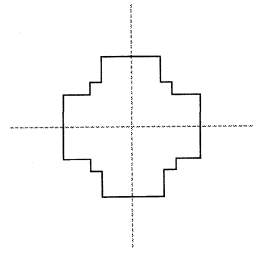
Ortho mode constrains all entities that you draw to 90° increments of the specified ortho angle, ensuring square corners and true horizontals and verticals.

When you use snaps, you don't need to know the precise location of a point you want to place. Instead, you define the relationship of the point to a point on an existing object. The new point snaps to the existing point.

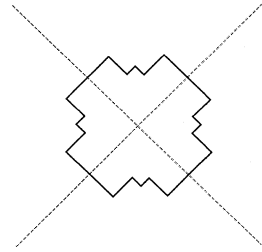
When you draw on a grid, you can snap points to the intersection of gridlines, ensuring that you place the points at regular intervals.

You can change ortho mode and grid settings on the Constraint tab of the Settings dialog box, but adjusting the settings in the speedbar and status bar as you work is often easier.

THE TOOLS FOR DRAWING EFFICIENTLY

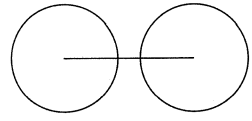


Ortho angle: 0.0

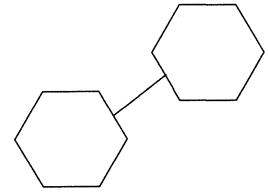


Ortho angle: 45.0

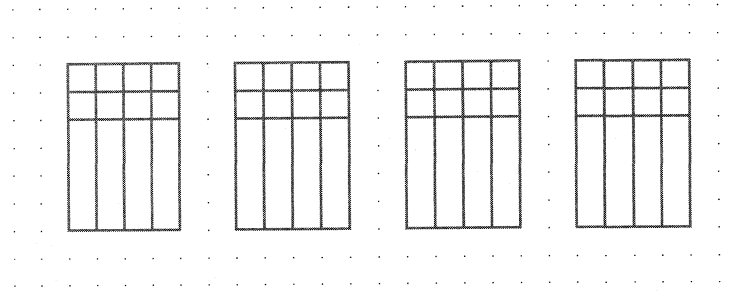
ORTHO MODE



SNAP CENTER



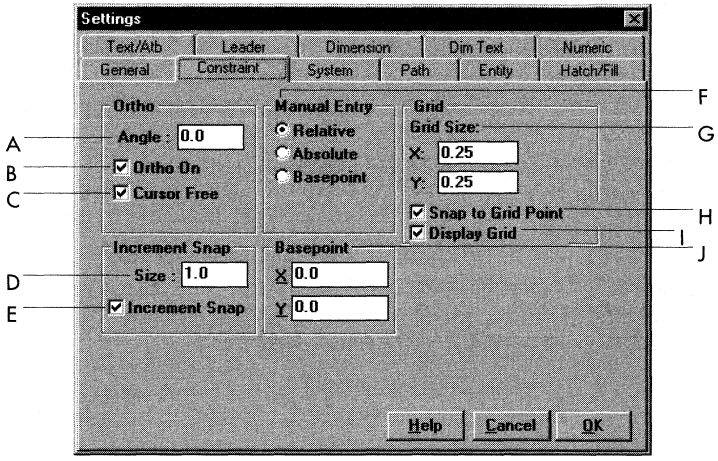
SNAP MIDPOINT



SNAP GRID

FOR MORE INFORMATION

CONSTRAINT SETTINGS

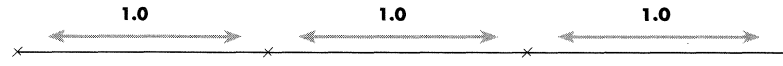


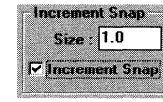
Opens when you click the Utilities menu, click Settings, and then click the Constraint tab.

- A Sets the base angle for drawing when ortho mode is on. All entities are placed in 90° increments of the specified angle.
- B Turns ortho mode on.
- C Frees the cursor to move anywhere on screen when ortho mode is on.
- D Sets the length of each increment snap.
- E Sets the cursor to move only in multiples of a predefined distance when ortho mode is on.
- F Sets the coordinate entry mode.
- G Sets the spacing between grid points.
- H When checked, constrains points to grid points.
- I Displays the grid.
- J Sets the basepoint coordinates for a temporary origin in the current drawing.

To draw in multiples of a specific length

With a tool selected and ortho mode on, click the Snaps menu, click Incr Snap, and then continue drawing.





Constraining lines and objects

In ortho mode, you constrain all points that you place to a perfect 90° relationship with each other. You can turn ortho mode on for one point, a series of points, or all points in your drawing.

Even when ortho mode is on, you can unlock the cursor from its constraint by making sure that when you click the right mouse button, Cursor Free is checked. With Cursor Free on, you can position the cursor over or snap to any point in the drawing, and Visual CADD places the closest point that is perpendicular to the cursor location on the constrained axis.

You can also set the ortho angle, which changes ortho mode's base orientation to the angle of the X axis relative to 0°.

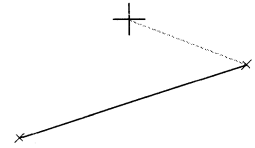
Ortho mode overrides all snaps except the constraining snaps; that is, points you place will constrain to the ortho axes even when you snap them to points not on the ortho axes. If you manually enter the point's coordinates, however, ortho mode is overridden, and the points are placed at the coordinates you enter.

TIPS

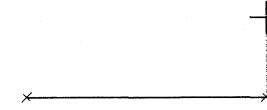
Turn on ortho mode temporarily by holding down CTRL as you place a point.

Set the ortho angle to align with an object in your drawing by clicking the Match button on the ortho-angle speed bar and then clicking the object you want to align the ortho angle to.

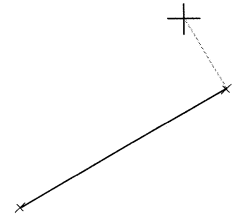
HOW ORTHO MODE CONSTRAINS ENTITIES



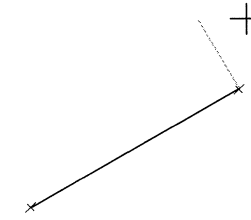
Ortho mode off



Ortho mode on, angle: 0.00, Cursor free off



Ortho mode on, angle: 30.00, Cursor free off

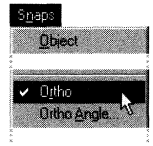


Ortho mode on, angle 30.00, Cursor Free on

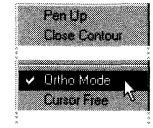
FOR MORE INFORMATION

Drawing efficiently 44
Drawing entities precisely with snaps 50
Setting properties and drawing tools by example 124

To turn ortho mode on and off



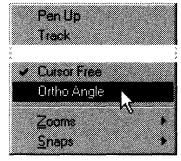
Click the Snaps menu, and then click Ortho. Ortho is checked when on.



or Select a tool, click the right mouse button, and then click Ortho Mode. Ortho is checked when on.

or On a high-resolution monitor, click the Ortho section of the status bar. With ortho on, the angle is displayed.

To change the ortho angle



1 Select a tool, click the right mouse button, and then click Ortho Angle.

2 Enter a value in the Ortho Angle box on the speed bar, and then click OK.

Working with a grid

When you are creating a drawing with entities spaced at regular intervals, such as a building's structural system or an electronic circuitboard, a grid can speed your drawing while maintaining precision. In Visual CADD, a grid is a nonprinting background of equally spaced dots. A grid enables you to snap the points you are drawing to grid points.

After you activate the grid, you can choose to display the grid or not. You can also change the distance between adjacent grid points.

By default, the grid aligns with the drawing origin's 0,0 location. You can move the grid, however, so that it aligns precisely with a specific point in your drawing.

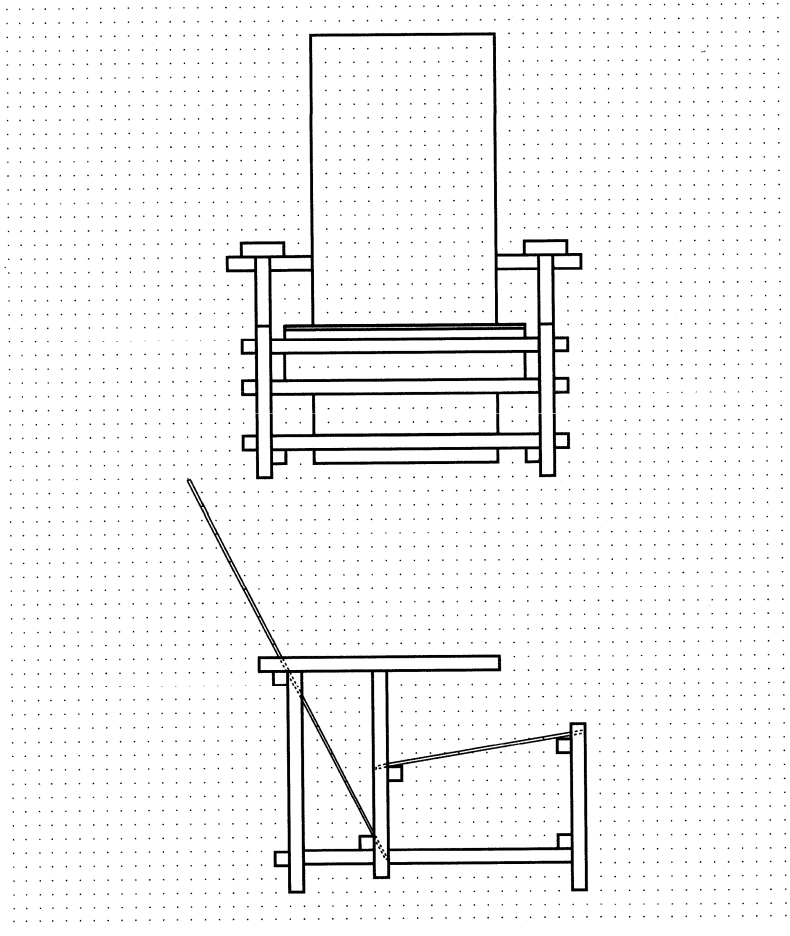
Note: Several actions and commands override the Snap Grid command:

- Manually entering coordinates to place a point
- A snap command
- Ortho mode

TIP

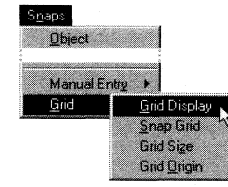
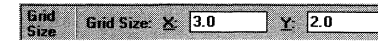
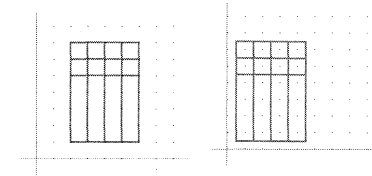
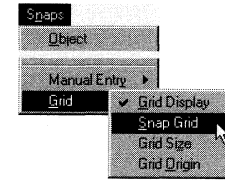
Place objects quickly and accurately at intervals of the X and Y distances from a specific point by aligning and sizing the grid and then turning on Snap Grid.

USING A GRID



Snap to a grid when you are drawing entities at regular intervals or designing in modules.

FOR MORE INFORMATION

To...	Do this...	
View the grid	Click the Snaps menu, click Grid, and then click Grid Display.	
Set the distance between grid points	Click the Snaps menu, click Grid, click Grid Size, enter X and Y grid intervals on the speed bar, and then click OK.	
Align the grid with a specific point	Click the Snaps menu, click Grid, click Grid Origin, and then place a grid point by manually entering its coordinates or by clicking your mouse. The grid will align with that point.	
Constrain point placement to the grid points	Click the Snaps menu, click Grid, and then click Snap Grid. When Snap Grid is checked, point placement is constrained.	

Drawing entities precisely with snaps

Snaps help you place points and objects precisely in relationship to other points, lines, and objects. Some snaps also serve as a drawing constraint. Used in combination with tracking, snaps can serve as a navigational tool for quickly finding points. You can use a snap any time you use a drawing tool or tracking, and you can use a combination of snaps together.

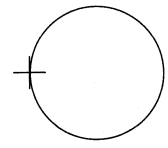
You can snap to the following:

- An object
- The midpoint of a line or arc
- The intersection of two entities
- The midpoint between points on two different objects
- The center of a closed object
- One quadrant point of a circle
- The point nearest the cursor
- The last point placed
- The closest construction point
- An object at a percentage of its length
- An object in a perpendicular, tangent, or parallel orientation

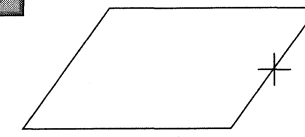
You can choose most snaps from the Snaps menu, from the toolbar, or by clicking the right mouse button and then clicking Snaps. You can also type a keyboard shortcut for all snaps.

Note: Ortho mode overrides all snaps except the constraining snaps.

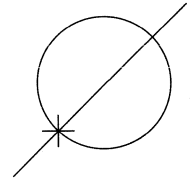
SNAPS THAT PROMPT YOU TO SELECT A POINT OR AN OBJECT



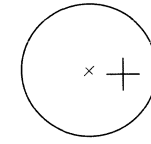
The snap-object tool finds the point on an object nearest where you click.



The snap-midpoint tool finds the midpoint of the line or arc you select.

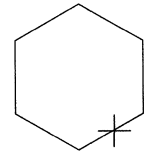


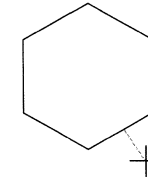
The snap-intersection tool finds the point nearest where you click at which two entities or objects intersect.



The snap-between-2-points tool finds the midpoint between two points or entities you select.

To draw using a prompting snap

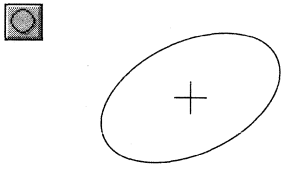




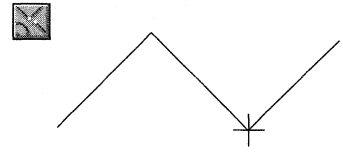
1. With a tool selected, click the Snaps menu and then click the snap you want, or click the snap tool that you want on the toolbar.
2. Click the point or entities to which you want to snap, and then continue drawing.

FOR MORE INFORMATION

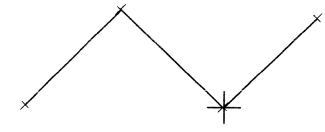
QUICK SNAPS



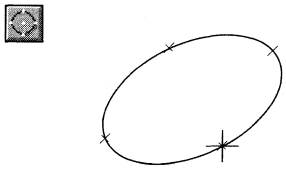
The snap-center tool finds the center of an arc, circle, ellipse, or polygon that you click.



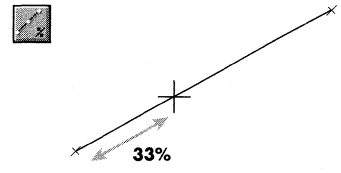
The snap-closest tool finds the construction point nearest where you click.



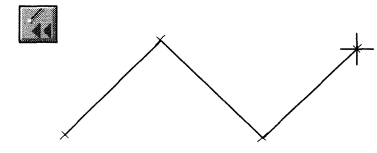
Snap near point finds the construction point nearest the location of the cursor when you type **NP**, or press Shift as you click the right mouse button.



The snap-quadrant tool finds the quadrant point nearest where you click on a circle, ellipse, or an elliptical arc. Quadrant positions are absolute, whatever the ortho angle, and correspond to a clock's 12:00, 3:00, 6:00, or 9:00 positions.

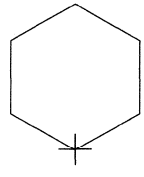


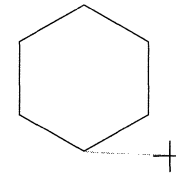
The snap-percentage tool finds the point at a distance from the end nearest where you click that is a percentage of the entity's total length that you specify. Use snap percentage with lines, arcs, and curves.



The snap-last tool finds the last point placed.

To draw using a quick snap

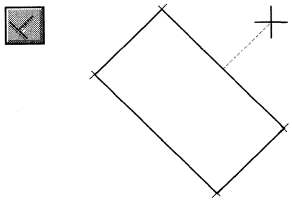




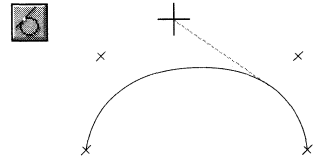
1 With a tool selected, type the two-letter command for the snap you want to use.

2 Continue drawing.

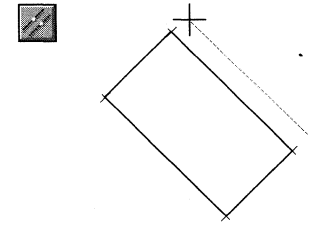
THE CONSTRAINING SNAPS



The snap-perpendicular tool sets a line or rectangle you are drawing to be perpendicular to another line, arc, curve, circle, rectangle, polygon, or ellipse.

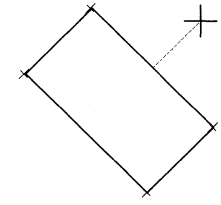


The snap-tangent tool sets a line or rectangle you are drawing to be tangent to another line, arc, curve, circle, rectangle, polygon, or ellipse.



The snap-parallel tool sets a line, rectangle, or polygon you are drawing to be parallel to another line, rectangle, or polygon.

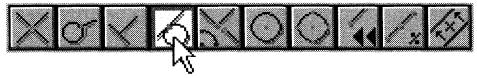
To draw a line perpendicular to an object

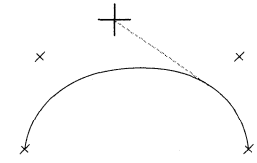


1 With a tool selected, click the snap-perpendicular tool.

2 Click the entity you want to snap to, and then continue to draw.

To draw a line tangent to an object

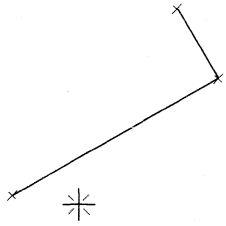




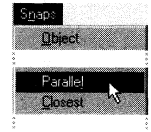
1 With a tool selected, click the snap-tangent tool.

2 Click the entity you want to snap to, and then continue to draw.

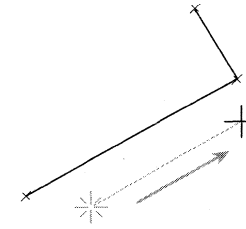
To draw a line parallel to an object



1 With either a straight-line tool or the 3-point-rectangle tool, place the starting point of the line.

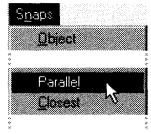


2 On the toolbar, click the snap-parallel tool.

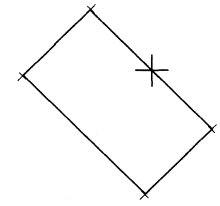


3 Click anywhere on the line you want the new line to be parallel to, and then continue to draw.

To draw a line parallel to an object at a specific distance

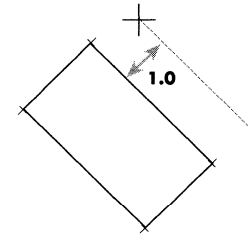


1 With a tool selected, click the snap-parallel tool.



2 Click the object you want to snap to.

3 Enter an offset value on the snap-parallel speed bar, and then click OK.



4 Place a starting point, and then continue to draw.

Making changes to objects

In conventional drafting, you hope to draw objects only once. In fact, the adage advises: draw no more in the morning than you can erase in the afternoon. It's quite different when drawing in CAD.

You tap the power of Visual CADD when you need to alter and manipulate objects. Once you have drawn basic shapes, you use the editing tools to refine them.

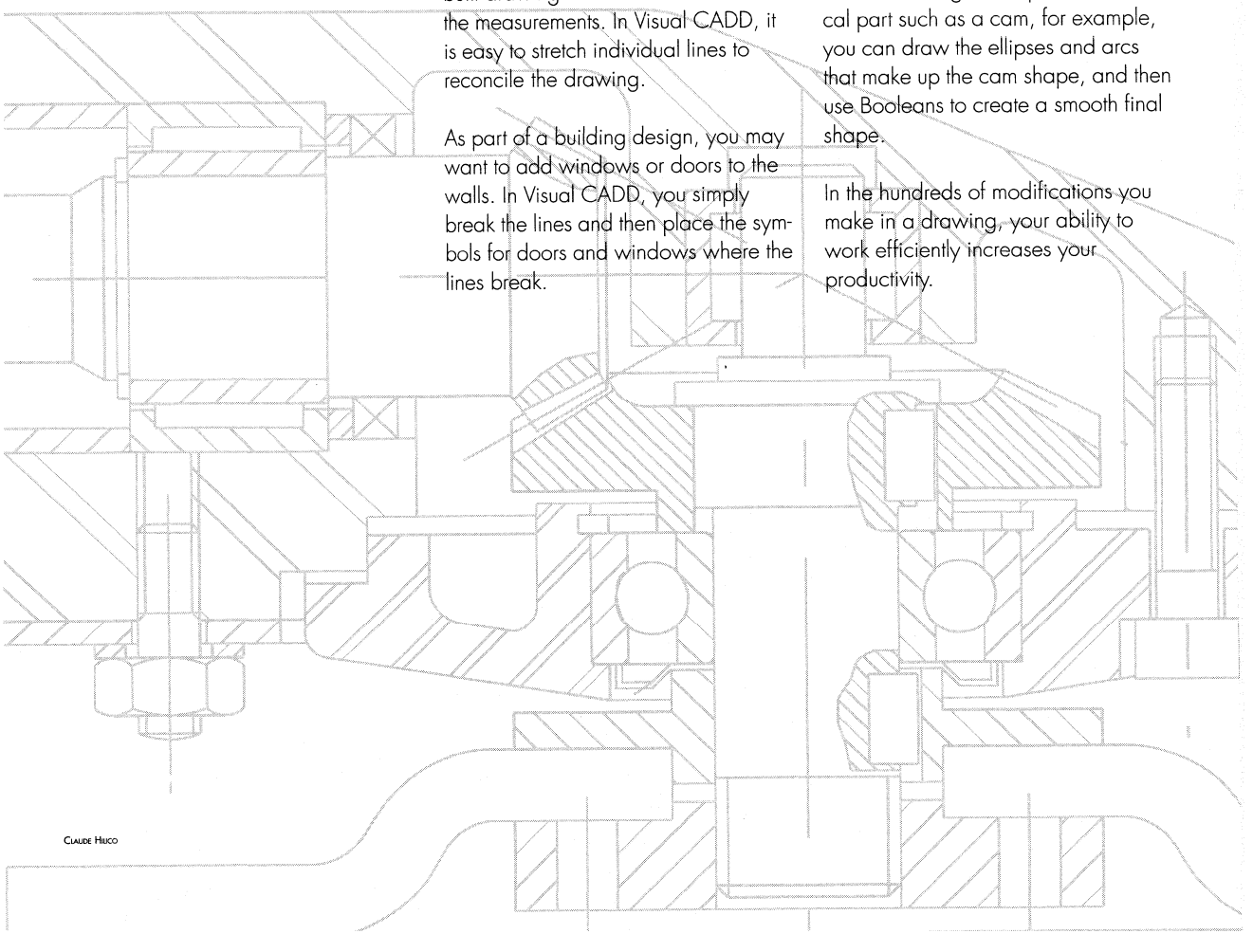
As part of a building renovation, for example, you need to complete as-built drawings and reconcile some of the measurements. In Visual CADD, it is easy to stretch individual lines to reconcile the drawing.

As part of a building design, you may want to add windows or doors to the walls. In Visual CADD, you simply break the lines and then place the symbols for doors and windows where the lines break.

Perhaps you're ready to draw building elevations. In Visual CADD, you simply extend construction lines from the plan into an adjoining area of the drawing to draw the elevations. Once completed, you can trim the construction lines back to the plan.

When drawing a complex mechanical part such as a cam, for example, you can draw the ellipses and arcs that make up the cam shape, and then use Booleans to create a smooth final shape.

In the hundreds of modifications you make in a drawing, your ability to work efficiently increases your productivity.



CLAUDE HELCO

CHAPTER

4

Selecting objects 56–61

Before you change an object, you must select it. Visual CADD provides a variety of ways to select only the objects with which you want to work.

Editing objects 62–77

Visual CADD offers tools to move, rotate, reshape, resize,and delete objects. You can break a line (for example, to insert a door in a wall), extend lines that are too short, trim lines that are too long, and clean up the intersection of multiple lines.

Filling and hatching objects and changing properties 78–81

Use color and pattern to render materials, indicate thickness of objects, and differentiate between objects at various distances from the viewing plane. You can change an object's properties, fill, and hatch pattern at any time.

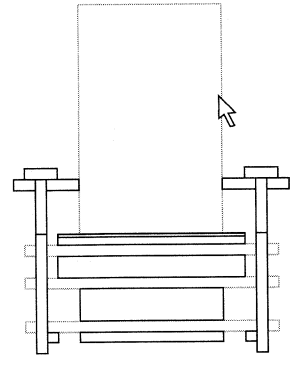
Selecting and deselecting objects

Visual CADD provides the tools to select only the objects you want to edit. You can select objects individually, in groups, and by type, properties, and layer by using the Selection tool, the Select commands on the Edit menu, or the buttons on the selection speed bar. Objects remain selected until you either edit them or explicitly deselect them. Select a set of objects to edit by repeating and combining selection commands.

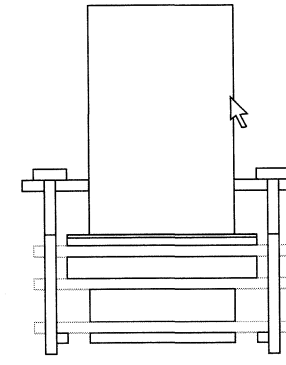
If you have not selected any objects before beginning to edit, Visual CADD prompts you to select an object and opens the selection speed bar. You can open the selection speed bar at any time by typing **S1**.

Note: With Selection Highlight on the Edit menu or on the selection speed bar unchecked, Visual CADD works slightly faster, but you won't be able to distinguish between selected and unselected objects on the screen.

USING THE SELECTION TOOL



Selecting with the selection tool The selection tool selects an object when you click the object. The selection tool selects several objects when you hold down CTRL or SHIFT as you click multiple objects, or when you draw a selection frame around a group of objects.

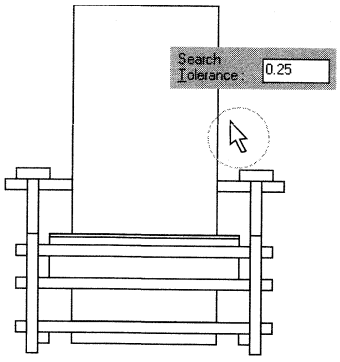


Deselecting with the selection tool The selection tool deselects objects when you click any unselected object or any empty portion of your drawing, or when you press CTRL as you click the selected object.

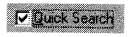
FOR MORE INFORMATION

Setting Visual CADD's defaults 13
Selecting objects by properties and type 60
Viewing the results of your edits 63

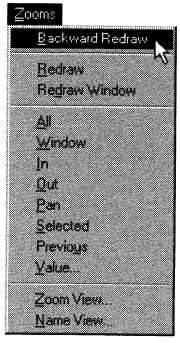
SELECTION SETTINGS



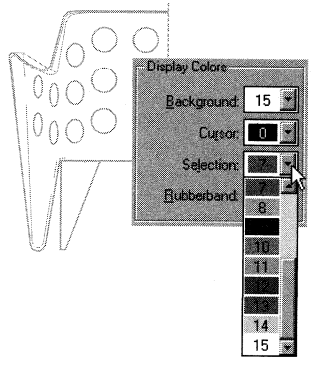
Search tolerance on the System tab in the Settings dialog box specifies the maximum distance the cursor can be from an object for Visual CADD to snap to or select it.



Quick Search With Quick Search checked on the System tab of the Settings dialog box, you select the object drawn earliest within search tolerance of the cursor when you click.



Backward Redraw With Backward Redraw checked on the Zooms menu and Quick Search checked on the System tab of the Settings dialog box, you select the object drawn most recently within a specific distance from the cursor when you click.



Selection color By default, objects appear gray when selected, but you can change the color of selected objects in the Display Colors box on the System tab in the Settings dialog box.

TIPS

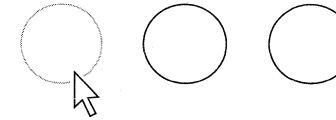
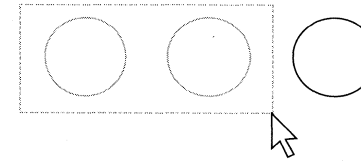
Perform additional editing operations on the same set of selected objects by clicking the Edit menu, clicking Select, and then clicking Last.

To select most of but not all the objects in your drawing, click the Edit menu, click Select, click Select All, and then deselect unwanted objects.

Selection tools available

This tool...		Does this...
All		Selects the entire drawing.
Clear List		Deselects all selected objects.
Invert List		Selects all unselected objects and deselects all selected objects.
Last		Selects the objects selected before the last editing operation or before the last selection was cleared using the clear-list tool.
Layer		Selects all objects on the layer you specify.
Filter		Opens the Selection Filter speed bar which you can use to select all objects of a certain type and with properties you specify.

Selection tools that prompt you to select

This tool...		Does this...	
Object		Selects an object that you click.	
Adjoining		Selects a series of objects that share end points after you click one of the objects. Press Tab to display branching selection alternatives, and then click to select.	
Window		Selects all objects fully enclosed in a selection frame that you draw by clicking its two opposite corners.	
Crossing		Selects all objects enclosed by or touching a rectangular selection frame that you draw by clicking its two opposite corners.	

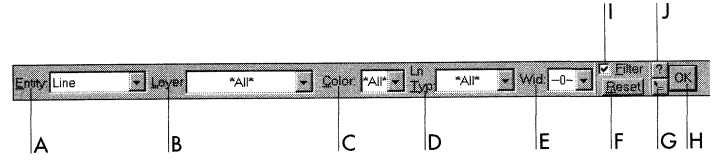
Selecting objects by properties and type

Using the selection filter speed bar, you can specify criteria for selecting objects. You can select objects based on type, color, line type, line width, and/or layer. For example, you can select a symbol or hatch by name to find all occurrences of that symbol or hatch.

When you check Use Filter on the selection speed bar, any selection command you use selects only those objects that meet all of the filter criteria you have set on the selection filter speed bar. For example, you can select all red entities on layer 5 or all Bézier curves with line type 8.

Use the selection filter speed bar to quickly select specific objects in a complex drawing.

SELECTION FILTER SPEED BAR



Opens when you click the Edit menu and then click Selection Filter.

A Sets the type of object as a search criterion.

B Sets a layer as a search criterion.

C Sets the color of objects as a search criterion.

D Sets the line type of objects as a search criterion.

E Sets the line width of objects as a search criterion.

F Resets the selection filter speed bar criteria to All.

G When clicked, sets the selection filter criteria to match the properties of the next object you click.

H Saves the new criteria and closes the selection filter speed bar.

I When checked, turns on the Selection Filter.

J Opens online Help.

TIPS

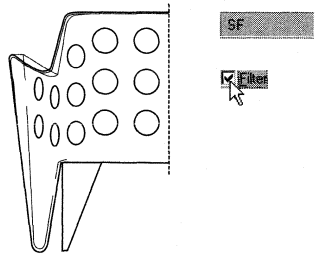
To select objects based on more than one set of criteria, set the selection filter for the first selection criteria, perform the selection, and then set the filter for another set of criteria and select again.

To use an object as a selection filter criterion, click the Match button  on the selection filter speed bar, and then click the object. Visual CADD will select all objects that exactly match the one you selected.

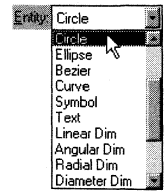
FOR MORE INFORMATION

Selecting and deselecting objects 56

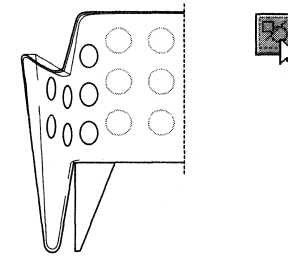
To select objects by properties and type



1 Type **SF**, and then click the selection filter button on the selection speed bar.



2 Select criteria from the drop-down lists, check Filter, and then click OK.



3 Use an appropriate selection command or tool, such as Select All, to select objects using the criteria of the selection filter.

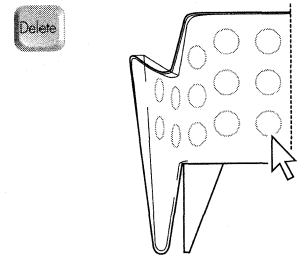
Deleting entities and undoing actions

In Visual CADD, you can delete a selected object or group of objects, the last point placed, the last object drawn, or the entire drawing.

You can also undo current drawing or editing actions. But if you accidently undo an action, you can redo it. Visual CADD doesn't limit the drawing actions you can undo and redo.

If you have chosen the Pack Data command, however, you can only undo and redo the drawing actions that followed the Pack Data command, which deletes all record of changes from the drawing database in order to free memory.

To delete an object or objects



Select the object or objects you want to delete, and then press Delete.

To...	Do this...
Delete the last object drawn or modified	Click the Edit menu and then click Erase Last.
Remove the last segment drawn while drawing a continuous line, irregular polygon, a boundary hatch, or a boundary fill	Click the right mouse button and then click Undo Vertex.
Undo your last operation	Click the Edit menu and then click Undo.
Cancel the last Undo	Click the Edit menu and then click Redo.
Delete the entire drawing	Click the Clear Drawing button on the main speed bar. **Caution:** Once you clear a drawing, you cannot retrieve any part of it.

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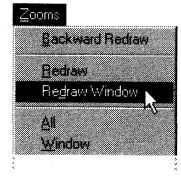
Viewing the results of your edits

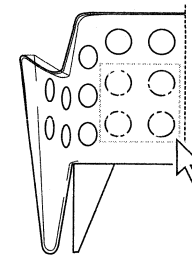
When you move, delete, or change an object that overlaps another object, the screen displays a break in the remaining object. Clicking the Zooms menu and then clicking Redraw updates the screen display.

Redraw time for a complex drawing can be lengthy, so you might want to redraw only the portion of the drawing on the screen.

By default, Visual CADD redraws objects in the order in which they were drawn. You can reset Visual CADD to redraw the newest objects first, which is useful when you want to update only the last few entities drawn.

To redraw a portion of a drawing



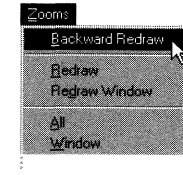


1 Click the Zooms menu, and then click Redraw Window.

2 Draw a selection frame around the area you want to update.

To redraw newest objects first

Click the Zooms menu, and then click Backward Redraw to check it.



TIP

Shorten the time it takes to zoom in on the most recently drawn entities in an area of your drawing by clicking the Zooms menu and then clicking Backward Redraw. When you next click a zoom command, press Esc to stop the redraw immediately after the object you want is redrawn.

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Breaking a line

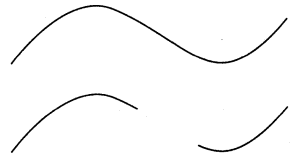
Use the Break command to create a gap in any object except text, dimensions, symbols, attributes, hatches, and fills. For example, you need to break the lines delineating a wall to insert a door symbol.

When you apply a break:

- Polygons and rectangles remain continuous lines.
- Circles become arcs.
- Ellipses become elliptical arcs.
- Open objects, such as continuous lines and arcs become two objects of the same type.

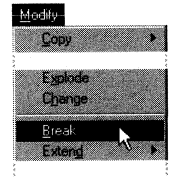
Break is a persistent command, enabling you to break any number of objects, one after another. Break repeats until you press Esc or choose another command or tool.

HOW FAR CAN A GAP SPAN?



In lines drawn with the continuous-line tool, the continuous-Bézier tool, and the spline-curve tool, you can begin a break at any point in one segment and end it at any point in another segment.

To insert a gap in a line



1 Click the Modify menu, and then click Break.

2 Click the object in which you want to create the break.

3 Place a point where you want the break to begin.

4 Place a point where you want the break to end. When finished, press Esc.

FOR MORE INFORMATION

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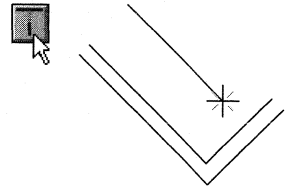
Extending lines to objects

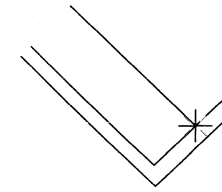
You can extend portions of already-drawn lines to an object. You can extend any line, arc, or complex curve to any entity except text and attributes. Separate tools enable you to extend one line or several lines at once.

The extend tools are especially useful when you want to quickly sketch a drawing, and then trim and extend lines as needed to finish.

To extend one line to an object

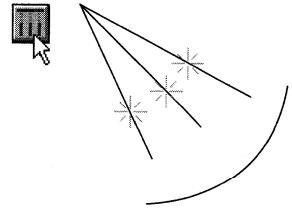


1 Click the extend-object tool, and then click the line you want to extend.

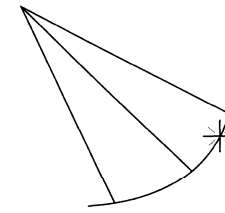


2 Click the object to which you want to extend the line.

To extend several lines to an object



1 Click the lines you want to extend, and then click the extend-multiple-lines tool.



2 Click the object to which you want to extend the lines.

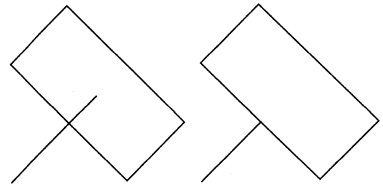
FOR MORE INFORMATION

Trimming lines

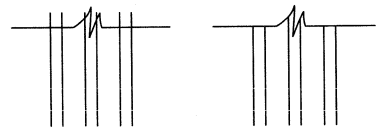
You can quickly erase a portion of a line that extends beyond another simple line or complex object. With the three trim commands, you can trim:

- One line at a time
- Several lines at a time
- The intersection of double lines

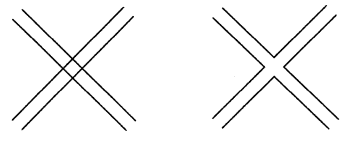
COMPARING THE TRIM COMMANDS



Trim Single deletes the portion of a line, arc, or complex curve that extends beyond another object. Trim Single is a persistent command, enabling you to trim a series of lines, one after another. Trim Single repeats until you press Esc or choose another command or tool.



Trim Multiple deletes the portions of more than one line, arc, or complex curve that extend beyond any other object except a complex curve.



Trim Intersection deletes the portions of crossing double lines that lie within their intersection.

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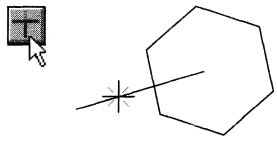
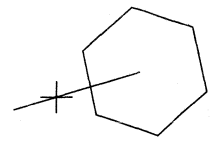
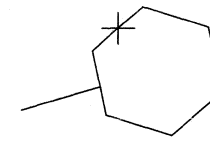
To trim a single line





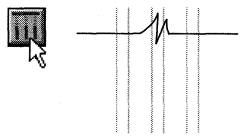


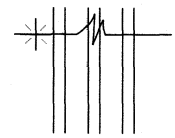
1 Click the trim-single tool from the toolbar, and then click the line you want to trim on the end you want to retain.

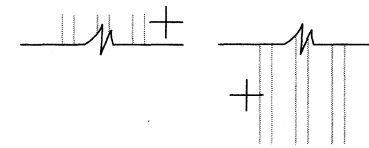
2 Click the object to which you want to trim the line.

3 Press Esc to finish trimming.

To trim several lines at once





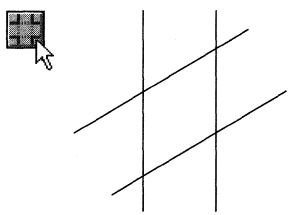


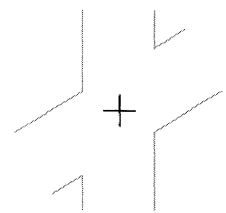
1 Select the lines you want to trim, and then click the trim-multiple tool from the toolbar.

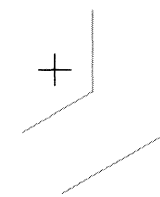
2 Select the object to which you want to trim the selected lines.

3 Move your cursor from one side to the other of the object for a preview of trim options, and then click when you see the option you want.

To trim double lines at their intersection







1 Click the trim-intersection tool from the toolbar.

2 Click inside the crossing of the two pairs of lines.

3 Move your cursor around the intersection for a preview of trim options, and then click when you see the option you want.

Reshaping and moving objects

In Visual CADD, you can reshape any object except a symbol. When you reshape an object, you move some of its construction points without moving others. When you stretch an object, all selected construction points move. You can also move an object without changing its shape.

You can change the shape of an object by selecting one construction point and moving it with the Move Point command. Using the Move Point command in combination with the Near Point snap, you can also change the shape of multiple objects at one time if they each contain a construction point at a common location. The Move Point command simply moves a symbol, however. You must explode a symbol before you can move points within it.

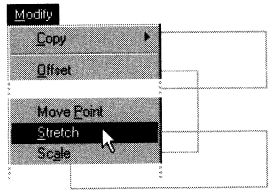
You can move an entire object precisely with the Move command. Move an object quickly but imprecisely by selecting it and then dragging it.

TIP

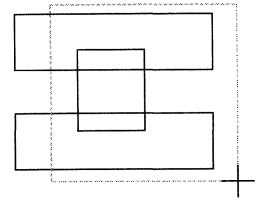
When moving and reshaping an object or a point, place the reference points with direct-distance entry, with a snap, by entering coordinates, or by clicking your mouse.

FOR MORE INFORMATION

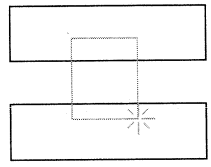
To reshape an object or objects by stretching



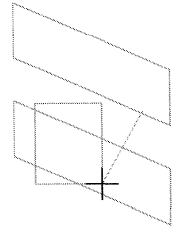
1 Select an object or objects, click the Modify menu, and then click Stretch.



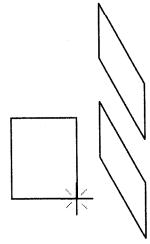
2 Draw a selection frame around the points you want to stretch.



3 Place a reference point establishing the move's starting point.

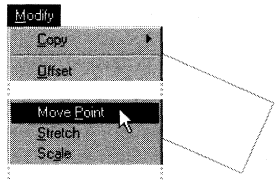


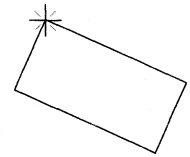
4 Move your cursor to preview stretch results.

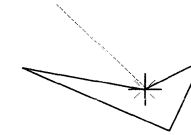


5 Place a reference point establishing the move's end point.

To reshape an object by moving a point





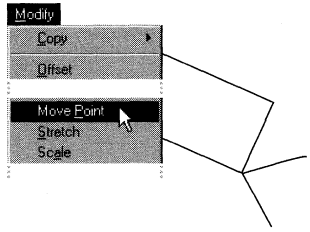


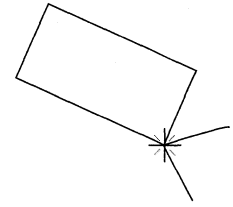
1 Click the Modify menu, click Move Point, and then select an object.

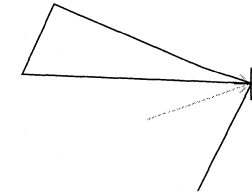
2 Place a reference point establishing the move's starting point.

3 Place a reference point establishing the move's end point.

To reshape multiple objects by moving a common point





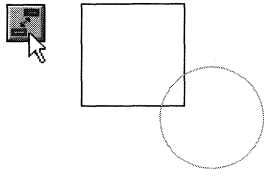


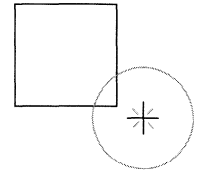
1 Click the Modify menu, and then click Move Point.

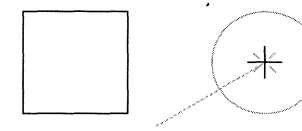
2 Move the cursor near the common point, and then type NP.

3 Place a reference point establishing the move's end point.

To move an object or objects to a precise location







1 Select an object or objects, and then click the Move tool.

2 Place a reference point defining the move's starting point.

3 Place a reference point defining the move's end point.

Resizing objects

When you draw an object at one size and later want to resize it, or when you merge one drawing into another and want them at the same scale, you can use two scale commands:

- The Scale command resizes an object at a ratio you specify.
- The Fit Scale command resizes one object to match the size of a second.

When you resize an object by using the Scale command, you can resize in both X and Y directions independently, thereby distorting the object. To avoid distortion and resize an object or group of objects proportionally, enter equal values for both X and Y axes.

Scale values of 1.0 are equivalent to full-size, or 100%. Values less than 1.0 reduce the selected object's size. Values greater than 1.0 increase the selected object's size.

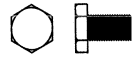
In most cases, changing the X scale value for an object resizes the object in the horizontal direction, and changing the Y scale value resizes it in the vertical direction. The X and Y scale values of rotated text and symbols, however, rotate with them. For example, text rotated to a 90° angle resizes in the vertical direction when you change the X scale value.

TIPS

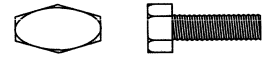
Mirror an object around its X axis by entering a negative X value. Mirror an object around its Y axis by entering a negative Y value.

Combine a real-world-scale drawing and a symbol that represents a drawing border by loading and placing the border symbol into the drawing, clicking the Modify menu, clicking Fit Scale, and then rescaling the border to fit around the drawing. When you print the drawing, choose the Fit to Paper option.

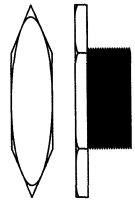
CHANGING SCALE VALUES



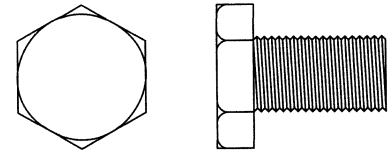
X: 1, Y: 1



X: 2, Y: 1



X: 1, Y: 4



X: 3, Y: 3

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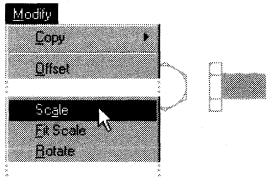
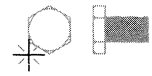
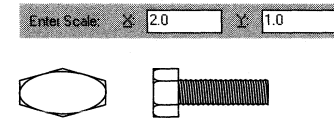
To resize an object by changing X and Y scale values





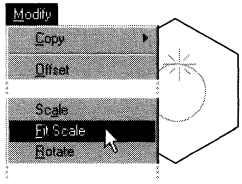


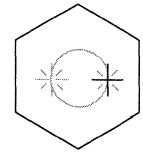
1 Select the object or objects, click the Modify menu, and then click Scale.

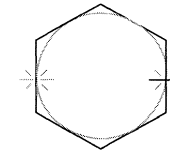
2 Place a reference point establishing the point from which the object(s) will be resized.

3 Enter scale values for X and Y factors on the scale speed bar, and then click OK.

To resize an object by example







1 Select the object or objects, click the Modify menu, and then click Fit Scale.

2 Place two reference points within the selection to serve as the base distance to resize.

3 Place two reference points anywhere in your drawing to establish the distance to which you want to resize the selected object or objects.

Rotating objects

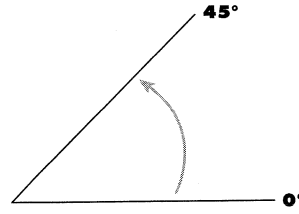
You can rotate an object or group of objects around any point that you place in your drawing. You can rotate precisely by entering an angle value; or, you can rotate by moving your mouse and then clicking when you see the preview option you want.

TIPS

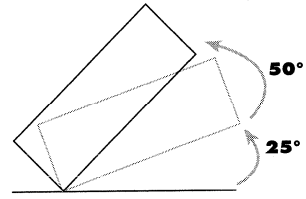
Realign an object to the ortho angle by clicking the Modify menu, clicking Rotate, and then making sure Ortho Mode is checked.

Set the rotation angle quickly by clicking the match-entity tool to select an object whose angle you want to match.

ROTATION ANGLE

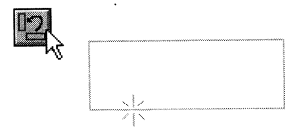


Rotation angle 0° measures counter-clockwise from a 3:00 position.



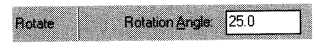
If the object you want to rotate lies at an angle, add the object's base angle to the angle of rotation for the new base angle.

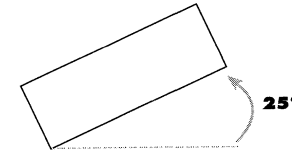
To rotate an object



1 Select an object or objects, and then click the Rotate tool.

2 Place a reference point at the center of rotation.





3 Enter a Rotation Angle value in the rotate speed bar, and then click OK.

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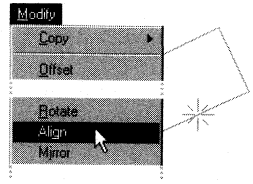
Moving and rotating objects in one operation

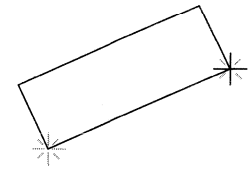
To move an object and change its angle at the same time use the Align command. The Align command changes the orientation of an object or a group of objects to match that of another object or angle you create.

Moving and rotating is a three-step process. First, you define reference points on the selected object, then you define a point to which the first reference point moves, and finally you define a point that establishes the new angle of the moved object.

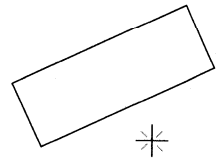
To move and rotate an object



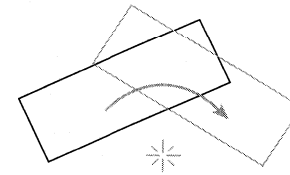
1 Select an object, click the Modify menu, and then click Align.



2 Place two reference points on the selected object to define the angle you want to change.



3 Place a reference point where you want to move the first reference point on the selected object.



4 Place a reference point that defines the new angle of the selected object.

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Inserting a chamfer or fillet between two lines

After you have drawn two lines that intersect, you can angle or curve the intersection. When you chamfer the intersection of two lines, a third, angled line joins them. When you fillet the intersection of two lines, an arc joins them.

You can also set the double-line tool to automatically fillet the joints between line segments as you draw. Do this by checking the Auto Fillet option on the General tab in the Settings dialog box.

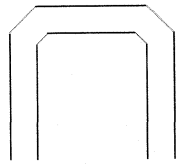
Note: Both the Chamfer and Fillet commands are persistent, repeating until you press Esc, select another tool, or choose another command.

TIPS

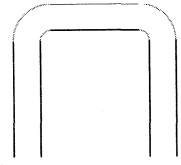
Turn on Auto Fillet without changing the fillet radius settings by clicking the right mouse button and then clicking Auto Fillet.

Trim two intersecting lines quickly by applying a fillet with fillet radius set to 0.

CHAMFERS AND FILLETS

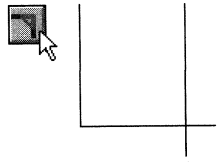


Chamfer inserts an angled line segment at the intersection of two lines or arcs. Set the distance of each end of the chamfer from the point of intersection in the Chamfer box on the General tab of the Settings dialog box.



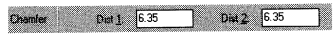
Fillet inserts an arc at the intersection of two lines or arcs. You set the radius of the fillet in a speed bar as you place it or preset it on the General tab of the Settings dialog box.

To insert an angled intersection between two lines

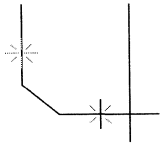


1 Select the chamfer tool.

2 Click the right mouse button, and then click Chamfer Dist.



3 On the Chamfer speed bar, set the Chamfer distance, and then click OK.



4 If only two entities intersect, click between them. If more than two entities intersect, click the two that you want to chamfer.

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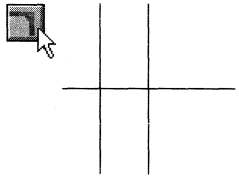
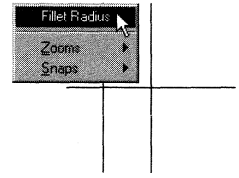
To insert a fillet between two lines

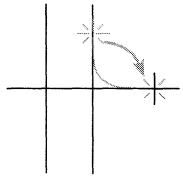


1 Select the fillet tool.

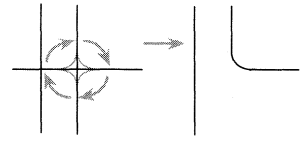


2 Click the right mouse button, and then click Fillet Radius.

3 On the fillet speed bar, set the fillet radius, and then click OK.



4 If only two entities intersect, click between them. If more than two entities intersect, click the two that you want to fillet.

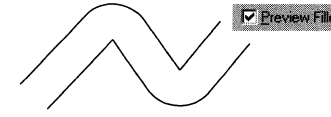


5 Move the cursor around the intersection for a preview of the fillet options, and then click when the preview fillet is located correctly.

To curve the vertices of a double line as you draw

1 Click the double-line tool.

2 Click the right mouse button, and then click Fillet Radius. Enter a value in the Fillet Radius box on the Fillet speed bar, and then click Auto Fillet to check it.



3 Click Preview Fillet to see the fillets as you draw, and then click OK.

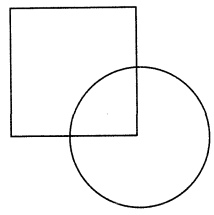
Combining entities to create a new object

Sometimes creating an object by combining existing entities is easier than drawing it from scratch. The Boolean command joins two selected, overlapping closed objects of any type into a mixed-entity path, enabling you to preview and choose from at least three variations. You preview these options by moving your cursor around the selected objects. The variations create an object that results from:

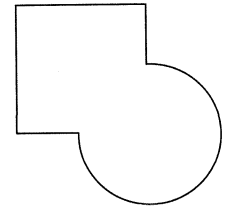
- The *union* of the selected objects, retaining all outer portions.
- The *intersection* of the selected objects, retaining only those portions that intersect.
- The *subtraction* of one object from the other, retaining portions that lie outside the limits of the other.

The Boolean command is useful in cases such as adding a room outside the existing perimeter of a house plan.

BOOLEAN VARIATIONS

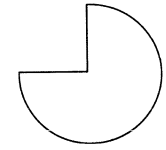


ORIGINAL OBJECTS



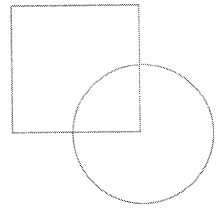
UNION

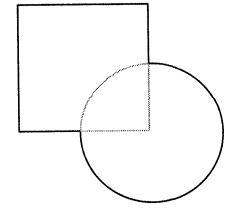
INTERSECTION



SUBTRACTION

To create a new object from existing entities





1 Select two overlapping closed objects, click the Modify menu, and then click Boolean.

2 Move your cursor around the objects for a preview of Boolean options, and then click when you see the option you want.

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Exploding objects into their basic components

When you want to edit an object contained in a compound object, such as a symbol, you must first explode the object. You can explode these entities:

- Continuous lines, rectangles, and polygons into collections of single lines
- Continuous Bézier and spline curves into collections of single Bézier curves
- Dimensions into lines, fills, and text blocks
- Vector text into lines and arcs
- Symbols and attributes into the objects and text blocks used to create them
- Hatch patterns into collections of individual lines
- Fills into the lines that bound them

You cannot explode basic objects, such as single lines, arcs, circles, ellipses, single Bézier curves, and TrueType text.

If the object you are exploding contains compound objects, you may need to explode it several times before you are able to select the entity you want to edit.

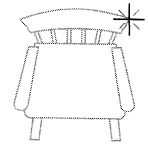
Note: Once you explode an object, you cannot regroup its components into a single object.

TIPS

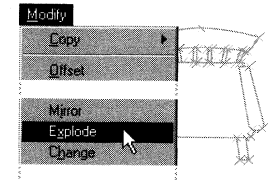
Select exploded objects easily with the select-adjoining tool.

Explode continuous lines as you draw them by making sure Explode Cont. Lines is checked on the System tab of the Settings dialog box.

To explode an object into its basic components



1 Select the object you want to explode.



2 Click the Modify menu, and then click Explode.

FOR MORE INFORMATION

Rendering objects with hatch patterns and solid fills

You use hatch patterns and fills to differentiate between solid and hollow objects, between materials, and between locations in an object. When you hatch or fill an object, you create a new entity with the same shape as the selected object. The hatch or fill entity has its own properties.

Hatches and fills are associative; that is, when you stretch or resize the object or boundary containing the hatch or fill, the hatch or fill follows the contours of the new shape.

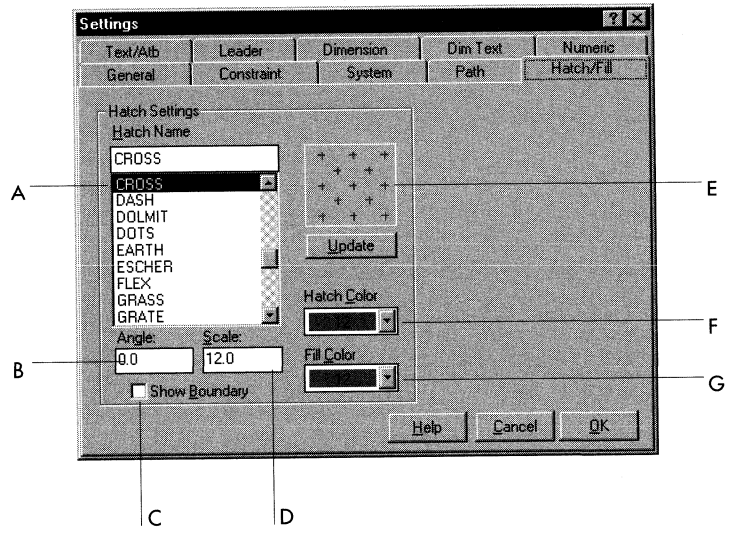
TIPS

When you want to stretch or resize an object that you have hatched or filled, select both the original object and the hatch or fill entity before stretching or resizing them.

To hatch or fill an irregular, closed shape, click the seed-hatch tool or the seed-fill tool, and then click inside the area you want to hatch or fill.

You can change the hatch settings and fill color that apply to the next objects you hatch and fill by changing settings on the Hatch/Fill tab of the Settings dialog box. You change the hatch pattern's line type and width and the fill color using the Change command on the Modify menu.

HATCH AND FILL SETTINGS



Opens when you click the Utilities menu, click Settings, and then click the Hatch/Fill tab.

A Lists the available hatch patterns.

B Sets the selected hatch pattern's angle. Type a positive or negative value between 0 and 360.

C Displays fill and hatch boundaries as a line on the screen.

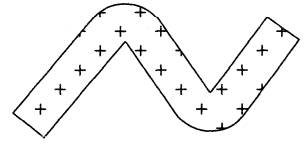
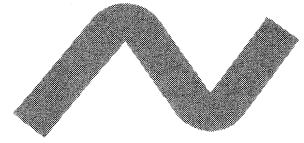
D Sets the scale of the selected hatch pattern. 1.0 = 100%.

E Displays a sample of the selected hatch pattern when you click Update.

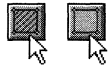
F Sets the hatch color, which applies to all hatch patterns.

G Sets the fill color, which applies to all fills.

FOR MORE INFORMATION

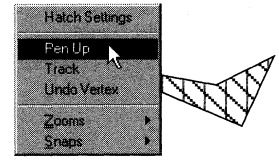
To...	Do this...	
Hatch a closed object with the current hatch pattern	Select an object or objects, and then click the hatch-selection tool.	 
Fill a closed object with a solid color	Select an object or objects, and then click the fill-selection tool.	 

Draw an irregular polygon and add a hatch pattern or solid fill



1 Click the hatch-boundary tool or the fill-boundary tool.

2 Place points defining the vertices of the enclosing polygon.



3 To finish, click the right mouse button, and then click Pen Up.

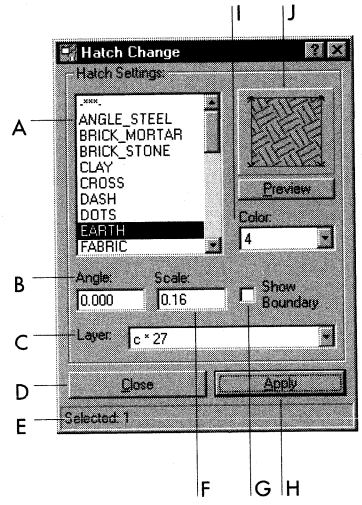
Changing the hatch pattern of a hatched entity

Materials and finishes frequently change in the life of a project. To change the rendered material in your drawing, you must change the hatch pattern or color. In Visual CADD, you can change the properties of a hatch pattern after it is applied to an object. You use the Hatch Change dialog box to change the pattern, angle, scale, or layer of the hatch pattern. You can change one or several properties of more than one hatched object, even if the objects contain different hatch patterns.

TIP

Change the color, layer, line type, and line width quickly by clicking the Modify menu and then clicking Change.

HATCH CHANGE DIALOG BOX



Opens when you click the Draw menu, click Hatch, and then click Hatch Change.

A Lists the available hatch patterns.

B Sets the selected hatch pattern's angle. Type a positive or negative value from 0 to 360.

C Sets the layer on which the hatch pattern will be located.

D Closes the Hatch Change dialog box.

E Displays the number of objects selected.

F Sets the scale of the selected hatch pattern. 1.0 = 100%.

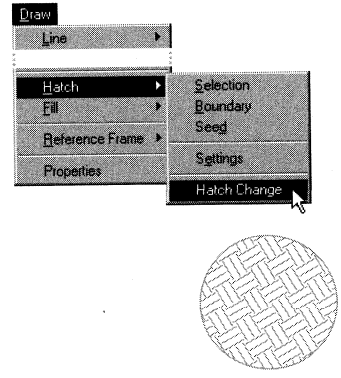
G Displays and prints the boundary of the hatch pattern as a line.

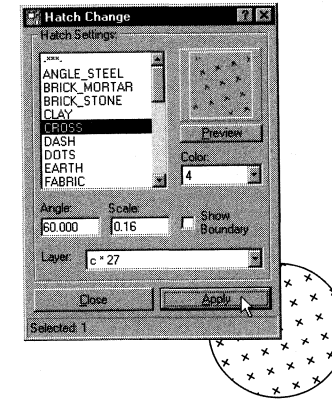
H Applies the settings to the selected entity.

I Sets the hatch color.

J Displays a sample of the selected hatch pattern when you click Preview.

To change the hatch pattern of a hatched entity





1 Select a hatched object or objects, and then click the Draw menu, click Hatch, and click Hatch Change.

2 Make changes to the hatch settings, and then click Apply.

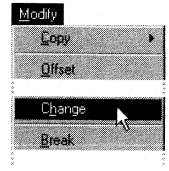
FOR MORE INFORMATION

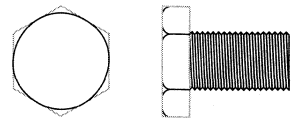
Changing the properties of objects

At any time while you are drawing or editing, you can change the color, line type, and line width of selected objects, as well as the layer on which they are located. If you select more than one object, you can change only those properties common to all selected objects.

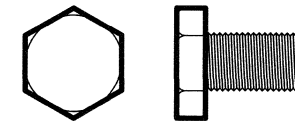
Use the Hatch Change dialog box to change all properties of a hatch pattern. Use the Text Editor to change all text properties. Use the Dimension Editor to change all properties of dimensions. Use the Layer properties speed bar to change the properties assigned to a layer.

To change the properties of an object





1 Select an object or objects, click the Modify menu, and then click Change.



2 Select options from the drop-down lists to set the new properties, and then click OK.

FOR MORE INFORMATION

Assigning properties to a layer 25
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Modifying existing dimensions 88
Adding text to a drawing 100
Setting properties and drawing tools by example 124

Dimensioning and measuring

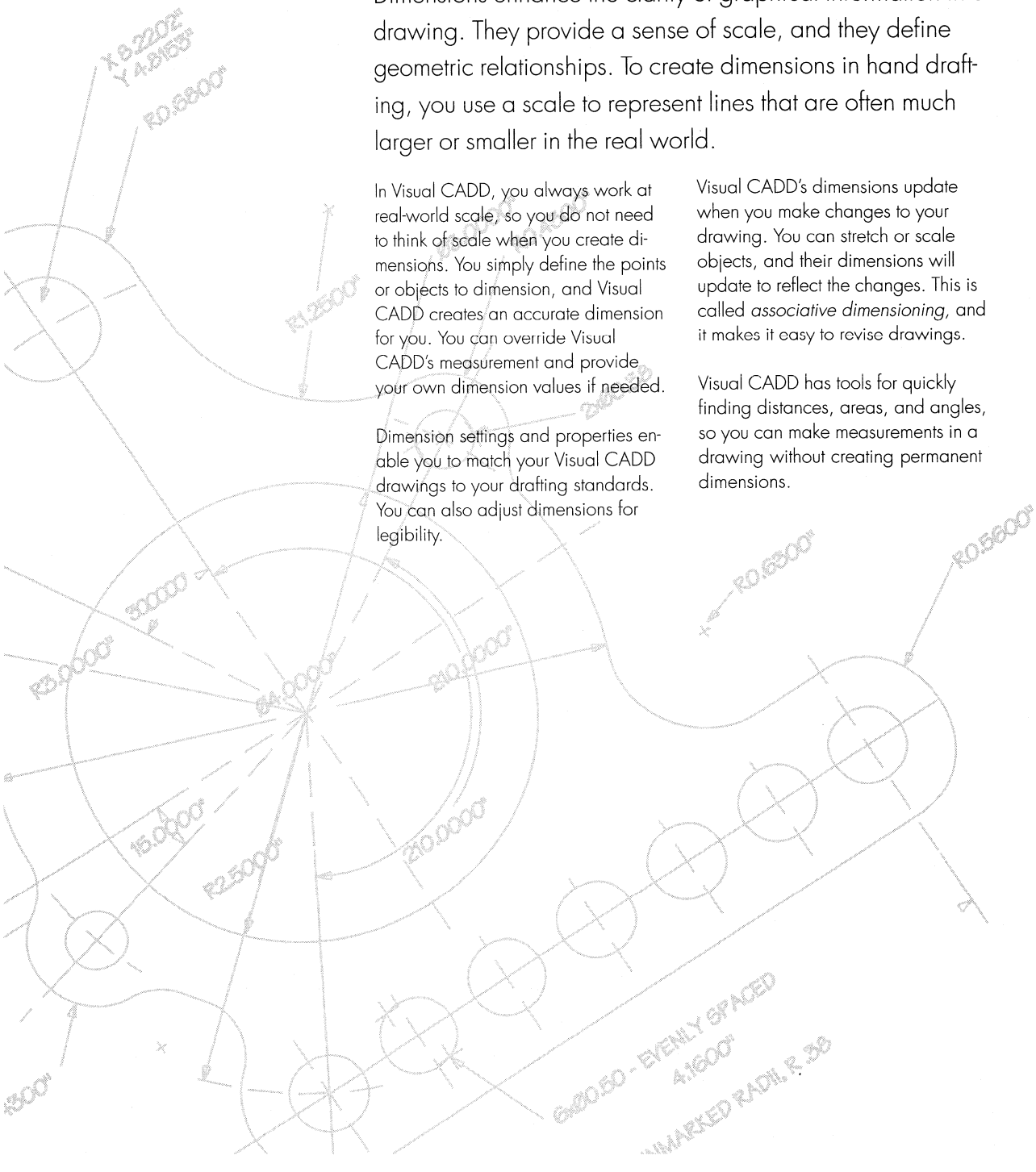
Dimensions enhance the clarity of graphical information in a drawing. They provide a sense of scale, and they define geometric relationships. To create dimensions in hand drafting, you use a scale to represent lines that are often much larger or smaller in the real world.

In Visual CADD, you always work at real-world scale, so you do not need to think of scale when you create dimensions. You simply define the points or objects to dimension, and Visual CADD creates an accurate dimension for you. You can override Visual CADD's measurement and provide your own dimension values if needed.

Dimension settings and properties enable you to match your Visual CADD drawings to your drafting standards. You can also adjust dimensions for legibility.

Visual CADD's dimensions update when you make changes to your drawing. You can stretch or scale objects, and their dimensions will update to reflect the changes. This is called *associative dimensioning*, and it makes it easy to revise drawings.

Visual CADD has tools for quickly finding distances, areas, and angles, so you can make measurements in a drawing without creating permanent dimensions.



CHAPTER

5

Adding dimensions 84–95

Dimensions describe drawing geometry. You can control the appearance of dimensions by adjusting their properties before, during, or after creating them.

Adding measurements 96–97

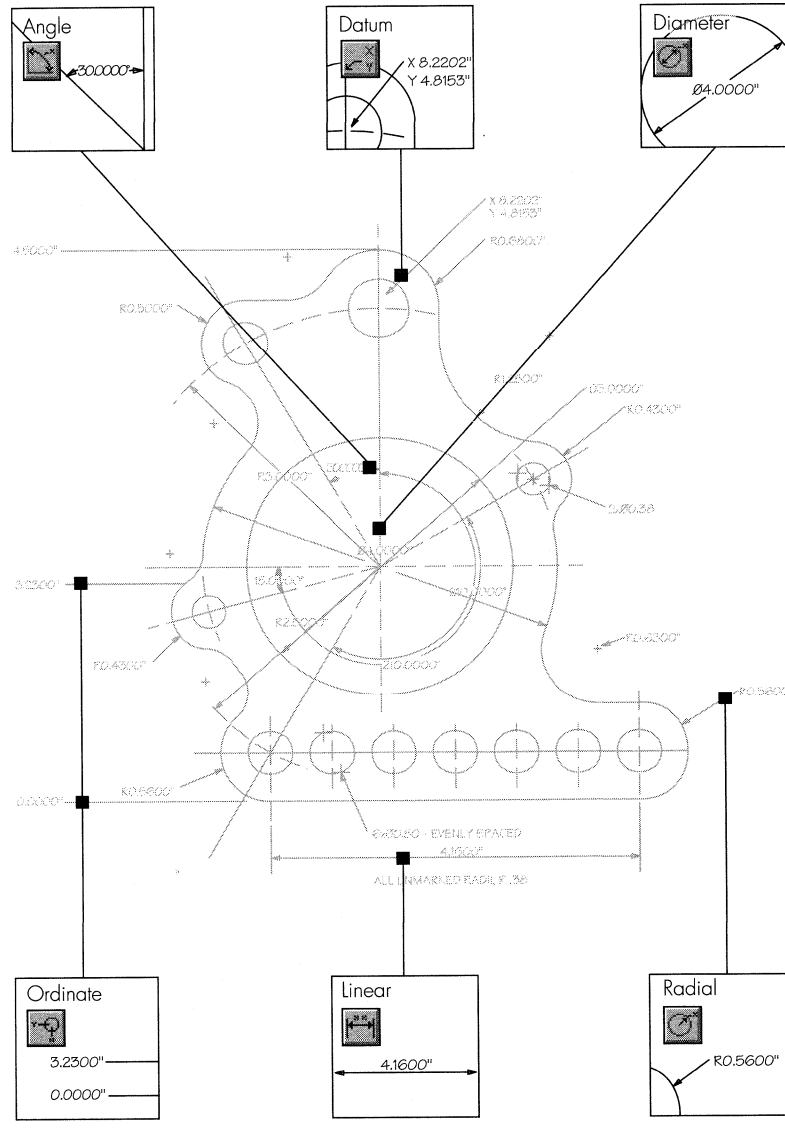
Measure areas, angles, distances, and arc lengths without creating permanent dimensions.

Adding dimensions to a drawing

Adding dimensions to your drawing doesn't require knowledge of dimension settings; you can use Visual CADD's default settings to get started, or change dimension settings before, during, or after creating a dimension.

Selecting points and objects precisely is crucial to accurate dimensioning. While creating dimensions, use snaps to select precise points on the drawing geometry rather than estimating endpoints or intersections.

CREATING DIMENSIONS



TIPS

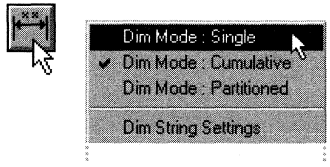
To change dimension options after selecting a dimensioning tool, click the option tabs on the speed bar or click the right mouse button.

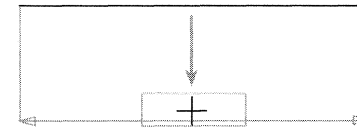
If you make a mistake while placing chained dimensions (ordinate, cumulative, or partitioned), delete the last dimension of the chain by pressing the Backspace key.

If ortho mode is constraining your leader line, click the right mouse button, and then click Ortho Mode to uncheck it. To temporarily turn ortho mode on or off, hold down CTRL while dragging the leader line.

FOR MORE INFORMATION

To dimension a single linear measurement



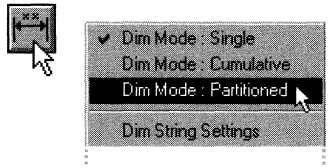


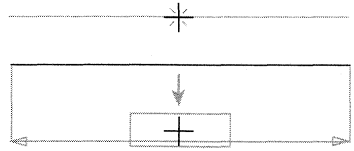
1 Click the linear-dimension tool, click the right mouse button, and then click Dim Mode : Single.

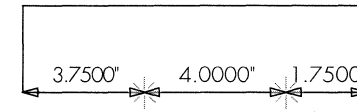
2 Click either the line or the start and end points for the dimension.

3 Click where you want to place the dimension line.

To create a chain of linear dimensions





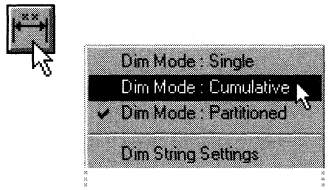


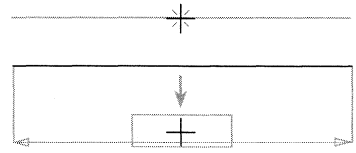
1 Click the linear dimension tool, click the right mouse button, and then click Dim Mode : Partitioned.

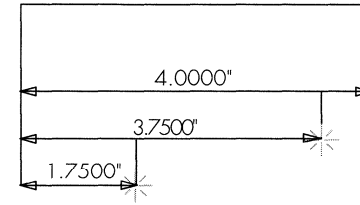
2 Click either the line or the start and end points for the first dimension of the chain, and then click where you want to place the dimension line.

3 Click the end points for the remaining dimensions.

To draw a baseline dimension





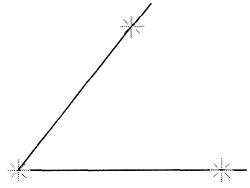


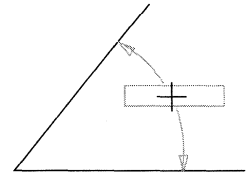
1 Click the linear dimension tool, click the right mouse button, and then click Dim Mode : Cumulative.

2 Click either the line or the start and end points for the first dimension, and then click where you want to place the first dimension line.

3 Click the end points and dimension line locations for the remaining dimensions, click the right mouse button, and then click Pen Up.

To dimension an angle



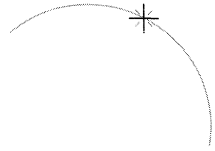


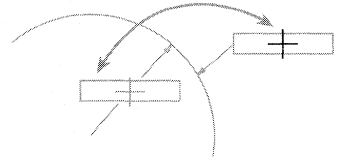
1 Click the angular-dimension tool.

2 Click the arc or the three points that define the angle.

3 Click where you want to place the dimension arc.

To dimension the radius of an arc or circle



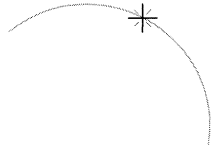


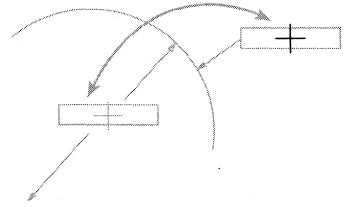
1 Click the radial-dimension tool.

2 Click the circle or arc to dimension.

3 Click where you want to place the dimension.

To dimension the diameter of an arc or circle



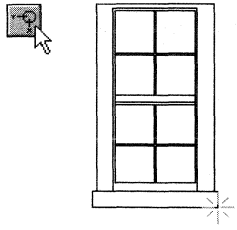


1 Click the diameter-dimension tool.

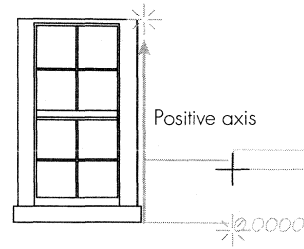
2 Click the circle or arc to dimension.

3 Click where you want to place the dimension.

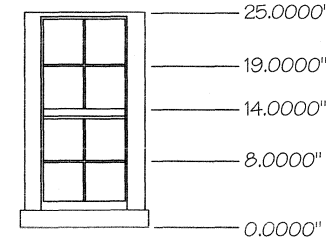
To dimension a series of offsets from a basepoint



1 Click the ordinate-dimension tool, and then click a basepoint from which to measure the dimensions.

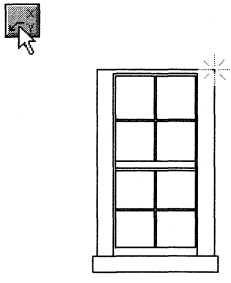


2 Click a point that describes the positive axis of your measurements, and then click where you want to locate the dimension lines.

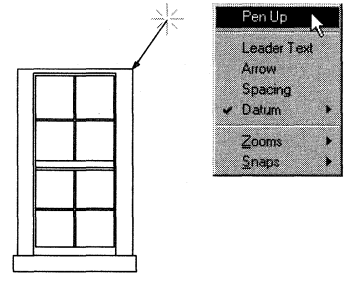


3 Click the points that you want to dimension, click the right mouse button, and then click Pen Up.

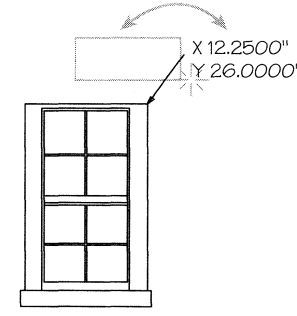
To create a datum dimension



1 Click the datum-dimension tool, and then click the point to dimension.



2 Click additional points to draw the leader, click the right mouse button, and then click Pen Up to complete the leader line.



3 Click to position the datum text.

TIP

To edit a datum dimension as you place it, click the right mouse button while placing the leader, click Datum, click Datum Edit, and then enter text and select options on the speed bar.

Modifying existing dimensions

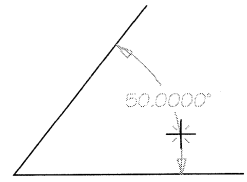
In addition to modifying the properties associated with a dimension, you can also move dimension lines and text.

To find the commands for modifying dimensions, click the right mouse button after you select the dimension you want to change. The options will vary according to the type and number of dimensions you select.

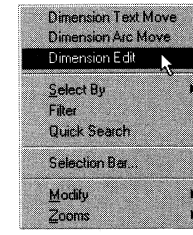
TIP

To reshape a leader line, click the Modify menu, click Move Point, and then select the leader segment you want to reshape.

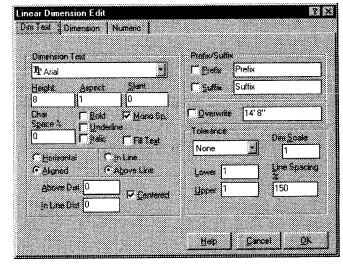
To modify dimension properties



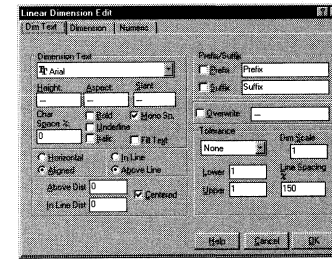
1 Select the dimension(s) you want to edit.



2 Click the right mouse button, and then click Dimension Edit.



3 Modify the dimension properties you want to change.



Note: When more than one dimension is selected, a dash (–) indicates that properties vary among the selected dimensions.

FOR MORE INFORMATION

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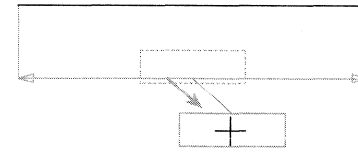
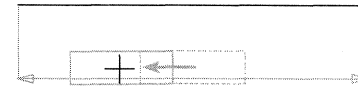
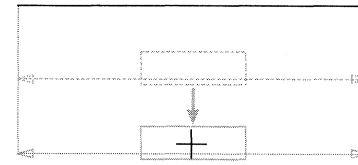
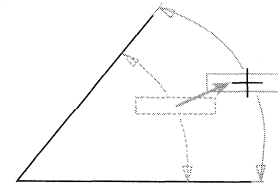
To...	Do this...	
Move dimension text	Select the dimension, click the right mouse button, and then click Dimension Text Move. Click where you want to relocate the dimension text.	
Slide dimension text along a dimension line	Select a linear, ordinate, diameter, angular or radial dimension, click the right mouse button, and then click Dimension Text Slide. Click where you want to relocate the dimension text.	
Move a dimension line	Select a linear dimension, click the right mouse button, and then click Dimension Line Move. Click where you want to relocate the dimension line.	
Move a dimension arc	Select an angular dimension, click the right mouse button, and then click Dimension Arc Move. Click where you want to relocate the dimension arc.	

Drawing leaders

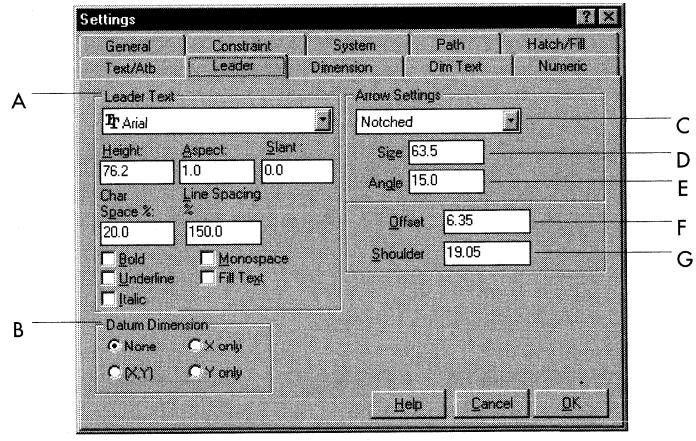
Leaders—arrows with text attached—allow you to visually connect text information to your drawing.

Use leader settings to adjust the text properties of the leaders in your drawing. Leader settings also control the format of datum dimensions.

TIP

If ortho mode is constraining your leader line, click the right mouse button, and then click Ortho Mode to uncheck it. You can also hold down CTRL while dragging the leader line to temporarily turn ortho mode on or off.

LEADER SETTINGS



Opens when you click the Utilities menu, click Settings, and then click the Leader tab.

A Sets standard text characteristics for leaders.

B Sets the datum dimension style to display X values, Y values, or both.

C Lists the types of arrowhead to place at the end of the leader.

D Sets the length of the arrowheads.

E Adjusts the shape of the arrowhead or the angle of the slash.

F Sets the distance between the tail-end of the leader and the leader text.

G Adjusts the size of the leader tail's horizontal segment that is nearest the text.

FOR MORE INFORMATION

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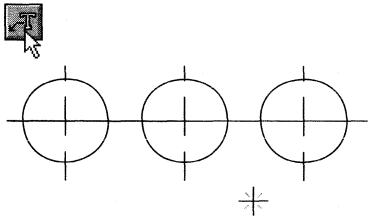
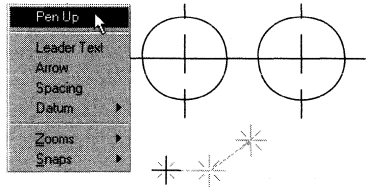
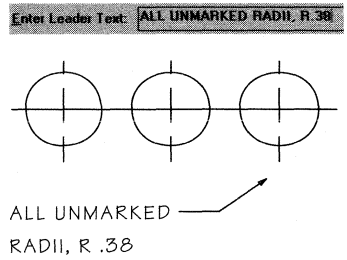
To draw a leader





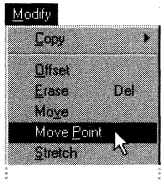


1 Click the leader tool, and then click the start point for the leader.

2 Click additional points to draw the leader, click the right mouse button, and then click Pen Up.

3 Type the leader text in the speed bar, click OK, and then click to locate the leader text.

To reshape a leader



1 Click the Modify menu, and then click Move Point.

2 Click anywhere on the leader line, and then click the point you want to move.

3 Click the new location of the point.

Adjusting dimension settings

Dimension settings control dimension and extension lines, dimension text, and numeric formats. Adjusting dimension settings enables you to match dimension objects to your drafting standards. Each element of a dimension can be modified individually.

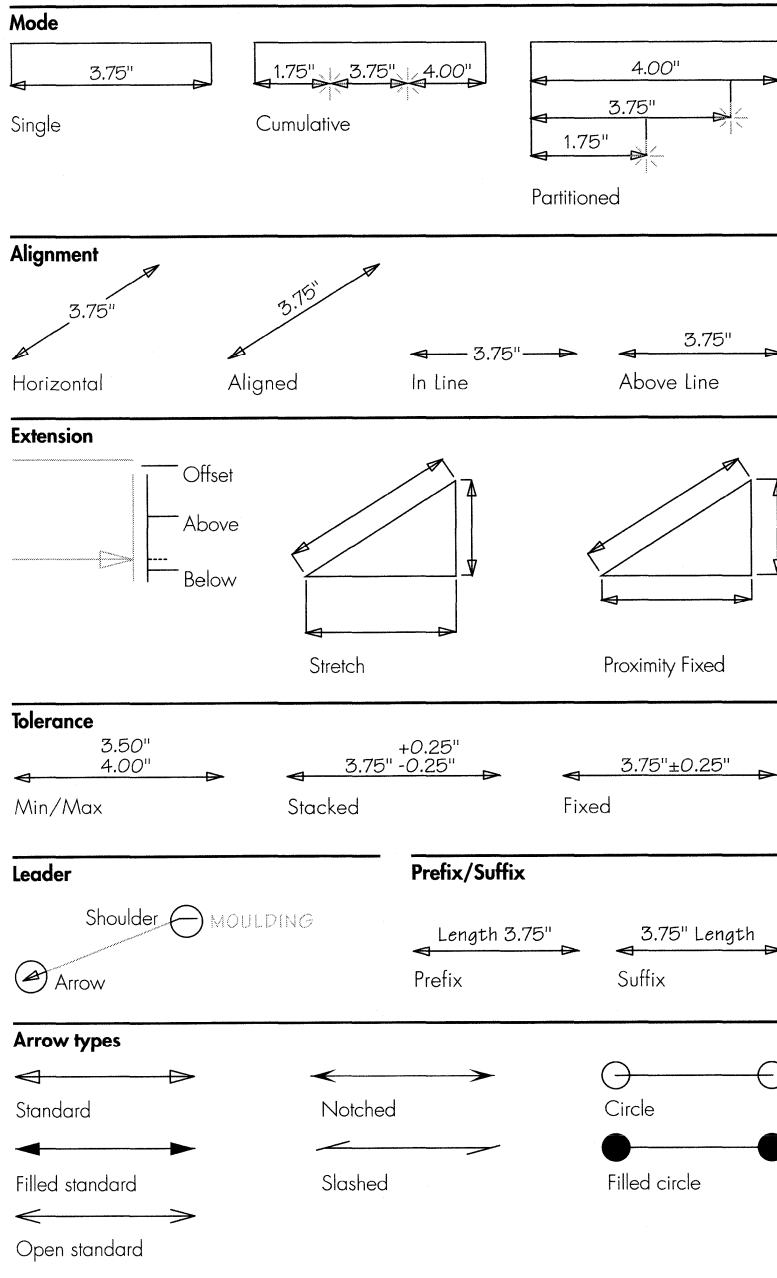
Because Visual CADD works in real-world scale, consider your final output when adjusting dimension text and elements. Set properties so that the elements are legible when they are scaled for printing.

By placing all dimensions on one layer, you can easily control when dimension information is visible.

Create dimensions that define an acceptable range of values by using dimension tolerances. You can specify the range as well as the way it is displayed.

You can store dimension settings with styles and then retrieve them to adjust settings for a particular dimension.

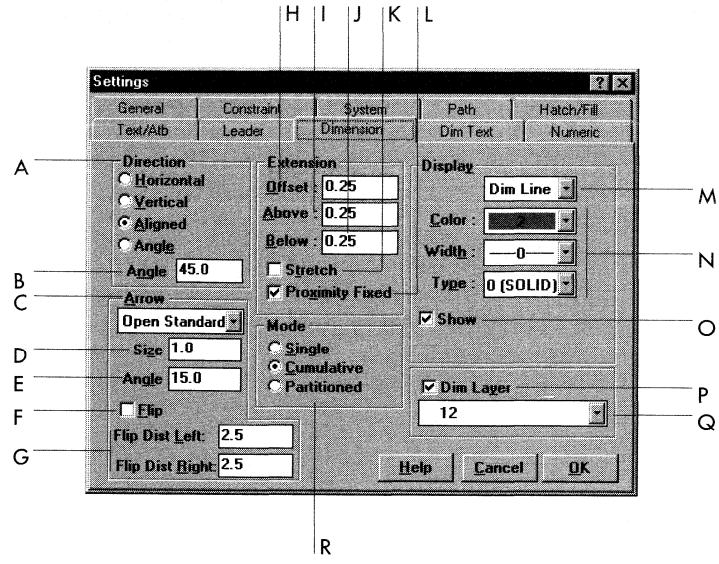
ANATOMY OF A DIMENSION



FOR MORE INFORMATION

DIMENSION AND EXTENSION LINE SETTINGS

Opens when you click the Utilities menu, click Settings, and then click the Dimension tab.



A Specifies the orientation of the dimension line. Use Align to match the dimension line with the angle of the dimensioned object.

B Sets the degree at which to slant an angled dimension line.

C Selects the type of arrowhead to place at the ends of the dimension line.

D Sets the length of the arrowheads.

E Adjusts the shape of the arrowhead or the angle of the slash.

F Flips the dimension line so that the arrows point inward. Use this option when the extension lines interfere with the dimension text.

G Set the length of the dimension lines that extend from the arrows when Flip is checked.

H Adjusts the space between the drawing object and the end of the extension lines.

I Adjusts the distance that the extension line extends past the dimension line and away from the drawing object.

J When Stretch is unchecked, adjusts the distance the extension line extends beyond the dimension line and toward the drawing object.

K When checked, the extension lines stretch to reach the drawing object (minus the Offset distance). When unchecked, the Offset setting is ignored, and the extension lines follow the setting of the Below distance.

L Places the dimension line at a fixed distance (Offset plus Below) from the drawing object.

M Selects the dimension element to adjust. Choosing All will affect all elements of a dimension.

N Set the color, line width, and line type for the selected element.

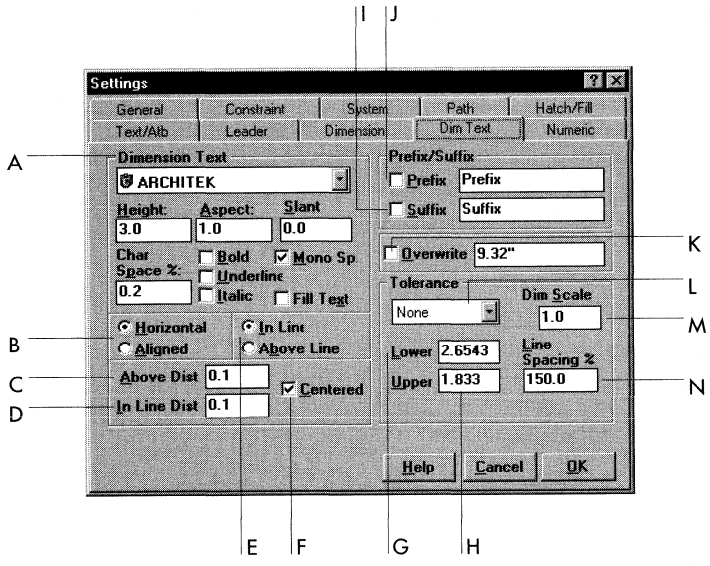
O Hides or displays the selected element.

P When checked, puts all dimensions on the layer specified.

Q Specifies the layer on which to place new dimensions when Dim Layer is checked.

R Creates individual, baseline, or chained dimensions.

DIMENSION TEXT SETTINGS



Opens when you click the Utilities menu, click Settings, and then click the Dim Text tab.

A Sets standard text characteristics for dimensions.

B Allows the dimension text to align with its dimension line or to display horizontally regardless of the line's orientation.

C Sets the distance at which dimension text is placed above the dimension line when Above Line is selected.

D Sets the distance between dimension text and dimension line segments when In Line is selected.

E Determines whether the dimension text appears above the dimension line or in the middle of it.

F When checked, centers dimension text in the dimension line. If unchecked, the dimension line and text are positioned using the same selection point.

G Sets the amount of tolerance below the dimensioned value. This amount is used in displaying tolerance values and is the only value used for the min/max display style.

H Sets the amount of tolerance above the dimensioned value. This amount is used in displaying tolerance values.

I Adds the text in the Suffix box to the end of the dimension text.

J Adds the text in the Prefix box to the beginning of the dimension text.

K Replaces the dimension text with the text you type here.

L Selects the style in which to display dimension tolerances.

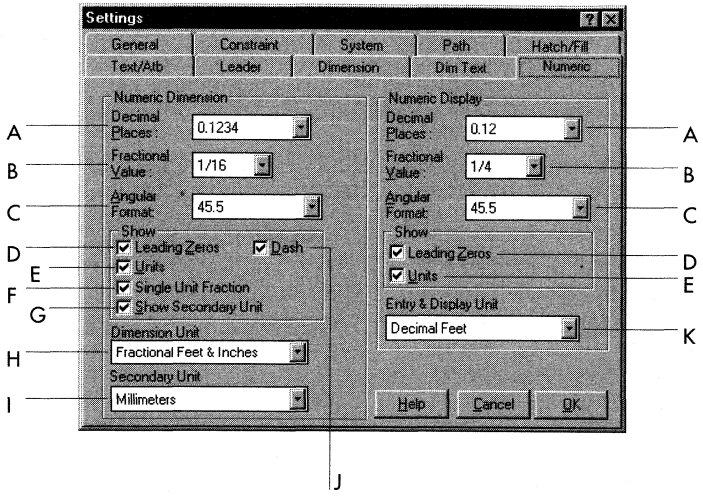
M Sets the factor by which dimension values are scaled from the standard drawing units. You can enlarge part of the drawing as a detail and set the Dim Scale multiplier so that dimensions will display properly.

N Sets the spacing (as a percentage of one line height) between lines of text for stacked tolerance display.

TIP

Snap dimension text back to the dimension line after you have moved it away by using the In Line or Above Line options. Similarly, return text to the center of the line by using the Centered option.

NUMERIC SETTINGS



Opens when you click the Utilities menu, click Settings, and then click the Numeric tab.

Numeric Dimension settings affect the number format of dimensions you create. Numeric Display settings affect the number format displayed on the status bar and used in direct-distance entry.

A Sets the number of digits to show to the right of the decimal in distances and angles.

B Sets the smallest fractional value permitted.

C Lists the format for displaying angles. If you select decimal degrees format, the Decimal Places setting determines the number of digits displayed to the right of the decimal.

D When checked, displays leading zeros in decimal measurements of less than 1 drawing unit (for example, 0.1234).

E When checked, displays the unit of measurement along with the value. If you are using feet and inches, the unit marks always appear.

F When checked, displays fractions as a single character (for example, ¼, ½, ¾). This option is only available when using a vector font for dimension text.

G When checked, displays dimensions in both the primary and the secondary units (H and I).

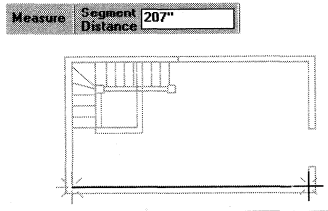
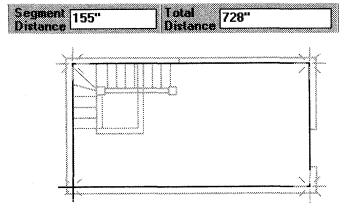
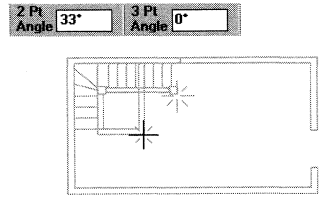
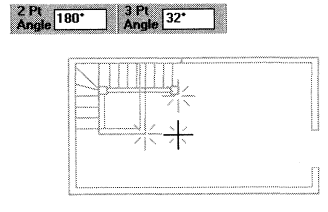
H Sets the primary unit of measurement in which to display dimensions.

I Sets an optional secondary unit of measurement in which to display dual-unit dimensions.

J When checked, displays a dash in dimensions using the feet and inches unit.

K Sets the unit of measurement used in the status bar display and in direct-distance entry.

Making measurements

To...	Do this...	
Measure the distance between two points	Click the Utilities menu, click Measure, and then click Measure Distance & Angles. Select two points, and then read the distance from the speed bar.	
Measure the distance along a perimeter	Click the Utilities menu, click Measure, and then click Measure Distance & Angles. Select points along the perimeter, and then read the distance from the speed bar.	
Measure the angle between two points	Click the Utilities menu, click Measure, and then click Measure Distance & Angles. Select two points defining the angle, and then read the angle from the speed bar.	
Measure the angle between two lines	Click the Utilities menu, click Measure, and then click Measure Distance & Angles. Select three points defining the angle to measure, and then read the angle from the speed bar.	

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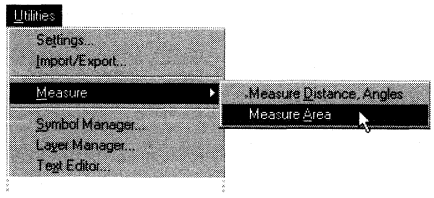
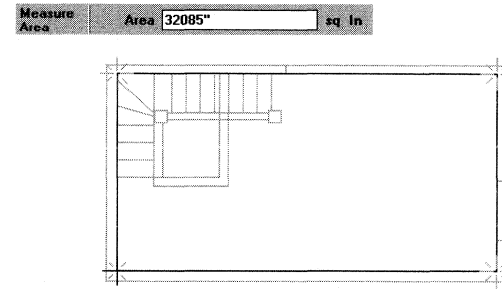
To measure the area of a region

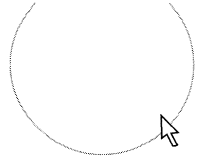


1 Click the Utilities menu, click Measure, and then click Measure Area.

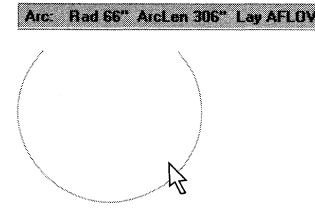


2 Select the points defining the perimeter of the area to measure, and then read the area from the speed bar.

To see basic information about an object



1 Select an object.



2 Place your pointer over the selected object, and then read the information displayed on the status bar.

TIPS

To avoid selecting more points than you want when measuring an area, select objects that define the perimeter of the area, click the Utilities menu, click Measure, and then click the Measure Area command.

To see detailed information regarding an object, select an object, click the Utilities menu, and then click Object Information.

Working with text

Drawings often use text to clarify details and to label graphics. Visual CADD offers an easy-to-use set of tools for creating and modifying text. Although each text tool addresses a different need, you can use each text tool to accomplish most tasks. For example, the text-line tool is designed to quickly enter text one line at a time, but you can use it to enter and edit blocks of text as well.

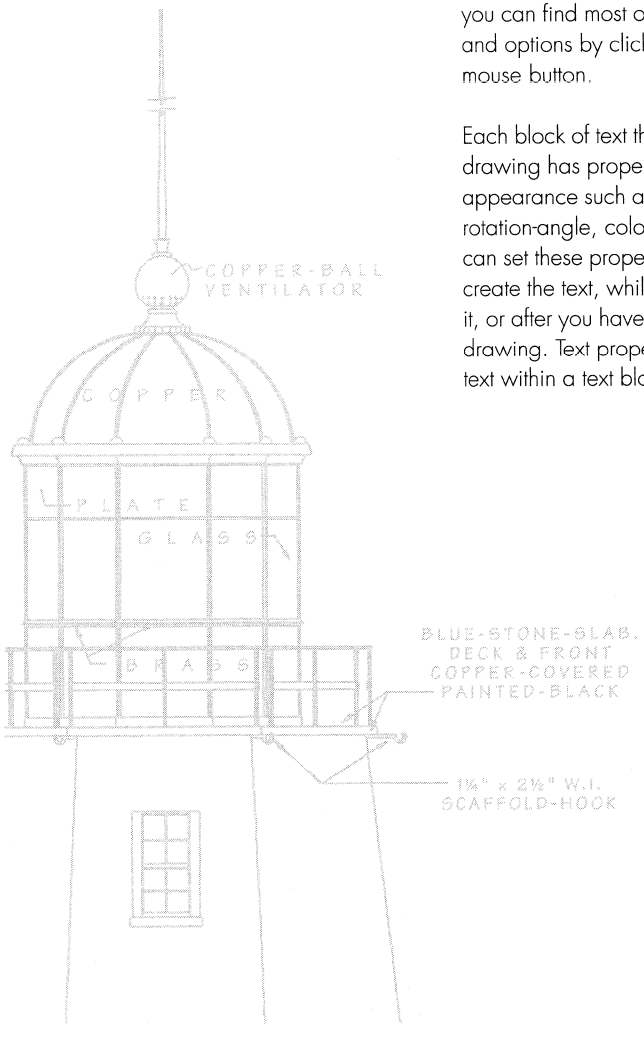
In addition to having more than one tool for a job, you have more than one way to find a tool. While creating text, you can find most of the commands and options by clicking the right mouse button.

Each block of text that you add to a drawing has properties that define its appearance such as font, size, rotation-angle, color, and layer. You can set these properties before you create the text, while you are creating it, or after you have added it to your drawing. Text properties apply to all text within a text block.

Visual CADD comes with a number of fonts that are optimized for plotters and other vector output devices. Although TrueType fonts tend to redraw faster than vector fonts, they are less accurate and can cause problems for some vector output devices.

Some text elements of your drawing may require specialized tools. Dimensions, leaders, and attributes include text, but you use dimensioning and attribute commands rather than text commands to create them.

If you want to use text that was created in another application, refer to "Using the Windows clipboard" in Chapter 9 for more information.



CHAPTER

6

Adding and modifying text 100–105

Choosing which text tool to use depends on how you want to view text as you create or modify it. You have great flexibility in controlling the appearance of text.

Choosing fonts 106–107

Although you can use standard Windows fonts, Visual CADD's vector fonts provide several advantages. You can also use AutoCAD and Generic CADD vector fonts with Visual CADD.

Adding text to a drawing

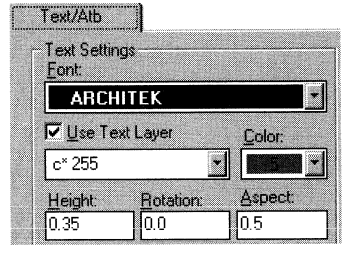
Two tools add text to a Visual CADD drawing: the text-line tool and the text-editor tool. Each tool displays the text in a different way.

- The text-line tool displays text one line at a time in the speed bar.
- The text-editor tool opens a dialog box that displays multiple lines of text along with their associated settings. This tool is good for editing longer text blocks. The text-editor tool also provides options for using text from sources outside Visual CADD.

While using either tool, you can adjust properties at any time. You can also set text properties before using either tool by clicking Settings from the Utilities menu and then making changes on the Text/Atb tab.

WORKING WITH TEXT

Use the text-line tool to open the text-line speed bar where you can change the properties for a block of text that you are creating or editing.



You can set Visual CADD to place all new text blocks on a specific text layer regardless of the active layer. By doing so, it is easier to view or hide the text to make redrawing and printing faster. Click the Utilities menu, click Settings, and then click the Text/Atb tab. Check Use Text Layer to create a text layer.

TIPS

To set values by example for the Height, Rotation, or Slant Angle boxes on the text speed bar, click the appropriate edit box, type **D** for a distance or **A** for an angle, and then click points in your drawing.

To create text containing tab characters, use the text-editor tool.

Locate text precisely by using snaps to select a start point.

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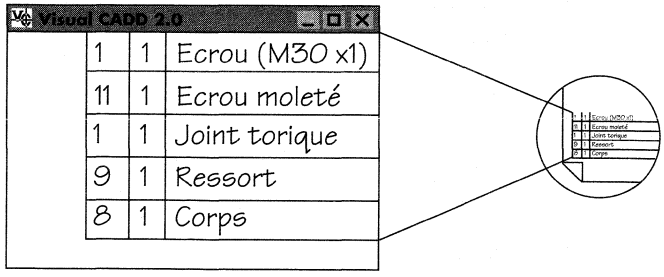
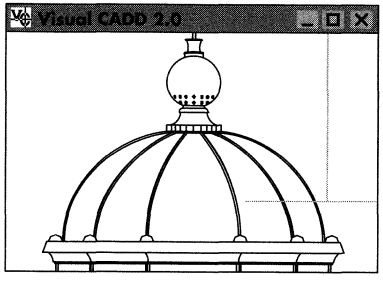
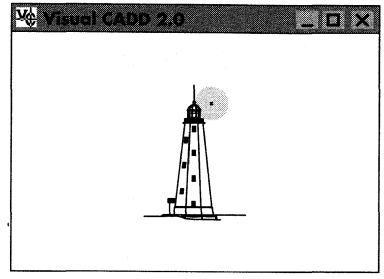
PLACING TEXT IN YOUR DRAWING



Drawing scale and text size Because text size is in real-world units, you will need to adjust it to be legible when plotted at small scales.



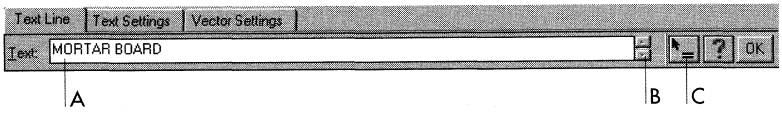
Text (represented by the I-beam) is too large when zoomed in too far.



Text is almost invisible when zoomed out too far.

Cursor size When you select a starting point for your text, the cursor becomes an I-beam. The size of the I-beam represents the actual size of the text. If you zoom in too far, the I-beam is too big to fit on the screen. If you zoom out too far, the I-beam appears as a tiny dot on the screen. Zooming in or out will make the I-beam and the text appear in a more manageable size.

TEXT PROPERTIES ON THE TEXT-LINE SPEED BAR

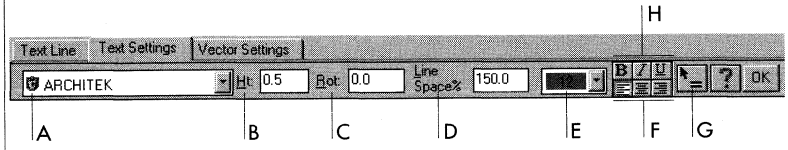


Opens when you click the text-line tool in a drawing. The Text tab contains the actual text of the text block.

A Enter the text to add to the drawing. Press ENTER for a new line, or press ENTER twice to end entry.

B Use the arrows to scroll through multiple lines of text.

C Matches the properties of an existing text block. Click the button, and then click the text block whose properties you want to match.



The Text-Settings tab contains some settings that affect all types of text and other settings that affect only text created using a TrueType font.

A Specifies the font of the current text block.

B Sets the height of the text.

C Adjusts the rotation angle of the text block.

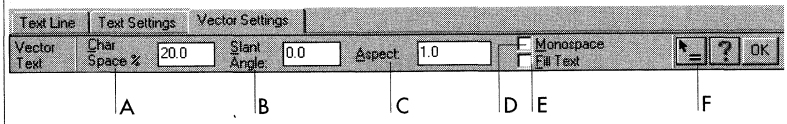
D Sets the line spacing of the text block as a percentage of the text height. For example, 200% yields double-spacing.

E Adjusts the color of the text.

F Adjusts the alignment of the text block to be flush left, centered, or flush right.

H Sets the style of TrueType fonts (bold, italic, underlined).

G Allows you to select an existing text block to which the properties of the newly entered text are matched.



The Vector-Settings tab contains settings that affect only text in a vector font.

A Adjusts the space between vector-font characters. Spacing is expressed as a percentage of character size. A value of 100% creates one full character width between each character in the text block.

B Adjusts the slant angle of vector-font characters. This is similar to italics but can slant backward.

C Adjusts the height-to-width ratio of characters in text blocks created using a vector font. An aspect of 1.0 yields characters that are as tall as they are wide.

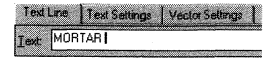
D When checked, all characters are the same width. When unchecked, wide characters (M, W) take up more space than narrow characters (I, L).

E When checked, characters are solid filled. When unchecked, characters are displayed in outline, making both redrawing and printing faster.

F Use to select an existing text block on which to match the properties of newly entered text.

To create a short text block

1 Click the text-line tool, and then click where you want to locate the text.

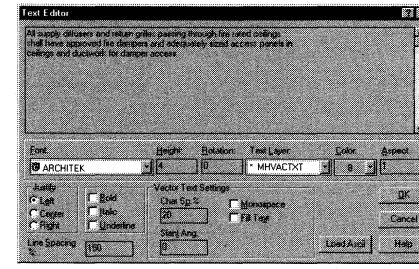


2 On the text speed bar, type the text. Press ENTER to add a new line. Press ENTER twice to end text entry.

Note: The speed bar displays your text one line at a time. Press the arrow keys to move the cursor through the text. An outline of the text block appears on the drawing as you type.

To create a longer text block

1 Click the text-editor tool, and then click where you want to locate the text.



2 In the text-editor dialog box, type the text to be added and adjust properties as needed.

Modifying text

When modifying existing text, you can change the words themselves, the style of the characters, or both. The same tools that you use to create text are also used to modify it.

TIPS

To see editing commands for text, select existing text, and then click the right mouse button.

To see cut, copy, and paste functions inside the text-editor tool, click the right mouse button inside the text-editor dialog box.

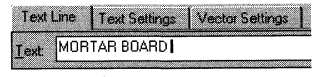
CHOOSING A TOOL TO MODIFY TEXT

The text-line tool modifies both text and properties of a single text block. It displays an outline of the text on the drawing as changes are being made. Text is displayed one line at a time.

Use the text-editor tool to modify single or multiple blocks of text. If a single block is selected, you can use the text-editor tool to change both text and properties. If multiple blocks of text are selected, the text-editor tool can change properties only.

To modify a short text block

MORTAR



1 Select the text block to modify, and then click the text-line tool.

2 In the text speed bar, make changes to your text and its properties.

Note: In the Text edit box, pressing ENTER starts a new line of text. Pressing the arrow keys moves the cursor through the text.

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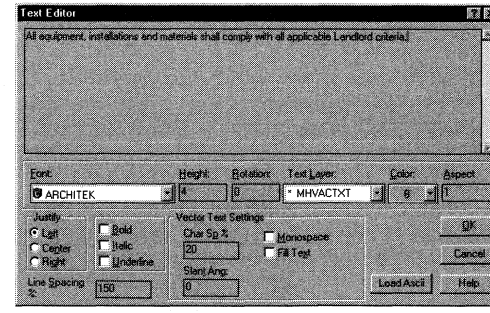
To modify a longer text block

All equipment, installations and materials shall comply with all applicable Landlord criteria.



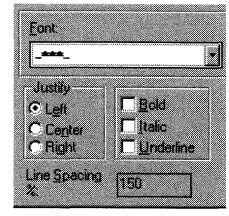
1 Select the text block to modify, and then click the text-editor tool.

2 In the Text Editor, make changes to text and properties.

To change the properties of multiple text blocks

All equipment, installations and materials shall comply with all applicable Landlord

Mount diffusers and grilles in upper grid ceiling in sales area.



1 Select the blocks of text you want to modify, and then click the text-editor tool.

2 Modify the text properties.

Note: The Text Property Change dialog box displays this symbol -***- for properties that vary among the selected text blocks.

Choosing fonts

In Visual CADD 2.0, you can use two types of fonts to create text: standard Windows fonts (TrueType and printer fonts) and vector fonts. Although TrueType fonts tend to redraw more quickly and to appear smoother on the screen, you may choose to use the vector fonts for several reasons.

Vector fonts are accurate. Vector fonts are defined by the same mathematics that define CAD entities. Fitting text within a particular space and aligning it precisely is much easier with vector fonts. With standard Windows fonts, what prints may not match what you see on the screen.

Vector fonts are flexible. You can control the spacing between vector font characters, adjust the fill of the text (solid, transparent, or color), assign a slant angle (simulating italics), and choose between fixed or proportional spacing. These options provide variations not possible with a standard Windows font.

Vector fonts are predictable. Some plotters and other vector output devices that use a shape description language—such as PCL or HP-GL/2—make a reasonably accurate representation of standard Windows fonts. Those using a vector language—such as HP-GL/2—attempt to assign each Windows font to an internal plotter font, which can have unpredictable results. To avoid discrepancies, use vector fonts in your drawing, and the vector ouput device will plot them accurately.

Visual CADD comes with a number of vector fonts. You can add more vector fonts to your system either by purchasing them from a third-party vendor or by translating fonts created for use with AutoCAD or Generic CADD.

TIP

Change the properties of multiple text blocks by selecting them, and then using the text-change tool.

TYPES OF FONTS

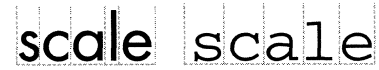
Visual CADD vector fonts

Slant is user defined.

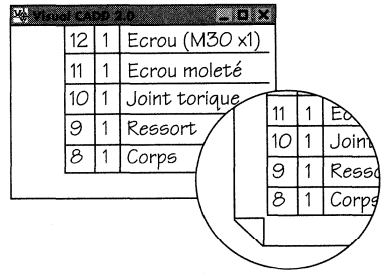
TrueType fonts

Slant is typeface defined.

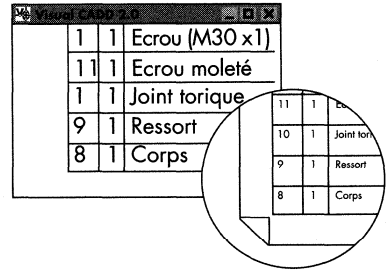
Choose between proportional and fixed spacing.



Proportional and fixed spacing are font dependent.



Print output is accurate and predictable.



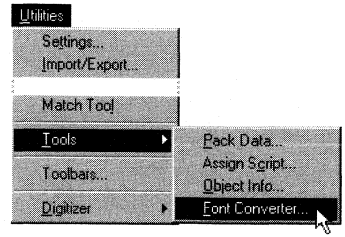
Print output is unpredictable on vector output devices.

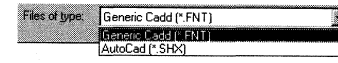
FOR MORE INFORMATION

Converting AutoCAD and Generic CADD fonts

If you have AutoCAD or Generic CADD fonts and want to use them with Visual CADD, you can use the Font Converter to install them. The Font Converter converts AutoCAD font files (.shx) or Generic CADD font files (.fnt), but it cannot convert AutoCAD's big-font files.

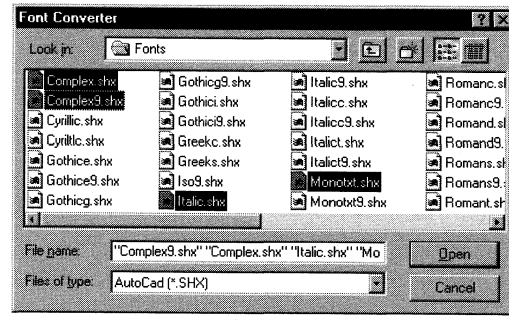
To convert an AutoCAD or Generic CADD vector font





1 Click the Utilities menu, click Tools, and then click Font Converter.

2 From the Files of Type list in the Font Converter dialog box, select the type of file you want to convert.



3 Select the font files to convert, and then click Open.

FOR MORE INFORMATION

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Working with AutoCAD and Generic CADD files 160
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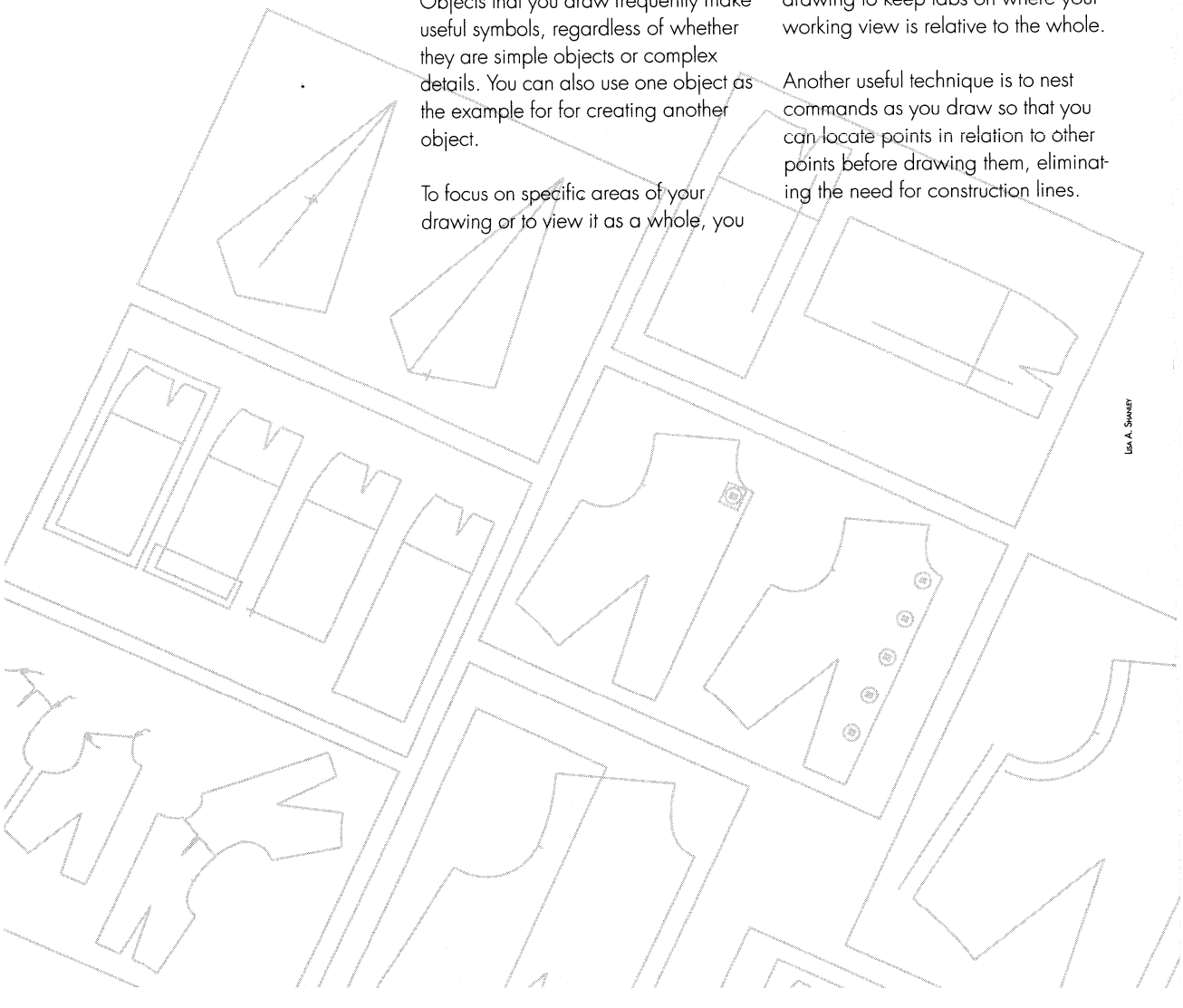
Practical techniques

In conventional drafting you employ a variety of techniques to draw quickly and efficiently. You might use templates, libraries of stock details, and tracing to quickly draw objects and details you've drawn before. Or you might paste copies of complex drawings or long text blocks into a new drawing.

In Visual CADD, you also employ a variety of methods to copy objects, including creating and using symbols. Objects that you draw frequently make useful symbols, regardless of whether they are simple objects or complex details. You can also use one object as the example for for creating another object.

To focus on specific areas of your drawing or to view it as a whole, you can quickly enlarge or reduce the drawing on the screen. You can also open a thumbnail view of the entire drawing to keep tabs on where your working view is relative to the whole.

Another useful technique is to nest commands as you draw so that you can locate points in relation to other points before drawing them, eliminating the need for construction lines.



LISA A. SHANLEY

CHAPTER

7

Making copies of objects 110–115

You have a variety of ways to copy objects. Several methods also arrange the copies as you create them. You can offset objects as well, creating another object that can be an enlarged or reduced copy or completely different from the original.

Working with symbols 116–123

Place symbols that you or others have created to speed the creation of your drawing. You can keep a collection of entities together by creating a symbol from them. Using attributes attached to symbols, you can create a bill of materials and equipment schedules as well.

Streamlining your work 124–135

Draw more quickly and efficiently by using another entity as an example, locating points before you draw them, changing the view of your drawing using zooms, working with multiple open windows, and running Visual CADD at top speed.

Creating copies of an object

Visual CADD provides you with several ways to copy a selected object or objects:

- Copying to and pasting from the Windows clipboard
- Dragging a copy from the original object
- Creating a mirror-image copy

The Cut and Copy commands on the Edit menu place copies of selected objects on the Windows clipboard. When you choose Paste from the Edit menu, Visual CADD copies the object from the clipboard to your drawing. These three commands are best used for copying objects to and from other applications and files.

Placement of an object from the clipboard is imprecise, whereas the copy tools—Mirror, Multiple Copy, Linear Copy, Radial Copy, and Array Copy—enable you to place copies of objects in your drawing precisely.

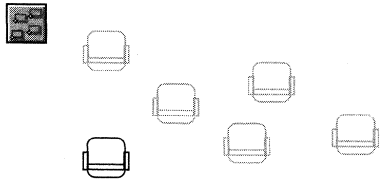
A mirror-image copy is useful when you're drawing an object with axial symmetry, such as the elevation of a classical building or a wine glass.

The other four copy commands—Multiple Copy, Linear Copy, Radial Copy, and Array Copy—can produce as many copies of an object or objects as your computer's memory allows. You provide spacing and angle parameters, and the objects are arranged as you specify.

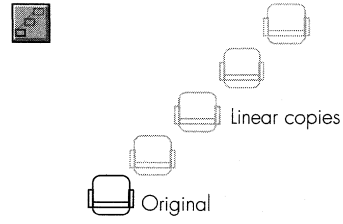
FOR MORE INFORMATION

Selecting and deselecting objects 58
Copying text and objects between Visual CADD and other applications 154

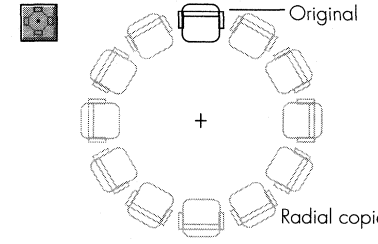
CHOOSING A TECHNIQUE FOR CREATING COPIES OF AN OBJECT



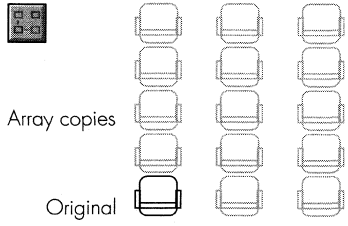
Multiple Copy places duplicates of selected objects wherever you click in your drawing. You continue placing copies of the original until you click Pen Up or press Esc.



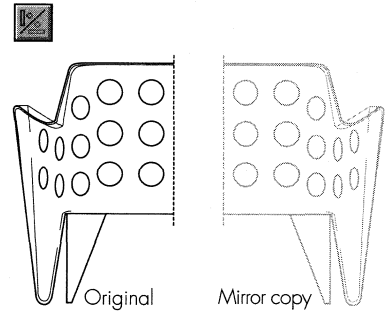
Linear Copy arranges duplicates of selected objects in a line. You set the number of copies you want, define the distance and direction between the original object and the first copy, and then set the distance between each object.



Radial Copy arranges duplicates of selected objects equidistant from a center point you place, rotating each to maintain the same relationship as the original object to the center point. You set the number of copies you want and the angle that the original and its duplicates will span.



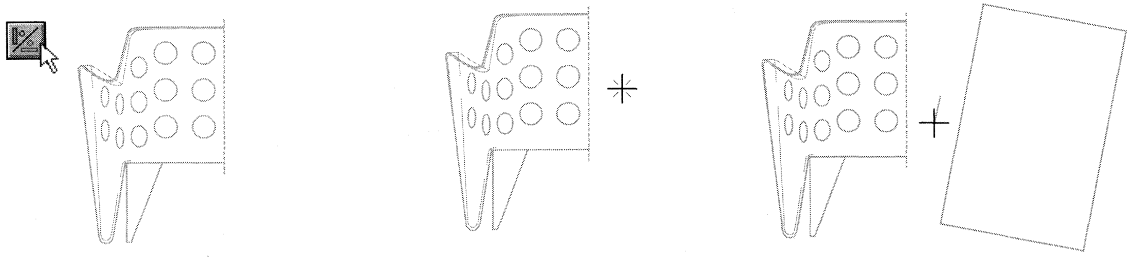
Array Copy arranges duplicates of selected objects in evenly spaced rows, forming a grid. You set the number of copies you want and the number of rows. You define the distance and direction between the original object and the first copy, and then define the distance and direction between rows.



Mirror creates a mirror-image copy of selected objects around an axis you define.

Creating copies of an object

To create a mirror-image copy

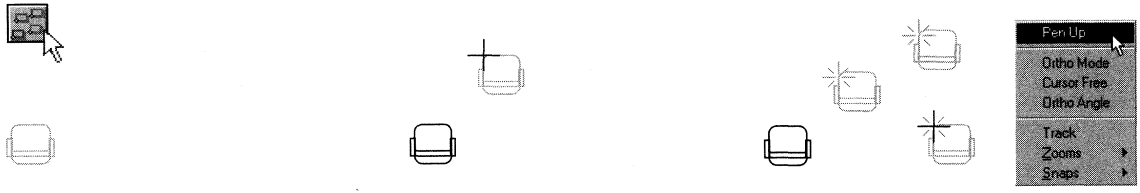


1 Select one or more objects, and then click the mirror tool from the toolbar.

2 Place a reference point defining the first point of the axis around which to mirror the object(s).

3 Place a reference point establishing the second point of the axis and setting the copy's position.

To create multiple copies

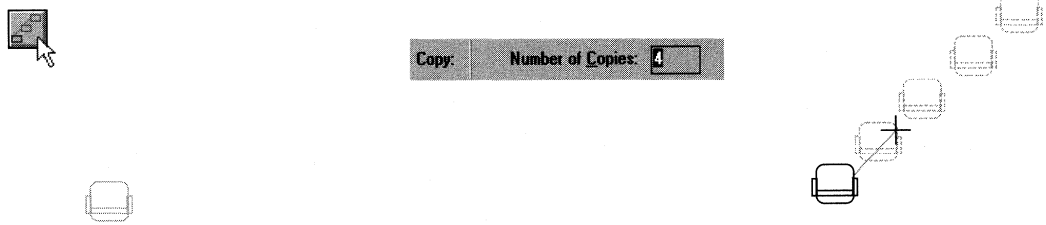


1 Select one or more objects, and then click the multiple-copy tool.

2 Place a reference point on the object(s) to create a handle.

3 Click in your drawing to place each copy. To finish, click the right mouse button and then click Pen Up.

To create a linear copy



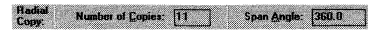
1 Select one or more objects, and then click the linear-copy tool from the toolbar.

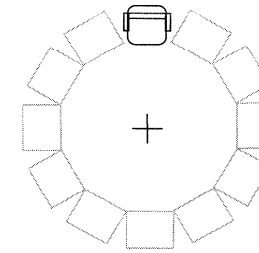
2 On the linear-copy speed bar, enter the number of copies (not including the original object), and then click OK.

3 Place two reference points anywhere in your drawing to establish the distance between each.

To create radial copies







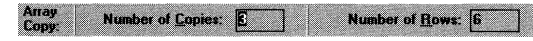
1 Select one or more objects, and then click the radial-copy tool from the toolbar.

2 On the radial-copy speed bar, enter the number of copies (including the original object) and the span angle, and then click OK.

3 Place a reference point anywhere in your drawing to define the center point of the circular array.

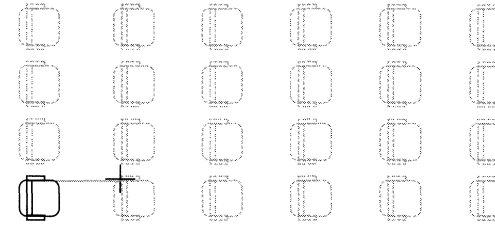
To create an array copy





1 Select one or more objects, and then click the array-copy tool from the toolbar.

2 On the array-copy speed bar, change the number of copies to be in each row (not including the original object) and the number of rows, and then click OK.

3 Place two reference points anywhere in your drawing to define the distance and direction between the original object and the adjacent copy.

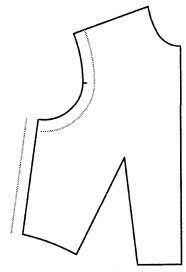
4 Place a reference point to define the distance between rows.

Offsetting objects from an original object

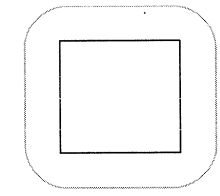
Use the offset tool to create an object that is parallel to and around or inside the selected objects. The offset tool often makes a scaled copy of the original object, but it also frequently results in an object that is quite different from the original.

You can offset a single entity or an object composed of multiple entities. You also can set the distance of the offset copy from the original visually or numerically.

TYPICAL OFFSET RESULTS



Use the offset tool to make scaled copies of many regular objects...



...or to create a variation of an object.

Entities that change type when they are offset

This entity...	Becomes...
Continuous Bézier	Single Bézier curves
Ellipse	Continuous Bézier curve
Elliptical arc	Continuous Bézier curve
Spline curve	Single Bézier curves

FOR MORE INFORMATION

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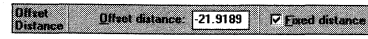
To offset an object



1 Select an object, and then click the offset tool.

2 To preview more than two offset options, make sure Fixed Distance is unchecked. To specify a precise offset, enter a value in the Offset Distance box, check Fixed Distance, and then click OK.

3 If Fixed Distance is unchecked, move the cursor inside and outside the original object to preview the offset options, and then click when you see the option you want.

Using symbols to keep drawing elements together

Symbols are objects that you can insert repeatedly in a drawing, which saves you from drawing the same objects over and over. You can add new symbols that you create to Visual CADD's symbol libraries, modify existing symbols, and create or purchase new symbol libraries.

Using symbols in your drawings gives you several advantages.

- Repeated use of the same symbol reduces the file size of your drawing because each symbol is defined only once in the drawing database, even when you insert it at different sizes and at different angles.
- A project team can work with a single library of symbols to standardize drafting conventions.
- In one step, you can delete all occurrences of a symbol in your drawing, or replace all occurrences of one symbol with another.
- You can attach attributes to a symbol (for example, manufacturer, model number, and size of equipment).

You can apply the current color to a symbol as you place it, but you cannot change the line type or line width. You can explode a symbol into its components, however, even as you place it. You then can edit the components, but you cannot attach an attribute to an exploded symbol. Exploded symbols also enlarge the drawing file.

Before placing symbols in your drawing, you must load them from their libraries. You can use loaded symbols in any drawing until you exit Visual CADD. Visual CADD saves symbols into the Symbols folder by default.

TIPS

Locate an exploded symbol on the layer on which the symbol was placed, on the layer that is current when you explode it, or on the layers where the entities were located at the time they were saved as a symbol by clicking the Utilities menu, clicking Settings, clicking the General tab, and then checking the corresponding option.

To snap the cursor to any point or entity within a symbol, click Settings on the Utilities menu, click the General tab, and then make sure Symbol Snap is checked.

Quickly place a copy of a symbol already placed in your drawing by clicking the Match tool, clicking the symbol in your drawing, and then placing the copy.

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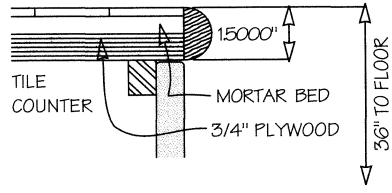
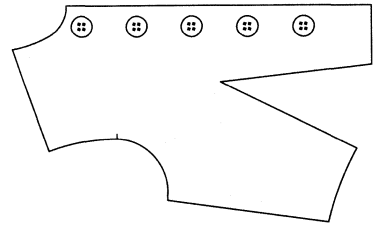
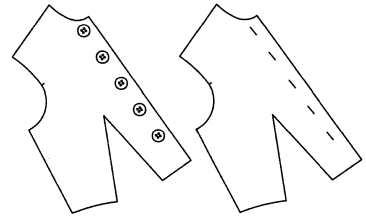
USING SYMBOLS IN YOUR DRAWING



Use a symbol as a company-standard detail.

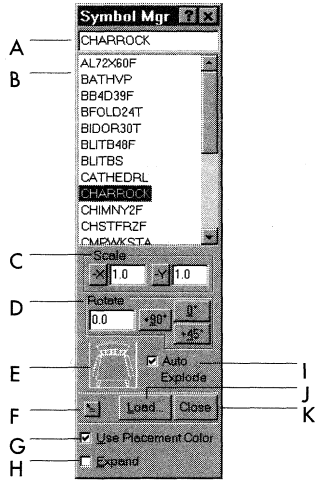


Use a symbol to group a collection of objects drawn frequently.



Use a symbol to search for and replace recurring objects.

SYMBOL MANAGER OPTIONS



Opens when you click the Utilities menu and then click Symbol Manager.

A Displays the selected symbol's name.

B Lists the symbols that are loaded in your drawing.

C Sets the size of the symbol when you place it. A value of 1.0 is equivalent to full size, or 100%.

D Sets the symbol's angle of rotation. Enter a value in the text box, or click buttons to add 45° or 90° to the current angle, or reset the angle to 0°. Positive values rotate symbols counterclockwise. Negative values rotate symbols clockwise.

E Previews the selected symbol.

F Sets the angle and properties of the selected symbol to match those of the next object you click in your drawing.

G When checked, applies the current color to the symbol you are placing.

H Sets the Symbol Manager to display only the list of symbols. Click the right mouse button to view all Symbol Manager options.

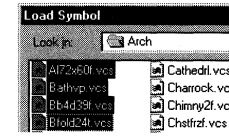
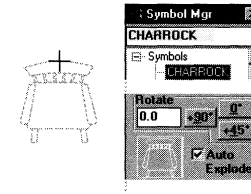
I Explodes symbols into their basic components upon placement.

J Opens the Load Symbol dialog box.

K Closes Symbol Manager.

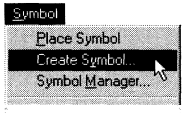
Note: Double-clicking the Symbol Manager title bar rolls it up or down.

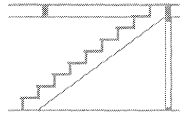
Symbol basics

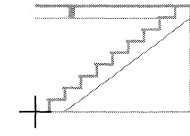
To...	Do this...	
Load symbols into your drawing	In Symbol Manager, click Load, click the symbols you want to load, and then click OK.	
Place a symbol in your drawing	In Symbol Manager, click a symbol from the List, change the scale and rotation angle if needed, and then place the symbol handle point in your drawing using any method for placing points.	

Creating and working with symbols

To create a symbol





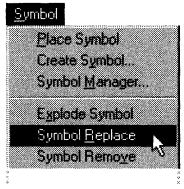


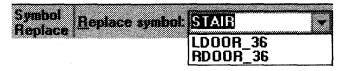
1 Select the objects with which you want to form a symbol, click the Symbol menu, and then click Create Symbol.

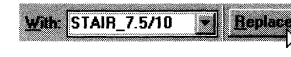
2 Enter a name for the new symbol in the Symbol Name text box.

3 Place a handle point for the symbol where you want to position the symbol.

To replace all occurrences of one symbol with another symbol







1 Click the Symbol menu, and then click Symbol Replace.

2 On the symbol-replace speed bar, select the symbol you want to replace from the Replace Symbol list.

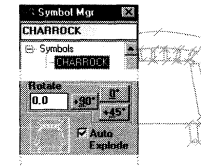
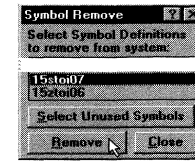
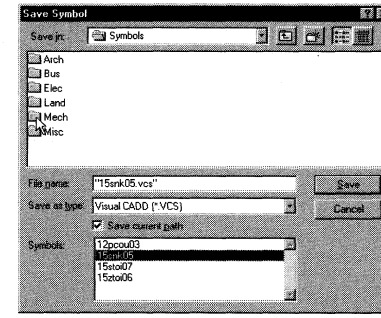
3 Select the name of the symbol with which you want to replace it from the With list, and then click Replace.

TIPS

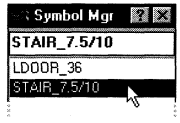
Replace only certain occurrences of a symbol by selecting each copy of the symbol you want to replace before clicking Replace on the symbol-replace speed bar.

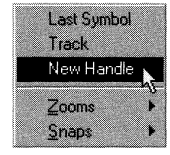
For quick object information about a symbol in the status bar, select the symbol, and then move your cursor over it.

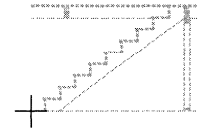
If you can only snap to a symbol's handle point but want to snap to another point within the symbol, check Symbol Snap on the General tab of the Settings dialog box.

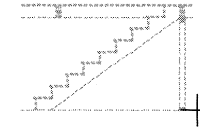
To...	Do this...	
Explode a symbol into its basic components upon placement	In Symbol Manager, select a symbol, check Auto Explode, and then place the symbol.	
Remove all occurrences of a symbol from all open drawings	Click the Symbol menu, click Symbol Remove, select a symbol from the list, and then click Remove.	
Save a symbol to disk for use in other drawings	Click the Symbol menu, click Save Symbol, select the symbol from the Symbols list that you want to save, and then select a folder to store it in.	

To place a symbol with a temporary handle









1 In Symbol Manager, click a symbol name from the list, changing scale and rotation angle if needed.

2 Click the right mouse button, click New Handle, and then place the symbol temporarily in your drawing.

3 Click to set the location of the new handle, and then place the symbol using the temporary handle.

Creating and attaching attributes to symbols

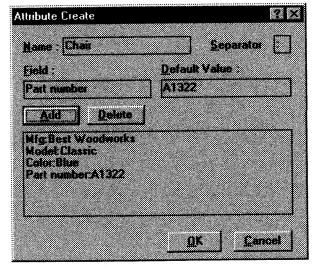
An attribute is textual information relating to a symbol. This information can be the model number, price, color, and manufacturer of the fitting, fixture, or equipment that the symbol represents. Only symbols can have attributes. You can display or hide attributes in the drawing.

You must create an attribute within the current drawing or load it into the drawing before you can attach it to a symbol. You can locate an attribute anywhere in your drawing. You can attach an attribute to one symbol in your drawing, or you can embed it in the symbol definition, so that it attaches automatically each time you place the symbol.

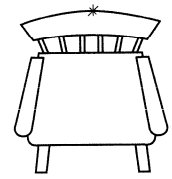
TIP

Delete an attribute field and default value when creating an attribute by selecting the field and then clicking Delete.

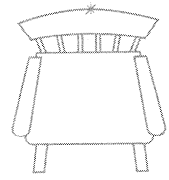
ATTRIBUTES IN YOUR DRAWING



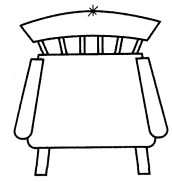
Parts of an attribute Each attribute can contain up to 128 lines of text. Each line consists of a field, a punctuation separator, and a value. Each field and value can contain up to 80 characters. Enter text in each edit box, and then add it to the preview below. You can edit an attribute at any time.

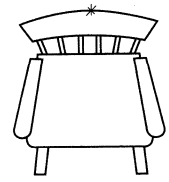


Displaying attributes Attributes are visible in your drawing when Display Attributes is checked on the Text/Atb tab of the Settings dialog box. Labels are visible in your drawing when both Display Attributes and Display Labels are checked on the Text/Atb tab of the Settings dialog box.



Positioning attributes Once you have attached an attribute to a symbol, it maintains its position relative to the symbol, even as you move the symbol, unless you move the attribute separately by clicking the Modify menu and then clicking Move Point.





Changing the properties of attributes When you change the font of attributes on the Text/Atb tab of the Settings dialog box, the change applies to all attributes in your drawing. Changes made to other properties apply only to those attributes placed after you have made the changes.

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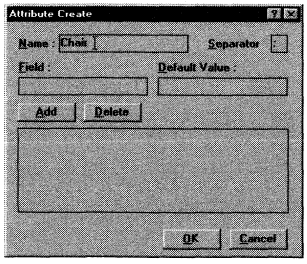
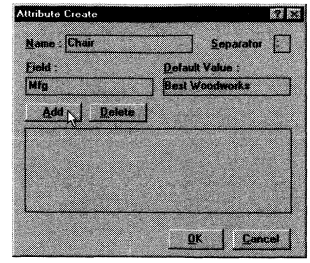
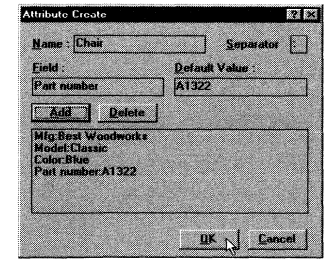
To create an attribute



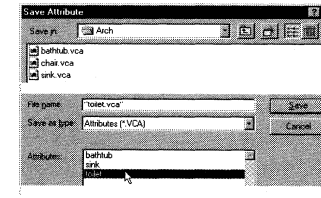
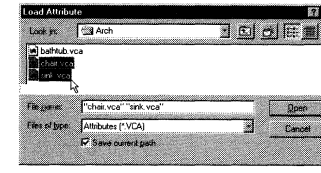
1 Click the Symbols menu, click Attributes, click Create Attribute, type an attribute name in the Name text box, and then type in a separator, if necessary.



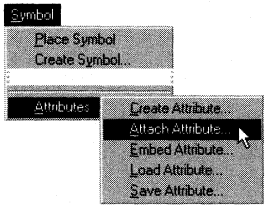
2 Type the name or label in the Field edit box, type a value in the Default Value edit box, and then click Add.

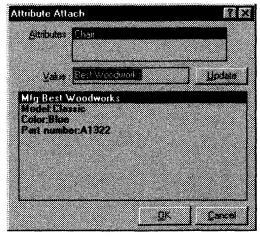


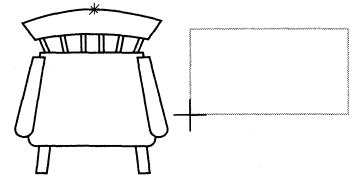
3 Continue adding fields and default values to the attribute, and then click OK.

To...	Do this...	
Save an attribute or attributes for use in other drawings	Click the Symbols menu, click Attributes, click Save Attribute, select an attribute file or files, locate the folder in which you want to save it, and then click OK.	
Load an attribute or attributes	Click the Symbols menu, click Attributes, click Load Attribute, select an attribute file or files, and then click OK.	

To attach an attribute to one occurrence of a symbol





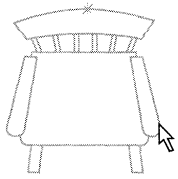


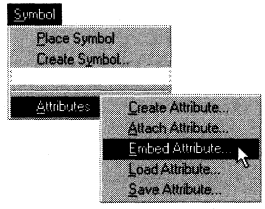
1 Select a symbol in your drawing, click the Symbol menu, click Attributes, and then click Attach Attributes.

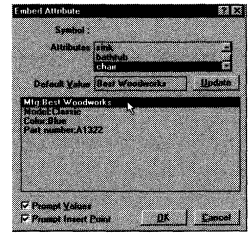
2 Select an attribute from the Attributes list. If necessary, make changes in the Values text box, and then click Update.

3 Click OK, and then place the attribute in your drawing.

To attach an attribute to a symbol definition



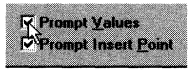


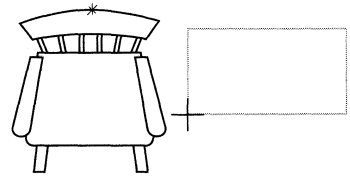


1 In your drawing, select the symbol you want to attach the attribute to.

2 Click the Symbol menu, click Attributes, and then click Embed Attribute.

3 Select an attribute from the Attributes list. If necessary, make changes in the Values text box, and then click Update.

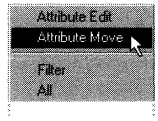


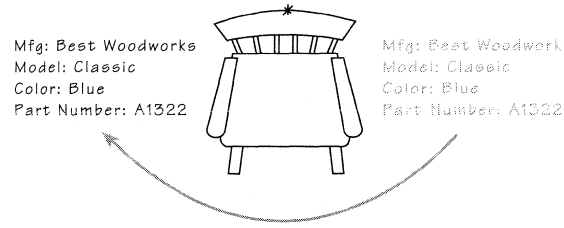


4 Check Prompt Values to be able to edit attribute values each time you place the symbol. Check Prompt Insert Point to manually position the attribute each time you place the symbol. Click OK.

5 Place the attribute in your drawing to set its default location, click the Symbol menu, click Save Symbol, enter a name for the symbol, and then select a folder to save it in.

To move an attribute after it is attached to a symbol

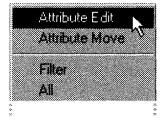


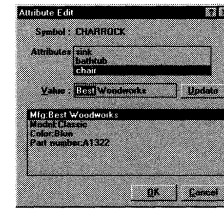


1 Click the attribute's symbol, click the right mouse button, and then click Attribute Move.

2 Click the attribute you want to move, and then drag it to a new location.

To edit an attribute after you have placed it





1 Select the attribute's symbol, click the right mouse button, and then click Attribute Edit.

2 Select a tag in the attribute display box, select the text you want to change in the Values edit box, and then make changes.

Setting properties and drawing tools by example

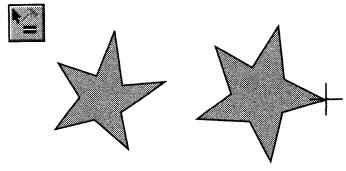
When you want to draw an object with the same properties as an existing object, you can use the existing object as an example, and set the drawing tool and properties in one step.

This is an easy way to set properties for all subsequent objects you draw. For example, if you select a line, you can use the match-entity tool to set the current layer and line color, type, and width. If you select a dimension, use the match-entity tool to make all dimension settings, including arrow types and styles, offsets, text size, aspect, and spacing.

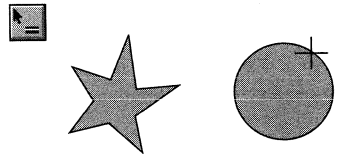
You also can use the match-entity tool to change all or selected properties for existing entities and to set an angle for the next object you draw.

Note: If you match the color, line type, or line width of an object whose properties are set to LP (layer properties), the properties may appear to be different than those of the object you match, depending on the layer properties of the new layer.

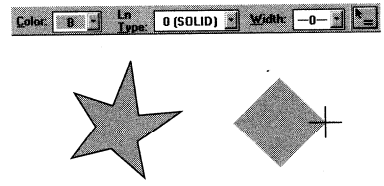
MATCHING PROPERTIES AND TOOLS



The match-tool tool sets the drawing tool and properties of the next entity you draw based on an entity you select.



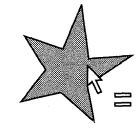
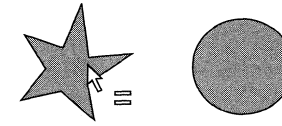
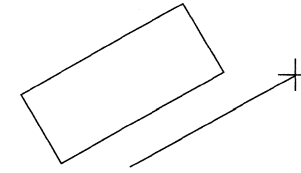
The match-entity tool sets the properties of the next entity you draw based on an entity you select.



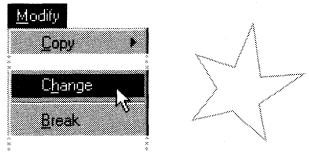
The match-entity tool on a speed bar sets the selected property of the entity you are drawing based on an entity you select.

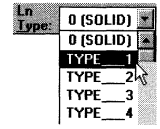
When you click the match-tool tool or match-entity tool, the cursor appears with an equals sign (=) until you click the entity you want to match.

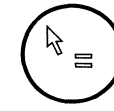
FOR MORE INFORMATION

To...	Do this...	
Set all properties of the entities you are about to draw to match another entity's properties	Click a drawing tool, click the Utilities menu, click Match Entity, and then click an entity whose properties you want to match.	
Change all properties of an existing entity or entities to match another entity's properties	Click the entity whose properties you want to change, click the Modify menu, click Change, click the match-entity tool, and then click an entity or entities whose properties you want to match.	
Use the angle of a line as an example for the line you are drawing	While drawing, click the Snaps menu, click Ortho Angle, click the match-entity tool on the ortho-angle speed bar, click the object whose angle you want to match, and then continue drawing.	

To reset one property of an entity to match another entity's property







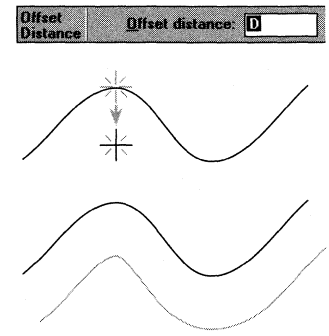
1. Select an object you want to change, click the Modify menu, and then click Change.
2. On the change speed bar, click the match-entity tool, and then click the property box you want to change in the speed bar.
3. Click an entity in your drawing whose property you want to match. You can reset additional properties you want to match if needed, and then click OK.

Setting distances and angles quickly

When you want to use an example to enter or change a setting for a length, height, or angle in a speed bar, but the object with the desired length, height, or angle doesn't exist in your drawing, you can use distance, angle, and vertex input options. You can only use distance, angle, and vertex input options within other commands, settings, and any distance or angle text boxes.

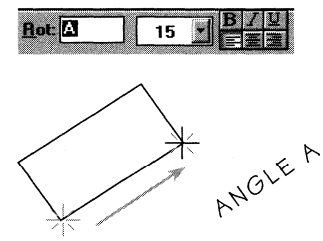
To enter a distance by creating an example

On a speed bar, click the distance, height, or length text box, type D, and then place two points in your drawing that establish the distance you want.



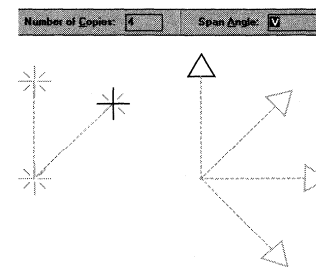
To enter an angle by creating an example

On a speed bar, click the rotation-angle text box, type A, and then place two points in your drawing that establish the angle you want.



To enter a vertex angle by creating an example

On the radial-copy speed bar, click the Span Angle text box, type V, and then place three points in your drawing that establish the vertex angle you want.



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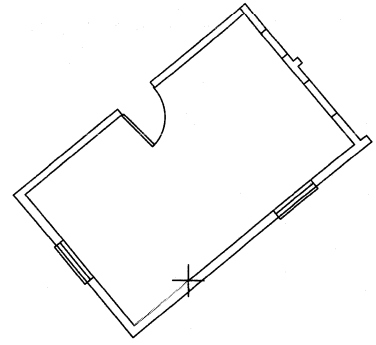
Finding coordinates before placing points

Frequently while drawing, you might start constructing an object but don't know the precise location of the second or third point you need to place. Rather than cancel the operation, you can nest a command that enables you to interrupt the action temporarily until you locate the point, and then continue drawing.

Tracking enables you to find the location of points before you place them. You can begin tracking at any time, whether or not you have placed any points.

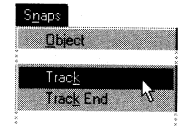
As soon as you click Track from the Snaps menu, you draw lines that will vanish when you use the Track End or Pen Up commands, leaving the cursor in a location ready to place the next point in your drawing. When tracking, you can use any coordinate system, drawing constraints such as ortho mode, and snaps to aid you in finding a location in your drawing.

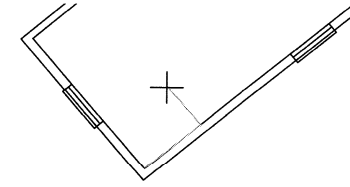
USING TRACKING TO LOCATE A POINT



Tracking is useful for finding the precise location where you want to break a wall to insert a window or door.

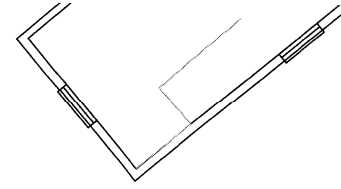
To locate a point in your drawing using tracking

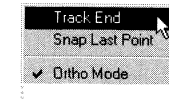




1 While drawing, click the Snaps menu, and then click Track.

2 Begin "drawing" from a known coordinate in your drawing, using snaps, constraints, and any other drawing tools as aids to locate the point.





3 When you locate the point, click the right mouse button, and then click Track End.

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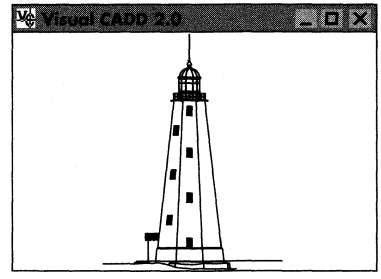
Viewing different areas of your drawing

Because of Visual CADD's limitless drawing area and real-world scale of drawing, you often need to view an area of your drawing that is not on your screen.

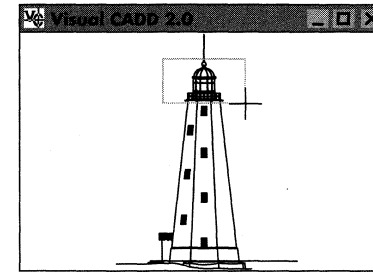
You have several options from which to choose on the Zooms menu for changing the view of your drawing:

- Zooming in and out to see more or less detail.
- Bird's-eye view, to open a thumbnail view of your entire drawing. You can leave Bird's-eye view open on your screen while you zoom in on a specific area.

THE ZOOM OPTIONS FOR VIEWING YOUR DRAWING



Zoom All fills the screen with the entire drawing. Zoom All is useful when you want an overview of your work and also to see remote objects you may have neglected.

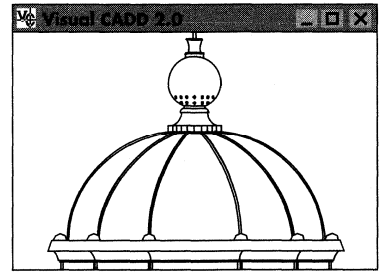


Zoom Window fills the screen with the objects contained in a selection frame you draw after choosing Zoom Window.

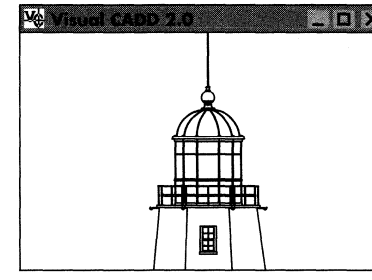
TIPS

Turn scroll bars on and off by clicking the right mouse button on the gray area of the speed bar and then clicking Scroll Bars.

With scroll bars turned off, move quickly around your drawing by pressing CTRL and an arrow key.



Zoom In enlarges the view by a multiplier equal to the Zoom In Factor, which you set on the System tab of the Settings dialog box. You can also set Visual CADD to prompt you to click the center of the new zoom view in your current view. If you do not, the new zoom view uses the same center of view as the current view.



Zoom Out reduces the view by the reciprocal of the Zoom In Factor, which you set on the System tab of the Settings dialog box. You can also set Visual CADD to prompt you to click in your current view to locate the center of the new zoom view. If you do not set the prompt, the new zoom view uses the same center of view as the previous view.

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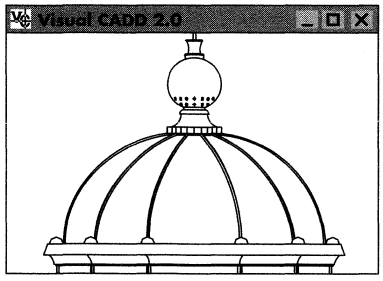
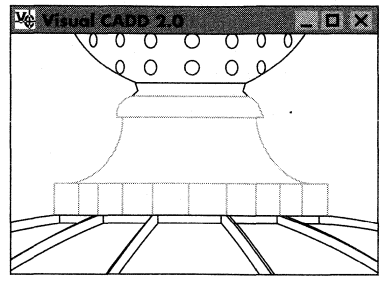
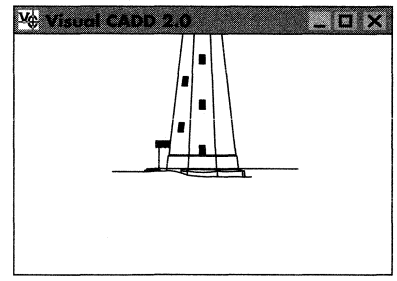


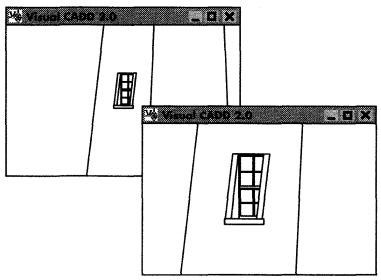
Zoom Previous returns to the previous view. Zoom Previous is useful when you are editing at a close-in view and want to return to an overview or adjacent view.



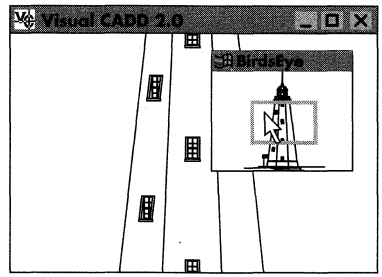
Zoom Selected fills the screen with the objects you have selected before clicking Zoom Selected from the Zooms menu. Zoom Selected is useful for focusing on specific objects.



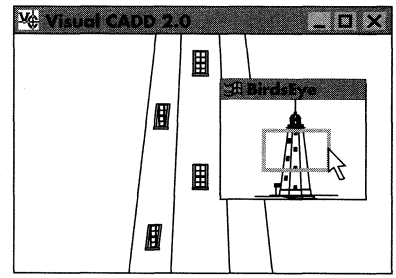
Zoom Pan centers the view around a point you locate after you click Pan from the Zooms menu. Pan does not change the magnification and is useful for viewing portions of your drawing that lie beyond your current view.



Zoom Value enlarges or reduces the view by a ratio of the actual size of the objects. You set the ratio on the zoom-value speed bar, and then locate the new center of the window. A value of 2, for example, displays objects at twice their size.



Bird's-eye view opens a small window displaying a thumbnail of your drawing. Click in the bird's-eye view window to center the new drawing view around that point. Click the right mouse button for other options.

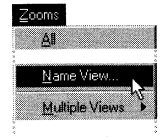


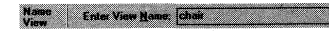
In the bird's-eye view window, draw a selection frame to fill the drawing area with the objects contained in the selection frame. You can drag the selection frame to a new location and resize it to enclose and view a different set of objects. Hold down SHIFT as you drag to draw a selection frame within the existing selection frame.

Naming views for easy reference

When you find yourself returning to the same view repeatedly, you can name the view so that you can select it from a list of view options.

To name the current view for later use

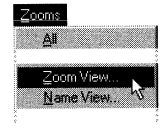


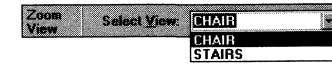


1 Click the Zooms menu, and then click Name View.

2 Enter a name in the text box, and then click OK.

To use a named view

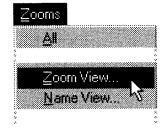




1 Click the Zooms menu, and then click Zoom View.

2 Select a named view from the list, and then click OK.

To delete a named view



1 Click the Zooms menu, and then click Zoom View.

2 Select a named view from the list, and then click Remove View.

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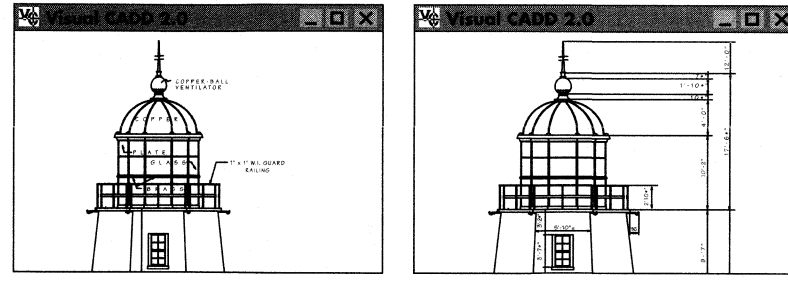
Viewing different areas of your drawing 128

Working on a drawing in multiple windows

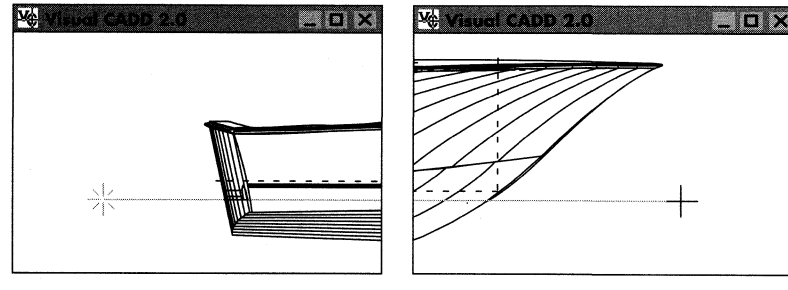
In Visual CADD, you can open multiple windows to work simultaneously with several views of your current drawing. You can open up to 64 windows, depending on the memory in your computer.

When you save a drawing, you also save the number and arrangement of windows as well as the view each window displays, including layer information. When you open the drawing file again it opens to the same set of windows.

USING MULTIPLE WINDOWS



Display different layer information in multiple windows.



Zoom in to see enough detail to snap points precisely to each of the two opposite ends of a drawing.

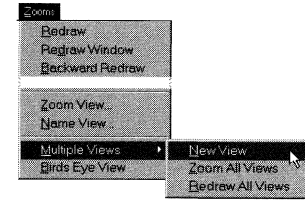
TIPS

To redraw objects in all windows at once, hold down SHIFT, click the Zooms menu, click Multiple Views, and then click Redraw All Views.

To arrange windows on your screen, click the Window menu, and then click Tile Vertical or Tile Horizontal.

To open a new window displaying the active drawing

Click the Zooms menu, click Multiple Views, and then click New View.



FOR MORE INFORMATION

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Drawing entities precisely with snaps 50
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Using a digitizer

With a digitizer, you can create drawings in Visual CADD based on pre-existing hand-drafted ones. You create the drawing by attaching the paper copy to your digitizer and tracing its geometry. For Visual CADD to read the input from your digitizer, you must first align your drawing.

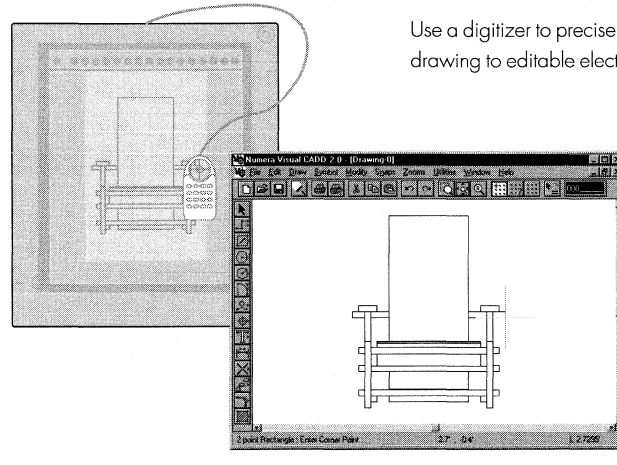
Your digitizer can be used as a normal Windows pointing device when Visual CADD is in mouse mode. When in tablet mode, however, you cannot point to screen menus or buttons with the puck. Instead, you must use digitizer buttons, function keys, keyboard commands, and scripts. Tablet mode only affects the digitizer's functioning in Visual CADD, not in other applications.

In addition to activating tablet mode, you must specify the relationships between the paper and the electronic drawings when using a digitizer.

You can adjust the digitizer scale by overriding the scale calculated by the Align Drawing command with a value of your own. For example, the Align Drawing command might calculate a scale of 47.65:1. If you assume that the actual scale is 48:1 (¼" = 1'-0"), you can override the calculated value.

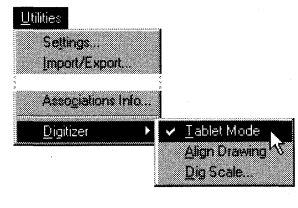
You must have a Wintab-compliant digitizer with a Wintab32 driver to use Visual CADD's tablet mode functions. Your digitizer manufacturer can supply the appropriate driver.

DIGITIZING A DRAWING

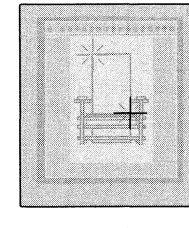


Use a digitizer to precisely transfer a paper drawing to editable electronic form.

To digitize points from a paper drawing



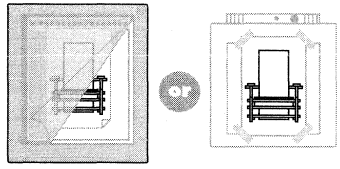
1 Click the Utilities menu, click Digitizer, and then click Tablet Mode.



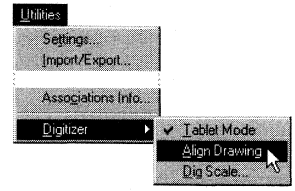
2 After using the Align Drawing or Digitizer Scale command, click points on the digitizer to draw scaled entities in Visual CADD.

FOR MORE INFORMATION

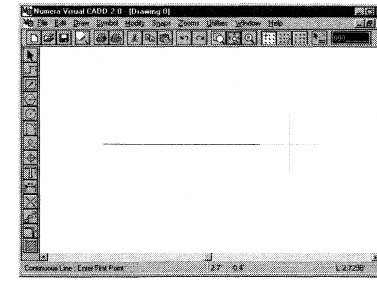
To align a drawing for digitizing



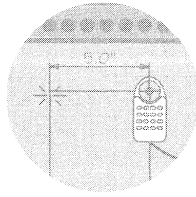
1 Securely attach your paper drawing to the digitizer.



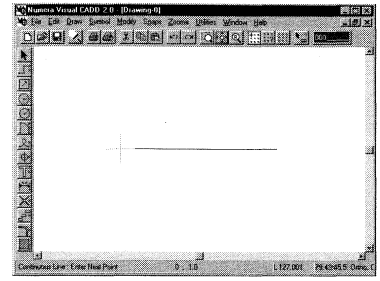
2 Click the Utilities menu, click Digitizer, and then click Align Drawing.



3 Draw a line that represents a line on the paper whose length and angle you know.



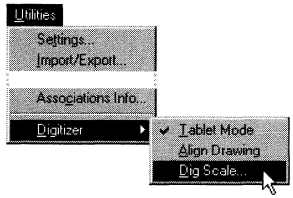
4 Align the digitizer crosshairs at each end of the line on the paper and click at the endpoints, noting the dimensioned length of the line.



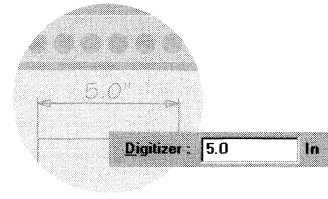
5 In Visual CADD, move the cursor to one end of the line, type **NP**, move the cursor to the opposite end of the line, and then type **NP** again.

6 Confirm the displayed scale value or type a new value on the trace scale speed bar.

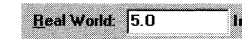
To adjust the digitizing scale



1 Click the Utilities menu, click Digitizer, and then click Dig Scale.

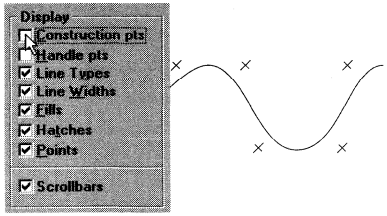


2 Enter a reference distance as measured on the surface of the paper.

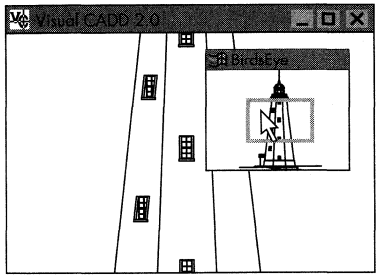


3 Enter the real-world distance represented by the previous measurement.

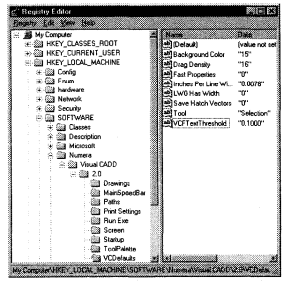
Running Visual CADD at top speed



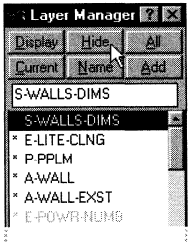
Turn off unnecessary display elements. Construction points, line types and widths, handles, fills, hatches, and fonts containing fills are all elements that must be redrawn. You can hide these elements on the System tab of the Settings dialog box. Turning off the display of attributes on the Text/Atb tab of the Settings dialog box can also speed redraw time.



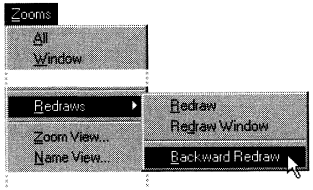
Work at a close-in view, and change views using bird's-eye view. Because Visual CADD searches through only the entities on the screen when you snap to a point or an object, the greater the magnification of your view, the faster Visual CADD will snap to a point. Draw a selection frame in the Bird's-Eye View window to select the view and level of magnification in one step.



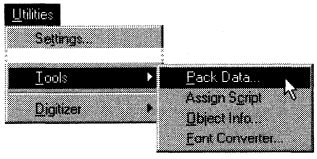
Raise the zoomed-out threshhold at which text displays as a line. When you zoom out, small text will display as a line, giving Visual CADD fewer entities to redraw. To change the text display threshold, click the Start menu, click Run, and then type RegEdit. Change VCFTextThreshold in the VCDefaults folder to a larger number (representing on-screen inches).



Hide layers containing objects you don't need to see. Only display the layers on which you need to work so that Visual CADD doesn't redraw unused objects.



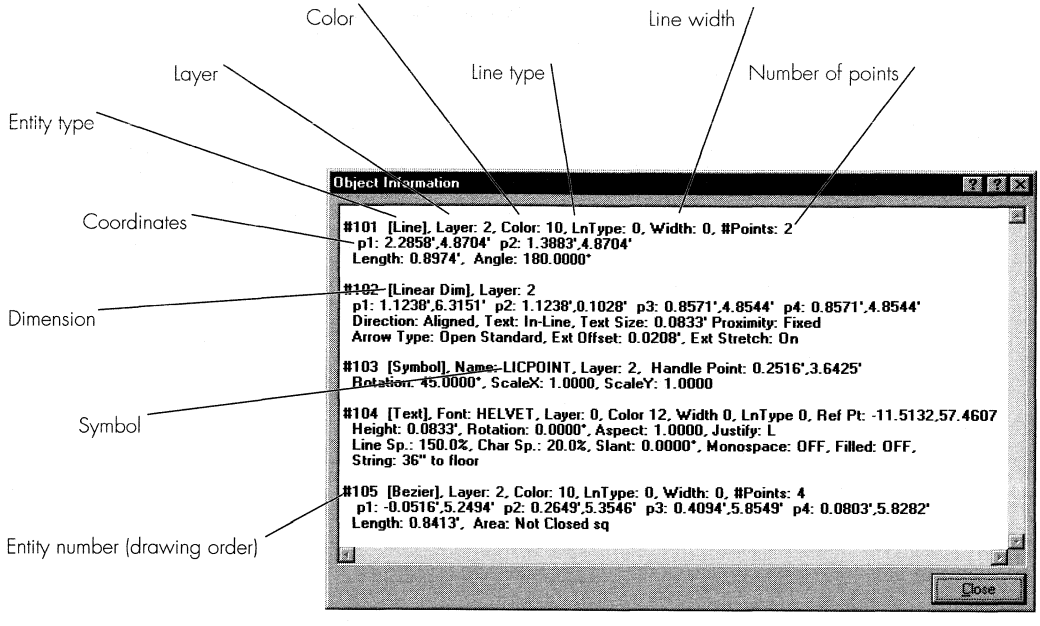
Stop redraw after Visual CADD redraws what you want to see. Shorten the time it takes to zoom in on the most recently drawn entities in your drawing by clicking the Zooms menu and then clicking Backward Redraw. When you next choose a zoom command, wait until what you want appears on the screen, and then press Esc to stop the redraw.



Use the Pack Data command periodically to delete all record of entities that have changed, freeing memory and restoring optimal system performance. Once you click Pack data on the Utilities menu, however, you cannot click the Undo command on the Edit menu to retrieve earlier iterations of your drawing.

FOR MORE INFORMATION

VIEWING A TEXT FILE OF OBJECT PROPERTIES



The Object Information window opens when you click the Utilities menu, and then click Object Info.

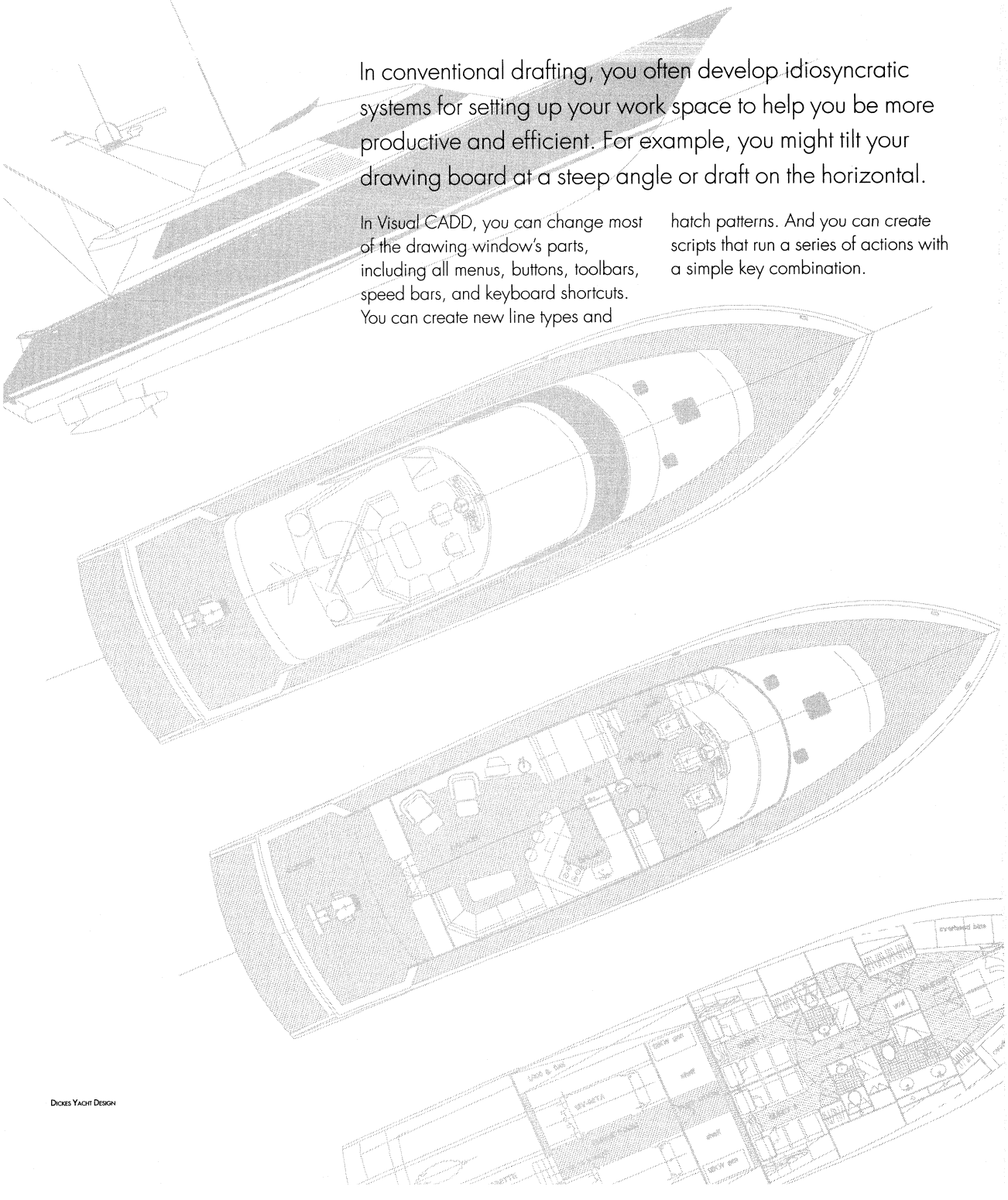
The Object Information window displays the order of each selected object in the drawing database, the coordinates of points, the lengths of segments, the object's area, and all object properties. For example, the object information for a circle includes not only coordinate data, layer, line type, and line width, but radius and diameter as well. The information for text includes its starting point (Ref Pt), font, and font height.

If no entities are selected, the Object Information window displays information about the current drawing window, such as the total number of entities in the drawing (including deleted entities available from memory for undo operations), valid (undeleted) entities in the drawing, and the extents of the drawing, with coordinates of the lower left and upper right corners.

Customizing Visual CADD

In conventional drafting, you often develop idiosyncratic systems for setting up your work space to help you be more productive and efficient. For example, you might tilt your drawing board at a steep angle or draft on the horizontal.

In Visual CADD, you can change most of the drawing window's parts, including all menus, buttons, toolbars, speed bars, and keyboard shortcuts. You can create new line types and hatch patterns. And you can create scripts that run a series of actions with a simple key combination.



Dickes Yacht Design

CHAPTER

8

Automating actions with scripts

If you repeat actions often when drawing in Visual CADD, you can write a script and assign it to a single keystroke or a combination of keystrokes. The next time you want to repeat those actions, you simply press the key combination you've assigned to it.

Using Script Editor, you can assign a script to function keys, key combinations, and mouse and digitizer buttons. Script Editor lists all Visual CADD native commands. To write a script, enter either the native command name or the equivalent keyboard shortcut.

Some of the symbols you need to add to scripts are not listed with the native commands in Script Editor; these you type directly in Script Editor's Edit Script text box. Be sure to write a script that incorporates all steps to complete an operation. For example, if the operation includes a prompt to place points, the script must include a notation for the prompt.

When you want to automatically load a set of symbols, for example, or start Visual CADD with the Drawing Setup wizard, you can initiate a script as Visual CADD starts by modifying the string value for the "Script" entry in Registry Editor: HKey_CURRENT_USER\Software\Numera\Visual CADD\2.0\Startup.

Script examples

This script...	Results in this action...
CS;SW;@;@;CO;	Copies objects you select using a selection window.
CS;SB;@;MV;@;@;	Moves an object.
SYMROT;$SYMROT+15;	Sets the angle for placing a symbol to 15°.

Common script elements

This element...	Does this...
;	Separates commands. Scripts must end with a semi-colon (;) as well.
@	Tells Visual CADD to wait for the coordinates of a point or a click on the drawing screen.
@@	Tells Visual CADD to wait for a Pen Up command to end drawing by a continuous tool before moving to the script's next command.
$	Retrieves the current value of the variable it precedes.
=;@	Matches the entity you select.
Uloff	Closes the specified speed bar.
Ulon	Opens the specified speed bar.
exename;<path to external application>;run;	Tells Visual CADD to run an external application from within Visual CADD.
dllname;<dll filename>;dllfunname;	Runs a function defined in a .dll file. <functionname within the dll>;dllrun;

Note: Scripts never contain quotation marks (").

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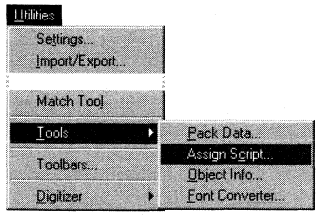
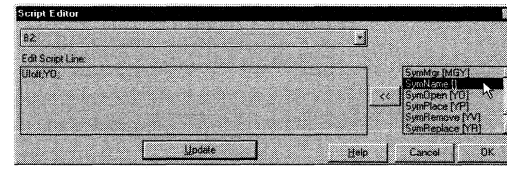
Keyboard shortcuts and native commands 180

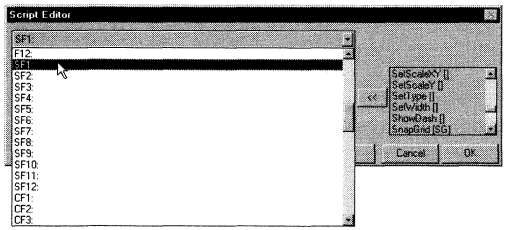
To assign a script



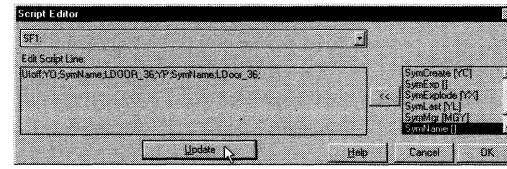
1 Click the Utilities menu, and then click Assign Script.



2 Select a command from the list of commands, and then click << to add it to the Edit Script Line text box; or type it directly in the Edit Script Line text box.

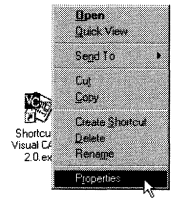


3 Select a key or key combination, such as SHIFT-F1, from the list to use for the script.

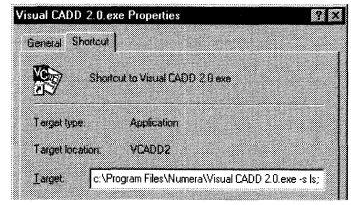


4 Click Update to add the script to the Script.def file.

To create a script to run when you start Visual CADD



1 Before starting Visual CADD, click the right mouse button on a Visual CADD shortcut icon, click Properties, and then click the Shortcut tab.



2 Type a script in the Target text box, starting with the Visual CADD path.

Note: Immediately after "Visual CADD 2.0.exe," type a space, "-s", another space, and then the script, ending with a semi-colon (;). A script for the single-line tool is shown. Your script will run when you next launch Visual CADD.

Creating custom menus

You can alter, add, or delete any menu in Visual CADD. You do this by editing a text file using an ASCII text editor, such as Windows Notepad. You can add commands to or remove them from any menu using native commands and custom commands created using scripts.

The menus are stored by default in the Visual CADD 2.0 Menu folder. The default menu file is Custom.mnu. To create a custom menu, create a text file and save it with an .mnu extension.

You can load custom menus either from within Visual CADD or by modifying Visual CADD's default startup menu in the Windows Registry. When you modify the Registry, the custom menu opens automatically.

Warning: Make changes in Windows Registry carefully to avoid disabling other applications or Windows 95. Edit string values only in the HKey_CURRENT_USER\Software\Numera\Visual CADD\2.0 key of the Registry. Make a backup copy of the Registry before modifying its contents, in case you later want to restore its original settings.

Anatomy of an .mnu file

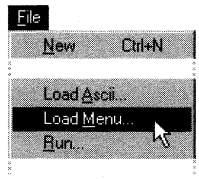
Text file	Description
POPUP "&File"	Defines the start of the File menu. "&" precedes the underscored character.
FileOpen	Native command name.
"&New File", FileNew	Overrides the menu description with the text in quotes.
Separator	Creates a separator line.
"&Close", 2405	Overrides the menu description with the text in quotes, using an associated Visual CADD native command numeric ID. Because they might change from version to version without notice, do not use Visual CADD numeric IDs.
Separator	Creates a separator line.
FileSave	Native command name.
FileExit	Native command name.
POPUPEND	Defines the end of the File menu.

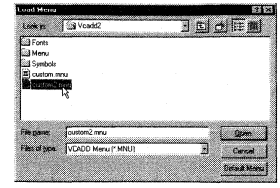
Note: If the item is a native command, Visual CADD stores the menu description internally. If the command is defined in the file Cmdext.def, Visual CADD takes the menu description from the menu description parameter of the custom command definition at the end of the Cmdext.def file.

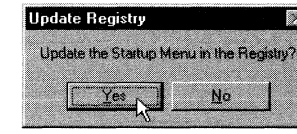
If after making changes to the Registry, you can't start Visual CADD, delete the contents of the 2.0 key of the Registry, and then start Visual CADD to restore the default settings.

FOR MORE INFORMATION

To load a custom menu from within Visual CADD





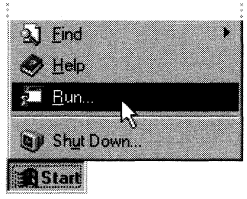


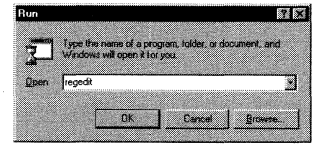
1 Click the File menu, and then click Load Menu.

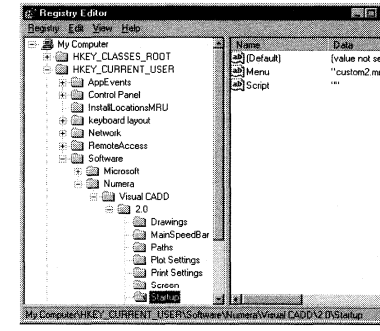
2 Select the menu text file (.mnu) you want to open, and then click OK.

3 Click Yes in the message box if you want this to be your default menu when you next start Visual CADD.

To set up Visual CADD to start with a custom menu



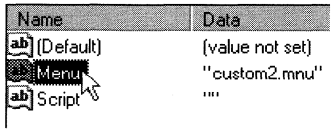


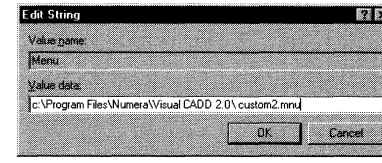


1 Click the Start button, and then click Run.

2 In the Open text box, type regedit, and then click OK.

3 Open the Startup key, which is in the HKEY_CURRENT_USER\ SOFTWARE\ Numera\Visual CADD \ 2.0 key.





4 Double-click Menu.

5 In the Edit String dialog box, type the full path and name of your menu text file in the Value Data edit box. At start-up, Visual CADD will load your custom file menu.

Customizing context-sensitive right mouse menus

You can modify or create the context-sensitive menus used with any tool by using an ASCII text editor, such as Windows Notepad, to edit text files with a .pop file extension stored in the Menus folder in the Visual CADD 2.0 folder.

The menu files with the .pop file extension contain the menu information and design for each tool's context-sensitive menu. A context-sensitive menu can contain native commands as well as custom com-mands created using scripts.

When you click the right mouse button while working in Visual CADD, Visual CADD reads the file Mousemnu.def to determine if a custom menu file (.pop) exists for the current tool. If one exists, Visual CADD loads the custom menu file. If one does not exist, Visual CADD loads the tool's default menu.

Save any changes you make to Mousemnu.def and the .pop files. The next time you start Visual CADD, the custom context-sensitive menus will be available.

ANATOMY OF MOUSEMNU.DEF

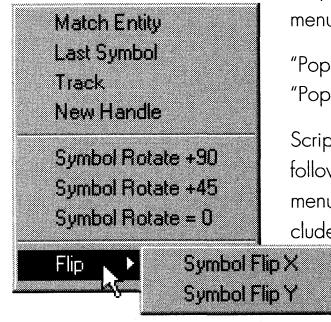
```
SymPlace,MENU\SYMPLACE.POP
LineCont,MENU\LINECONT.POP
LineSingle,MENU\LINESING.POP
LineDbl,MENU\LINEDBL.POP
Point,MENU\POINT.POP
```

Each line in the file Mousemnu.def directs Visual CADD to a file for the different right-mouse menus contained in text files with .pop file extensions. Each line contains the native command in effect when the context-sensitive menu is available and the path that locates the .pop file, separated by a comma (and no space).

Note: If you do not specify the full path of the menu file, Visual CADD starts searching in the System folder.

SAMPLE MENU FILE (.POP)

```
MatchEnt
SymLast
Track
NewHandle
Separator
SYMROT90
SYMROT45
SYMROT0
Separator
POPUP "Flip"
SYMFLIPX
SYMFLIPY
POPUPEND
.CmdExt
SYMROT90,,,Symbol Rotate +90,Rotate +90,SymRot;$SymRot+90;
SYMROT45,,,Symbol Rotate +45,Rotate +45,SymRot;$SymRot+45;
SYMROT0,,,Symbol Rotate = 0,Rotate = 0,SymRot;0;
SYMFLIPX,,,Symbol Flip X,Flip X,SymScX;-$SymScX;
SYMFLIPY,,,Symbol Flip Y,Flip Y,SymScY;-$SymScY;
```



A .pop menu file can contain native commands as well as user-defined scripts.

"Separator" creates a separator line in the menu.

"PopUp" defines the start of a submenu. "PopUpEnd" defines the end of the submenu.

Script definitions for custom commands must follow the line ".CmdExt" at the end of the menu. Custom commands defined here include three symbol rotation commands.

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Customizing keyboard shortcuts

When you start Visual CADD, it reads a text file called Alias.cmd. This file includes a list of all Visual CADD native commands and their keyboard shortcuts, if any exist. Visual CADD also reads the text file Cmdext.def, which contains full definitions of custom commands you create.

When you type a keyboard shortcut in Visual CADD, Visual CADD checks both the Alias.cmd and Cmdext.def files for a match. If it finds one, it runs the native command associated with that shortcut. If it finds no match, Visual CADD ignores your entry.

To customize the keyboard shortcuts, open Alias.cmd in an ASCII text editor, such as Windows Notepad. Note that each line in the file starts with the currently assigned keyboard shortcut, followed by a comma (and no space), and then the native command.

You can change the keyboard shortcut to any two- or three-letter text, but make sure that these letters do not start any other keyboard shortcut or native command name. You can check for conflicts by using the search command in your text editor. Save any changes you make to the ASCII text file. The next time you start Visual CADD, you will be able to use the new keyboard shortcuts.

CUSTOMIZED KEYBOARD SHORTCUTS

PO,Point	▶	PT,Point
LS,LineSing		LS,LineSing
LC,LineCont		LC,LineCont
R2,Rect2	▶	RE2,Rect2
R3,Rect3	▶	RE3,Rect3

To change the keyboard shortcut for a command, edit the command's text line in the Alias.cmd file. Here, for example, we've edited the keyboard shortcuts for the Point, 2-point-rectangle, and 3-point-rectangle tools.

Note: If you need descriptions of the native commands, you can open the file Native.txt in the Visual CADD folder by using an ASCII text editor. Native.txt lists all native commands and descriptions of each.

TIP

To restore the original shortcuts, delete the Alias.cmd file, and then restart Visual CADD.

FOR MORE INFORMATION

Keyboard shortcuts and native commands 180

Customizing toolbars and speed bars

You can add buttons for commands to the toolbar by editing the text file Toolpal.cst in the Visual CADD folder using an ASCII text editor, such as Windows Notepad.

You make changes to the main speed bar by editing the file Mainsbar.cst in the Visual CADD folder. The next time you start Visual CADD, the new toolbar and/or speed bar will be available.

To create several toolbars and choose which one to load any time you work within Visual CADD, you can create a text file with a .cst extension for each toolbar, and then in the Windows Registry create a copy of the string value, edit it, and save it with a new name and menu item for loading it.

Warning: Make changes in the Windows Registry carefully. You could easily disable other applications or Windows 95. Edit string values only in the HKey_CURRENT_USER\SOFTWARE\Numera\Visual CADD\2.0 key of the Registry. Make a backup copy of the Registry before modifying it, in case you later want to restore it to its original settings. If you can't start Visual CADD 2.0 after making changes, delete the contents of the 2.0 key of the Registry, and then restart Visual CADD to restore the default settings.

TIPS

If you want Visual CADD to ignore one of its native commands (but don't want to delete the command from the .cst text file), type two forward slashes (//) before the command.

After you create custom toolbar files, you can update the toolbars without restarting Visual CADD by clicking the right mouse button while the cursor is on the speed bar or the toolbar and then clicking the name of the custom toolbar file.

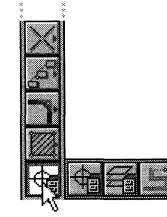
FOR MORE INFORMATION

CUSTOMIZED TOOLBARS AND SPEED BARS

Toolbar When you edit the text file Toolpal.cst, type commands in native command format, as shown below for the result shown on the right. Separate each native command by a comma (and no space) to create a string of flyout buttons. You can rearrange and add as many buttons to the toolbar as will fit on your screen.

```
Copy,RadCopy,ArrayCopy,Move,Rotate,Mirror,
Fillet,Chamfer,Trim,MTrim,Extend,MExtend,IntTrim,
HatchBnd,HatchSel,FillBnd,FillSel
SymMgr,LayMgr,Track
```

TEXT FILE

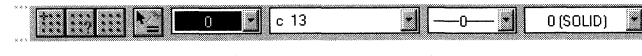


TOOLBAR RESULT

Speed bar When you edit the file Mainsbar.cst, type commands in native command format. "SEPARATOR" inserts a small space between buttons. You can create a string of flyout buttons comparable to those in the toolbar by separating each native command by a comma (and no space). You can rearrange and add as many buttons to the speed bar as will fit on the screen.

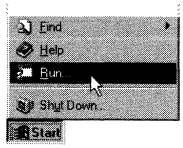
```
MatchTool
SEPARATOR
ColorProp
SEPARATOR
LayerProp
SEPARATOR
WidthProp
SEPARATOR
TypeProp
```

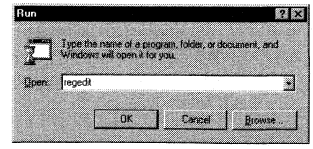
TEXT FILE

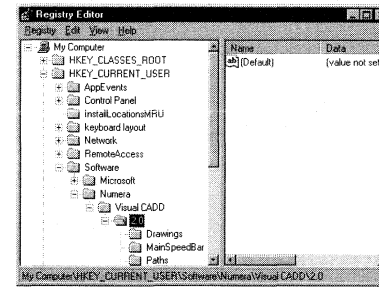


SPEED BAR RESULT

To create a new menu item for loading a custom toolbar or speed bar



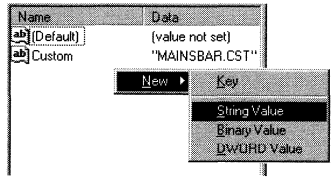


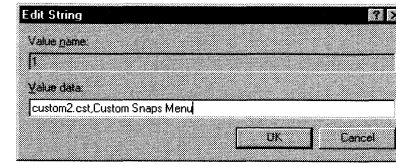


1 Click the Start button, and then click Run.

2 Type **regedit**, and then click OK.

3 Open the 2.0 key, which is in the HKEY_CURRENT_USER\ SOFTWARE\ Numera\Visual CADD key.





4 Open the MainSpeed Bar key (to create a new speed bar menu item) or the ToolPalette key (to create a new toolbar menu), click the right mouse button in the blank area of the right half of the window, click New, click String Value, and then type a number, in sequence.

5 Double-click the new string, enter the filename of the custom menu (.cst text file) in the Value Data edit box followed by a comma and the name you want to appear in the menu, and then click OK.

Creating new hatch patterns

You can rename, modify, or create hatch patterns for use in Visual CADD. To customize hatch patterns, open the file Hatches.hat located in the Visual CADD folder using an ASCII text editor, such as Windows Notepad.

The file Hatches.hat contains all hatch pattern definitions. The hatch pattern definitions don't need to appear in any particular order.

One or more lines of text within the Hatches.hat file may define a hatch pattern. A Visual CADD hatch definition describes a family of dashed or solid lines, each at a user-defined angle. Visual CADD copies and extends these dashed or solid lines to fill the hatched area. A single line of text defines the angle, line pattern, and spacing of each dashed or solid line. The hatch definitions define the hatch pattern as it would appear if drawn at a hatch scale factor of 1 and hatch rotation angle of 0°.

Save any changes you make to the file Hatches.hat, and then restart Visual CADD to use the new hatch patterns.

ANATOMY OF A HATCH PATTERN DEFINITION

HatchName, VERBATIM, Comment
StartX,StartY,Angle,OffsetX,OffsetY,Dash1Length,SpaceLength,Dash2Length,Space2Length,Ö

HatchName and comment line precede the actual hatch line definitions. A hatch definition can have any number of lines, but the more lines in the definition, the longer Visual CADD takes to hatch.

HatchName is the word of up to 32 characters (none of which can be a space) that will appear in the hatch list box.

VERBATIM is an optional message to Visual CADD to interpret the definition literally rather than reduce the longest line segment in the hatch pattern to a length of 1 inch and scale all other line segments proportionally. VERBATIM must be uppercase and include the space before and comma after.

Comment is an optional note for describing the hatch pattern definition. You see it only within the text file. Separate any comment from the hatch name with a comma.

StartX is the X coordinate (in inches) of the start of the line relative to the origin of the hatch pattern.

StartY is the Y coordinate (in inches) of the start of the line relative to the origin of the hatch pattern.

Angle is the angle of the line (with 0° corresponding to a clock's 3:00 position, and 90° corresponding to the 12:00 position).

OffsetX and OffsetY are the X and Y components (in inches) of the offset between copies of the line within the orientation of the hatch pattern. For example, if you rotate the hatch 90° (to vertical) when applying it to an object, the X offset of the line within the hatch pattern offsets the line in the Y direction of the drawing's absolute coordinate system.

DashLength is the length of a solid-line segment that makes up part of the line pattern. A positive value creates a dash.

SpaceLength is the distance between the end of one dash segment or dot and the beginning of the next. A negative value creates a space.

Both DashLength and SpaceLength are measured in inches. A value of 0 creates a dot. The total number of dashes, dots, and spaces cannot exceed 16. You can start any line with a dash, dot, or space. The pattern begins at the starting point of the line you draw.

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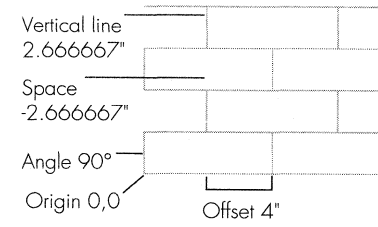
HATCH PATTERN EXAMPLE

StdBrickRun, VERBATIM, Standard brick (running bond, 8i long, 2-2/3i high)
0,0, 0, 0,2.666667
0,0, 90, 2.666667,4, 2.666667,-2.666667

The first line defines the continuous, solid horizontal line creating the top and bottom of the bricks. The first line starts at the hatch pattern's origin at an angle of 0° (horizontal). This line repeats every 2.666667" in the Y direction only.

The second line defines the vertical faces of the bricks. This line starts at the hatch pattern's origin at an angle of 90° (vertical), with a length of 2.666667". The 4 offsets the line horizontally every 4". The line's equal solid and space segments (2.666667 and -2.666667) create dashes.



Creating new line types

You can rename or modify the line types that come with Visual CADD or create your own. To customize line types, open the file Linetype.Int in the Visual CADD folder using an ASCII text editor, such as Windows Notepad.

The file Linetype.Int contains all line type definitions. The line type definitions do not have to appear in order in the linetype.Int file. Linetype.Int can contain up to 512 line types.

Save any changes you make to the ASCII text file Linetype.Int, and then restart Visual CADD to use the new line types.

ANATOMY OF A LINE TYPE DEFINITION

PL___X1, 19, D+U, 1.00, -0.08, 0.13, -0.08, 0.13, -0.08 PL___X1

A single line of text defines each line type in the following format:

LineName,Index,Reference,Dash1Length,Space1Length,Dash2Length,Space2Length,Ö

LineName is the word of up to 32 characters (no spaces) that will appear in the line type list box. You do not have to specify a LineName to use a line type in Visual CADD. If the LineName occurs more than once in Linetype.Int, Visual CADD uses the definition associated with the most recent appearance of the LineName.

Index is the unique line type number (0–511).

Reference is the frame of reference for measuring the lengths of the solid and blank segments of a custom line type. The W or world frame of reference measures line segment lengths in the same units as the drawing objects, scaling them as you zoom in or out and plot or print at different scales. The D or device frame of reference measures line segment lengths in the same units as the computer screen, plotter, or printer, keeping them at a fixed size regardless of magnification or print scale.

To prevent a space in a line type from falling at the end of a line, add "+U" (for Unbalanced) to the Reference definition. Visual CADD will always end the line with a solid segment, regardless of where the line ends in the line definition.

DashLength is the length in inches of a solid-line segment that makes up part of the line pattern. A positive value creates a dash.

SpaceLength is the distance between the end of one dash segment or dot and the beginning of the next. A negative value creates a space.

The length is measured for both DashLength and SpaceLength in real-world inches if the world reference is set, and in screen or printed inches if the device reference is set. A value of 0 creates a dot. The total number of dashes, dots, and spaces cannot exceed 16. You can start any line with a dash, dot, or space. The pattern begins at the starting point of the line you draw.

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LINE TYPE EXAMPLES

TYPE___2, 2, D+U, 0.65, -0.35

DASHDOT, 5, D+U, 0.60, -0.20, 0, -0.20

DIVIDE, 7, D+U, 0.60, -.133, 0, -.134, 0, -.133

BORDER, 14, D+U, 0.60, -0.20, 0.60, -0.20, 0, -0.20

PAVED_RD, 28, D+U, 1.22, -0.08

GRAVELRD, 29, D+U, 0.72, -0.08

Customizing Visual CADD's settings

Visual CADD stores its default settings in the Windows Registry. Because Visual CADD refers to these settings at start up, they play a central role in its performance. You can change most settings from within Visual CADD, but you can change those described here only in the Registry.

The key that contains these settings within the Windows Registry is: HKey_CURRENT_USER\Software\Numera\Visual CADD\2.0. All settings are string values.

Warning: Make changes in Windows Registry carefully. You could disable other applications or Windows 95. Edit string values only in the 2.0 key of the Registry. Make a backup copy of the Registry before modifying its contents, in case you need to restore its original settings. If, after making changes, you can't start Visual CADD 2.0, delete the contents of the 2.0 key of the Registry, and then restart Visual CADD to restore the default settings.

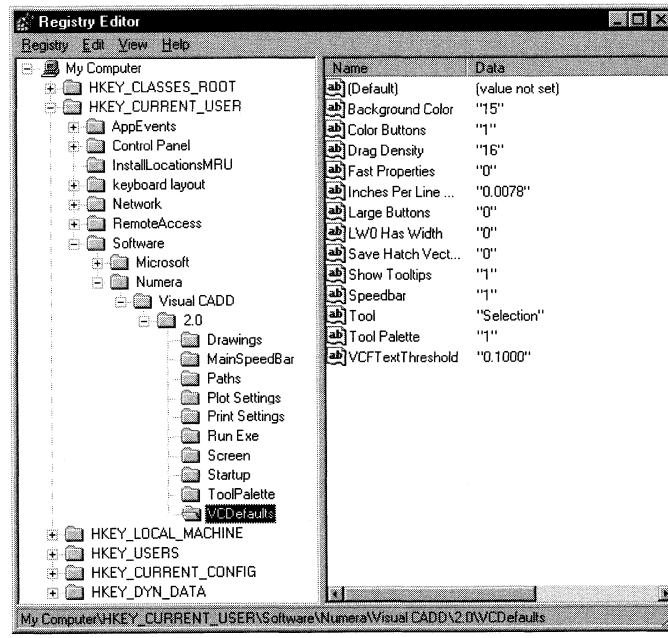
TIPS

Open the Windows Registry by clicking the Start menu, clicking Run, and then typing Regedit in the Open edit box of the Run dialog box.

Change settings in the Registry by double-clicking the name.

In all data fields containing only a 0 or 1, 0 turns off the setting, and 1 turns the setting on.

VISUAL CADD'S LOCATION IN THE WINDOWS REGISTRY



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Windows Registry default settings for Visual CADD 2.0

Key name...	Contents include...
Drawings	The last files opened in Visual CADD, as shown at the bottom of Visual CADD's File menu. You redefine the number of files listed by changing the Max value.
MainSpeedBar and ToolPalette	References to default and custom speed bar and toolbar files.
Paths	The default paths to Visual CADD system files. The extension .ext represents the last drawing file you opened or saved.
Print Settings	The current settings in the Print dialog box. TT Rotate is the only setting you cannot change in the Print dialog box. TT Rotate controls the orientation of TrueType text. When Windows TrueType text rotates 180° in print from what you draw on the screen, set the TT Rotate value to 1.
Plot Settings	These settings, which you can change only in the Registry: *Curve smoothness* sets the number of short straight lines making up a curve printed or plotted by an output device that does not support HP-GL/2. Setting curve smoothness to a higher value causes Visual CADD to use fewer lines and create a rougher curve but increases plot speed. Use values from 1 to 100. *Timeout* sets the duration, in seconds, after clicking the Plot button, for notifying you that no plotter is available.
Screen	The last positions of the Bird's-eye view window, Layer Manager, and Symbol Manager on the screen, so that they open next time in the same place.
Startup	References to the menu file and all scripts with which Visual CADD starts.
VCDefaults	*Background color* sets the background color of Visual CADD's drawing board. *Drag Density* sets the on-screen appearance of objects as you drag them. The higher the value (in on-screen inches) entered here, the more dots are used to create the drag line and the longer it takes to draw the dragged object. *Fast Properties* switches between the Properties and the Fast Properties speed bars. *Inches per line width* controls the line width of Visual CADD printed output. The data field value is a multiplier of the base-line width. *LWO Has Width* sets the finest line width in Visual CADD drawings. When this setting is 0, Visual CADD prints the smallest line width in a drawing with the finest line your printer can print. When it's set to 1, Visual CADD prints the smallest line width with the same width as line width 0. *Save Hatch Vectors* speeds up the loading of hatches in your drawing but substantially increases the file size of the drawing. *Tool* sets the default tool, both of which you can set easily on the System tab of the Settings dialog box. *VCF Text Threshold* sets the minimum size, measured in on-screen inches, of text before it is displayed as an illegible gray band.

Working with other applications

In hand drafting, using information from one drawing in another involves duplicating a portion of the drawing and then cutting and pasting on a layout table. Visual CADD provides tools that allow you to share information with other applications inside and outside the Windows environment quickly and easily.

To take full advantage of CAD, it is necessary to share information among applications. You can exchange drawings with users of AutoCAD and Generic CADD, and you can use Visual CADD to create illustrations for other applications. You can also add text to your Visual CADD drawings that was created in other applications.

Using the Windows clipboard is the easiest way to share information among applications. Using object linking and embedding (OLE), you can link Visual CADD drawings to files in other applications so that changes you make to the Visual CADD file are updated in the file it is linked to as well and vice versa.

Linda Joy Weinstein, AIA

CHAPTER

9

Copying text and objects between Visual CADD and other applications

Incorporating parts of Visual CADD drawings into other Windows-based documents is simple. You can use the standard cut, copy, and paste commands to add Visual CADD illustrations to your spreadsheets or word-processing documents. You can also use layout and publishing applications to combine Visual CADD drawings with other text and graphic elements for visually appealing publications.

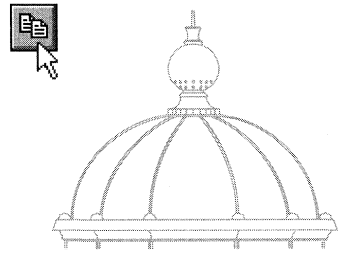
Because objects copied into another application are stored in the other application's document file, you may want to insert a Visual CADD object as a picture. This format takes less space, but is not updated when you edit the original Visual CADD object.

To share text between Visual CADD and other applications, you must use the text-editor tool.

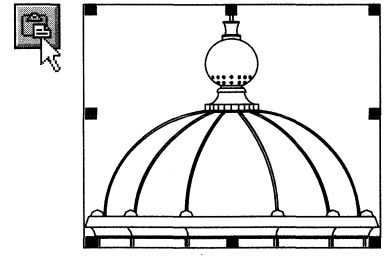
TIP

Hold down Ctrl and press X, C, or V instead of clicking Cut, Copy, or Paste respectively.

To copy Visual CADD objects into another application



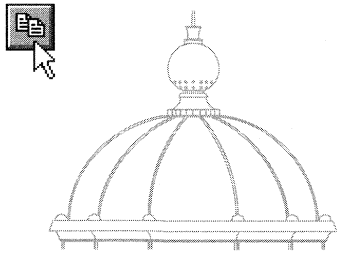
1 Select the Visual CADD entities you want to use in another application, and then click the copy button.

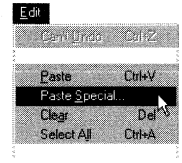


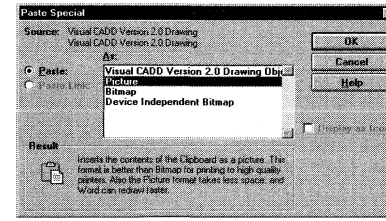
2 Switch to the other application, click the Edit menu, and then click paste.

FOR MORE INFORMATION

To copy Visual CADD objects as a picture





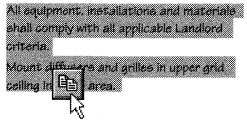


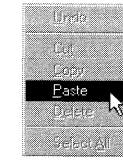
1 Select the Visual CADD entities you want to use in another application, and then click the copy button.

2 Open the other application, click the Edit menu, and then click Paste Special.

3 Select Picture as the format in which to paste.

To import text from another application



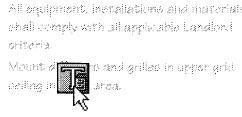


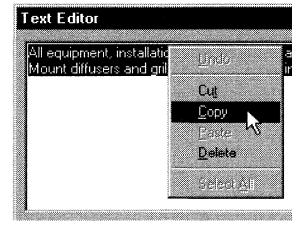
1 Within the other application, select the text you want to use, and then click the copy button.

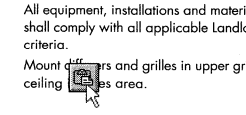
2 Switch to Visual CADD, and then click the text-editor tool.

3 Select a point on the drawing at which to insert the text, click the right mouse button in the text-editor box, and then click Paste.

To use Visual CADD text in another application







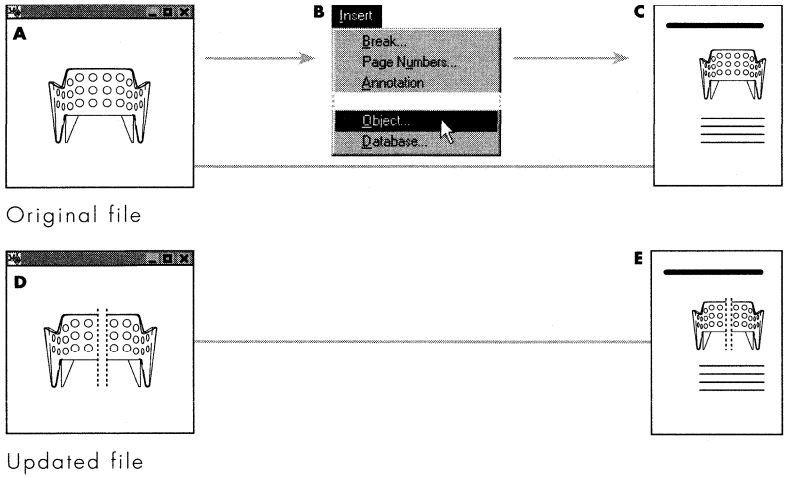
1 In Visual CADD, select a text block, and then click the text-editor tool.

2 In the text-editor box, select the text you want to use, click the right mouse button, and then click Copy.

3 Open the other application, and then click the paste button.

Linking and embedding objects

LINKING AN OBJECT USING OLE

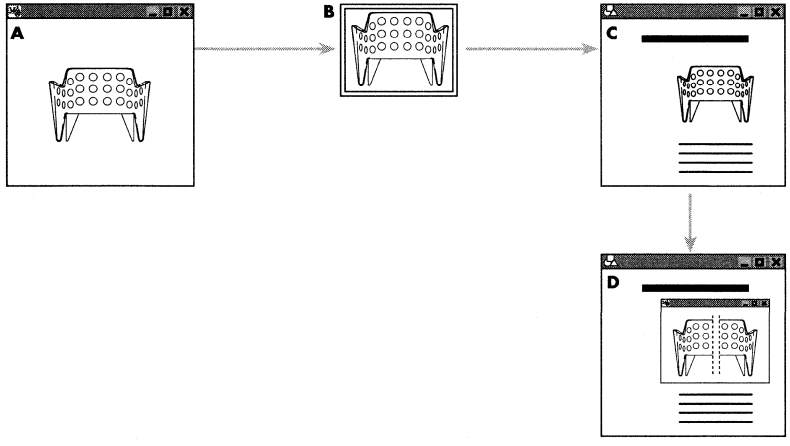


Link to a Visual CADD object from another application when you want the imported object to update automatically after making changes to it in Visual CADD, and to keep both files small.

After creating an object in Visual CADD **A**, you can import it into a file in another application with the Insert Object command **B**. Windows creates a link **C** between the object and its original file.

When you double-click the Visual CADD object to edit it, Windows automatically opens the object in Visual CADD **D**. When you've finished editing, Windows updates the file that contains a copy of the object **E**.

EMBEDDING AN OBJECT USING OLE



Embed a Visual CADD object in a file in another application when you want a self-contained drawing, and when you don't need to update the image.

After creating an object in Visual CADD **A**, you can import it into a file in another application by copying or cutting it to the clipboard **B**, and then choosing Paste or Insert to paste the object from the clipboard into the host application, or Paste Special to choose among all import options in the host application **C**.

When you double-click the Visual CADD object to edit it, you can use most of Visual CADD's tools to edit it in place **D**.

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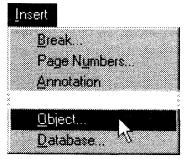
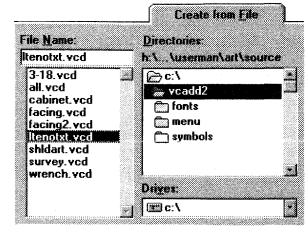
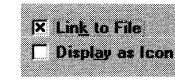
To link to an existing Visual CADD file from within another application





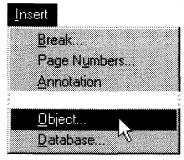


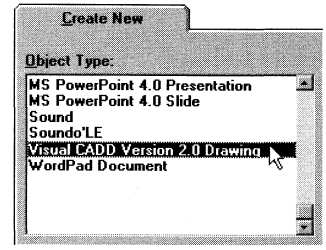
1 In the application in which you want the drawing, click the Insert menu, and then click Object.

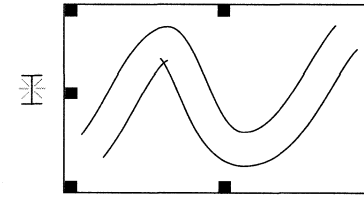
2 Click the Create from File tab, and then enter the file name of an existing Visual CADD drawing.

3 Check Link to File if you want the OLE object to reflect subsequent changes to the drawing file.

To embed a Visual CADD drawing into another application file





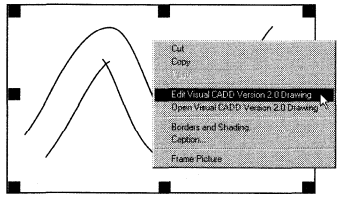


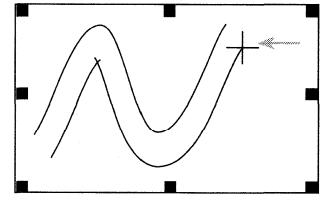
1 In the application in which you want to insert the drawing, click the Insert menu, and then click Object.

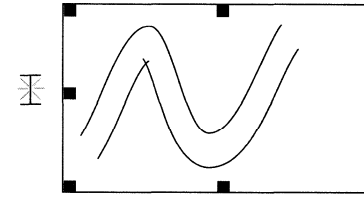
2 Click the Create New tab, click the Visual CADD Drawing object type, and then click OK.

3 Use the Visual CADD tools and menus to create your drawing, and then click outside the drawing frame to return to your application.

To edit a Visual CADD object in another application







1 Double-click the OLE object or use the right mouse button to click the object, and then click Edit Visual CADD Version 2.0 Drawing.

2 Edit the entities using the Visual CADD tools and menus that appear.

3 Click outside the drawing frame to return to your document.

Creating export files

If the application you are using does not support OLE or the Windows clipboard, you can create an export file containing the graphic information, and then place that file in your application.

The Windows metafile format is the most flexible format, as it supports lineweight control as well as filled entities and displays on the screen what you will see in print. The encapsulated Postscript (EPS) format allows lineweight control and filled entities, but most applications will not display the graphic object until you print. The Hewlett Packard graphics language (HP-GL) format is the least flexible as it does not support lineweight control and it uses parallel lines to approximate filled entities. HP-GL also cannot be displayed for preview by most applications but may be used if the other formats are not available in your application.

Creating a metafile is as simple as saving a file in the proper format. Creating EPS and HP-GL files requires that you set Windows to send printer or plotter output to a file.

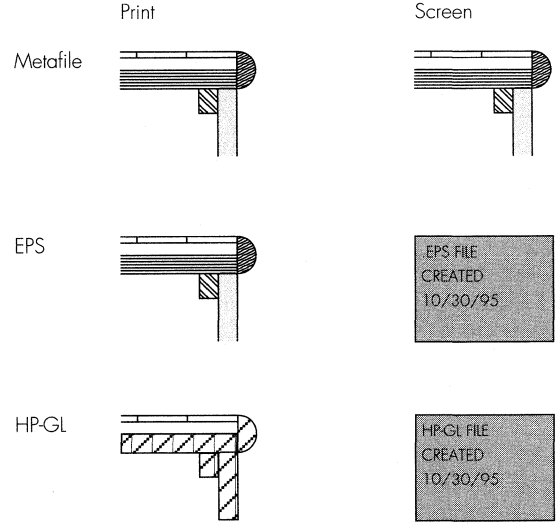
If you use this method, remember to set printer output back to a port when you are done, or you will be unable to print. You can simplify this process by setting up a separate printer description for each type of output that you use by using the Add Printer wizard in Windows. For further information about setting up different printers, see your Windows documentation.

TIPS

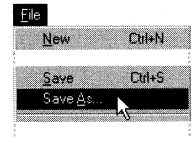
To export only part of a drawing, select those entities to be included in the export file, and then check Save Selected in the Save As dialog box.

To print only the entities you select, check Selection Only in the Print dialog box before you print.

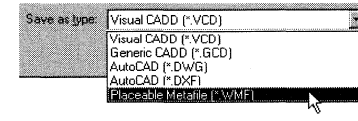
FILE FORMAT DIFFERENCES



To create a Windows metafile



1 Click the File menu, and then click Save As.



2 Specify a file name, and then click Placeable Metafile (.wmf) from the Save As Type list.

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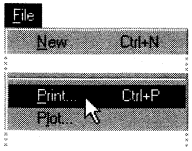
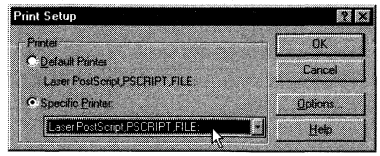
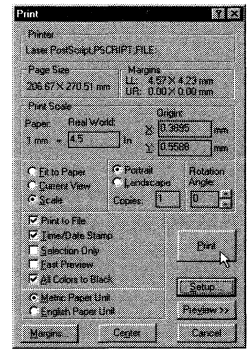
To create an encapsulated Postscript (.eps) file





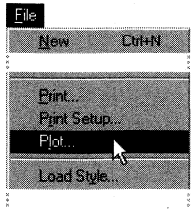


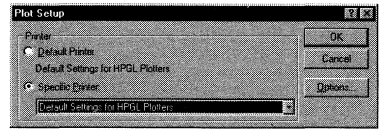
1 Click the File menu, click Print, and then click Setup.

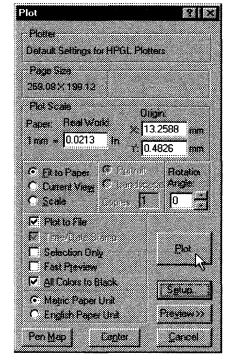
2 Select a printer that supports Postscript, and then click OK.

3 Check Print to File, click Print, type a filename, select a file type, and then click Save.

To create an HP-GL file







1 Click the File menu, click Plot, and then click Setup.

2 From the Specific Plotter list, select a plotter that supports HP-GL, and then click OK.

3 Check Plot to File, click Plot, type a filename, select a file type, and then click Save.

Sharing files with AutoCAD and Generic CADD

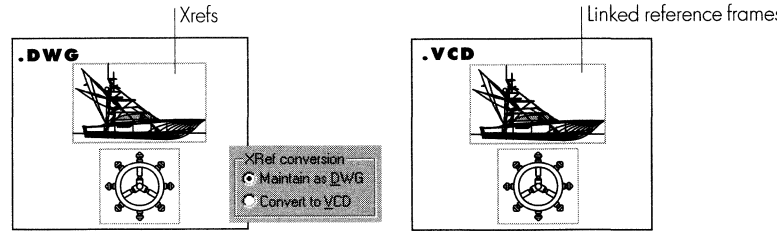
Visual CADD can read and write both AutoCAD and Generic CADD file formats. Because the three applications support different features, some objects and information might not be translated. AutoCAD's paper space viewports are imported into Visual CADD as bound reference frame entities. Although Visual CADD does not use AutoCAD's Extended Entity Data, it will store it with a drawing and include it if the drawing is translated back into the AutoCAD format. You have access to Extended Entity Data through Visual CADD's programming interface.

When you open an AutoCAD or Generic CADD drawing in Visual CADD, Visual CADD does not modify the original file. If you make changes, you must decide whether to save the changes in the original file format or into a *new* Visual CADD file. If you open an AutoCAD drawing with external references, both the drawing and external reference files must be located in the same folder.

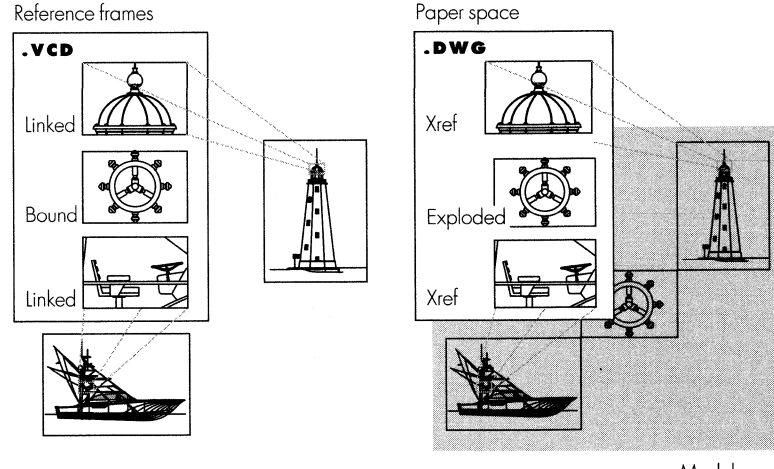
If you are working with a drawing created in another application and want to keep it in its original file format, it is best to save it as a Visual CADD drawing while you work on it. When you finish editing it, convert it back to its original format.

FOR MORE INFORMATION

CONVERTING AUTOCAD XREFS AND PAPER SPACE

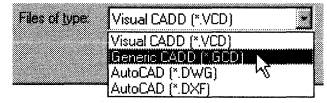


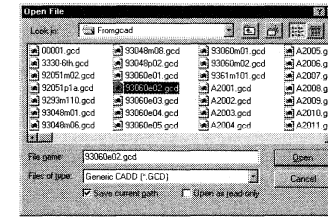
External references are imported into Visual CADD as linked reference frames, with the option of converting all linked files to .vcd format, or maintaining them as .dwg files.



Linked reference frame entities export to AutoCAD as Xref insert entities. Bound reference frame entities are simply exploded. If the view of any reference frame is not the entire source object, all reference frame entities also map into AutoCAD viewport entities in paper space and the entities are tiled in model space. The AutoCAD paper space drawing appears the same as the original Visual CADD drawing.

To open an AutoCAD or Generic CADD drawing in Visual CADD



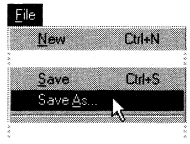


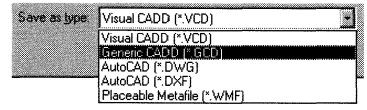
1 Click the open-drawing button.

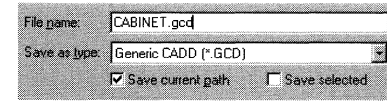
2 Click the Files of Type list, and then click either AutoCAD (.dwg) or Generic CADD (.gcd).

3 Click the file to open, and then click Open.

To save a drawing in the AutoCAD or Generic CADD format





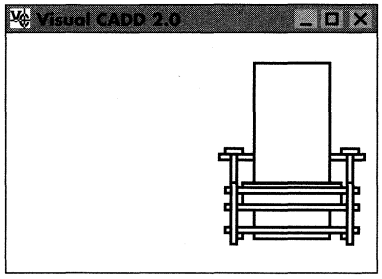


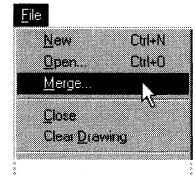
1 Click the File menu, and then click Save As.

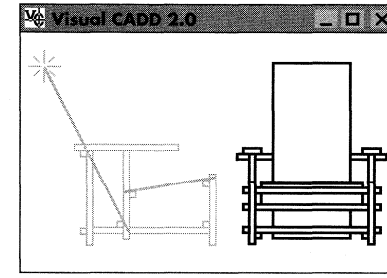
2 Click the Files of Type drop-down list, and then click either AutoCAD (.dwg) or Generic CADD (.gcd).

3 Specify the folder and filename in which to save the file.

To merge the contents of two Visual CADD drawings







1 Open a Visual CADD drawing file into which another will be inserted.

2 Click the File menu, and then click Merge.

3 Specify a file name, click OK, and then click the point at which to insert the other drawing.

AutoCAD and Generic CADD translation settings

Although Visual CADD, AutoCAD, and Generic CADD operate differently, you can adjust the way in which certain entities and properties are translated.

Duplicating Generic CADD hatch patterns

Because of the way Generic CADD defines hatch patterns, they cannot be used in Visual CADD. You can have Visual CADD hatch the area either with a symbol that looks like the Generic CADD hatch pattern or with a default hatch pattern of its own. If a symbol is used, it will match the Generic CADD drawing, but the hatch pattern will not be associative and will not update if you stretch or modify its boundaries. If you choose a default hatch pattern, the Generic CADD hatch pattern will be lost, but the hatch will be associative, and you can easily change it to a Visual CADD hatch pattern.

Setting AutoCAD measurement units

When opening AutoCAD drawings, you must identify the unit of measurement in which they are drawn because it is not stored in the drawing files.

Translating AutoCAD colors

In AutoCAD, a numbered screen color is mapped to a plotter pen. On pen plotters, the color number is more important than the displayed color. If you use an inkjet plotter, the displayed color itself is often more important. In Visual CADD, you must decide whether to maintain the color number or the actual color from the AutoCAD drawing.

Converting AutoCAD XRefs

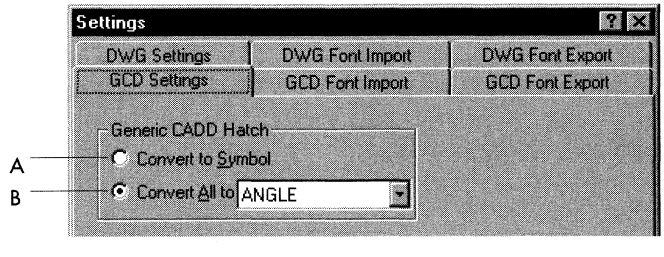
When you open an AutoCAD drawing that contains XRefs, you must decide how to treat the drawings to which the XRefs refer. You can either leave them in the AutoCAD format or you can convert them to the Visual CADD format.

Importing fonts correctly

When you import a drawing, you can choose "Keep," and all of the font names used in the drawing will be unchanged in the Visual CADD drawing. You must use the Font Converter to translate AutoCAD or Generic CADD fonts into a form that Visual CADD understands before you import the drawing. Once done, the same font names can be used in both versions of a drawing.

If you do not have the original AutoCAD or Generic CADD font files, or you do not want to convert them, you can translate AutoCAD or Generic CADD font names into names of fonts available to Visual CADD.

GENERIC CADD TRANSLATION SETTINGS



Opens when you click the Utilities menu, and then click Import/Export.

A When selected, indicates that Generic CADD hatches are converted to a symbol that looks like the original hatch.

B When selected, indicates that all Generic CADD hatches are converted to the selected Visual CADD hatch pattern.

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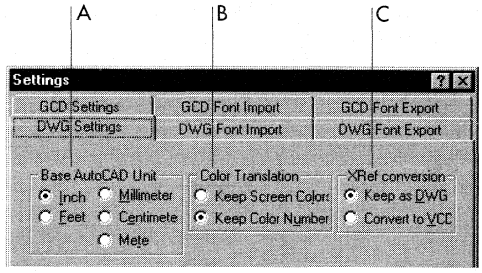
Adjusting font mapping 163

AUTOCAD TRANSLATION SETTINGS



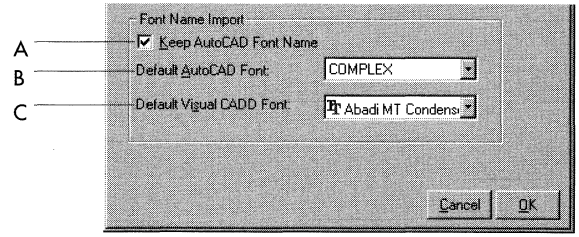
Opens when you click the Utilities menu, click Import/Export, and then click DWG Settings.

A Selects the unit of measure in which the AutoCAD drawing was created.

B Specifies whether AutoCAD screen colors or color numbers will be retained during translation.

C Specifies whether drawings referenced by AutoCAD XRefs are translated into Visual CADD format or left in AutoCAD format.

FONT NAME TRANSLATION SETTINGS



Opens when you click the Utilities menu, click Import/Export, and then click GCD Settings or DWG Settings. Appears as the bottom half of either of the two tabs.

A When checked, retains original font names during importing rather than using font name mapping.

B During export, specifies the font name to be used when no mapping exists for a font.

C During import, specifies the font name to be used when no mapping exists for a font.

Adjusting font mapping

Visual CADD, AutoCAD, and Generic CADD do not use the same fonts. When converting a file between formats, Visual CADD must decide how to substitute for fonts that exist in one application but not in the other. To control that substitution, you need to create font maps.

Font maps simply describe the way in which font names are translated during file conversion. For example, you can specify that every time the font named "TXT" is found in an AutoCAD drawing, it is replaced with the font named "Engineer" when the drawing is loaded into Visual CADD.

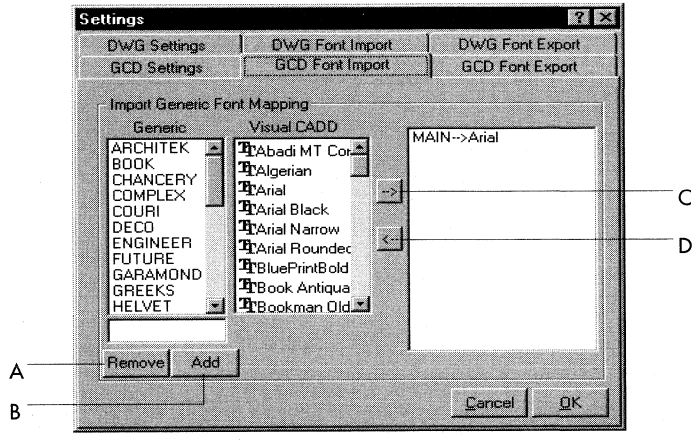
The default font on the GCD or DWG Settings tabs specifies which font is used when no substitution is found.

If you have an AutoCAD or Generic CADD font that is not shown on the font list, you can click the Add button to add a new name. The number of fonts on the list is limited to 32.

TIP

To avoid the need to translate font names, use the Font Converter to translate your AutoCAD and Generic CADD fonts.

FONT NAME MAPPING



Opens when you click the Utilities menu, click Import/Export, and then click the GCD Font Import tab.

A Removes the selected Generic CADD font name from the list of available fonts.

B Adds a new Generic CADD font name to the list of available fonts.

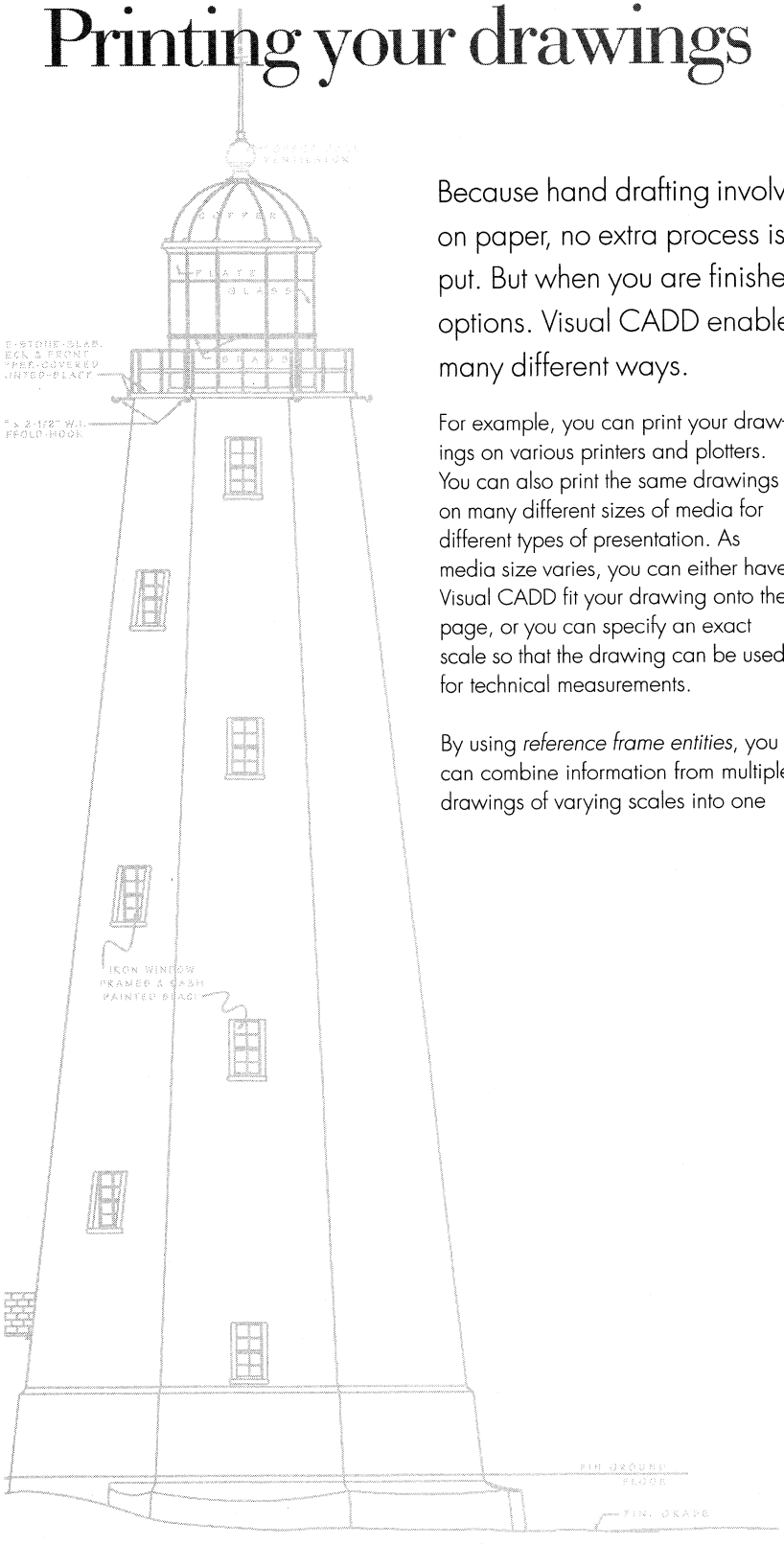
C Adds a new font name mapping to the list using the selected Visual CADD and Generic CADD font names.

D Removes the selected font mapping from the list.

Note: All four Font Import and Export tabs contain controls similar to those found on the GCD Font Import tab.

FOR MORE INFORMATION

Printing your drawings



Because hand drafting involves recording information directly on paper, no extra process is required to create printed output. But when you are finished, you have few presentation options. Visual CADD enables you to present drawings in many different ways.

For example, you can print your drawings on various printers and plotters. You can also print the same drawings on many different sizes of media for different types of presentation. As media size varies, you can either have Visual CADD fit your drawing onto the page, or you can specify an exact scale so that the drawing can be used for technical measurements.

By using *reference frame entities*, you can combine information from multiple drawings of varying scales into one drawing. You can control the scale and the layer visibility of each reference frame entity to use them in a variety of ways. For example, you can use them to create a detail within a drawing or even to draw your own overlays of drawings created in AutoCAD or Generic CADD.

If you have a vector-output device like a plotter, Visual CADD provides you with control over line weight that is not available by using the standard Windows printer drivers.

CHAPTER

10

Printing your drawing

Printing is as simple as sizing your drawing to your paper and then clicking Print. If you have more complex requirements, you can adjust Visual CADD's print settings to print your drawing just as you want it.

Print settings give you control over the size and orientation of your drawing on the page as well as a variety of options for expediting printing and controlling various printers.

Visual CADD's preview option allows you to see how your output will look before you send it to the printer and to make changes to margins, orientation, and scale as you do so.

If your computer is not connected to the printer you want to use, you can store the output in an export file for later use.

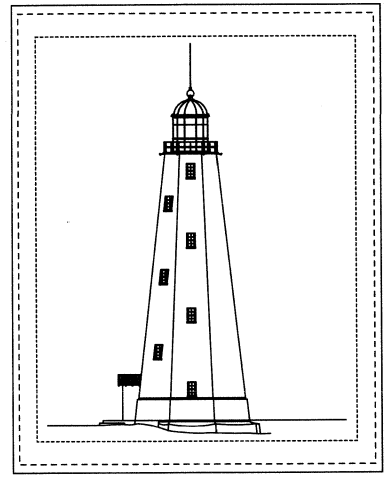
TIPS

Reduce the time required to preview large drawings by checking Fast Preview in the Print dialog box.

To update the print preview after you have changed print settings, click Update.

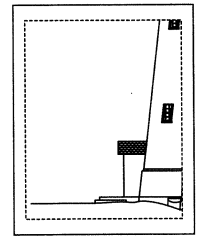
FOR MORE INFORMATION

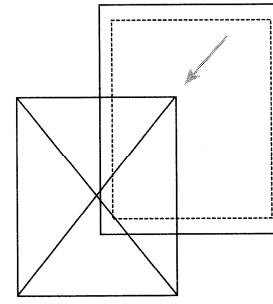
ADJUSTING PRINT SETTINGS AND OPTIONS

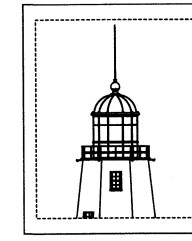


You can change the size of margins by clicking Margins from the Print dialog box, and then typing new values. Margins can be no larger than the maximum print area defined by your printer. The printer cannot print to the edge of the paper. Often the printer can print no less than ¼" from the edge of the paper. Page Size is, at maximum, the actual usable area on the paper.

After changing the Print Scale of your drawing, you can drag the preview into position to display the portion you want to print.



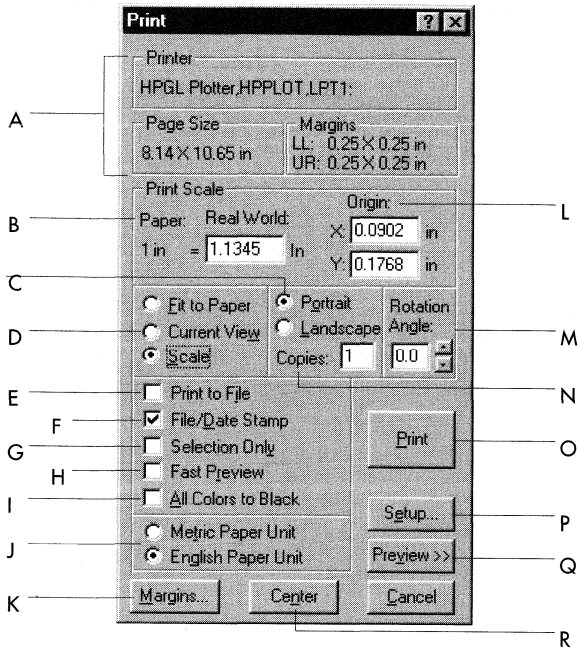




Common print scales

Print scale...	Real world value...	Print scale...	Real world value...
3/32" = 1'	1" = 128"	½"= 1'	1" = 24"
⅛"= 1'	1" = 96"	¾"= 1'	1" = 16"
3/16"= 1'	1" = 64"	1"= 1'	1" = 12"
¼"= 1'	1" = 48"	1½"= 1'	1" = 8"
⅜"= 1'	1" = 32"	3"= 1'	1" = 4"

PRINT SETTINGS



Opens when you click the File menu and then click Print.

A Displays the current printer, paper size, and margin settings.

B Sets the factor by which the drawing will be scaled when the Scale option is checked.

C Selects between horizontal (landscape) and vertical (portrait) page orientation.

D Specifies whether the drawing is to be scaled to fit the paper size, scaled so that the displayed view will fill the page, or scaled to a factor that you enter.

E Sends the output to a file on the disk drive.

F Check this box to include a time and date stamp on the printed page's edge.

G Prints only the selected entities on a drawing.

H Shows only an outline of the drawing during the print preview to speed the preview of large or complex drawings.

I Prints all entities in black, regardless of their color.

J Selects the unit in which to measure paper size and margins.

K Opens the Margin Settings dialog box.

L Sets the drawing origin relative to the lower left corner of the margin. The units of measure are those of the page size and not of the drawing.

M Sets the rotation angle of your drawing relative to the paper.

N Sets the number of copies to print.

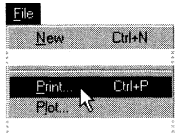
O Prints your drawing.

P Opens the Print Setup dialog box.

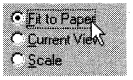
Q Shows a preview of the drawing as it will print using the current print settings.

R Centers the drawing on the printed page.

To print an entire drawing

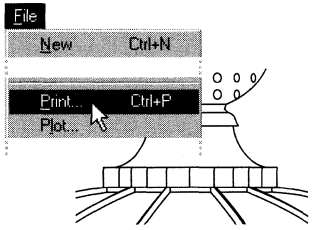


1 Click the File menu, and then click Print.

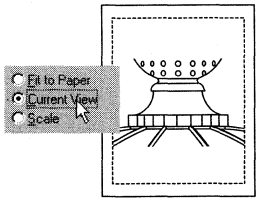


2 Click Fit to Paper under Print Scale, and then click Print.

To print a portion of a drawing

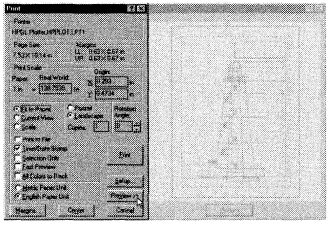


1 Zoom to display a portion of your drawing, click the File menu, and then click Print.

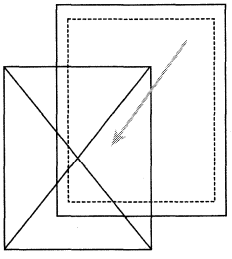


2 Click Current View under Print Scale, and then click Print.

To preview and realign your drawing before you print

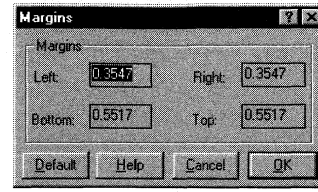


1 Click the File menu, click Print, and then click Preview.



2 Drag the image of the drawing to locate the portion you want to print within the margins, and then click Print.

To set printing margins

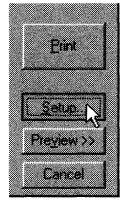


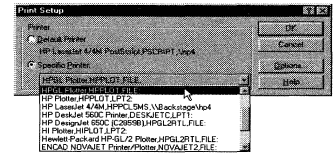
1 Click the File menu, click Print, and then click Margins.

2 Adjust the margin settings. Note: Clicking Default returns the settings to their default values.

3 Click OK to return to the Print dialog box.

To select a different printer



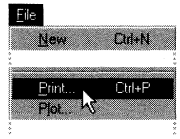


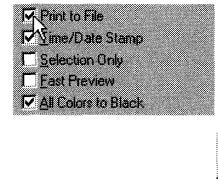
1 Click the File menu, click Print, and then click Setup.

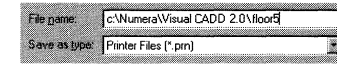
2 Click either Default Printer or Specific Printer. If you select Specific Printer, choose a printer from the list.

3 Click OK to return to the Print dialog box.

To create a print file







1 Click the File menu, and then click Print.

2 Check Print to File, and then click Print.

3 Type a name and folder in which to save the print file, select a file type, and then click Save.

Adding reference frame entities to your drawing

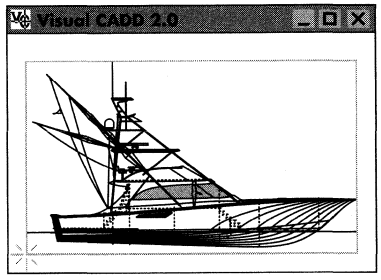
Like windows into other drawings, reference frame entities enable you to display the contents of one drawing inside another. You can use the frames to lay out drawings for printing or to create overlays. You cannot edit the contents of the frame, but you can control its display characteristics, and you can snap to it while drawing new entities.

When you add a reference frame, you can either define the size of the frame and the drawing will fit into it; or, you can specify the location of the frame and it will be sized to display the whole drawing at full scale.

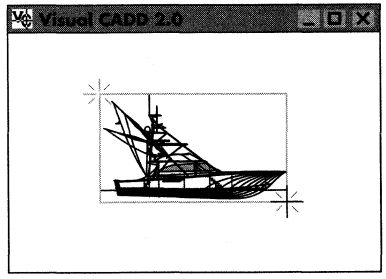
You can also choose either Bind Data to insert the contents of the referenced drawing into the current drawing or Link File to link the contents. If you link to the referenced drawing, any changes made to it will be reflected in the reference frame. Linking will also greatly reduce the size of the drawing containing the reference frame.

You can use reference frames to display the contents of other CAD files as well. Linking a reference frame to an AutoCAD or Generic CADD file will provide an updated image of that drawing each time you open your Visual CADD drawing.

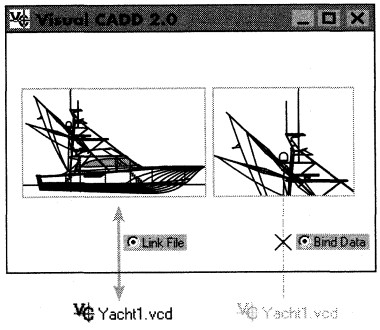
USING REFERENCE FRAMES TO DISPLAY INFORMATION



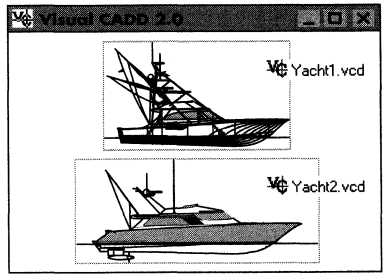
Placing a reference frame Click one corner and size a frame to the full-scale drawing.



Creating a reference frame Click both corners to size the drawing to the frame.



Reference frames are useful for collecting information from multiple drawings into one layout for display and printing. Binding a reference frame into a drawing inserts the contents into the drawing directly.



Linking a reference frame into a drawing keeps drawing file size to a minimum and updates the contents of the frame based on the contents of its origin.

TIPS

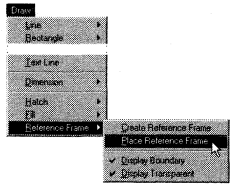
Draw an overlay by inserting a drawing using the Place Reference Frame command and then drawing on top of it using snaps.

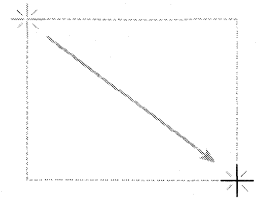
To update the contents of a reference frame, click the reference frame, click the right mouse button, and then click Update File Link.

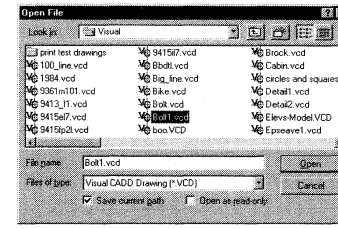
FOR MORE INFORMATION

Drawing entities precisely with snaps 50
Working with AutoCAD and Generic CADD files 160
Adjusting a reference frame entity 174

To create a reference frame in which to fit a drawing





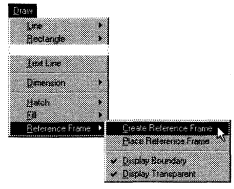


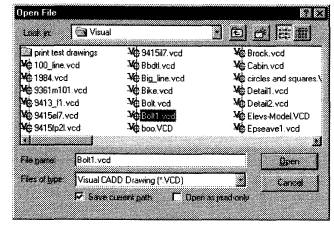
1 Click the Draw menu, click Reference Frame, and then click Create Reference Frame.

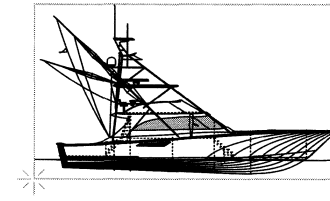
2 Click two points representing the corners of the reference frame.

3 Click Browse to find a drawing, or if you have multiple drawings open, click the MDI tab.

To place a reference frame that displays a whole drawing at full scale





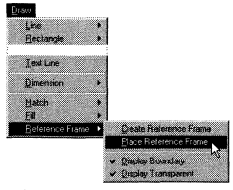


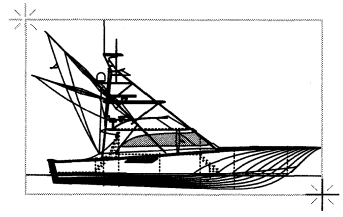
1 Click the Draw menu, click Reference Frame, and then click Place Reference Frame.

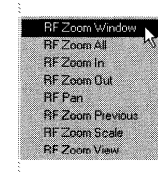
2 Click Browse to find a drawing, or if you have multiple drawings open, click the MDI tab.

3 Click the point representing the lower left corner of the frame.

To add a detail by using a reference frame entity







1 Click the Draw menu, click Reference Frame, and then click Create Reference Frame.

2 Click the two corners of the reference frame, check Bind Data on the speed bar, click Browse to find the drawing, and then click OK.

3 Click the right mouse button, and then click the appropriate zoom command to select a view and scale for the area to detail.

Adjusting a reference frame entity

Once you add a reference frame entity to your drawing, you control its contents.

You can zoom in or out by using the reference frame zoom commands. Or use a Bird's-eye view to display the whole drawing to which a reference frame is connected.

If you don't want to see all of the drawing information within a reference frame, you can hide layers without affecting the original drawing.

You can also change an existing reference frame to display a different drawing. This is useful if a drawing name changes during revision.

TIPS

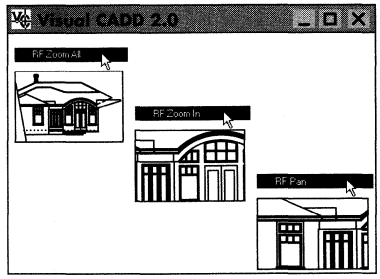
See reference frame entity properties and options by selecting an existing reference frame and clicking the right mouse button.

To create accurate details within a drawing without copying or recreating entities, set a reference frame to display at an exact scale by using the RF Zoom Scale command.

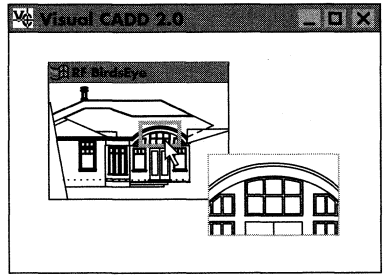
FOR MORE INFORMATION

Using layers to simplify a complex drawing 20
Viewing different areas of your drawing 128
Adding reference frame entities to your drawing 172

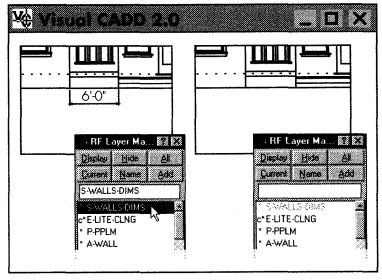
CONTROLLING WHAT IS VISIBLE IN A REFERENCE FRAME



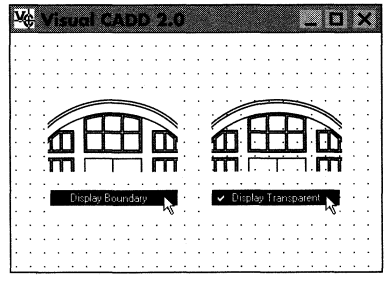
Use RF zoom commands to control the scale at which reference frames are displayed.



Use the RF Bird's-eye view to quickly zoom to a different part of the reference frame.

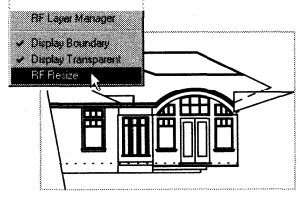


Use RF Layer Manager to display and hide layers of information within a reference frame.



Uncheck Display Boundary on the right mouse menu to hide the reference frame border. Check Display Transparent so that entities and grid marks show through a reference frame.

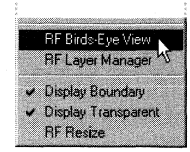
To resize a reference frame

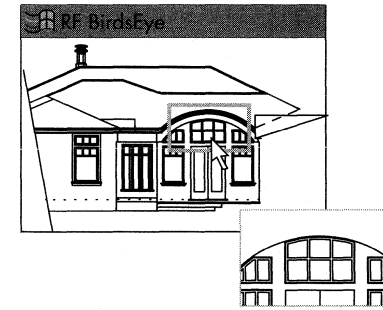


1 Select a reference frame, click the right mouse button, and then click RF Resize.

2 Click a new upper-right corner for the reference frame.

To change the zoom scale or view of a reference frame



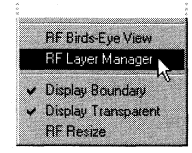


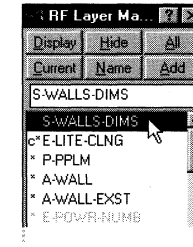
1 Select an existing reference frame, and then click the right mouse button.

2 Click an RF Zoom command or RF Bird's-Eye View command.

3 Reposition the contents of the reference frame as you would with any zoom command.

To adjust layer visibility in a reference frame



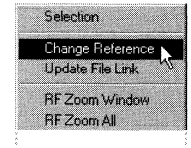


1 Select an existing reference frame, and then click the right mouse button.

2 Click RF Layer Manager.

3 Use RF Layer Manager to display or hide layers within the reference frame.

To change the file that a reference frame displays



1 Select an existing reference frame, and then click the right mouse button.

2 Click Change Reference.

3 Select the new file that you want the reference frame to display.

Using direct plotting

If you have a vector output device such as a plotter, you can use Visual CADD's direct plotting features to enhance control over the device. By using direct plot, you bypass the Windows plotter drivers and send information directly to the plotter. Using direct plot provides you with control over pen mapping.

Direct plot also allows you to use plotters for which you do not have a Windows driver.

When you use pen mapping, remember that many ink jet plotters use pen numbers to refer to different line types. Certain pen numbers or ranges of pen numbers may refer to lines that are not solid or that are created using a gray-scale fill. If you check All Colors to Black in the Plot dialog box, all colors will plot using the pen mapped to color 0, which is normally black.

Visual CADD increases plotting speed by using pen sorting and motion optimization. Both features take more memory, but they increase overall plot speed. Pen sorting reduces pen changes by assuring that each pen is used only once. Motion optimization improves plot performance by minimizing unnecessary pen movements and drawing from one end of the sheet to the other whenever possible.

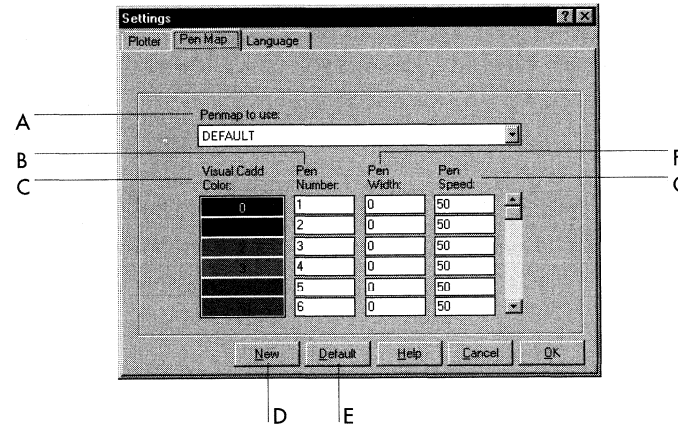
To use direct plotting, make sure that your plotter is configured for hardware flow control.

Note: Direct plotting does not support TrueType fonts. Vector fonts will be substituted for TrueType fonts when direct plotting is used.

TIP

Decrease overall plot time by checking Optimize Plotting and Sort Colors on the Plotter tab of the Plotter Settings dialog box.

PEN MAPPING SETTINGS



Opens when you click the File menu, click Plot, click Setup from the Plot dialog box, click Options from the Plot Setup dialog box, and then click the Pen Map tab.

A Lists the pen-mapping set you can use.

B Specifies the pen numbers assigned to individual colors.

C Specifies the color assigned to each pen.

D Click to create a new set of pen mappings.

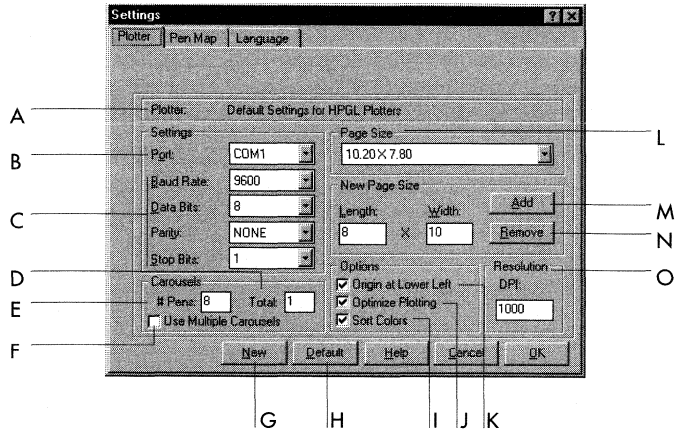
E Resets the pen map to its original settings.

F Adjusts the width of a pen. The width setting is used to create solid fills and is measured in millimeters.

G Sets the speed at which the pen moves across the paper. Speed is measured in millimeters per second. Specifying too high a speed may result in damage to the pen tip.

FOR MORE INFORMATION

PLOTTER CONNECTION SETTINGS



Opens when you click the File menu, click Plot, click Setup from the Plot dialog box, click Options from the Plot Setup dialog box, and then click the Plotter tab.

A Displays the plotter for which the current settings are listed.

B Specifies the communications port to which the plotter is connected.

C Specifies the communication settings.

D Sets the number of pen carousels to be used.

E Sets the number of pens in each carousel.

F Check this box if you are using more than one carousel.

G Creates a new plotter definition.

H Sets the current plotter driver as the default for direct plotting.

I When checked, activates pen sorting.

J When checked, activates motion optimization.

K When checked, places the origin in the lower left corner of the media. When unchecked, places it in the center of the media.

L Selects the size of the plotter media. Page size reflects the printed area on the page and not the actual sheet size.

M Adds a new page size to the Page Size list by using the values set in the Length and Width boxes.

N Deletes the current page size from the list.

O Specifies the maximum resolution of the plotter in dots per inch.

Making a custom plotter configuration

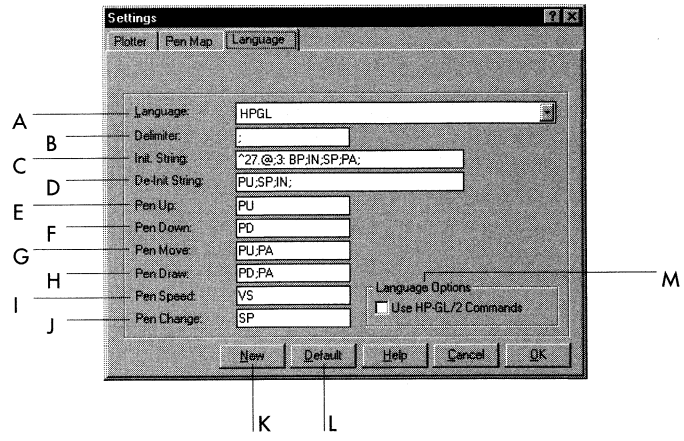
If your plotter language is not already supported by Visual CADD, you can easily create a custom plotter definition. Use the control codes listed in your plotter's documentation to set up your plotter type.

TIPS

To add a control character to a field, type the caret symbol (^) followed by a letter or up to three numerals. For example, type ^27 for ESC, or ^M for ENTER.

To adjust the number of seconds that Visual CADD will wait for your plotter to respond, change the timeout value in the Windows Registry.

PLOTTER LANGUAGE SETTINGS



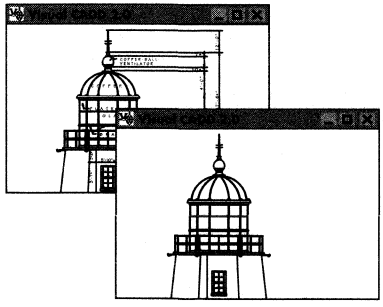
Opens when you click the File menu, click Plot, click Setup from the Plot dialog box, click Options from the Plot Setup dialog box, and then click the Language tab.

- A Specifies the plotter language for the current settings.
- B Specifies the character that separates commands sent to the plotter. This field can be left blank.
- C Describes the commands that are sent to the plotter to initialize it.
- D Describes the commands that are sent to the plotter after a plot is completed.
- E Specifies which characters raise the pen from the paper.
- F Specifies which characters lower the pen to the paper.
- G Specifies the characters that signal the plotter to move the pen from one location to another in the up position.
- H Specifies the characters that signal the plotter to move the pen from one location to another in the down position.
- I Specifies the characters that set the pen speed for the current pen.
- J Specifies the characters that signal the plotter to change to a different pen.
- K Creates a new plotter language setting.
- L Restores the plotter language settings to the default values for an HPGL (Hewlett Packard graphics language) plotter.
- M Enables the use of HP-GL/2. Check this box to improve the quality of arcs and circles and decrease plot time if your plotter supports the HP-GL/2 graphics language. If you check this box, you need to provide an Init String that tells your plotter to recognize HP-GL/2 commands.

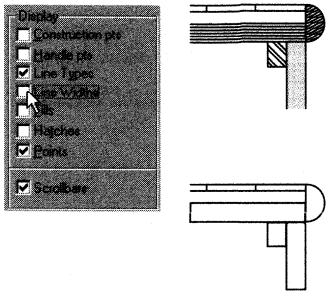
FOR MORE INFORMATION

Using direct plotting 176

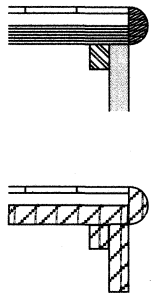
Increasing printing and plotting speed



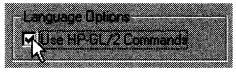
Turn off unnecessary layers. By eliminating information such as text or dimensions that you don't need in your plot, you can substantially reduce plot time. By using the same layer for all text and the same layer for all dimensions, you can easily hide these complex entities to enhance printing speed. Control the visibility of layers by clicking the Utilities menu, and then clicking Layer Manager.



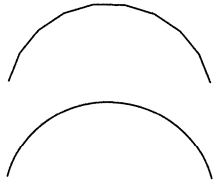
Turn off the display of fills, hatches, and line widths. By printing only the boundaries of fills and hatches, and by removing line widths, substantially less information is printed and printing time is reduced. You can access these settings by clicking the Utilities menu, clicking Settings, and then clicking the System tab.



Use simple hatch patterns. Avoid using hatch patterns that have many small elements. The smaller or more complex they are, the longer they take to plot.



Use HP-GL/2 if you can. HP-GL/2 reduces plot time by reducing the amount of information that needs to be sent to the plotter. You can access this setting from the Plot dialog box by clicking Setup, clicking Options, and then clicking the Language tab.



Set curve smoothness to a high value. Setting curve smoothness to a higher value causes Visual CADD to use fewer lines and create a rougher curve, but it increases plot speed. You can change Visual CADD's setting for curve smoothness in Visual CADD's Plot Settings folder in the Windows Registry. The value can range from 1–100.

FOR MORE INFORMATION

Appendix: Keyboard shortcuts and native commands

Command	Shortcut	Native command
Align	AG	Align
Aligned Dimension	D3	DimAlign
All Layers Edit	AL	AllLayEdit
All Layers Edit Off		AllLayEdOff
All Layers Edit On		AllLayEdOn
All Layers Snap On		AllLaySnOff
All Layers Snap On		AllLaySnOn
Angular Dimension	DA	DimAng
Arrange Icons		WinArrange
Array Copy	AC	ArrayCopy
Assign Script	AS	ScriptAssign
Attribute Attach	TA	AttAttach
Attribute Create	TC	AttCreate
Attribute Embed	TD	AttEmbed
AutoFillet	AF	AutoFillet
AutoFillet Off		AutFilletOff
AutoFillet On		AutFilletOn
Backward Redraw	BA	BackRD
Bird's-Eye View	ZB	BirdsEye
Boundary Fill	FB	FillBnd
Boundary Hatch	HB	HatchBnd
Break	BR	Break
Cascade Windows		WinCascade
Chamfer	CH	Chamfer
Chamfer Distance	CO	ChamferDist
Chamfer Distance 1	CML	ChamDist1
Chamfer Distance 2	CMR	ChamDist2
Change	CG	Change
Circumscribe Regular Polygon		RPolyCrcm
Clear Drawing	. DX	Clear
Clear Selection List	CS	SelClear
Close	FC	FileClose
Close Contour	CC	CloseContr
Color Property	CP	ColorProp
Constraints Tab	TBC	TabCnstrnt
Continuous Bézier Curve	BC	BezierCont

Command	Shortcut	Native command
Continuous Line	LC	LineCont
Copy to Clipboard		CBCopy
Copy to Current Layer	CY	CopyLay
Create Symbols	YC	SymCreate
Cursor Free	CF	CursorFree
Cursor Free Off		CursFreeOff
Cursor Free On		CursFreeOn
Cut to Clipboard		CBCut
Datum Dimension	DU	
Datum Dimension Off		DatumOff
Datum Dimension On		DatumOn
Datum X		DatumX
Datum XY		DatumXY
Datum Y		Datum Y
Diameter Circle	CD	CircDiam
Diameter Dimension	DD	DimDia
Digitizer Mode	GM	DigMode
Digitizer Scale	GZ	DigScale
Dimension Arrow Settings	DMA	DimArrowSet
Dimension at Angle	D4	DimAtAngle
Dimension Display Settings	DMD	DimDispSet
Dimension Extension Settings	DMX	DimExtSet
Dimension Leader Settings	DME	DimLeadSet
Dimension Line Settings	DML	DimLineSet
Dimension: Proximity Fixed	PF	ProxFixed
Dimension Scale	DMZ	DimScaleSet
Dimension String Settings	DMS	DimStrSet
Dimension Tab	TBD	TabDim
Dim Text Alignment	DMG	DimTextAlign
Dimension Text Settings	DMT	DimTextSet
Dimension Text Tab	TBX	TabDimText
Dimension Tolerance Settings	DMO	DimTolSet
Dim Mode: Single	D5	DimSingle
Dim Mode: Cumulative	D6	DimCumul
Dim Mode: Partitioned	D7	DimPart
Display Construction Pts	DC	ConstPts

Command	Shortcut	Native command
Display Handle Pts	DH	HandlePts
Display Layer	YD	LayDisplay
DLL Command Line variable		DllCmdLine
Dll Function Name variable		DllFunName
Dll Name variable		DllName
Dll Run		DllRun
Double line	LD	LineDbl
Double Line Fill Off		SolidOff
Double Line Fill On		SolidOn
Double Line Offset 1	WWL	WallWidth1
Double Line Offset 2	WWR	WallWidth2
Double Line Settings	DB	DBSet
Drawing Align	GA	DigAlign
Edit Dimension	DE	DimEdit
Ellipse	EP	Ellipse
Elliptical Arc	EA	EllArc
Entity Tab	TBE	TabEntity
Erase	ER	Erase
Erase Last	EL	EraseLast
Exename Variable		ExeName
Exit	FX	FileExit
Explode	EX	Explode
Explode Continuous Line	CX	ContLineEx
Explode Symbols		SymExplode
Extend Multiple	XM	MExtend
Extend Single	XT	Extend
File Run	FU	FileRun
Fill Color		FillColor
Fill Selected	FS	FillSel
Fillet	FI	Fillet
Fillet Radius	FR	FilletRad
Fillet Radius Variable		FilletRVar
Fit Scale	FT	FitScale
Fixed Offset Off		OffsetFixOff
Fixed Offset On		OffsetFixOn
Flip Arrow	AR	ArrowFlip

Command	Shortcut	Native command
Font Converter		FontConv
General Tab	TBG	TabGeneral
Grid Display	GR	GridDisp
Grid Origin	GO	GridOrg
Grid Size	GS	GridSize
Grid Size X		GridSizeX
Grid Size Y		GridSizeY
Hatch Color		HatchColor
Hatch Name		HatchName
Hatch Rotation		HatchRot
Hatch Scale		HatchScale
Hatch Selected	HS	HatchSel
Hatch Settings	HT	HatchSet
Hatch/Fill Tab	TBH	TabHatch
Help		Help
Help, About		HelpAbout
Help, Index		HelpIndex
Help, Technical Support		HelpTech
HelpKey		Help, Search
Hide Layer	YH	LayHide
Horizontal Dimension	DI	DimHorz
Import/Export Tab		TabImpExp
Incremental Snap		IncSnap
Incremental Snap Off		IncSnapOff
Incremental Snap On		IncSnapOn
Incremental Snap Size		IncSnapSize
Inscribe Regular Polygon		RPolyIn
Intersection Trim	IT	IntTrim
Invert Selection List	IS	SelInvert
Irregular Polygon	IP	IPoly
Last Symbol	YL	SymLast
Layer Manager	MGL	LayMgr
Layer Property	CL	LayerProp
Layer Properties speed bar	LYP	LPDlg
Leader	LE	Leader
Linear Copy	CO	Copy

Command	Shortcut	Native command
Linear Dimension	DL	DimLin
Linetype Property	TP	TypeProp
Linetype Scale Device		LTScaleD
Linetype Scale World		LTScaleW
Linewidth Property	WP	WidthProp
Load ASCII Text	LA	LoadAscii
Load Attribute	TO	AttOpen
Load Menu	LM	LoadMenu
Load Style	TY	LoadStyle
Load Symbol	YO	SymOpen
Manual Entry: Absolute	MO	Absolute
Manual Entry: BasePoint	MB	Basepoint
Manual Entry: Relative	MR	Relative
Match Entity	ME	MatchEnt
Match Tool	MT	MatchTool
Measure Area	MA	MeasArea
Measure Distance	MD	MeasDist
Merge	FM	FileMerge
Midline tool	ML	Midline
Mirror	MI	Mirror
Move	MV	Move
Move Dimension Text		DimMoveTxt
Move Point	MP	MovePt
Multiple Copy	MC	MultiCopy
Name File		FileName
Name View	NV	NameView
New	FN	FileNew
New Symbol Handle	NH	NewHandle
Number of Copies		NumCopies
Number of Regular Polygon Sides		RPolySides
Number of Rows		NumRows
Numeric Tab	TBN	TabNumeric
Object Information	OI	ObjInfo
Offset	OF	Offset
Offset Distance		OffsetDist
Open	FO	FileOpen

Command	Shortcut	Native command
Ordinate Dimension	DO	DimOrd
Ortho Angle Variable		OrAngVar
Ortho Mode	OR	OrthoMode
Ortho Off		OrthoOff
Ortho On		OrthoOn
Ortho Settings	OA	OrthoSet
Pack Data	PD	PackData
Pan	PA	ZmPan
Paste from Clipboard		CBPaste
Path Tab	TBP	TabPath
Path: Sys		SYSPath
Path: DWG		DWGPath
Path: DXF		DXFPath
Path: VCD		VCDPath
Path: VCF		VCFPath
Path: VCS		VCSPath
Pen Up	PU	Penup
Place Symbol	YP	SymPlace
Plot	PL	FilePlot
Point	PO	Point
Preview Fillet Off		FilletPrvOff
Preview Fillet On		FilletPrvOn
Print	PR	FilePrint
Print Setup	PT	PrintSetup
Properties Button	PP	Properties
Quick Search	QS	QSearch
Radial Copies		RadCopies
Radial Copy	RC	RadCopy
Radial Dimension	DR	DimRad
Radial Span Angle		RadSpanAngle
Redo	RE	Redo
Redraw	RD	Regen
Redraw Window	RW	RegenArea
Reference Frame: Create	RF	RFCreate
Reference Frame: Display Boundary	RB	RFDispBd
Reference Frame: Place	RP	RFPlace

Command	Shortcut	Native command
Reference Frame: Transparent	RT	RFTransparent
Regular Polygon-Center	PC	RPolyCen
Regular Polygon-Circumscribed		RPolyCrcm
Regular Polygon-Side	PS	RPolySide
Reset		Reset
Rotate	RO	Rotate
Run "EXENAME"	RUN	WinExec
Save	DS	FileSave
Save As	FA	FileSaveAs
Save Attribute	TS	AttSave
Save Environment	EN	SaveEnv
Save Style	TV	SaveStyle
Save Symbol	YS	SymSave
Scale	SZ	Scale
Seed Fill	FD	SeedFill
Seed Hatch	HD	SeedHatch
Select Adjoining	SJ	SelAdj
Select All	SA	SelAll
Select Crossing	SX	SelCross
Select Last	SL	SelLast
Select Layer	SY	SelLay
Select Object	SB	SelObj
Select Window	SW	SelWin
Selection Filter	SF	Filter
Selection Highlight On/Off	HI	Hilite
Selection Set		SelSet
Selection Speed Bar	S1	SelRibalog
Selection Tool	SE	Selection
Send	SD	FileSend
Set Angle		SetAngle
Set BasePoint	BP	SetBasePt
Set Current Color		SetColor
Set Current Layer		SetLayer
Set Current Linetype		SetType
Set Current Linewidth		SetWidth
Set Rotation Angle		SetAngle

Command	Shortcut	Native command
Set Scale X		SetScaleX
Set Scale X & Y		SetScaleXY
Set Scale Y		SetScaleY
Single Bézier Curve	BS	BezierSingle
Single line	LS	LineSingle
Snap between 2 Points	S2	SnMid2Pts
Snap Center	SN	SnCenter
Snap Closest Point	SC	SnClosestPt
Snap Grid	SG	SnapGrid
Snap Intersection	SI	SnIntersect
Snap Last Point	LP	SnLastPt
Snap Layer	NL	SnapLayer
Snap MidPoint	SM	SnMidPt
Snap Near Point	NP	SnNearPt
Snap Object	SO	SnObject
Snap Parallel	LL	SnPara
Snap Percentage	SR	SnPercent
Snap Percentage Value		PercSnapVal
Snap Perpendicular	SP	SnPerp
Snap Quadrant	SQ	SnQuad
Snap Tangent	ST	SnTangent
Snap to Curve Tangent Pts	DV	CurveTanPts
Spline Curve	CV	Curve
Stretch	SS	Stretch
Symbol Count		SymCount
Symbol Explode	YX	SymExp
Symbol Manager	MGY	SymMgr
Symbol Name		SymName
Symbol Remove	YV	SymRemove
Symbol Replace	YR	SymReplace
Symbol Rotation		SymRot
Symbol Scale		SymScale
Symbol Scale X		SymScX
Symbol Scale Y		SymScY
Symbol Snap	YN	SymSnap
Symbol Snap Off		SymSnapOff

Keyboard shortcuts and native commands

Command	Shortcut	Native command
Symbol Snap On		SymSnapOn
System Tab	TBS	TabSystem
Tab Settings	TBO	TabOptions
Text Aspect		TextAspect
Text Bold		TextBold
Text Character Spacing		TextChSp
Text Color		TextColor
Text Editor	TE	TextEditor
Text Font		TextFont
Text Height		TextHeight
Text Italic		TextItalic
Text Justification		TextJust
Text Layer		TextLay
Text Line	TL	Text
Text Linespacing		TextLnSp
Text Rotation		TextRot
Text Settings	TT	TextSet
Text String		TextStr
Text Tab	TBT	TabText
3-Point Arc	A3	Arc3
3-Point Circle	C3	Circle3
3-Point Rectangle	R3	Rect3
Tile Windows Horizontally		WinHoriz
Tile Windows Vertically		WinVert
Track	TK	Track
Trim Multiple	TM	MTrim
Trim, Single	TR	Trim
2-Point Arc	A2	Arc2
2-Point Circle	C2	Circle2
2-Point Rectangle	R2	Rect2
Undo	OO	Undo
Undo Dimension	UD	UndoDim
Undo Vertex	UV	UndoVertex
Update Dialog		Update
User Interface Off		UIOff
User Interface On		UIOn

Command	Shortcut	Native command
Vertical Dimension	D2	DimVert
Window: Horizontal		WinHoriz
Window Stretch	WS	WinStretch
Window: Vertical		WinVertZoom All
ZA	ZmAll	
Zoom In	ZI	ZmIn
Zoom Out	ZO	ZmOut
Zoom Previous	ZP	ZmPrev
Zoom Selected	ZS	ZmSel
Zoom Value	ZU	ZmValue
Zoom View	ZN	ZmView
Zoom Window	ZW	ZmArea

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Colophon

The books in this set were created electronically using Numera Visual CADD, Adobe PageMaker, Adobe Photoshop, Macromedia FreeHand, and ULead Systems ImagePals for Windows and Macromedia FreeHand on an Apple Macintosh. The Futura and Bauer Bodoni families of typefaces, from the library of Adobe Systems, Inc., are used throughout the books.