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View Window

Vellum allows you to look at your project from more than one point of view at a time. The view may be of your entire drawing or of a detail of the drawing. It may be of a different scale or orientation from the original drawing.

The new view appears in its own window, which you may modify using the view window's pop-up submenu. You may also move and resize the window in the same manner as other windows are modified.

To access the pop-up submenu, click on the button in the corner of the view window and hold down the mouse button.

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Properties

This command allows you to move the location of the view window, resize the window, change the scale of the view in the window, name or change the name of the view.

A dialogue box will appear when you choose this command.

"Name"

This command allows you to name or rename the view window by typing in this box. Click on "OK" or hit "Enter" when you are done typing.

"Scale"

This command changes the scale of the geometry shown in the view window. The status line box will show the current scale, which is a ratio to the existing scale, i.e., a scale of 1 is the same as the current view, .75 is three quarters the size of the current view, and 2 is twice the size of the current view. Click on "OK" or hit "Enter" when you have chosen the scale you want.

You can also use the "Zoom" commands within a view window to change the scale of your drawing. See below.

"Left, Right, Top, Bottom"

These properties define the corners of the view window as coordinates from the center of the sheet. A negative number would be to the left or below the center while a positive number would be to the right or above the center. Setting these values can resize as well as move the view window.

You may also resize or move the view window in two other ways. To resize using the "Resize" command, see below.

You may also move and resize the view window by moving the cursor to the outer edges of the window. When the cursor appears as a double-pointed arrow on either side or the bottom of the window, you may resize it. When the cursor appears as a double-pointed arrow at the top of the window you may move the window without changing its size.

Cut

This command deletes the active window, but saves it on a clipboard. The window can be pasted elsewhere, until a new cut is made, replacing the original cut on the clipboard. To recover the window, use the "Paste" command from the "Edit" menu. The new window will appear a little offset from the original location and can be dragged to another location or moved using the [Properties](#) command. See "Cut" and "Paste" under the "Edit" menu.

Copy

This command allows you to duplicate a view window and all its attributes. Select the copy command and then use the "Paste" command from the Edit menu. The new window will appear a little offset from the original location and can be dragged to another location or moved using the [Properties](#) command.

Delete

This command deletes the view window. It can not be undone. You may be able to recreate a view accidentally deleted using the trackball pop-up submenu or the "Views" command pop-up submenu if the view was one of those in the submenu.

Pan

This command allows you to move the geometry within a view window. When you choose this command the cursor will change to a hand icon, which you can use to drag the drawing to the desired location in the view window.

Resize

This command allows you to resize or move a view window. When you choose the command the icon will look like the movie camera icon from the "Detail View" command. Click in one corner of the desired view location and drag to the opposite corner.

You can also use the "Zoom" commands within a view window to change the scale of your drawing. To make the commands apply to the view window, hold down the shift key while choosing the desired command from the "Arrange" menu using the mouse.

Pattern

This command changes the appearance of selected lines. If no lines have been selected, the new pattern will be applied to any new lines drawn. To select lines, chose the select tool and click and drag the mouse to draw a box around the lines you want to change.

To use this command, choose it from the "Pen" menu. A submenu will appear. Move the cursor to the desired pattern and release the mouse button. A check mark will appear beside the pattern currently in use.

Choices are: Solid, Dotted, Center, Dashed, Hidden, Phantom, Dashdot, Border, Divide, Cutting, Stitch.

Menus

This Help section describes the commands on Vellums menus available from the menu bar:

File Edit Layout Arrange Pen Text Dimension Views 3D Help

These commands perform filing tasks or help you create and configure your drawings. The commands are generally available, even when you are using the tools from the palette. For example, when you are in the process of constructing connected lines, you can create a custom construction line by choosing Construction from the Layout menu.

When you display menus, you may notice that some commands are displayed in dimmed text which indicates that these commands are not currently available. For example, Undo is not available unless you have previously done something that can be undone.

Some commands toggle between conditions, such as Show Grid and Hide Grid. To switch to the other condition, simply choose the command again. Other commands, such as Align Left from the Text menu, display a check mark to indicate which condition is in effect.

File Menu

[New](#)
[Open](#)
[Close](#)
[Save](#)
[Save As](#)
[Symbol](#)
[Import](#)
[Export](#)
[Preview Layout](#)
[Print Setup](#)
[Print](#)
[Exit](#)

File	
N ew	Ctrl+N
O pen...	Ctrl+O
C lose	Ctrl+F4
S ave	Ctrl+S
Save A s...	
Symbol...	
Import...	
Export...	
Preview L ayout	
P r int Setup...	
P rint...	Ctrl+P
E xit	Ctrl+Q

The commands on this menu allow you to perform document-level activities such as opening and saving files. These documents can be Vellum files, parametric Vellum symbol files, or files from other applications. You will find more information on using the standard Windows commands on this menu in the ***Microsoft Windows User's Guide***.

New

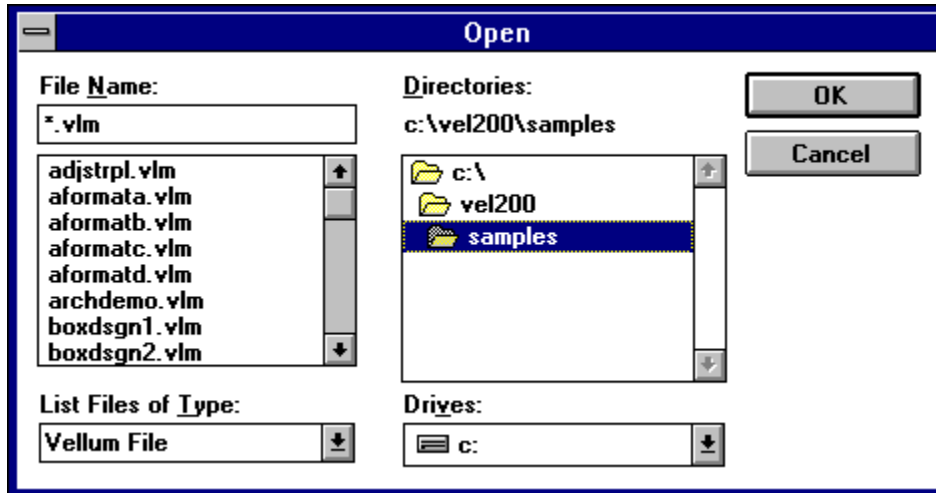
This command creates a new Vellum document.

The new document is untitled (the title bar displays a name such as ***Untitled 1***), and the default (preset) options, such as pen style or Grid Off, are set.

If you open more than one new document, the subsequent documents are numbered sequentially until you name them by saving.

Open

This command opens an existing Vellum document. The document is displayed in the drawing area maintaining the same settings as the last time you saved it. If you made any changes to default settings, such as editing pen styles or hiding layers, those conditions remain in effect.



The dialog box allows you to specify the document you want and lets you change directories and disks, if necessary.

To open a document:

1. Double-click the appropriate **disk** and/or **directory** in the **Directories** list box.
2. Double-click the **filename** you want to open in the **Files** list box.

You can also type the name of the file you want and use wild cards to display matching names in the files list box.

You can also choose the **Open** command by holding down the Ctrl key while pressing the letter **o**.

Close

Closes the current Vellum document (the one displayed in the top window); if only one document is open, the **Close** command also quits from Vellum and returns you to Windows. If other Vellum documents are open, they are unaffected by closing the current document. If you have made any changes since you last saved the current document, a dialog box allows you to save the current version. You can close the document with or without saving the changes.

You can also choose the **Close** command by holding down the Ctrl key while pressing the letter **w** or by double-clicking the Control Menu box at the left side of the title bar.

Save

Saves the current Vellum document to its original directory and disk. If you want to save it to a different directory or disk or with a different name, choose **Save As....** If you choose **Save...** and the document has not been saved previously, the **Save As...** dialog box displays automatically, allowing you to name the document and specify the disk and directory in which to save it.

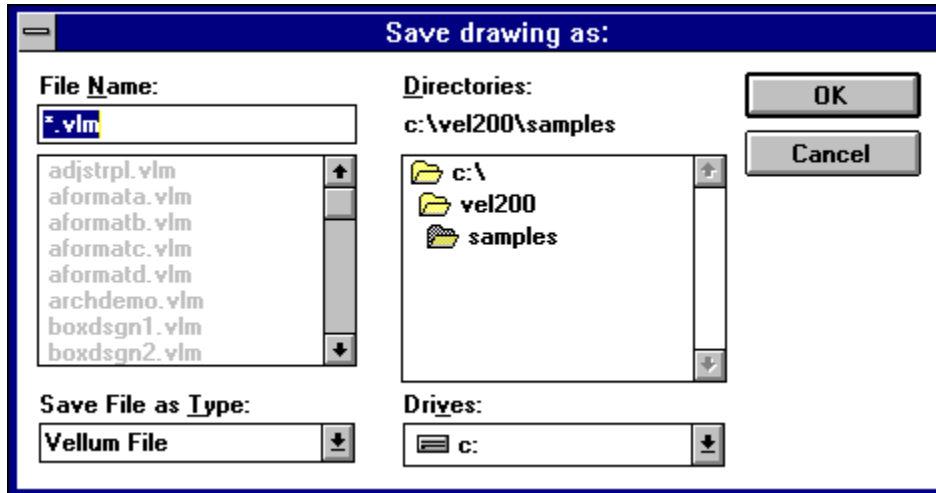
If you have previously named and saved the document, no messages appear when you choose **Save**, but the program pauses while it updates the information on the disk.

You should save frequently. *Even though your drawing appears on the screen, it is not stored on the disk until you save it; therefore, hours of work could be lost because of a power failure. It is especially important to save before performing any intricate, multi-step maneuver. In that way, if the result is not exactly what you had hoped, you can abandon the file by closing without saving and try again.*

You can also choose the **Save** command by holding down the Ctrl key while pressing the letter **s**.

Save As

Saves the current document. A dialog box appears so that you can name the current document, give it a different name, or save it to a different disk or directory.



To save in the current directory:

1. Type the **name** you want to use in the **Filename** box.
2. Either press **Return** or click **Save**.

The filename extension **.VLM** is automatically appended to the filename.

To save the document in a different directory or disk drive:

1. Double-click the **directory** or **disk drive** listed in the **Directories** List box.
2. Enter the **name** of the document in the **Filename** box.
3. Click **Save**.

You can use the **Save As...** command to make a backup of a document. You should back up documents at least once a day. It is best to save the backup document on a different disk. If you are using a hard disk and do not back up individual documents on floppy disks, be sure to back up your hard disk often.

Symbol

Copies the specified document into the current document.

Create symbol documents for generic parts so that you can use them repeatedly, modifying the dimensions each time. Use the dialog box that resembles the **Open...** dialog box.

To create a document to use as a symbol:

1. Construct the geometry.
2. Dimension all aspects of the part by entering a variable in the text box of each dimension.
3. Choose **Save As...** from the **File** menu.
4. Name the document.

To display the symbol:

1. Choose the **Symbol...** command from the **File** menu.
2. Specify the symbol file you want to use.

An outline of the geometry and variable entry boxes are displayed. To see an enlargement of any part of the geometry, move the pointer to the area you want to see and press the mouse button. The enlargement reduces when you release the mouse button.

3. Enter values into the variable boxes.
4. Click **OK**.

The geometry is copied into the current document as you specified.

Smart Symbols

The directory named Architect in the Samples directory contains smart symbols for doors and windows. When you add one of these symbols to a smart wall, it breaks into the wall in the location you specify.

Import

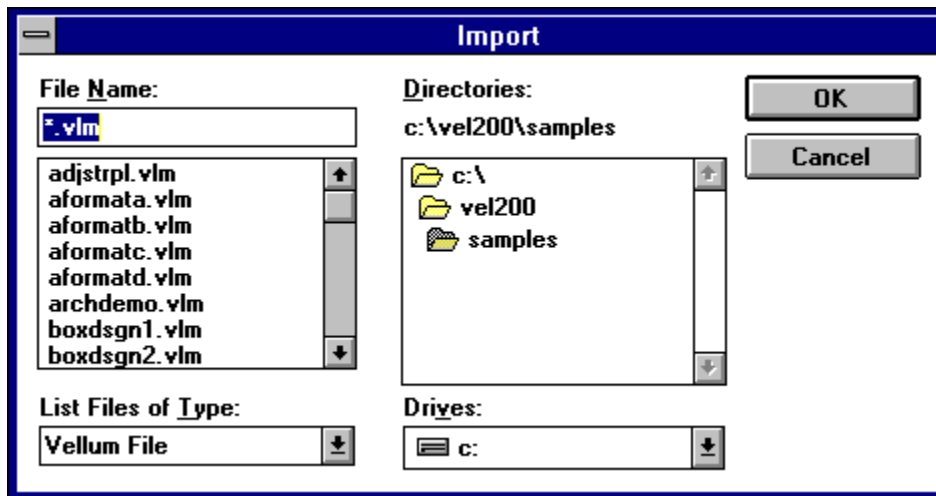
This command imports a document or drawing into your current Vellum file. The names of documents or drawings that are in a format Vellum can read are displayed in the dialog box that appears when you choose this command.

The available file types and available drive choices will be displayed in drop down menus. To use the boxes, click on the arrow in the upper right corner. Move the cursor, or scroll, until your choice is highlighted and then click on that choice.

To import a drawing from another document:

1. Choose **Import...** from the **File** menu.

The dialog box displays.



2. Go to the bottom left corner drop-down list and choose the types of files you wish to import from the list of those Vellum can read. The menu box in the upper left corner will list all files of that type.
3. Choose the file you wish to import from the upper menu box by double-clicking on the file name. If there are no files of the type you chose, no file names will appear in the box. If you don't see the file you want, try changing the drive using the right drop down menu.
4. Click the **OK** button.
5. Another dialog box will appear. Choose any options you wish for the file you are importing by clicking in the box next to that option. If the option has been selected an "X" will appear in the box next to it. To de-select it, click in the box again.

Available options are:

"Group" (combines all imported items and treats them as one item).

"Geometry Only" (imports only lines and curves, not text or dimensions or other non geometric information).

"Onto Work Layer" (puts the imported items onto the current work layer; otherwise the program will try to find the logical layer to place it on).

"Unscale" (changes the scale of the imported document to fit your current scale; useful for importing drawing formats)

6. Click the **OK** button.

The file you have chosen will be imported into your current file.

Text will be entered in a text box in the drawing.

Export

Saves a document in the format you specify.

1. Choose **Export** from the **File** menu.
2. Click the **button** to indicate the format for saving the document.
3. Click **OK**.

Saving Vellum files in any other format will cause you to lose some data from your file.



Vellum

The format for Vellum documents.

To save a portion of your current document, select the geometry you want to save and choose Export, specifying the Vellum format and Only Selected.

DXF

The format used by AutoCAD and other CAD applications for microcomputers. If you choose the DXF format, the filename extension .DXF is automatically appended to the current filename. If you change the name of the file, be certain that you include the .DXF extension.

IGES

Initial Graphics Exchange Specification, version 4, the National Bureau of Standards format to standardize the exchange of graphics.

Text

ASCII characters. This option is available only if the Text tool is the active tool.

META

The format used by the Windows clipboard, an interchange format specifically used for graphics commands. A document saved in this manner contains only the routines to reproduce the drawing as it appears on the screen. The filename extension .WMF is automatically appended to the filename.

Bitmap

Microsoft Windows bitmap file format.

If you specify **Only Selected**, only those objects that are selected in your drawing will be saved in the export format. If you do not specify **Only Selected**, all objects are saved for export.

Preview Layout

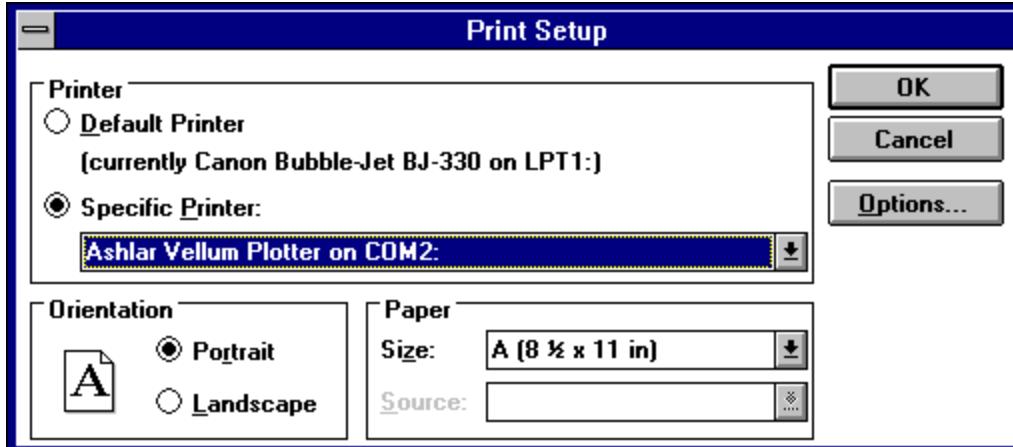
Shows the size and orientation of the printing area as you have specified in [Print Setup](#). If you have not specified a printing/plotting size, the default size is used.

If the drawing does not fit the paper size, use the [Drawing Size...](#) command on the Layout menu to scale the drawing.

Print Setup

This command allows you to select or change the printer or plotter used to print a Vellum drawing. You may also use this command to change the orientation of paper in the printer, the paper size, paper source, or to adjust other options specific to the printer being used such as darkness or quality of shading.

The dialog box that displays when you choose "Page Setup..." is shown below:



Choose the printer you want to use by clicking on the button in front of either "Default Printer" or "Specific Printer."

The "**Default printer**" is the printer you have selected in the Windows Control Panel using the Printer icon. If you have selected this option and you later change your default printer in the Control Panel, the printer used with your Vellum documents will automatically change.

If you choose "**Specific Printer**" the printer you specify will be used whenever you print this Vellum file, unless you change it using the "**Page Setup**" command. To make the chosen printer the one used with all your Vellum files, use the "**Save Preferences**" command in the "**Layout**" Menu.

To see what printers are available, click on the arrow in the upper right corner of the drop down menu below the "**Specific Printer**" button. Move the cursor until the printer of your choice is highlighted and then click on that choice.

If the printer or plotter you wish to use is not listed, you must install it using the printer section of the Windows Control Panel.

The "**Orientation**" option allows you to change the orientation of the printed image on the paper to "**Portrait**", with the short side of the paper on top, or "**Landscape**", with the long side of the paper on top. The graphic with the option will change to show the orientation you have chosen.

The "**Paper**" option allows you to select the paper size or source that will be used with the chosen printer. The options available with your printer will be displayed in a drop down menu. To use the box, click on the arrow in the upper right corner. Move the cursor until your choice is highlighted and then click on that choice. If there is no choice in paper size or paper source with the chosen printer, the drop down menu for that option will appear in light type and can not be used.

Click the "**Options**" button to display options specific to the chosen printer.

Consult the manual for the printer or plotter or the *Windows User's Manual* for details concerning the available options.

When you have finished setting your print options, click on the "OK" button to put your choices into effect.

Print

This command prints or plots your file.

The dialog box called up when you choose this command shows the printer you have chosen using **Print Options**. If you wish to change the printer or other options not in this dialog box click on "**Print Options**" to call up that dialog box.

The "**Print range**" command allows you to choose the number of pages of your Vellum file you wish to print. You may specify any page to start printing on and any page to print to. If the number you type in is not valid, you will receive an error message.

The "**Print Quality**" command allows you to specify the resolution (dots per inch) you wish the file to be printed at. The options available with your printer will be displayed in a drop down menu. To use the box, click on the arrow in the upper right corner. Move the cursor until your choice is highlighted and then click on that choice.

The "**Print to file**" command allows you to save a file that has been formatted for printing for later use with a printer.

The "**Copies**" command specifies how many copies you want printed. One is the default number, if you want more type in the number you want."

"**Collate copies**" can be toggled on or off if you are printing multiple copies of a multiple page document. If the box next to the command has an "X" in it, collating is on; if it does not it is off. Click in the box to toggle it from on to off.

Click on the "**Set-up**" button if you wish to change printing options not shown in this dialog box.

The area printed or plotted is the part of the drawing chosen using the "**Drawing Size**" command from the "**Layout**" menu to scale your drawing to the appropriate size. You can specify tiling (printing on several pages to be pasted together) through the **Drawing Size** command.

You can also choose the **Print...** command by holding down the Ctrl key while pressing the letter **p**.

Exit

Closes Vellum and returns to Windows. If you have made changes since you last saved, a dialog box allows you to save the changes. If you have more than one document open, you are asked to save each unsaved document.

Edit Menu

[Undo](#)
[Redo](#)
[Cut](#)
[Copy](#)
[Paste](#)
[Delete](#)
[Select All](#)
[Selection Mask](#)
[Selectable Points](#)
[Linear Duplicate](#)
[Polar Duplicate](#)
[Edit Objects](#)
[Resolve](#)

Edit	
U ndo	Ctrl+Z
R edo	Shift+Ctrl+Z
C ut	Ctrl+X
C opy	Ctrl+C
P aste	Ctrl+V
D elete	
S elect All	Ctrl+A
S election M ask...	
<input checked="" type="checkbox"/> S electable Points	
L inear Duplicate...	
P olar Duplicate...	
E dit Objects... Ctrl+I	
R esolve...	

The commands on this menu allow you to make changes to the text and geometry you create.

Undo

Reverses editing. You can Undo and Redo any of the last eight editing actions. For example, if you delete an object, you can choose Undo to restore it. You can then use Redo to return to the deleted version.

You can undo actions that create and edit geometry and text, but not actions that do not change the contents of the drawing, such as re-sizing the window or quitting.

If you are using a tool which involves a multi-step process, such as constructing connected lines or creating a 3-point fillet, choosing Undo reverts to the beginning of the process. Some activities, can be stopped by pressing the Esc key.

You can also choose this command by holding down the Ctrl key while pressing the letter z.

Redo

Reinstates the last action reversed by Undo. You can Undo and Redo any of the last seven editing actions.

Cut

This command deletes the selection and places it onto the Clipboard. Each addition to the Clipboard replaces the previous Clipboard contents.

1. Select the **object** to be cut.
2. Choose **Cut** from the **Edit** menu.

Once a selection is cut, you can paste it into a document. You can use **Cut** and **Paste** to move geometry or text around the document, from one sheet to another, or from one document to another. You can even paste the cut selection into a document created with a different application.

You can also erase selections with the Delete or Backspace key or by choosing Delete from the Edit menu. Selections erased in this way are not placed on the Clipboard and, therefore, cannot be pasted. They can be restored, however, by using **Undo**.

Moving geometry with the Cut command:

1. Select the **objects** you want to move.
2. Choose **Cut** from the **Edit** menu.
3. Display the destination on the screen.
4. Choose **Paste** from the **Edit** menu.

The object is pasted in the center of the screen. The object is selected so that you can move it, if you want.

If you want to move an object from one layer to another, use the **Edit Objects...** command on the Edit menu.

Copy

This command places a copy of the selection onto the Clipboard without deleting the original. You can paste the copy elsewhere in the current document or into a different document. You can even paste the copied selection into a document created with a different application.

You can also create a copy of an object or text with the Selection tool. First select the object, then hold down the Ctrl key and drag a copy of the object to a new location.

Paste

Pastes a copy of the Clipboard contents onto the center of the screen on top of other geometry or text. The Clipboard contents are not changed when you use the **Paste** command.

If the object is cut or copied from a Vellum document, the pasted object remains on the layer it was on when it was cut or copied. A cut or copied object retains its original attributes (layer and pen color, weight, and pattern) when it is pasted. If the pasted object was cut or copied from a non-Vellum document, it will be placed on the work layer.

You can also paste the selection into another document or into a document created with another application.

If you want to create an even distribution of geometry, you can use **Linear Duplicate...** or **Polar Duplicate...** on the Edit menu.

Delete

Deletes a selection without placing it onto the Clipboard. You can undo this deletion, but you cannot paste what is deleted. In addition to this command, you can use the Delete or Backspace key to delete a selection.

Erasing Geometry

You can use several methods to erase selections.

You can press the **Delete** or **Backspace** key. You can retrieve what you deleted with the **Undo** command. As many as eight undo actions can be performed.

You can choose the **Delete** command from the Edit menu. You can retrieve what you deleted with the **Undo** command.

You can choose **Cut** from the Edit menu. You can retrieve what you cut with the **Paste** command, *as long as you have not cut or copied anything else.*

Select All

Selects all objects except those on a hidden layer or excluded by the Selection Mask. This command is useful if you want to make a global change in a drawing, such as changing the width of all lines.

If you choose Select All while using the Text tool, all characters in the current text area are selected.

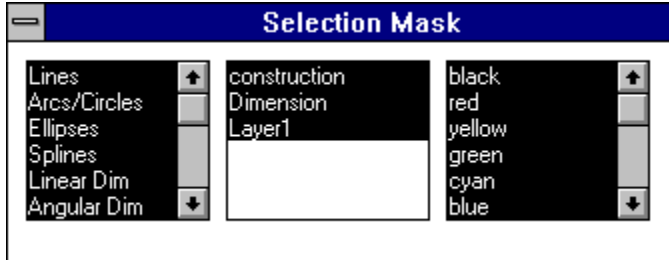
You can set the appearance of a selected object on the Selections submenu of the Preferences command in the Edit menu where you can specify blinking, color, or both to indicate that the object is selected.

You can specify that only certain objects, layers, or colors are selected by setting a selection mask with the Selection Mask... command.

You can also choose **Select All** by holding down the Ctrl key while pressing the letter **a** or by double-clicking the Selection tool.

Selection Mask

Allows you to limit selection by object type, layer, and color. Only those specifications that are highlighted will be selected by the various selection methods. You can select the entire category or individual specifications.



When an item in the list is highlighted you can select only that type of item. If circles are not highlighted in the Selection Mask dialog box, and you choose Select All from the Edit menu, everything but the circles will be selected. In this way, you can select such choices as only blue splines or only the red objects on a particular layer.

Using the Selection Mask

1. Choose **Selection Mask...** from the **Edit** menu.

The Selection Mask dialog box appears. The highlighted items will respond to all selection methods and can be detected by the Drafting Assistant.

2. Click the items you want to be able to select so that they are highlighted.

You can select or de-select groups of items by holding down the Shift key and clicking the first and last item in the group.

You can select and de-select non-contiguous items by holding down the Ctrl key and clicking the items.

While the dialog box is displayed, you can select, draw, and edit geometry. You can move the dialog box if it covers geometry you want to work on. The Selection Mask remains in effect even after you close the dialog box until you select a tool.

When an item in the dialog box is not highlighted, the Drafting Assistant and all the tools cannot detect it, even though it is visible to you on the screen.

This feature is useful when you have created a complex drawing and want to change particular groups of objects. You can use this feature to export some but not all geometry. You can also use the Selection Mask... to change all dimensions from parametric variables to real values, as described in the chapter on **Parametrics** in the **User Guide and Reference Manual**.

Selectable Points

Allows you to be able to select points. If this setting is off, you cannot select points by any method. If Selectable Points is on and if points are displayed, simply click the point to select it. If points are not displayed, you can select a point by dragging a selection fence around it.

Linear Duplicate

Creating an array of copies along a straight line

1. Select the **object** to be duplicated.
2. Choose **Linear Duplicate...** from the **Edit** menu.

A dialog box appears.

The dialog box titled "Linear Duplicate" contains the following fields and options:

- Number Per Row:** Input field with the value 2.
- X Length*:** Input field with the value 0.
- Y Length*:** Input field with the value 0.
- Total Lengths:** Radio button (selected).
- Step Lengths:** Radio button (unselected).
- Number Of Rows:** Input field with the value 1.
- Offset*:** Input field with the value 0.
- Total Offset:** Radio button (selected).
- Step Offset:** Radio button (unselected).
- Buttons:** Cancel and OK.

An asterisk (*) appearing next to an entry field in a dialog box indicates that the information can be filled in by using a mouse click to indicate the appropriate coordinate in the drawing area. This is a very handy and accurate way to specify this information.

3. Enter the **number of objects** per row.
4. Specify **Total** or **Step** Lengths by clicking a radio button.
5. Click the **X Length*** box.
6. Move the pointer to the drawing area and drag to indicate the length of the row of copies.

Both the X and Y values are filled in.

7. If you want more than one row, enter the number of rows, and click a radio button to indicate the type of offset.
8. Click **OK**.

The Linear Duplicate dialog box contains the following settings:

Number Per Row	The total number of objects you want in each row. Be sure to include the selected object itself in this total count.
X Length/Y Length	The X length value determines the horizontal length of a row. The Y length value determines the vertical rise of the row.
Total Lengths	The X and Y length is the distance from the selected object to the last copy.
Step Lengths	The X and Y length is the distance from the selected object to the first copy.
Number of Rows	The total number of rows that you want.

Total Offset	The perpendicular distance between the row containing the selected object and the last row.
Step Offset	The perpendicular distance between the row containing the selected object and the next row.
Offset	The perpendicular distance setting for the Total Offset or Step Offset.

Polar Duplicate

Allows you to copy and rotate the selected geometry.

You can copy the objects upright or rotated.

1. Select the **object** to be duplicated.
2. Choose **Polar Duplicate...** from the **Edit** menu.

A dialog box appears.

The dialog box titled "Polar Duplicate" contains the following elements:

- Number:** Input field with value 4.
- Center X*:** Input field with value 0.
- Center Y*:** Input field with value 0.
- Total Angle:** Radio button selected, input field with value 360°.
- Step Angle:** Radio button unselected, input field with value 90°.
- Rotated Objects:** Radio button selected.
- Upright Objects:** Radio button unselected.
- Ref X*:** Input field with value 1.
- Ref Y*:** Input field with value 0.
- Buttons:** Cancel and OK.

An asterisk (*) appearing next to an entry field in a dialog box indicates that the information can be filled in by using a mouse click to indicate the appropriate coordinate in the drawing area. This is a very handy and accurate way to specify this information.

3. Enter the **number of objects** in the circular array.
4. Click the **Center X *** box.
5. Move the pointer to the drawing area and click to indicate the center for the array of copies.

The values for Center X and Center Y are filled in.

6. Specify **Upright** or **Rotated** objects by clicking a radio button.
7. If you specified Rotated, click a location for the **reference point** on the drawing area.
8. If you do not want the copies in a complete circle, click the **Total** or **Step** Angle radio button.
9. Enter a **value** for the specified angle type.
10. Click **OK**.

The dialog box contains the following settings:

- Number:** The total number of objects you want. Be sure to include the selected object itself in this total count.
- Center X/Center Y:** The center X and Y values determine the center of the circular array you will create.
The center X and center Y can be set by entering values, or by clicking the mouse in the drawing area.
- Total Angle:** The distance between the center of the selected object and the center of the last copy.

Step Angle: The distance between the center of the selected object and the center of the first copy.

Rotated Objects: Copies are rotated relative to the angle between the copies.

Upright Objects: Copies are upright with respect to the selected object. This option requires a reference point.

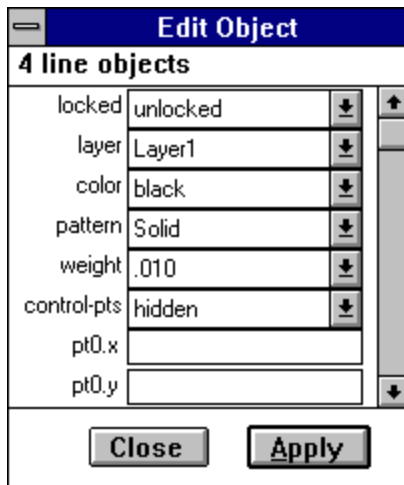
Ref X / Ref Y The reference point for the Upright Objects option.

Ref X and Ref Y determine an imaginary point that will be duplicated as specified around the center. The selected objects will be reproduced in the same position relative to each imaginary point that is duplicated.

The Ref X and Ref Y can be set by entering a value, or by clicking the mouse in the drawing area.

Edit Objects

You can edit selected objects by either changing individual characteristics, such as layer or line style, or by changing the specifications in the Edit Objects dialog box.



The only way to edit the measurements of an existing object is through the Edit Objects... command. This dialog box is a fast, easy way to make several changes at once. You can change a single characteristic faster by simply selecting the object and changing the characteristic. But if you are changing several characteristics or any measurements or positions, you should use the Edit Objects... command.

1. Select the **object** to be edited.
2. Choose **Edit Objects...** from the **Edit** menu.
3. Change the information in the entry boxes.

Double-click the entry box and type a new entry (do **not** press Enter).

Many boxes have drop-down menus for selection. Click the down arrow and then use the scroll bar to display the selection you want, then click the item you want to specify.

4. Click the **Apply** button.
5. Make any other changes you want.
6. Click the **Apply** button.
7. Double-click the **Control Menu** button to close the Edit Objects box.

Changes made through this dialog box can be reversed with the Undo and Redo commands.

The topics of information shown in this box depend on the type of object selected, and include at least the following:

Number (or type) of objects

Lock status

Current layer

Current color

Current pattern

Current weight

Status of control points.

The measurements reflect the settings of the Units option in the Preferences submenu. If you want to prevent changes to an object, you can specify "locked" in this dialog box, or you can select the object and choose Lock from the Arrange menu.

If you will be doing a lot of editing, you may want to leave the Edit Objects dialog box displayed. In that way you can select the object, make the changes in the dialog box, click Apply, and then go on to the next object.

Leaving the Edit Objects dialog box displayed during construction is not recommended.

If you select several objects and then choose Edit Objects..., all of the objects will reflect the changes you make in the dialog box. For example, if you select all dimensions and then change the text entry in the Edit Objects dialog box to "2" all of the dimensions will display a 2.

Changing an individual characteristic:

1. Select the **object** to be changed.
2. Choose **Edit Objects...** from the **Edit** menu.
3. Click the **arrow** for the characteristic you want to change.
4. Click the new **characteristic** from the list.
5. Click **Apply**.

Moving an object to a different layer:

1. Select the **object(s)** to be moved.
2. Choose **Edit Objects...** from the **Edit** menu.
3. Click the **arrow** for the layer entry box.
4. Click the **layer** from the list.
5. Click **Apply**.

Resolve

Allows you to re-draw geometry automatically to fit specified dimensions. This parametric feature enables you to draw a geometric shape without regard to measurements and then have Vellum re-draw the same shape to the values you specify.

To use parametrics:

1. Create the geometry.
2. Dimension all geometry by using variables, constant values, and expressions. (You must dimension the essential, related geometry so that Vellum can reconstruct your selection. **Note that Vellum can identify parallel but not collinear lines.**)

Select the appropriate dimensioning tool.

Click the geometry to be dimensioned.

Type an expression (such as 1.5, x, $x+3*y$) into the text box on the status line and press Return.

3. Select the geometry and dimensions to be resolved.
4. Choose **Resolve...** from the **Edit** menu.
5. Enter the **values** you want to assign to the variables.
6. If you want to indicate a point in the drawing to remain fixed, click that point.
7. Click the **OK** button.

The geometry is re-drawn to the dimensions you specify.

If you have not given all of the dimensions required to draw the geometry, the geometry is divided into groups that can be defined. The missing information will determine how the groups relate to one another.

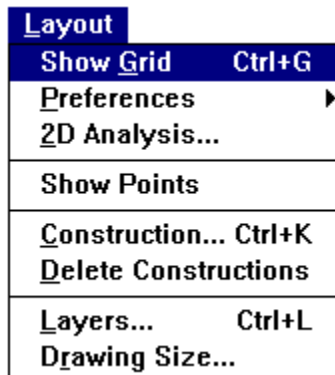
An alert box tells you how many groups are formed. To see each group, click the Next button. Examine the groups shown to determine why the position of each one is not fully defined relative to any other. A group consisting of a single point is particularly telling. (This investigation can require some clever thought because it may not be obvious.)

The **Parametrics** chapter of the **User Guide and Reference Manual** gives example and describes problems you might have with parametrics.

Note: The *Resolve* command does not apply to geometry created with the *Ellipse* or *Spline* tools.

Layout Menu

[Show/Hide Grid](#)
[Preferences](#)
[Snap](#)
[Grid](#)
[Units](#)
[Selection](#)
[DAssistant](#)
[Save Preferences](#)
[2D Analysis](#)
[Show/Hide Points](#)
[Construction...](#)
[Delete Constructions](#)
[Layers](#)
[Drawing Size](#)



The commands on this menu allow you to set the specifications for the drawing area and customize Vellums functionality to your specific needs. They also allow you to create construction lines, and perform 2D analysis.

Show/Hide Grid

Displays the grid. The command name toggles between Show Grid and Hide Grid. The grid is made up of horizontal and vertical lines of dots. The spacing of the lines of the grid and the number of tick marks between the intersections of the grid lines are set in the Grid submenu of the Preferences command on this Layout menu.

When the grid is displayed, the Drafting Assistant snaps to the grid spacing. In other words, if the Grid is set to .25 inch spacing, an object cannot be constructed closer than .25 inch to another object. If you attempt to place it closer, it will *snap* to touch the object.

You can also choose this command by holding down the Ctrl key while pressing the letter **g**.

Preferences

Displays a submenu for setting the following specifications:

[Snap](#)

[Grid](#)

[Units](#)

[Selection](#)

[DAssistant](#)

[Save Preferences](#)

Snap

Sets the hit radius and alignment angles for the Drafting Assistant.

Hit Radius

The detection distance in pixels. The pointer is detected when it is within the specified distance from an object. When the pointer is within the hit radius, the Drafting Assistant notations are displayed or the object is selected if the mouse is clicked. The default is 12 pixels.

If you are unable to specify locations that are close together because the Drafting Assistant snaps to an existing control point, you can do any of these things:

- Decrease the hit radius.

- Zoom in so that more pixels separate the existing point and the point you want to select.

- Lock on a point by pressing the mouse button and then typing the letter that represents the point (M for midpoint, for instance).

To turn off the Drafting Assistant, set the Hit Radius to zero.

Alignment Angle

The angles for the construction lines which the Drafting Assistant automatically displays. If you want to change the orientation of your drawing, you can change these specifications; for example, you could set these angles to 30° and 150° for an isometric drawing. The defaults are 0° (horizontal) and 90° (vertical). The values must be separated by a semicolon.

Line Angle

The angles of the Drafting Assistant lines between the horizontal and vertical construction lines defined by the alignment angle. The defaults are 45° and 135°. The values for multiple construction lines must be separated by a semicolon.

To display a construction line through a point, move the pointer to the point to activate the point (a diamond displays); construction lines automatically display through the active point. You can have as many as eight active points. When you activate the ninth point, the first one in the series is deactivated.

% Point

The divisions of a line for Drafting Assistant notations. If you want the midpoint notation to divide the line into quarters, use a 25 specification. The default is 50, which shows the halfway point of lines.

Grid

Determines the spacing of the grid lines.

When you choose this command a dialog box will appear asking for "Grid Spacing" and "Number of subdivisions." Type the desired numbers in the box. When finished, click on "OK" and the grid will automatically be turned on in the desired configuration.

The grid spacing will be in the units you set with the "Units" submenu of the "Preference" command. When you change the units, the appearance of the grid spacing remains the same, and the values listed here change automatically.

Units

Determines precision, American or metric units, and whether or not leading or trailing zeros are displayed.

When you change the units, existing dimensions are updated to reflect the change.

Precision	Sets the number of decimal places of the precision. The default is 0.001. If you want values for measurements and x,y coordinates displayed in other than three decimal places, change this specification accordingly.
inches, feet	Displays measurements in American units.
mm, meters	Displays measurements in metric units.
Leading 0	Determines whether or not the zero to the left of the decimal point is displayed.
Trailing 0s	Determines whether or not zeros to the right of the decimal point are displayed. If set, the number of trailing zeros is determined by the precision.
Fractions	Displays measurements as fractions rather than decimals. You can set the format for fractions on the Linear submenu of the Dimension menu.

When you change the unit type, the values for the grid lines change, but the appearance does not. All geometry reflects the new unit measurement.

Selection

Determines how selected objects are indicated. You can select blinking, color, or both.

If you specify blinking, selected objects blink at the rate set on the Control Panel on the Windows desktop.

To choose a color, move the pointer to the color box, and click the arrow to display the menu choices. Drag to the color you want and release. Once the selection color is changed, all current and future selected items are displayed in the new color.

DAssistant

This command toggles on and off Vellum's Drafting Assistant. When the command is checked, the Drafting Assistant is on.

To use this command, choose it from the Preference submenu in the Layout menu. To turn the Drafting Assistant on or off, click on the command. The Drafting Assistant can also be considered off if Hit Radius in the Snap submenu in Preferences is set to zero.

Save Preferences

This command saves the current setting of all preferences (snap, grid, units, selection, display list and DAssistant). The settings will remain until you change the preferences and use this command again.

To use this command, choose it from the Preference submenu in the Layout menu.

2D Analysis

Provides statistics on the length of the perimeter, the area, the center of gravity (centroid), and inertia of the selection. The selection must be a closed figure.

2D Analysis			
Tolerance	<input type="text" value=".01"/>	Perimeter	3.14
Wt. Per Area	<input type="text" value="1"/>	Area	.78
		Weight	.78
		Centroid X	.41
		Centroid Y	-.84
		Inertia IXX	.05
		Inertia IYY	.05
		Inertia IXY	0
<input type="button" value="CALCULATE"/>			

1. Select the **geometry** that defines a closed boundary.
2. Choose **2D Analysis** from the **Layout** menu.
3. If necessary, enter **changes** for the tolerance and weight per area value.
4. Click the **Calculate** button.

The analysis is performed and the values are displayed in the dialog box.

The calculations apply to a closed area such as the area that could be crosshatched. If you are in doubt about the area, simply crosshatch the geometry to see if it represents the area you want to analyze.

Tolerance affects the accuracy of curved boundaries. The smaller the tolerance, the higher the accuracy.

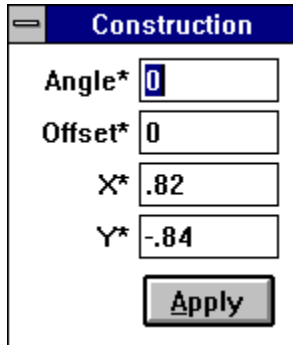
Show/Hide Points

Toggles the display of the control points (endpoints, midpoints, center points, and knot points) for selected and future objects. When points are displayed (and Selectable Points is set on the Edit menu), you can select a point by clicking it. If points are not displayed (but Selectable Points is set on the Edit menu), you can select a point by dragging a selection fence around the location of the point.

To show points for an individual selection, use the Edit Objects... command on the Edit menu.

Construction...

Creates a construction line which is automatically placed on the construction layer of the current document. Subsequent geometry is placed on the work layer.



Angle*	0
Offset*	0
X*	.82
Y*	-.84
Apply	

You can specify the angle of the construction line, the offset from the reference point, and the coordinates of the reference point. An asterisk indicates values that can be specified with the mouse in the drawing area.

The coordinates of the last point you specified on the drawing are displayed in the X and Y boxes.

To specify the angle with the mouse:

1. Click the **Angle** box.
2. Drag a vector in the drawing area.

The angle of the vector line is entered in the box.

To specify the offset with the mouse:

1. Click the **Offset** box.
2. Drag the offset distance in the drawing area.

The distance you drag is entered in the offset box.

To specify x,y coordinates:

1. Click the **X** box.
2. Either enter coordinates or click the location you want in the drawing area.

The coordinates are entered automatically for both x and y.

To create multiple construction lines through the same point, enter the angles you want, separated by semicolons.

To create parallel construction lines, specify a single angle value and different offsets. You can also create parallel lines by creating one construction line using this box, then make the construction layer the work layer and drag new lines with the Parallel Line tool.

You can also choose the **Construction...** command by holding down the Ctrl key while pressing the letter **k**.

To create temporary construction geometry, such as arcs or circles, make the construction layer the work layer, create the geometry to be used for construction, then switch to another layer to continue your work. You can use the Construction pen style if you prefer. Once you no longer need the construction geometry, choose Delete Constructions from the Layout menu; all geometry on the construction layer is deleted, regardless of the pen style used.

Caution: The Delete Constructions command deletes all geometry on the construction layer,

so do not create geometry that you want to save on the construction layer.

Delete Constructions

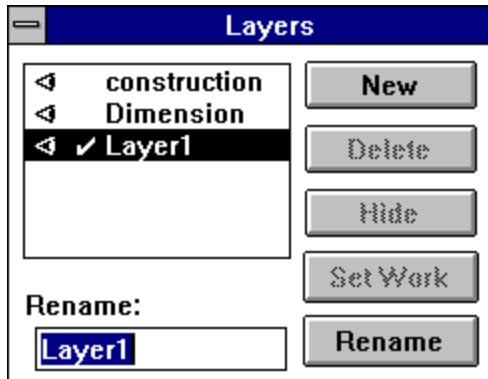
Deletes all construction lines. The Drafting Assistant's dynamic construction lines are not affected by this command. Any geometry on the construction layer (regardless of the pen style used) will be deleted by this command.

You can retrieve deleted construction geometry within the limits of the Undo command.

Layers

Used for creating, deleting, and renaming layers as well as setting layer specifications. The work layer is the active layer -- the layer on which the current construction is created.

The "eye" indicates visible layers, and the check mark indicates the work layer.



Construction lines are automatically placed on the construction layer. If you create a Dimension layer, all dimensions added after you create the layer are placed on the dimension layer.

You can edit any geometry or text that is visible, regardless of which layer it is on. If you want to make some geometry unselectable, use the "**Selection Mask...**" command

To set the layer of an object, use the "**Edit Objects**" command.

You can use layers for complex drawings so you can turn off the display of some components to simplify the drawing visually. You can think of visible layers as transparent pages and hidden layers as invisible pages. Although you cannot see objects on hidden layers, they do exist. The layer must be visible for you to select objects on the layer. For example, if there are objects on a hidden layer and you choose "**Select All**", then delete all objects, those objects on the hidden layer will not be deleted.

You can also specify the selectability of layers with the "**Selection Mask..**". command on the Edit menu.

To create a new layer:

Click **New**.

The new layer (named Layer #) is created on top of all other layers. You can type the name of the new layer and click **Rename**. As many as 256 layers are allowed.

To rename a layer:

1. Select the **name** of the layer from the list box.
2. Type the **new name**.
3. Click **Rename**.

To delete a layer:

1. Select the **name** of the layer from the list box.
2. Click **Delete**.

(Caution: The layer and everything on that layer is deleted. A warning is displayed

before this choice is executed, however.)

To make a layer the work layer:

1. Select the **name** from the list.
2. Click the **Set Work** button.

The layer must be visible before you can make it the work layer.

You can also select the layers name from the pop-up menu in the lower-left corner of the Vellum window. This menu displays all the visible layers.

To hide a layer:

1. Click the **layer** you want to hide in the list.
2. Click **Hide**.

The "eye" beside the name disappears.

The work layer cannot be hidden.

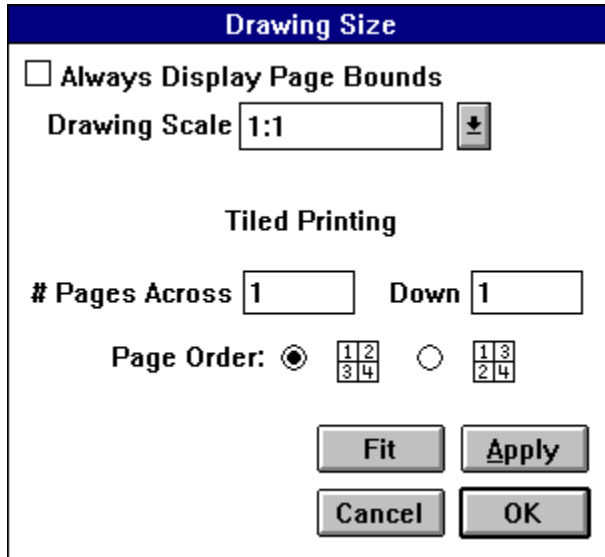
To display a hidden layer:

1. Click the **name** of the layer you want to make visible.
2. Click **Show**.

The layers name is checked in the list.

Drawing Size

Allows you to see the size of the paper relative to the drawing and to specify the scaling of the drawing so that it fits the paper size and orientation set by the Page Setup command.



When you choose Drawing Size..., a gray rectangle displays in the drawing area. This rectangle represents the page so that you can see how your drawing fits the paper.

To display the page size at all times, click **Always Display**.

To specify scale, use one of two methods:

Enter a scale in the Scale box and click Apply to see the effect of the scaling you specified.

Click the arrow beside the Scale box and select one of the standard ratios from the menu; the drawing area is scaled as you specify.

The ratio represents drawing size to paper size.

To scale the drawing to fit the paper, click Fit; the drawing is automatically scaled to fit the paper. The geometry does not change scale, only its ratio to the paper size or drawing area changes.

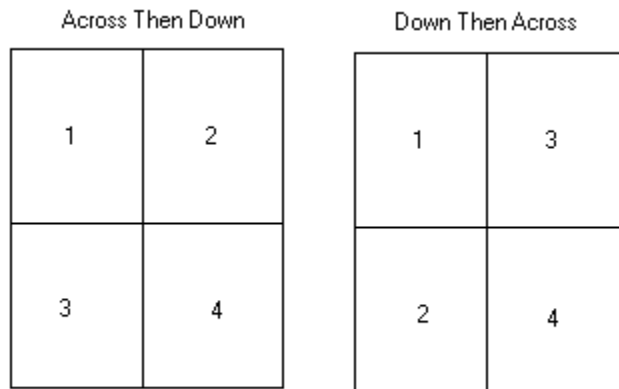
To change the paper orientation while the Drawing Size box is displayed, choose Page Setup from the File menu and enter the specifications you want. When you click OK, the new specifications are shown in the drawing area. You can relocate the drawing boundaries by dragging the boundary.

Tiled Printing

If you want to use a printer but the drawing is too large for the paper size you have available, you can print the drawing on multiple pages and then tape the pages together.

1. Choose **Drawing Size** from the **Layout** menu.
2. Specify the **number** of pages across for the final drawing.
3. Specify the **number** of pages down for the final drawing.

4. Specify the **order** for printing the pages (Across Then Down or Down Then Across).



5. Click the **Fit** button.

Your drawing will be scaled to fit the number of pages and layout, as you have specified.

Arrange Menu

[Zoom In](#)
[Zoom Out](#)
[Zoom All](#)
[Zoom Previous](#)
[Redraw Screen](#)
[Divide](#)
[Group](#)
[Ungroup](#)
[Lock](#)
[Unlock](#)

Arrange	
Z oom In	Ctrl+]]
Zo o m O u t	Ctrl+[[
Zo o m P r ev i ous	
Zo o m A ll	Ctrl+F F
R edraw Screen	Ctrl+R R
D i vide...	
G roup	Ctrl+Y Y
U ngroup	
L o ck	
U n lock	

The commands on this menu allow you to zoom in and out on your constructions, group and lock selections, and divide lines and curves into equal segments.

Zoom In

Enlarges what is shown in the active window by a factor of 2. To enlarge a particular area of your drawing, use stroke commands or the "Magnifying" tool for zooming.

To use a stroke command to zoom in, hold down the "Shift" and "Control" keys at the same time, and drag the cursor that appears diagonally from the upper left to lower right of the area you wish to enlarge. When you release the mouse button that area will be magnified.

To reverse the zoom-in stroke, hold down the "Shift" and "Control" keys and drag the cursor from the lower right to upper left of the screen. The object will appear at its previous magnification.

Zoom Out

Reduces what is shown in the active window by a factor of one-half. You cannot specify the area of reduction. If you want to reduce a particular area, use stroke commands or the "Magnifying" tool for zooming.

To use a stroke command to zoom-out, hold down the "Shift" and "Control" keys at the same time, and drag the cursor that appears diagonally from the upper right to lower left of an area the size you wish the image reduced to. When you release the mouse button the image will be reduced.

To reverse the zoom-out stroke, hold down the "Shift" and "Control" keys and drag the cursor from the lower left to upper right of the screen. The object will appear at its previous magnification.

Zoom All

Zooms in or out to make all parts of the drawing visible on the screen.

Zoom Previous

Zooms to the last magnification you specified.

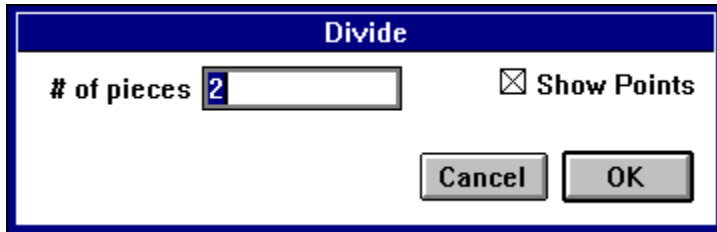
Redraw Screen

Refreshes the screen to eliminate extraneous geometry and points.

You can also choose this command by holding down the Ctrl key while pressing the letter r.

Divide

Divides the selected geometry into the specified number of equal parts.



The image shows a dialog box titled "Divide". It has a blue header bar with the title "Divide" in white. Below the header, there is a label "# of pieces" followed by a text input field containing the number "2". To the right of the input field is a checked checkbox labeled "Show Points". At the bottom of the dialog box, there are two buttons: "Cancel" and "OK".

You can see the divisions when you display the points by choosing Show Points from the Layout menu.

Group

Combines selected objects to function as a single object.

1. Select the **objects** to be grouped.
2. Select **Group** from the **Arrange** menu.

Once geometry is grouped, you cannot edit the individual objects within the group unless you ungroup them.

If you move the group, all components move together. If you change the size of a group, the individual objects change proportionally.

Use Ungroup to separate the group into individual objects.

Groups can be grouped. When you use multiple groupings, you must ungroup each group in the order that they were grouped.

To create temporary groups, you can drag a selection fence around several objects and treat them as a single unit while they are selected. Only those objects that are completely inside the selection fence are selected.

Ungroup

Separates selected grouped objects into their individual components.

1. Select the **group**.
2. Choose **Ungroup** from the **Arrange** menu.

Lock

Prohibits editing or movement of selected geometry.

1. Select the **objects** to be locked.
2. Choose **Lock** from the **Arrange** menu.

Although you cannot move or change a locked object, you can copy, group, and select it.

To prevent accidental changes to objects, lock the objects.

To prevent changes to an entire document, choose the Select All and Lock commands.

Selected objects can also be locked or unlocked by clicking the Locked box in the Edit Objects... dialog box.

Unlock

Removes the lock on the selection. You can change and move the unlocked objects.

1. Select the **locked** object(s).
2. Choose **Unlock** from the **Arrange** menu.

Selected objects can also be locked and unlocked by clicking the Locked box in the Edit Objects... dialog box.

Pen Menu

[Style](#)
[Edit Style](#)
[Outline](#)
[Visible](#)
[Hidden](#)
[Dashed](#)
[Center](#)
[Phantom](#)
[Dimension](#)
[Construction](#)
[Color](#)
[Weight](#)
[Pattern](#)
[Arrow at Start](#)
[Arrow at End](#)
[Crosshatch](#)
[Hatch](#)



When Vellum loads, the pen style is set for the Outline pen style solid, black lines .01 inch wide. The pen style resets when you begin a new document. The Pen menu includes different styles of pens. The default settings for pen styles are described in this section. You can permanently change the characteristics of a style with the Edit Style... command or change the characteristic of a selection using the Color, Weight, and Pattern settings on the Pen Menu. Crosshatching is also on this menu.

Style

[Edit Style](#)

[Outline](#)

[Visible](#)

[Hidden](#)

[Dashed](#)

[Center](#)

[Phantom](#)

[Dimension](#)

[Construction](#)

Edit Style

Sets the specification for the pen styles.

To change the specification of a pen style:

1. Choose **Edit Style...** from the **Pen** menu.
2. Select the **pen style** you want to change.
Click the arrow beside the Style entry box. When the list of pen styles displays, click the style you want to change.
3. Specify the **characteristics** you want for that style.
Click on the arrow beside the characteristic you want to change. When the list of characteristics displays, click the characteristic you want.
4. Click **OK** to set the specifications.
5. Click **Apply** to put the specifications into effect for one pen style.

The dialog box remains open so that you can make changes to another pen style.



When you edit the style, the new specifications are in effect, so that whenever you choose a pen style from the Pen menu, the pen reflects the new characteristics. The specification remains in effect until you change it again with the Edit Style... command.

Style

Selects the pen style to be edited.

The check mark indicates the current style.

Color

Sets the on-screen pen color for the selected pen style. The check mark indicates the current color.

Weight

Sets the pen width for the selected pen style. The check mark indicates the current width.

Pattern

Specifies the line pattern for the selected pen style. The check mark indicates the current pattern.

Outline

Sets the current, on-screen pen to solid, .01 inch, black lines. This is the default pen setting. To change the characteristics of this pen style, choose Edit Style..., specify Outline, and set the characteristics you want. When you click OK, the characteristics of this style are changed.

Visible

Sets the current, on-screen pen to solid, .02 inch, black lines.

To change the characteristics of this pen style, choose Edit Style..., specify Visible, and set the characteristics you want. When you click OK, the characteristics of this style are changed.

Hidden

Sets the current, on-screen pen to hidden pattern, .016 inch, red lines. Hidden lines are actually visible and are used to represent lines that would be hidden in a solid object.

To change the characteristics of this pen style, choose Edit Style..., specify Hidden, and set the characteristics you want. When you click OK, the characteristics of this style are changed.

Dashed



Sets the current, on-screen pen to dashed, .016 inch, yellow lines.

To change the characteristics of this pen style, choose Edit Style..., specify Dash, and set the characteristics you want. When you click OK, the characteristics of this style are changed.

Center

— — — — —
Sets the current, on-screen pen to center, .01 inch, green lines.

To change the characteristics of this pen style, choose Edit Style..., specify Center, and set the characteristics you want. When you click OK, the characteristics of this style are changed.

Phantom

—————
Sets the current, on-screen pen to phantom, .01 inch, cyan lines.

To change the characteristics of this pen style, choose Edit Style..., specify Phantom, and set the characteristics you want. When you click OK, the characteristics of this style are changed.

Dimension

Sets the current, on-screen pen to solid, .01 inch, blue lines.

To change the characteristics of this pen style, choose Edit Style..., specify Dimension, and set the characteristics you want. When you click OK, the characteristics of this style are changed.

Note: *Dimension lines will always be .01 inch lines, regardless of the weight you specify.*

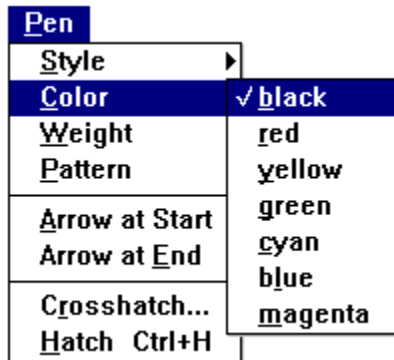
Construction

.....

Sets the current, on-screen pen to dotted, .01 inch, magenta lines. ***This pen style does not create construction lines***; it only uses the line style of construction lines. True construction lines are placed on the construction layer and are deleted when you choose Delete Constructions from the Layout menu. To change the characteristics of this pen style, choose Edit Style..., specify Construction, and set the characteristics you want. When you click OK, the characteristics of this style are changed.

Color

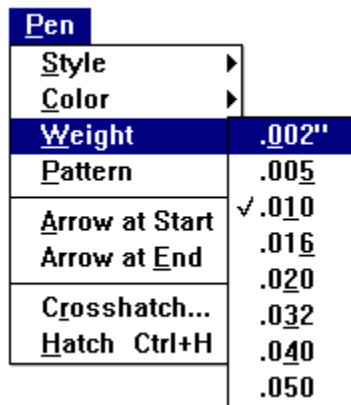
Specifies the pen color of selected and future lines on screen:



To specify the color of the current pen, drag to Color on the Pen menu. When the submenu displays, drag to the color you want and release. The check mark indicates the current color. This overrides the current Color in Pen Style when selected.

Weight

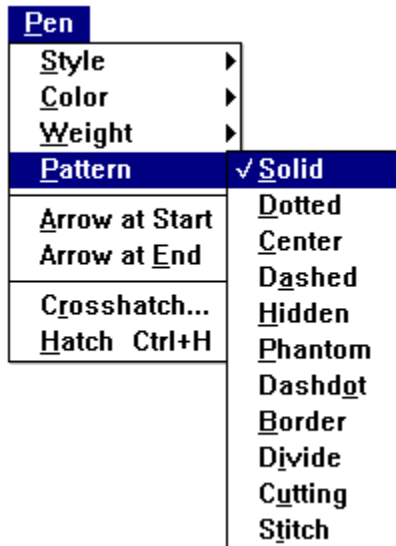
Sets the pen width of selected and future lines in inches:



To specify the weight of the current pen, drag to Weight on the Pen menu. When the submenu displays, drag to the weight you want and release. The check mark indicates the current weight. This overrides the current Weight in Pen Style when selected.

Pattern

Sets the pattern of selected and future lines:



To specify the pattern of the current pen, drag to Pattern on the Pen menu. When the submenu displays, drag to the pattern you want and release. The check mark indicates the current pattern. This overrides the current Pattern in Pen Style when selected.

Arrow at Start

Draws an arrow at the beginning of the selected lines or lines to be constructed. The type of arrowhead is determined by the selection on the [Arrowhead](#) submenu on the [Dimension menu](#).

If both Arrow At Start and Arrow At End are checked, all constructed lines will have arrows at both ends.

Arrow at End

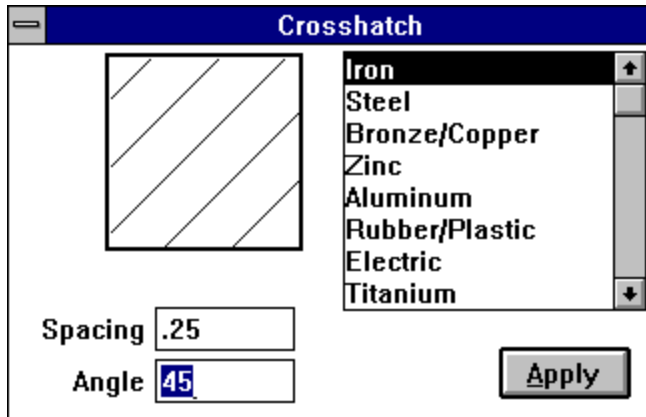
Draws an arrow at the end of the selected lines or lines to be constructed. The type of arrowhead is determined by the selection on the [Arrowhead](#) submenu on the [Dimension menu](#).

If both Arrow At Start and Arrow At End are checked, all constructed lines will have arrows at both ends.

Crosshatch

Crosshatches the selection with one of the industry standard crosshatching patterns.

1. Select the objects which represent a closed boundary. The closed boundary may include cutout areas such as a hole.
2. Choose **Crosshatch** from the **Pen** menu.



3. Select the **pattern** you want from the crosshatch list.

The pattern you select is displayed in the Pattern area, just as it will appear on the drawing.

4. Change the **spacing** and **angles**, if you want.

As you make changes to the angle or spacing, the pattern box shows the pattern exactly as it will be when the specified object is crosshatched.

5. Click **Apply**.

The selected geometry is crosshatched as specified.

*To crosshatch a figure that is not a closed figure, you must create closed figures by adding hidden lines or by segmenting lines. In this way, you can crosshatch individual sections of the geometry. See **Crosshatching** in the **Advanced Features** section of the **Tutorial** for a step-by-step example.*

Hatch

Crosshatches the selected objects using the current crosshatch pattern. If you want to change the crosshatch pattern, choose Crosshatch... and make your selection from the dialog box.

Text Menu

[Font](#)
[Size](#)
[Style](#)
[Align Left](#)
[Align Right](#)
[Align Middle](#)
[Justified](#)
[Single Space](#)
[1-1/2 Space](#)
[Double Space](#)
[Indentation](#)



The commands on this menu allow you to set the font, text size, alignment, line spacing, and indentation for the text you use. Both annotations and dimensions are affected by the settings on this menu.

Font

This command allows you to change the font, either for a selected block of text or for future text entries in the current document. When you dimension your drawing, the dimension text is displayed in the currently selected font.

You may change the font in two ways, by clicking on the font you want from the "Font" menu or by using the "More" command in the "Font" menu. The "More" command brings up a "Character Format" dialog box that allows you to change the font, the font size and the style (bold or italic) of the characters.

To use the "Character Format " dialog box, first select it from the "Font" menu. The available fonts and available font sizes will be displayed in drop down menus. To use the boxes, click on the arrow in the upper right corner. Move the cursor until your choice is highlighted and then click on that choice.

The text may be made bold or italic by clicking in the box next to that option. An "X" will appear in the box if bold or italic has been selected. To return to plain text, click again in the box.

Double click in the upper left corner of the dialog box to close it after you have made your choices.

If you change the font while you are using the Text tool, only the selected characters are changed. If you are not using the Text tool, font changes affect the text of selected and future dimensions.

The fonts listed on the menu are those installed when you installed Windows, as well as the plotter font. You must use the plotter font if you will plot the drawing. All other fonts convert to plotter font when text is rotated.

The font you choose stays in effect for the current document until you choose another font.

Size

Changes the font size for the selected text or for future text entries in the current document. In addition, when you create a dimension, the text is displayed in the current font size.

You can select preset font sizes in points, inches, or millimeters, or you can choose Other..., which allows you to specify a custom size in units.

The font size can also be specified in inches or millimeters.

To specify a nonstandard size:

1. Choose **Other** from the **Size** submenu on the **Text** menu.
2. Type the **size** you want.
3. Click **OK**. If you want to specify a point size, type the number followed by *pt*, otherwise, the text size is displayed in current units.

The size you specify stays in effect until you choose another size.

Style

Changes the style (such as bold or underlined) for selected text or future text entries in the current document. In addition, when you create a dimension, the text is displayed in the current font style.

The style you specify stays in effect until you choose another style.

Align Left

Aligns the selected and future text at the left margin of the text area. If you are using the text tool, only selected paragraphs are aligned.

The check mark indicates which alignment is in effect.

Align Right

Aligns the selected and future text at the right margin of the text area. If you are using the text tool, only selected paragraphs are aligned.

The check mark indicates which alignment is in effect.

Align Middle

Centers the selected and future text in the text area. If you are using the text tool, only selected paragraphs are aligned.

The check mark indicates which alignment is in effect.

Justified

Aligns both the left and right margins of the selected and future text in the text area. If you are using the text tool, only selected paragraphs are aligned.

The check mark indicates which alignment is in effect.

Single Space

Sets the spacing of selected and future text so that each line occupies one space in the text area.

The check mark indicates which spacing is in effect.

1-1/2 Space

Changes the spacing of selected and future text so that each line occupies one and a half spaces in the text area.

The check mark indicates which spacing is in effect.

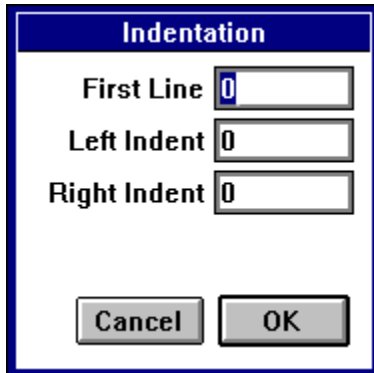
Double Space

Sets the spacing of selected and future text so that each line occupies two spaces in the text area.

The check mark indicates which spacing is in effect.

Indentation

Sets the indentation of paragraphs for the selected text area as defined by the units set on the Preferences submenu.

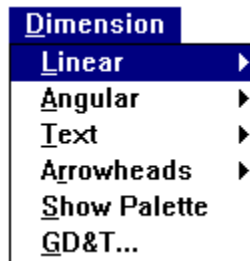


The image shows a dialog box titled "Indentation". It contains three input fields, each with the number "0" entered. The fields are labeled "First Line", "Left Indent", and "Right Indent". At the bottom of the dialog are two buttons: "Cancel" and "OK".

- First Line** Sets the number of units that the first line of each paragraph is indented.
- Left Indent** Sets the number of units that the left margin of each paragraph is indented.
- Right Indent** Sets the number of units that the right margin of each paragraph is indented.

Dimension Menu

[Linear](#)
[Angular](#)
[Text](#)
[Arrowheads](#)
[Show/Hide Palette](#)
[GD&T](#)



This menu allows you to set specifications for format and tolerance of dimensions, specify alignment of dimension text, and choose the types of arrows for dimensions. It also provides dimension and GD&T (geometric dimensioning and tolerancing) tools.

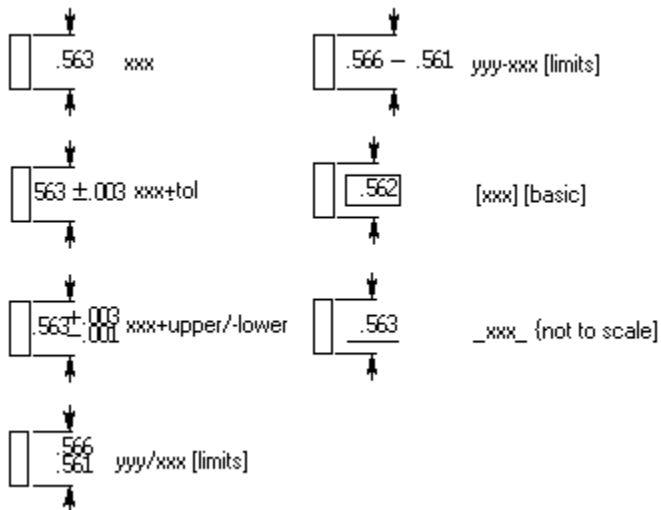
Linear

Allows you to set the number of decimal places and the format of tolerance and limits for linear dimensions. *Do not set these parameters smaller than your Precision in the Units submenu of Preferences.*

Dimension	
Linear	√ No Decimals
Angular	.x
Text	.xx
Arrowheads	.xxx
Show Palette	.xxxx
GD&T...	.xxxxx
	√ xxx xxx±tol xxx+upper/-lower yyy/xxx (limits) yyy-xxx (limits) [xxx] (basic) _xxx_ (not to scale)

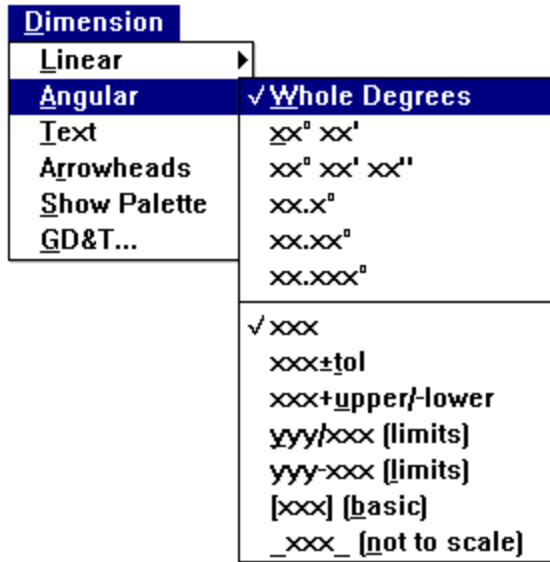
The default is three decimal places with no tolerance or limits.

Use the [xxx] (basic) format to create dimensions to be used with GDT symbols.

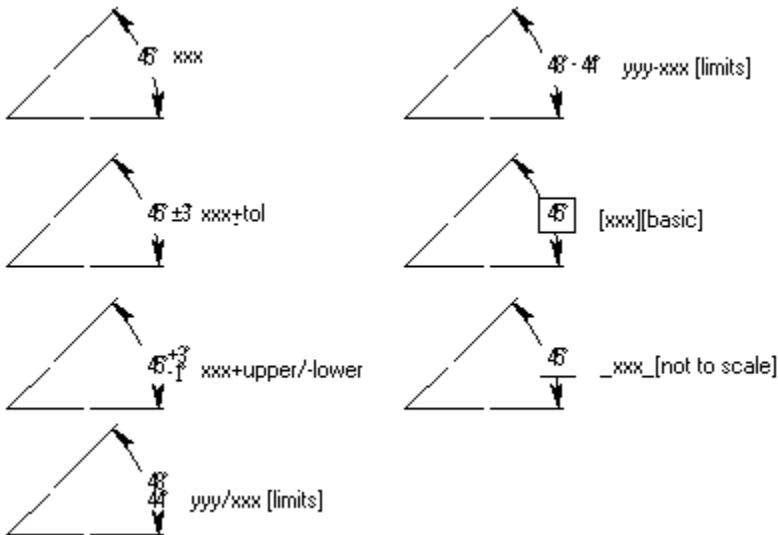


Angular

Allows you to set the format for degrees, minutes, and seconds as well as tolerance and limits for angular dimensions.



The default is degrees with no minutes or seconds and no tolerance or limits.



Use the [xxx] (basic) format to create dimensions to be used with GDT symbols.

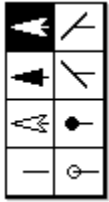
Text

Allows you to set the orientation for text used in dimensions.

You have three choices, "**ISO Style**", "**Aligned**" or "**Horizontal**." To change the current orientation, double click on your choice.

Arrowheads

Allows you to specify the arrowheads that appear on the dimension leader line.



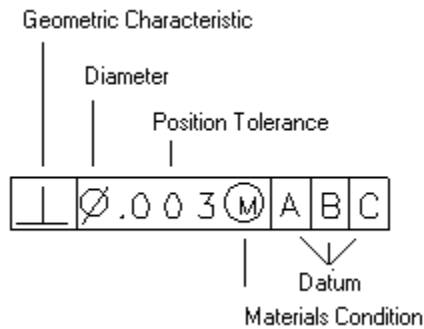
Show/Hide Palette



Displays the dimensioning tool palette. The dimension tools are described in the [Dimension Tool Palette](#) section of Help.

GD&T

Creates a label, a feature control frame for indicating dimensions, alignment, and tolerances.



Geometric Characteristic

The first section of the feature control frame contains the geometric characteristic symbol; for the frame shown above the characteristic being described is Perpendicularity. The available characteristics are as follows:

- **Straightness** The axis of an item of revolution or element of a surface is a straight line.
- **Flatness** All elements of a surface are in one plane.
- **Roundness** All points of a surface of revolution are equidistant from an axis that is intersected by a plane perpendicular to that axis.
- ⊘ **Cylindricity** All points of a surface of revolution are equidistant from a common axis.
- ⌒ **Profile of Line** A 3D element is projected onto a plane.
- ⌒ **Profile of Surface** A 3D surface is projected onto a plane.
- ∠ **Angularity** An axis or surface is at a specified angle from another axis or datum plane.
- ⊥ **Perpendicularity** An axis or planar surface is at a right angle to a datum axis or datum plane.
- // **Parallelism** An axis or planar surface is equidistant along its length to a datum axis or datum plane.
- ⊕ **True Position** A zone of tolerance exists for a center axis or center plane.
- ◎ **Concentricity** The axis of all elements in a cross-section shares a common datum axis.
- ≡ **Symmetry** The location of a feature relative to a center plane.
- ↗ **Circular Runout** Circular elements of a surface of revolution associated with a datum axis.
- ↗ **Total Runout** All elements of a surface of revolution associated with a datum axis.

Creating a GD&T label

1. Choose **GD&T...** from the **Dimension** menu.

The GD&T dialog box appears.

2. Click a **radio button** (Stand Alone, Arrow Line, or Witness Line) to indicate the way you want the GD&T symbol connected to the geometry you are labeling.
3. Enter the appropriate data to create the GD&T label.

There are two types of entries that are made in this dialog box in order to create a complete GD&T label:

- a. Selecting from pop-up menus that appear when you press the pop-up menu buttons.
- b. Entering text in the rectangular boxes by clicking to place a text cursor, and then typing the entry.

4. Indicate the location of the label.

If you are using a Stand Alone label, click the location for the upper-left corner of the GD&T label. If you are using a label connected with an arrow line or witness line, click on the geometry you wish to label, and then click on the drawing area to indicate the position for the label.

Editing a GD&T label

If you have already created a GD&T label, you may wish to make changes.

1. Select the **GD&T label** you wish to edit.
2. Choose **GD&T...** from the **Dimension** menu.

The GD&T dialog box appears with the selected text filled in.

3. Make the desired changes in the GD&T dialog box.
4. Click **Edit**.

The changes are made in the GD&T label.

Using an editing shortcut

When you are editing a GD&T label or creating a second label, you may want to remove every entry in Line 1 or Line 2.

1. Press on the first box, **Geometric Characteristic**.

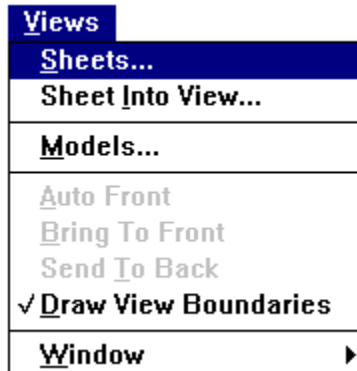
The symbol menu displays.

2. Drag to the **X** symbol.

All entries in the line are deleted.

Views Menu

[Sheets](#)
[Sheet Into View](#)
[Models](#)
[Auto Front](#)
[Bring To Front](#)
[Send To Back](#)
[Draw View Boundaries](#)
[Windows](#)



The commands on the Views menu allow you to create and manipulate a view for placing your construction in a drawing format, as well as to create multi-page documents. Review the information in the **Advanced Drafting** chapter of the **User Guide and Reference Manual** for more information on drafting techniques.

Sheets

Names and renames the sheets. The check mark indicates the work sheet.

You can use several sheets for construction of various related components. A detail view on one sheet is associated with the work view from which it was derived, even if that view is on a different sheet.

To create a new sheet, click **New**.

To rename a sheet:

1. Select the name of the sheet from the list box.
2. Type the new name.
3. Click Rename.

To delete a sheet:

1. Select the name of the sheet from the list box.
2. Click Delete.

To make a sheet the current sheet:

1. Select the name from the list.
2. Click the Current button.

An asterisk indicates the current sheet.

Sheet Into View

Displays the contents of a sheet in a view, scaled as you specify in the scale box on the status line. You can think of a sheet as an infinitely large piece of paper. Depending on how large your geometry is, you may need to scale the view of it so that it will fit on the size paper you want for plotting or printing. For example, you can draw a 6-foot part full-size on a sheet, then scale it by putting it into a view.

Models

Allows you to specify models which are essential for creating detail views on multi-page documents. A model can be related or unrelated geometry.

To rename a model:

1. Select the name of the model from the list box.
2. Type the new name.
3. Click **Rename**.

To delete a model:

1. Select the name of the model from the list box.
2. Click **Delete**.

To make a model the current model:

1. Select the name from the list.
2. Click the **Current** button.

A check mark indicates the current model.

Auto Front

Overrides the **Bring to Front** or **Send to Back** settings, thus enabling clicking to make a view active.

Bring To Front

Brings the specified view to the front.

1. Select the **view**.
2. Choose **Bring To Front** from the **Views** menu.

This command counteracts the Auto Front command, so that you can select objects within the view by clicking rather than activating a view which may be overlapped by the top view.

Send To Back

Sends the specified view to the back.

1. Select the **view**.
2. Choose **Send To Back** from the **Views** menu.

This counteracts the Auto Front command.

Draw View Boundaries

Shows the boundaries of all detail views that are not active.

Windows

This command allows you to move from one open Vellum file to another. When you choose the command a submenu showing the open files will appear. The current file will have a check mark. Move the cursor to the file you want and release it. That file will be brought to the top of the screen.

3D Menu

[Views](#)

[Define View...](#)

[View the Plane](#)

[Unfold View](#)

[Flatten View](#)

[Show/Hide Trackball](#)

[Planes](#)

[Define Plane](#)

[Set Origin](#)

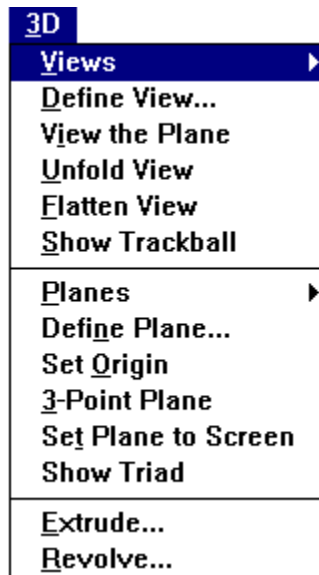
[3-Point Plane](#)

[Set Plane to Screen](#)

[Show/Hide Triad](#)

[Extrude...](#)

[Revolve...](#)



Views

Vellum 3D allows you to look at your project from more than one point of view at a time. The view may be of your entire drawing or of a detail of the drawing. It may be of a different scale or orientation from the original drawing. You may look at many different views at once.

The "View" command allows you to choose from a preset pop-up submenu to change the orientation of a view you have already created. You can add your own personalized viewpoints to the menu using the [Define View](#) command.

The active window, the one you last took action in, will be affected by any choices you make.

Most changes made in one view window are "associative" and will be reflected in all others. *Exceptions are: dimensions, text and cross-hatching*

Submenu choices include Isometric, Right, Front, Top and Trimetric plus any other views you have defined using the [Define View](#) command.

Isometric -- shows your drawing in three dimensions. Same length lines along the x, y and z axes are shown as the same length. This view can cause problems when drawing certain geometry, however, because a 45 degree angle from the x or y axis lines up exactly with the z axis.

Trimetric-- shows your drawing in three dimensions. Differs from isometric in that same length lines are not shown as the exact same length on the screen. This avoids alignment problems in drawing 45 degree angles.

Top-- Shows your drawing in two dimensions from the x, y plane.

Right -- shows your drawing in two dimensions from the y, z plane.

Front -- shows your drawing in two dimensions from the x, z plane.

The view window itself may be moved or changed in size by using the menu which appears when you activate the view by clicking in it and then clicking on the button in its upper left corner. See [View Window](#).

To create a new view window, use the [Detail View](#) tool, which looks like a camera, from the View Control Tools palette.

Once a view window is created you may not change the orientation of your main sheet.

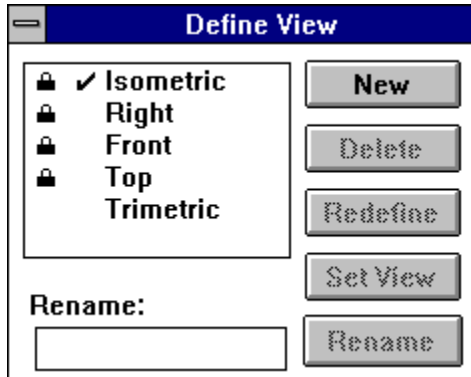
You may also create additional view windows by using the pull-down menu associated with an existing view window. See [View Window](#).

Define View...

This command allows you to add a new point of view to the "Views" command and trackball submenus. Once you define a new view you can change any view window to your new point of view by choosing it from either of the submenus.

You can also modify existing views with this command.

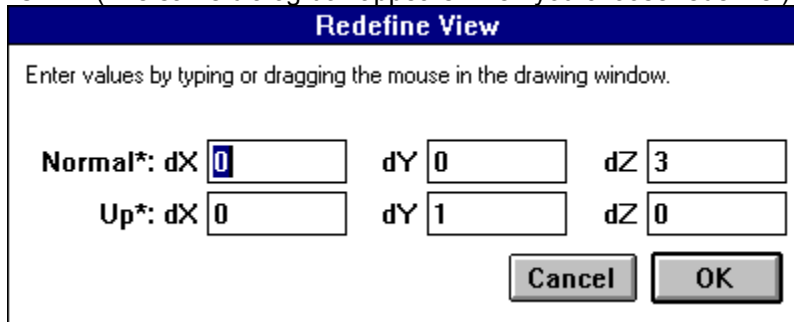
When you choose the Define View command the following dialog box appears:



The dialog box also contains a list of predefined views. These views may be "locked", meaning they can not be changed. If they are locked, a lock icon will appear in front of the view name. To "unlock" a view so it can be changed, simply click on the lock icon. To "lock" an unlocked view, click in the spot where the lock icon will appear.

If a view is locked, only the "New" and "Set View" commands can be used. Only an unlocked view may be deleted, redefined, or renamed.

To create a new view, click on the "New" button. A dialog box will appear which contains the values of the currently active view window. If you wish the view in that window to be your new view, simply click on "OK." (The same dialog box appears when you choose redefine.)



To enter new values, either type them in the appropriate fields or click the mouse in the appropriate areas of your active view:

The "**N**ormal" coordinates define the line of sight that will run from the view to the viewer. To set the coordinates, move the mouse to the window and drag a line through the part of the view you want to face straight toward you.

The "**U**p" coordinates define which part of your drawing will point up in the new view. To define the coordinates, move the mouse to the view window and drag a line in the direction you want to point up in the new view.

The new view will automatically be named "View 1" (or View 2 or View 3, etc., if the other names are already in use.) To give it your own name: either type in the name you want in the "Rename" box of the original dialog box before you click on "New", or choose rename from the commands in the dialog box after you have created the new view.

Remember, you may not be able to see the name of your new view in the submenu because it will be placed after all existing view names. Scroll down to see the entire

submenu.

"Delete" - removes a view from the trackball and view submenus (but not from the screen). The view still exists, it just can't be called up using these submenus.

To use the delete command, click on the file you want to delete and then on "Delete".

"Redefine" - changes the attributes of a view you have named, using the same procedure as the "New" command.

"Set View" changes the active view to a different point of view. To set a view, click on a view title in the dialog box and then on "Set View." The active view will change to one with the attributes defined for that view name. This is the default command in the Define View submenu, which means it will be highlighted when the submenu opens and will be activated by hitting either "Enter" or "Set View".

"Rename" gives a new name to an already defined view. The old name will no longer exist. To rename a view, click on the view title you wish to change, then type the new name in the "rename" box. Then either type an enter or click on "Rename" and the new name will replace the old name.

View the Plane

This command changes the view so you are looking at your drawing with the x axis horizontal in the view window, the y axis vertical in the view window and the z axis coming out toward you. It changes only the orientation of the view and does not affect the work plane.

Unfold View

This command allows you to create a new view along a specified fold line, as if unfolding the drawing at that point to look at a drawing from a new point of view. The new view is "unfolded" 90 degrees from the fold line.

To use this command, first make sure the view containing the geometry you want to unfold is active by clicking in it, then click on Unfold View. Next, specify the line you wish to fold along by clicking on the start and end points. Messages in the Help Line will guide you. Once you have chosen the fold line a new window will appear containing the folded geometry.

You may manipulate the new view window using the commands explained under [View Window](#).

Flatten View

This command allows you to make changes in a view of your project without affecting other views or the main model. For example, you may want to remove a hidden line (which may not be hidden in another view).

Flatten view "disassociates" the view from other views, so changes in the flattened view affect it only. Changes in other views won't be reflected in the flattened view, either.

When you chose this command all overlapping lines of equal length and all lines parallel to the line of sight are removed. Any dimensions will be removed because they no longer apply to the flattened view.

To use the command, first make sure the view you want to flatten is active by clicking in it, then click on Flatten View.

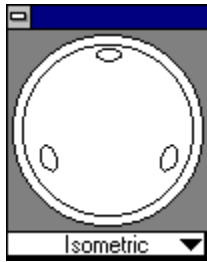
A warning message will advise you that this command will disassociate the view. Click "OK". The flattened view will appear on the main sheet.

If another message appears warning that the current view (which you are flattening) and the draft view have the same model, you must change the active model on the sheet.

To do so, first click on "OK" in the warning message. Then make the main sheet the active view by clicking on it. Next, choose the "Models" command from the Views menu. Click on "New" and then on "Current."

Then return to the view you wish to flatten and follow the original procedure.

Show/Hide Trackball



This command toggles on and off to show or hide a trackball that can be used to rotate the geometry within a view. The trackball also contains a submenu that can be used to change the orientation of views to preset viewpoints.

There are two ways to use the trackball. First, turn it on by clicking on the "Show Trackball" command. Move the cursor to the central round "trackball" and rotate it. The active view should rotate as the trackball rotates.

You may also use the trackball's pop-up submenu, which is identical to the submenu which appears when you use the "Views" command.

You can add your own personalized viewpoints to the submenu using the "[Define View](#)" command.

Submenu choices include Isometric, Right, Front, Top and Trimetric plus any other views you have defined using the [Define View](#) command.

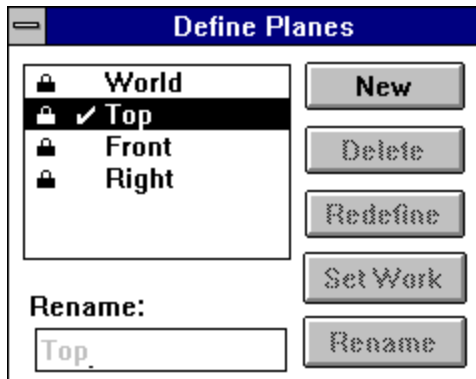
When you choose one of the submenu view names, the geometry in the active window will automatically change to the orientation which had previously been defined for that view name.

The trackball window can be moved by clicking in the title bar area of the window and dragging it to the desired location.

To hide the trackball, either chose the "Hide Trackball" Command or double click on the title bar in the trackball window.

Planes

Normally using Vellum 3D does not require changing the work plane, the two-dimensional slice of the three-dimensional universe you are working in. Occasionally, however, changing the work plane makes it easier to draw geometry on an angled surface that is not parallel to the work plane. You may also want to change the work plane to add text on an angled surface.



It is usually necessary to change the work plane to add a center point circle, a rectangle or two-point ellipse to an angled surface because each is drawn by specifying only two points. Geometry drawn by specifying three points can usually be done on an angled surface without changing the work plane.

The Planes command allows you to choose from a preset pop-up submenu to change the work plane. To change the work plane, chose the Planes command. Move the cursor to the submenu, highlight the name of the plane which you wish to make the work plane and release the mouse button.

To view the new work plane, choose the [View the Plane](#) command from the 3D menu.

To return to the original work plane, chose "World" from the submenu.

You can add your own personalized plane coordinates to the menu using the [Define Plane](#) command.

If you have not defined any new plane coordinates the pop-up submenu will contain "World, Top, Front, and Right" as choices.

World -- is the plane used when the file was originally created. The coordinates of this plane can not be changed.

Top-- The x, y plane.

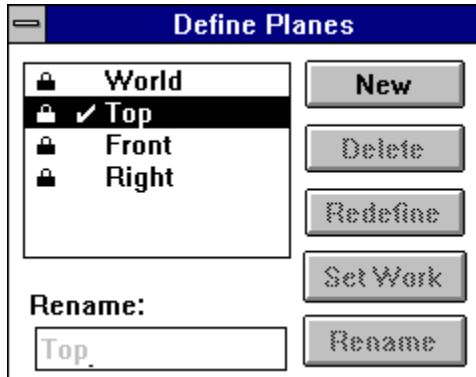
Right -- The y, z plane.

Front -- The x, z plane.

Define Plane

This command allows you to add coordinates for new work planes to the "Plane" submenu. Once you define a new plane you can make it the work plane at any time by choosing it from the submenu. You can use this command to create a new work plane, rename a temporary plane or change an existing plane.

When you choose the Plane Define command the following dialog box appears



The dialog box also contains a list of predefined work planes. These planes may be "locked", meaning they can not be changed. If they are locked, a lock icon will appear in front of the plane name. To "unlock" a plane so it can be changed, simply click on the lock icon. To "lock" an unlocked plane, click in the spot where the lock icon will appear.

If a plane is locked, only the "New" and "Set Work" commands can be used. Only an unlocked plane may be deleted, redefined, or renamed.

The current work plane will be marked with a check mark. You may not "delete" the current work plane.

The **"New"** command is used to create a new work plane. A dialog box will appear which contains the values of the current plane (probably the World plane, the plane you originally began work on, unless you have selected a new work plane). You may either type in the coordinates for the new plane or select them by clicking the mouse buttons on a view of your drawing. You may need to move the dialog box to have access to the drawing.

You must first choose the origin, the point on your plane that will have the coordinates of 0,0,0 where the x, y and z axes will all begin.

Next choose the direction that will be right (horizontal) in your plane and then the direction that will be up (vertical). You must be very precise in your coordinates for the new work plane to be oriented properly.

To enter new values, either type them in the appropriate fields or click the mouse in the appropriate areas of your drawing. When you have set the new coordinates, click on "OK".

The new plane will automatically be named "Plane 1" (or Plane 2 or Plane 3, etc., if the other names are already in use.) To give it your own name: either type in the name you want in the "Rename" box of the original dialog box before you click on "New", or choose rename from the commands in the dialog box after you have created the new plane.

Remember, you may not be able to see the name of your new plane in the submenu because it will be placed after all existing plane names. Scroll down to see the entire submenu.

"Delete" - removes a pre-defined plane name from the submenu. To use the delete command, click on the file you want to delete and then on "Delete".

"Redefine" - changes the attributes of a plane you have already defined, using the same procedure as the "New" command.

When you use this command, the values that appear in the dialog box are the coordinates of the plane you are redefining, and not of the "World" plane you originally began work on. Therefore, any new coordinates you type in should reflect changes from the plane as it exists, not as changes from the world plane (as would be appropriate when originally defining a plane).

To enter new values, either type them in the appropriate fields or click the mouse in the appropriate areas of your active view.

"Set Work" changes the current work plane. To set a new work plane, click on a plane title in the dialog box and then on "Set Work." The work plane will change to one with the attributes defined for that plane name.

"Rename" gives a new name to an already defined plane. You can also use this command to make a temporary plane created using the [3-Point Plane](#) command permanent. In any case the old name will no longer exist.

To rename a plane, click on the plane title you wish to change, then type the new name in the "rename" box. Then either hit return or click on "Rename" and the new name will replace the old name.

To view the new work plane, choose the ["View the Plane"](#) command from the 3D menu.

Set Origin

This command is used to change the origin, the 0,0,0 coordinate at which the x, y and z axes begin. This can be helpful in measuring distances. For example, moving the origin from the center of the sheet to a corner of your geometry can make it easier to measure distances on your geometry.

To change the origin on the work plane, choose this command, move the cursor to the point where you want the new origin and click.

To change the origin on a new work plane, or one not currently used, use the [Define Plane](#) command.

Only the origin will be changed. The orientation of the x, y and z axes remains the same.

3-Point Plane

This command is a short-cut for creating a new work plane. [Planes](#) created using this command will appear on the "[Planes](#)" submenu as TempPlane. Vellum 3D will save TempPlane specifications only until a new TempPlane is created using the 3-Point Plane command. To save a plane created using this file permanently, use the [Define Plane](#) command and choose "Rename" from the submenu.

To create a 3-point plane, choose the 3-Point Plane command. The help line will prompt you to choose the origin of the plane. Move the mouse to that spot and click. The help line will then prompt you to choose the direction of first the x axis and then the y axis by clicking on a point on each axis.

To view the new work plane use the [View the Plane](#) command from the 3D menu.

Set Plane to Screen

This command changes the work plane so the origin is the center of the screen, the x axis runs horizontally across the screen, the y axis runs vertically from the top to bottom of the screen and the z axis comes out toward you.

It is especially helpful for adding text, details or balloons to a drawing.

To set the work plane to this plane, simply choose Set Plane to Screen from the 3D menu. A check mark will appear beside the command to show it is in effect.

To return to the original "World" plane, click on the command again. The check mark will disappear.

To save this plane in the [Planes](#) submenu for further use, use the [Define Plane](#) command and choose "New" from the submenu, then click on "OK" when the dialog box. The plane will appear as Plane 1 (or Plane 2, or Plane 3 if the other names are already in use). The name may then be changed using the "Rename" command. See [Define Plane](#).

Show/Hide Triad

This command displays the Triad symbol in the upper-left corner of the active view or the sheet. The orientation of the symbol shows the orientation of the work plane and changes when you change the work plane.

To show the triad, simply choose the command from the 3D menu.

The command toggles on and off. When you choose Show Triad the Hide Triad command will replace it in the menu. When you choose Hide Triad the Show Triad command will return.

The long arm of the symbol represents the direction of the x axis, the short arm the direction of the y axis and the line off the triangle the direction of the z axis.

The symbol will also automatically appear at the origin (0, 0, 0 coordinate) point when a view is rotated with the on-screen trackball.

Extrude...

This command allows you to make a 2D object into a 3D object by adding depth to it, for example by making a square into a cube.

To use the command, first select the geometry you wish to extrude. When you choose the extrude command, a dialog box will appear.

You may enter the values for the amount of extrusion you want in two ways. You may use the dialog box to type in the number of units you want to extrude the object along each axis. Or you may use the mouse to drag a line from the object to show how far and in what direction you want it extruded.

When you type an "enter" or click on "OK" the object will be extruded.

Which axis to extrude the part along is determined by the plane it was drawn on. For example, an object originally drawn on the x, y axis should be extruded in the z direction, while an object drawn on the x, z plane should be extruded in the y direction.

Revolve...

This command is used to transform a 2D object into a 3D wire frame object by rotating it along a specified axis. This command can turn a circle into a sphere or a torus; a cross section into a 3D object.

When you choose "Revolve" a dialog box will appear asking for information.

The "**sweep angle**" is how far you want the object rotated: 360 degrees to rotate the object in a full circle, 180 degrees for a half circle, etc.

"**Number of steps**" is how many faces you want on the 3D object. The more steps used, the closer the final object is to a solid object. But using more steps may slow the computer because the model may become quite complex.

"**Origin**" is the starting point of the axis around which your object will rotate.

"**Axis**" is the ending point of the rotational axis. Instead of typing in coordinates for the origin and axis you may click or drag the mouse on the desired axis. You may have to move the dialog box to do this if it appears on top of the object you want to rotate.

Once the desired coordinates have been entered, type an "enter" or click on "OK" to rotate the object.

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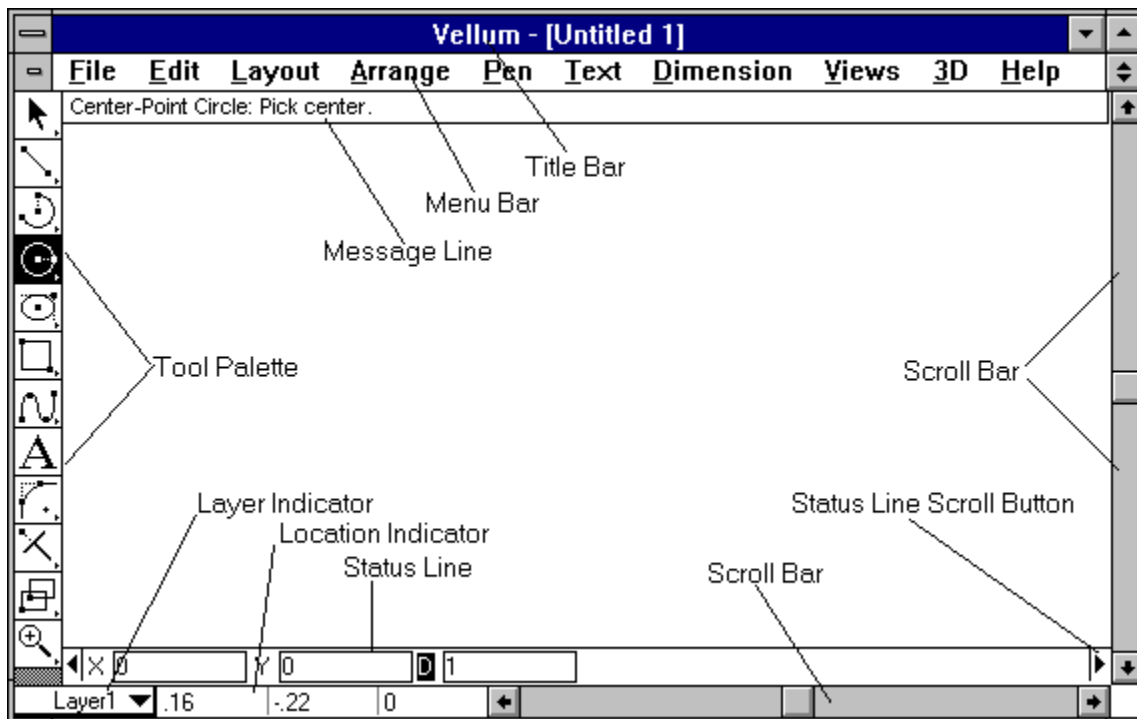
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The Vellum Window

Title Bar	Includes the title of the document and three buttons for controlling the window.
Menu Bar	Contains the Vellum menus of commands and settings.
Message Line	Displays the name and step-by-step instructions for using the current tool.
Tool Palette	Contains the drawing, text, and editing tools you'll use for constructing and annotating geometry.



Scroll Bars	Allow you to move around the document.
Layer Indicator	Displays the name of the active layer. Pressing the mouse button while the cursor is over this area brings up a popup menu of layers. Select a layer from the menu to change the active layer.
Location Indicator	Displays the coordinates of the pointer location.
Status Line	Indicates the coordinate location(s) and geometric measurement(s) of the current tool.
Status Line Scroll Button	Click to scroll status lines for more parameters.

Glossary

Accelerators	Keys and letters that invoke commands rather than using the mouse to choose from menus.
Alignment	The justification of text with even left and/or right margins or centered within the designated text space. Alignment is set in the Text menu.
Alignment Angle	The angle indicated by the Drafting Assistant that falls between horizontal and perpendicular. The specification is set in the Preferences Snap submenu.
Arc Tool	Draws a circle or part of a circle.
Associativity	A link between an object and its dimensions or between views. If the object is changed, the dimension automatically changes to match. In associative views, such as a detail view, when geometry is changed in one view, all associated views are changed accordingly.
Attributes	The layer and pen style (color, weight, and pattern) of an object.
Backspace key	The keyboard key which removes the character to the left of the cursor in text. The Backspace key also works like the Delete key by deleting selected objects.
Bezier curve	A free form curve. NURB splines are a superset of Bezier curves.
Border	A frame showing the boundary of a view.
Boundary	The limiting area of a view, or the geometry that defines the limits for operations such as trimming, relimiting, or crosshatching.
CAD	An acronym for Computer-Aided Design.
CADD	An acronym for Computer-Aided Design and Drafting.
CAE	An acronym for Computer-Aided Engineering.
CAM	An acronym for Computer-Aided Manufacturing.
Chamfer Tool	The tool that constructs beveled or sloping edges between two lines.
Characteristics	The specifications for pens and text. For pens, the characteristics are color, weight, and pattern. For text, the characteristics are font (Helvetica, Times, Roman, etc.), size, and style (bold, underline, italic).
Circle Tool	Draws circles specified by the radius, the diameter, or tangency to other entities.
Circumference	The length of an arc.
Circumscribed	Enclosing a circle. In circumscribed polygons, the midpoint of each side touches an imaginary circle.
Click	To press and release the mouse button. When you are told to click an object, move the pointer to the object and press and release the mouse button.
Clipboard	The memory buffer where selections are stored when the Cut or Copy command is used.

Construction	Creating or drawing geometry.
Construction Lines	Lines, displayed as dotted or gray lines, that you use for exact alignment. The Drafting Assistant creates dynamic, temporary construction lines, and you can create permanent construction lines, which can be used in the geometry or used for alignment and then deleted.
Control Point	The endpoint or midpoint of an object or knot point defining a spline. The Drafting Assistant indicates these positions when the pointer is moved near them.
Control Menu Button	The button in the upper-left corner of the title bar. When you double-click the button, the window closes. If the window is a document you have not saved, you will be asked if you want to save. Clicking the button once displays the Window Control menu.
Coordinates	Positions on axes that specify the locations of a point. Two-dimensional graphics have x,y coordinates; three-dimensional graphics have x,y,z coordinates.
Copy	The command that places a duplicate of the selected geometry on the Clipboard. See the descriptions of the Polar Duplicate... and Linear Duplicate... commands and Transformation tools for additional copying methods.
Corner Tool	An editing tool that trims lines extending past their intersection or extends lines to form a corner.
Crosshatching	The filling of closed geometry with a pattern.
Cursor	The I-beam position indicator in the text tool and boxes which use text. Elsewhere, the position indicator is called a pointer.
Curve	A line, arc, or spline.
Cut	The command to delete selected entities. The selection is placed on the Clipboard and can be pasted into the same or different documents or into documents created by other applications.
Dash	A pen style or line pattern made up of dashes.
Default	Built-in settings that are used by the system if you do not specify a value or choice.
Delete	The command to erase selected geometry. The selection is not placed on the Clipboard; however, it can be retrieved within the limits of the Undo command.
Delete Key	The keyboard key which erases selections. The selection is not placed on the Clipboard, but the action can be retrieved within the limits of the Undo command. The Backspace key and the Delete key perform the same action on selected objects.
Delta	A change, usually in position.
Detail View Tool	The tool that creates a view of the geometry you indicate at the scale you specify.

Dialog Box	A specification box that appears in response to your choosing a command. A dialog box allows you to provide information that qualifies the execution of a command.
Diameter	The distance across an arc, passing through the center.
Dimension	The measurements of an object. Also a pen style.
Divide	To divide a line or curve into equal parts.
Double-click	To press and release the mouse button twice, quickly.
Drafting Assistant	A unique Vellum feature which displays feedback notations and construction lines. The patent pending Drafting Assistant facilitates exact construction without requiring you to be exact. When the pointer is close enough to display feedback, the Drafting Assistant locks onto the exact location for you.
Drag	To press and hold the mouse button, move the pointer to a new location, and release the mouse button.
DXF	The format of AutoCAD [®] files--Data Exchange Format.
Ellipse Tool	The tool for drawing ellipses from rectangle or parallelogram specifications.
Endpoint	The first and last point of a line or curve. The Drafting Assistant indicates these positions when the pointer is moved near the endpoint of an object.
Export	To save a document in a file format that can be used by a different application program.
File	An individual document.
Fillet Tool	The tool that allows you to create an arc of a specified radius tangent to entities.
Font	The assortment of type used in text.
GD&T	Drafting notations for Geometric Dimensioning and Tolerancing.
Geometry	The objects used to construct parts.
Grid	The rectangular array of lines that facilitates measurement and alignment. The grid display can be turned on or off, and the spacing can be specified through the Layout menu.
Group	To specify several entities as one unit that will be treated as a single object.
Hidden	A line pattern or pen style used to draw lines that represent lines not visible in a solid. Drafters traditionally use hidden lines for geometry that is behind other geometry.
Hit Radius	The distance, in pixels, that is detectable by the Drafting Assistant between the object and the pointer.
Import	To load a non-Vellum file.
Inscribed	Within a circle. For polygons, all vertices touch the (imaginary) circle.

Intersection	The position where two lines or curves meet. The curves may actually touch or only intersect when they are extended. The Drafting Assistant indicates only actual intersections.
Knot Points	The points defining a spline, indicated as vertex points by the Drafting Assistant.
Layer	Analogous to transparent media used in conventional manual drafting. Parts can be constructed on several layers which you can make visible or invisible.
Line Tool	The tool that enables you to draw single, connected, or parallel lines.
Linear Duplicate	The command that copies an object in a line or in an array of multiple lines.
Lock	The command to render a selection unchangeable.
Mask	To select entities as a group, masking out all others.
Message Line	The top line of the drawing area. It names the current tool and provides instructions for using it.
META	The file format used by the Windows Clipboard.
Mnemonics	The key sequence which invokes a command from a menu. Hold down the Alt key and type the menu name letter, then type the letter for the command you want.
NURB	Non-Uniform Rational B-splines--the type of splines Vellum creates. NURB splines are a superset of Bezier curves. NURB splines provide designers with two interrelated functions. First, curvature continuity remains intact even when the curve is changed, so that kinks won't develop as the spline is altered. Second, localized control of a complex curve is provided, so that you can isolate an area for changes which do not affect the remainder of the spline.
Object	An individual piece of geometry, such as a line, arc, or circle.
Origin	The 0,0 location on the drawing area. When a new document is opened, 0,0 is located in the middle of the screen. The coordinate symbol displays at the origin when the grid is turned on.
Palette	A group of tools. The general tool palette is always displayed to the left of the drawing area. The dimension tool palette is selectable from the Dimension menu
Parametrics	The integrated Vellum feature available through the Resolve and Symbol commands. Once you create geometry, you add dimensions as text variables (rather than as actual measurements). When you choose Resolve (or Symbol), you can enter specific dimensions for the variables, and the object is drawn to your specifications.
Part	A collection of entities representing an object or structure.
Paste	To place the contents of the Clipboard in the specified location.
Perpendicular	At a 90° angle. The Drafting Assistant displays a notation when the current construction is at a 90° angle to an object.
Pick	To select a location or object by clicking.

Plot	To draw a part on paper using a plotter.
Plotter	A computerized drawing device for hardcopy output.
Point	An object or location.
Pointer	The position locator similar to a cursor. When a tool is in effect, the pointer takes on a representative shape while in the drawing area.
Polar Duplicate	To copy a selection rotated around a reference point.
Polygon Tool	The tool that draws regular polygons, that is, polygons with equal sides. You can specify rectangles and inscribed or circumscribed polygons.
Press	To press and hold down the mouse button. This action is most commonly used to view the contents of a menu.
Radius	Half the distance across an arc, starting from the center.
Redo	The command that reverses the action of the Undo command. You can undo and redo the last eight creating and editing actions.
Redraw	To refresh the screen, recreating all objects.
Relative Position	A location specified as a certain distance from another location. It is often called the delta position.
Relimit	A trim tool that lengthens or shortens lines to the specified object.
Resolve	The command that draws geometry according to the precise measurements you indicate. See <i>Parametrics</i> .
Segment Tool	The tool used to divide a line or curve into unequal parts.
Selection Tool	The arrow tool that selects an object or entities. Usually, subsequent actions affect only the selection.
Sheet	An unbounded region for drawing. You can think of a sheet as one page in a set of blueprints.
Slope	The change of x relative to y between two points on a line. In a spline, slope defines the vector of a line tangent to the spline at a particular knot point.
Snap	The Preferences command choice that allows you to set the specifications for the Drafting Assistant.
Spline	A smooth, free-form curve passing through specified points.
Status Line	The line at the bottom of the drawing area in which you can enter the specifications for the geometry you are creating or editing.
Symbol	The command that retrieves a predefined drawing. This command may use Vellums parametric feature.
Tangent	The point where a line or curve touches a curve without intersecting it. The Drafting Assistant displays the tangent notation of a curve when the pointer nears it.
Text Tool	The tool for drawing annotation. It allows you to specify a text box the area where text is displayed, and then enter text.

Toggle	To switch between two conditions, for example, Hide Grid/Show Grid.
Transformation	The tools to move, rotate, shrink or expand,
Tools	or mirror an object or group of entities.
Trim	To shorten or remove a portion of a line.
Undo	The command that reverses the last editing or creation action. You can Undo (and Redo) the last eight actions.
Units	Measures used for construction (U.S. or metric).
Vertex	The point at which the sides of an angle intersect, or a knot point of a spline.
View-Dependent	Geometry that appears only in the view in which it was created. Crosshatching, text, GD&T symbols, and dimensions in a detail view are view-dependent.
Visible	A pen style.
Zoom	The tool or command that allows you to magnify or reduce an image.

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Stopping Construction During Creation of Geometry

If you are using a tool that requires multiple steps, such as the Connected Lines tool, you may want to stop during the construction.

Press the Esc key to revert to the previous step of the current tool. You can also use the Undo command to revert to the beginning step.

Using the Mouse to Choose Commands

To choose from a menu by dragging:

1. Move the pointer to the menu name.
2. Press and hold the mouse button.
3. When the menu drops, drag until your choice is highlighted.
4. Release the mouse button to choose the highlighted menu item.

To choose from the submenu:

1. Move the pointer to the command to display the submenu
2. Move the pointer to the right and down the submenu until your choice is highlighted.
3. Release the mouse button.

To choose from a menu by clicking:

1. Move the pointer to the menu name.
2. Click the mouse.

The menu drops down.

3. Click the menu item you want to choose.

If the item you choose is a submenu, the submenu will display and you can click the item you want from the submenu.

Using the Keyboard to Choose Commands

You can use the keyboard to choose any command, either with an Alt key sequence or for those commands displaying a Ctrl key sequence, you can use the Ctrl key.

To choose a command from the keyboard using the Alt key sequences:

1. Press the **Alt** key.
2. Press the **letter** that is underlined in the menu name.
3. Press the **letter** that is underlined in the command name.

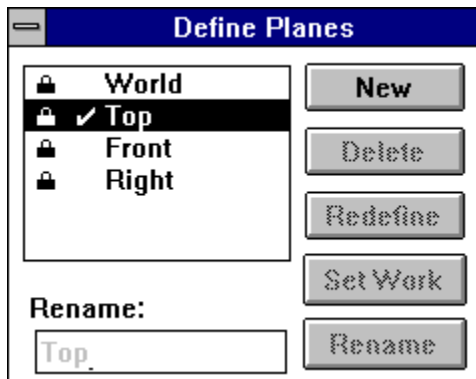
To choose a command from the keyboard using the Ctrl key sequences:

1. Hold down the **Ctrl** key.
2. Press the appropriate **letter** or **key**, as displayed beside the command on the menu.
3. Release the keys.

Using Dialog Boxes

When you choose a command that includes an ellipsis, such as Open..., a dialog box displays that containing additional choices. You can make selections and entries within the dialog box and close the box by clicking a completion button (such as OK).

If the dialog box displays scroll arrows such as the arrows in the box below, you can click an arrow to display the choices. Click the item you want to select and it will appear in the entry box.



You can close the dialog box by double-clicking the Control Menu button.

Copying and Pasting "Clip Art"

The Template, Symbols, and Samples directories included on the Vellum Samples Disk contain drawings of parts that you can copy and paste into your document. These can be used as clip art; or you can use the Symbol... command for the parts in the Symbols directory to resolve them parametrically, as described in the chapter on ***Parametrics*** in the ***User Guide and Reference Manual***.

Using Mathematical Operators

You can use mathematical, exponential, and trigonometric expressions in many entry boxes. For example, if you have selected a circle, you can make its diameter half the size by clicking after the entry in the diameter box and typing $/2$. When you click Apply, the circle will be located with the center in the same location and the diameter will be half as large. **Appendix A** of the *User Guide and Reference Manual* provides examples of valid operators.

Units of Measure

You can use different units of measure such as inches ("), feet ('), inches and feet (x'y"), millimeters (mm), and meters (m). You can also mix the units in the mathematical expression as long as you label the units properly for example, you could enter 10" + 25.4 cm. **Appendix A** of the **User Guide and Reference Manual** provides examples of valid operators.

Entering Characters from the Extended Character Set

You can enter characters from the windows extended character set by holding down the ALT key, entering the four digit code, then releasing the ALT key. The Windows' "Character Map" program displays the extended character set for each font. Click on the character to display its four digit code at the bottom right of the "Character Map" window. For example, to enter a degree sign you would press and hold the ALT key, enter 0176, then release the ALT key.

Listed below are some common codes:

code	character	
0176	°	<i>degree sign</i>
0177	±	<i>plus minus sign</i>
0216	∅	<i>degree sign</i>

Scrolling to the Edge of the Drawing Area

--- Obsolete ---

Hold down the Ctrl key as you click the arrows at the ends of the scroll bars. You must have specified the drawing size for this option to work.

Selecting a Single Object

1. Click the **Selection** tool in the tool palette.
2. Move the pointer to an object and click.

The object is selected, and previously selected objects are deselected.

Selecting Multiple Objects

Clicking

1. Click the **Selection** tool in the tool palette.
2. Move the pointer to an object and click.
3. Press and hold down the **Shift** key.
4. While holding down the **Shift** key, click other objects to be selected.

The objects clicked are selected. If you click any selected objects, they will be deselected.

5. Release the Shift key.

Dragging

If you want to select more than one object, you can drag a selection fence around the objects.

1. Click the **Selection** tool in the tool palette.
2. Drag a **selection fence** around the objects you want to select.

All objects that are completely inside the selection fence are selected. If a portion of an object is outside the region you dragged, that geometry will not be selected but the control points of that geometry which lie inside the selection fence will be selected.

If you want to select most of the objects within an area, you can drag to select all of the objects, and then deselect the objects you do not want selected by holding down the Shift key and clicking them.

Selecting All Objects

To select everything in the Vellum document, choose Select All from the Edit menu. Pressing Ctrl + A also executes this command.

You can also double-click the Selection tool to select everything.

Deselecting Objects

To deselect an object, click anywhere in the drawing area where there is no object, or click any tool in the tool palette.

You can deselect an object that was selected in a multiple selection operation. While the objects are still selected, hold down the Shift key and click the objects you want deselected.

Selecting Points

Selecting points differs from selecting objects because points are not always displayed.

Displaying points

The Show/Hide Points setting in the Layout menu controls the display of control points for selected geometry.

1. Select the geometry.
2. Choose **Show/Hide Points** from the **Layout** menu to toggle the display of points on and off.

Once the points for an object are displayed, you must select the object again and choose either Hide Points from the Layout menu or the Control Points option in the Edit Objects... dialog box to turn off the point display.

Showing and hiding points with strokes:

When you hold down the Shift + Ctrl key and click on an object, the display of an object's points toggles on or off. If the points are hidden when you click the object, the points will be displayed, and vice versa.

Selectable Points

Selectable points are useful in two situations: stretching selected geometry (described later in this chapter) and control point selection for transformations. When you drag a selection fence around objects, all geometry that is completely within the fence is selected. If geometry is partially within the fence, only the control points inside the fence are selected, and the geometry is not selected. The following example illustrates using Selectable Points.

Selecting points

You can select a control point whether the points are displayed or not; however, Selectable Points in the Edit menu must be on (displaying the _ symbol in the menu).

If points are not displayed (the selection in the Layout menu is Show Points):

1. Click the **Selection** tool.
2. Drag a **selection** fence around the location of the point.

The selected point displays as a square.

If points are displayed (the selection in the Layout menu is Hide Points):

1. Click the **Selection** tool.
2. Click the point.

The selected point displays as a square.

Drawing a Line Perpendicular to Another Line

1. Construct the line.
2. Move the pointer to the line until a Drafting Assistant *on* notation appears.
3. Move straight out from the line to display the *perpendicular* notation.
4. Drag the extent of the line.

Drawing a Tangent or Perpendicular to a Curve

1. Construct an arc, circle, or ellipse.
2. Choose the **Single Line** tool.
3. Move the pointer to the arc until a Drafting Assistant **on** notation appears. The Drafting Assistant notation must be *on* rather than *endpoint* or *quadrant*.
4. Drag in the appropriate direction (straight out for perpendicular and at an angle for tangent) until the Drafting Assistant **tangent** or **perpendicular** notation appears.
5. When the Drafting Assistant locks on to perpendicular or tangent, you can then drag the line around the arc to the location you want and extend the line to the length you want.

Using Tools

To use the tool indicated on the palette, move the pointer to the icon and click. The tool becomes active when you move the pointer to the drawing area and the pointer takes on a shape representing the active tool.

Most of the tool icons have subpalettes to provide related functionality. The tools with subpalettes are designated by the arrow in the lower-right corner of the icon.

Clicking the icon activates the tool displayed on the palette. If you press (move the pointer to the icon and press and hold the mouse button) an icon, the subpalette displays the additional tools. For example, the line tool allows you to draw lines between two points. One of the subpalette choices allows you to draw connected lines, in which the endpoint of one line is the beginning point of another. The other line tool subpalette choice allows you to construct lines parallel to existing lines.

Default Tool

The icon displayed on the palette is the *default* tool; if you simply click the icon, that tool is activated. When you select a different tool from a subpalette, it becomes the default tool the one visible on the palette when the subpalette isn't in view. The order of the tools on the subpalette is always the same, regardless of which tool is the default tool.

Changing Tools

When you start Vellum, the Single Line icon is displayed on the tool palette. If you drag to select the Connected Line tool from the subpalette, it becomes the active tool, and its icon displays on the tool palette. If you change tools and then return to the Line tools, clicking the icon activates the Connected Line tool. At this point, if you want to use the Single Line tool, you must select it from the subpalette.

Drafting Assistant

The Drafting Assistant indicates endpoints, midpoints, center-points, intersections, tangents, quadrants, and perpendiculars when you move the pointer near existing objects. It also displays on-the-fly dynamic construction lines as the pointer nears the horizontal, vertical, and 45° angular locations (or those angles specified on the Snap submenu of the Preferences command).

See [Snap Points](#) in this Task section.

Snap Points

If the geometry is dense and points are close together, the Drafting Assistant may *snap* between the points. If you want to select one of several points indicated by the Drafting Assistant, you can lock on a specific point by holding down the mouse button and pressing the key of the first letter of the points name. For example, if the Drafting Assistant is snapping between *midpoint* and *endpoint*, you can hold down the mouse button and press the **m** key to lock on the midpoint. They are as follows:

c	=	center
e	=	endpoint
g	=	grid
i	=	intersection
m	=	midpoint
n	=	no snap
o	=	on
p	=	perpendicular
q	=	quadrant (12,3,6,9 o'clock on an arc or circle)
t	=	tangent
v	=	vertex
%	+	% point

Pressing the Spacebar restores the original functionality of the Drafting Assistant.

The Drafting Assistant is set to detect control points within 12 pixels. This means that if you are trying to indicate a location within 12 pixels of an existing point, the Drafting Assistant will snap to the existing point. You can use either of two methods to remedy this situation:

Zoom in to enlarge the area. In this way, the location you want to choose will be more than 12 pixels away from the existing point.

Reduce the *hit radius* that determines the detection distance for the Drafting Assistant. The Hit Radius is found on the Snap submenu of the Preferences command on the Layout menu. Setting the hit radius to zero turns off the Drafting Assistant.

Dynamic Construction Lines

To display the on-the-fly construction lines, move the pointer to a control point on an existing object to activate it (the object displays a diamond). You can have as many as eight active control points. When you activate the ninth point, the first one is deactivated.

Displaying the Center of a Circle or Arc

To display the center of an arc or circle, move the pointer to the arc or circle until the *on* notation appears and then move to the center to display the center-point.

Creating Offset Geometry

You can construct geometry that is offset from a control point.

1. Select the **tool** you want to use.
2. Move the pointer to the control point you want to be near.
3. Once you click to lock on the control point, click after the existing text in the appropriate X or Y box in the status line.
4. Type the **offset value**.
5. Press **Return**.

For example, if you want the offset to be .060 inches from an existing control point, click to the right of the existing entry in the X status box, type +.06, and when you press Return, the point will be offset in the drawing area.

Keyboard Variations While Using Tools

When used with the geometry creation tools, the Ctrl key generally allows you to make a copy of the previous geometry in a new location. Occasionally it has other purposes, such as drawing an arc rather than a line segment with the Connected Line tool or specifying tangency for 3-Point arcs and circles.

When using the editing tools, the Ctrl key allows you to specify *No Trim* with fillets and chamfers or to create a copy of transformed geometry.

The Shift key allows you to select additional boundaries when you are using the Trim tools or to create a fillet, chamfer, or corner trim with a single click.

Using the Status Line

The status line at the bottom of the drawing area consists of one or more boxes and displays when you choose a tool. It provides information on x,y coordinates and measurements of the object. It also gives you the opportunity to alter the geometry being constructed and to create additional geometry.

You can use the status line in three ways to assist in construction:

Construct exact geometry in the drawing area, observing the status line for information on location, line length, and angles.

Create geometry in the shape you want without concern for location or measurements. Then, edit information in the boxes in the status line to redraw the geometry to the specifications you enter.

Create geometry without using the pointer by entering the appropriate data in the status boxes.

The letter of the active status box is highlighted. You can move activity from one status box to another by pressing the Tab key, by clicking the label of the box, double-clicking the box itself, or by clicking once to place the cursor within the box.

The specifications of the last object you created remain in the status line until you construct a new object with that tool. This allows you to create other objects with the same specifications, even though you may have used a different tool or command in the interim. It also allows you to construct objects with slightly different specifications.

The number of decimal places displayed in the boxes is determined by the Precision setting on the Units submenu of the Preferences command on the Layout menu.

Modifying the Objects During Construction

When you select a tool and draw an object, the object is selected and the status line is activated and linked to the geometry so that you can refine the specifications for that object. When you make entries in the status boxes, pressing the Tab key or clicking to move from one box to another, and then press the Return key, the object is redrawn to the new specifications. The status line is no longer linked to the original geometry. From that point, entries in the status line will create new geometry when the Return key is pressed.

Defining Additional Objects

Once you have created an initial object, you can create additional objects each time you press the Return key. To create additional objects, move from box to box, making the changes that describe the new object, and then press the Return key when the specifications are complete.

Caution: This is a useful feature, but you can create unwanted geometry inadvertently. For example, if you draw a single line, type 1.5 for the length, and then press Return, a 1.5-inch line is drawn. If you then decide that the line is not long enough, and you type 2.0 and press Return, the 2-inch line is drawn on top of the 1.5-inch line. Although this geometry is not visible, it can cause problems later on. See the next topic for information on making changes once you have pressed the Return key.

Editing Existing Geometry

Once geometry is constructed, you may want to make changes. If you want to move the object or change the position of a control point, you can use the Selection tool. See the description of the Selection tool for instructions. You can make broader changes by changing entries in the Edit Objects... box. See [Edit Objects...](#) on the Edit menu in the previous chapter. You can also use the tools on the Edit menu.

Layer Box

The layer box at the lower-left corner of the Vellum window displays the name of the work layer, and the pop-up menu displays the names of all visible layers. The layer name shown in the layer box indicates the work layer the layer on which the current construction is drawn.

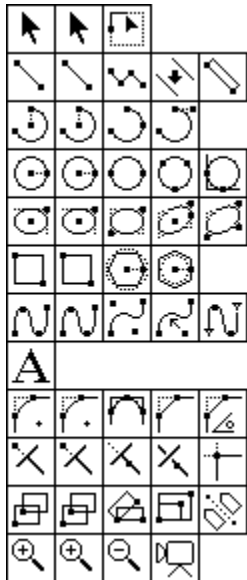
To specify a different layer as the work layer, move the pointer to the layer box, press the mouse button to display the pop-up, drag to the layer you want to be the work layer, and release the mouse button.

Tool Palette

The tool palette contains the drawing, text, editing, and view control tools. It is displayed along the left side of the drawing area.



Tool Palette



Selection Tools

Selection Tool

Tracer Tool

Line Tools

Single Line

Connected Lines

Parallel Lines

Smart Wall Tool

Arc Tools

Center-Point Arc

3-Point Arc

Tangent-Point Arc

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Fillet and Chamfer Tools

2-Entity Fillet

3-Entity Fillet

2-Entity Chamfer

Angular Chamfer

Trim Tools

Simple Trim

Relimit

Segment

Corner Trim

Transformation Tools

Move

Rotate

Expand/Shrink

Mirror

View Control Tools

Zoom In Tool

Zoom Out Tool

Detail View

Selection Tools



These tools are used to select objects, boundaries of objects, lines and points.

Selection Tool



To select an object, begin by clicking the Selection tool in the tool palette. You can click one object, shift-click more than one object, or drag a selection fence around a group of objects.

Selecting a single object:

1. Click the **Selection** tool in the tool palette.
2. Move the pointer to an object and click.

The object is selected, and previously selected objects are deselected.

Selecting multiple objects by clicking:

1. Click the **Selection** tool in the tool palette.
2. Move the pointer to an object and click.
3. Press and hold down the **Shift** key.
4. While holding down the **Shift** key, click other objects to be selected.

The objects clicked are selected. If you click any selected objects, they will be deselected.

5. Release the Shift key.

Selecting multiple objects by dragging:

If you want to select more than one object, you can drag a selection fence around the objects.

1. Click the **Selection** tool in the tool palette.
2. Drag a **selection fence** around the objects you want to select.

All objects that are completely inside the selection fence are selected. If a portion of an object is outside the region you dragged, that geometry will not be selected but the control points of that geometry which lie inside the selection fence will be selected.

If you want to select most of the objects within an area, you can drag to select all of the objects, and then deselect the objects you do not want selected by holding down the Shift key and clicking them.

Selecting all objects:

To select everything in the Vellum document, choose Select All from the Edit menu. Pressing **Ctrl + A** also executes this command.

You can also double-click the Selection tool to select everything.

Deselecting

To deselect an object, click anywhere in the drawing area where there is no object, or click any tool in the tool palette.

You can deselect an object that was selected in a multiple selection operation. While the objects are still selected, hold down the Shift key and click the objects you want deselected.

Selecting Points

Selecting points differs from selecting objects because points are not always displayed.

Displaying points

The Show/Hide Points setting in the Layout menu controls the display of control points for selected geometry.

1. Select the **geometry**.
2. Choose [Show/Hide Points](#) from the **Layout** menu to toggle the display of points on and off.

Once the points for an object are displayed, you must select the object again and choose either Hide Points from the Layout menu or the Control Points option in the Edit Objects... dialog box to turn off the point display.

Showing and hiding points with strokes

When you hold down the Shift + Ctrl key and click on an object, then the display of an object's points toggles on or off. If the points are hidden when you click the object, the points will be displayed, and vice versa.

Selectable Points

Selectable points are useful in two situations: stretching selected geometry (described later in this chapter) and control point selection for transformations.

When you drag a selection fence around objects, all geometry that is completely within the fence is selected. If geometry is partially within the fence, only the control points inside the fence are selected, and the geometry is not selected. The following example illustrates using Selectable Points.

Selecting points

You can select a control point whether the points are displayed or not; however, Selectable Points in the Edit menu must be on (displaying the _ symbol in the menu). If points are not displayed (the selection in the Layout menu is [Show Points](#)):

1. Click the **Selection** tool.
2. Drag a **selection** fence around the location of the point.

The selected point displays as a square.

If points are displayed (the selection in the Layout menu is Hide Points):

1. Click the **Selection** tool.
2. Click the **point**.

The selected point displays as a square.

Tracer Tool

This tool helps you to easily trace the boundaries, both outside and inside, of an object. This is helpful in crosshatching and when using area properties.

To use the tool, select it by holding down the mouse button when the cursor is over the select tool, moving the cursor until the tracer tool icon is highlighted and then releasing the mouse button.

When you select the Tracer tool the cursor will appear as an arrowhead.

Where you click the **cursor** determines which boundary is traced. If you click inside the object, inside boundaries will be traced; if you click outside, outside boundaries will be traced.

A small square cursor will trace the boundary which is being selected. The lines which have been selected will be highlighted.

The lines will become deselected when you use the Tracer Tool again unless you hold down the shift key when you select additional lines.

To select additional boundaries, such as in a part with a hole in it, hold down the shift button and click near the additional boundaries you wish to select. Repeat the process, holding down the shift key, until you have selected all the boundaries of the object.

Note: *When the Tracer Tool selects a boundary that includes only part of a line or curve, it does so by creating temporary geometry. This temporary geometry will disappear when it is deselected unless you save it by moving it to another layer.*

Line Tools



The line tools create line segments, connected lines, lines parallel to existing lines, and smart walls. As you create line segments, the coordinate locations, line length, and angle from horizontal are displayed in the status line.

The line is drawn according to the pen specifications for **color**, **weight**, and **pattern**.

Single Line



Draws a line between two points. **Two methods for drawing a line are available:**

Click two locations to indicate the endpoints of the line.

Drag to indicate the endpoints of the line; press at the beginning and release at the end of the line. As you drag, you will see a rubberband line that previews your construction.

Once you construct a line, you can create a copy of the line by holding down the Ctrl key and clicking where you want the first endpoint of the copy to appear.

The status line allows you to specify the x,y coordinates of the beginning, the relative location of the end (delta x and delta y), the line length, and the angle from horizontal. Once a line is drawn, line length is the default status box selection.

Drawing a line perpendicular to another line

1. Construct the line.
2. Move the pointer to the line until a Drafting Assistant **on** notation appears.
3. Move straight out from the line to display the **perpendicular** notation.
4. Drag the extent of the line.

Drawing a line tangent to or perpendicular to a curve

1. Construct an arc, circle, or ellipse.
2. Choose the **Single Line** tool.
3. Move the pointer to the arc until a Drafting Assistant **on** notation appears. The Drafting Assistant notation must be **on** rather than *endpoint* or *quadrant*.
4. Drag in the appropriate direction (straight out for perpendicular and at an angle for tangent) until the Drafting Assistant **tangent** or **perpendicular** notation appears.
5. When the Drafting Assistant locks on to perpendicular or tangent, you can then drag the line around the arc to the location you want and extend the line to the length you want.

Connected Lines



Draws lines in which the endpoint of one line segment is the beginning point of the next.

1. Use separate clicks or drag to indicate the **endpoints** of the lines.

If you click a point and then change your mind, press the Escape key to remove the last line. Pressing the Delete key removes all connected lines in the current construction.

2. Indicate the last point by double-clicking or by choosing another tool.

If you want to create an arc as one of the connected lines, hold down the Ctrl key before clicking the endpoint of the segment. One line must have been created before you can use the arc function.

The status line allows you to specify the x,y coordinates of the beginning, the relative location of the end (delta x and delta y), the line length and the angle from horizontal. Once the line segment is created, line length is the default status box selection.

Parallel Lines



Once you have created a line, you can create a duplicate parallel to it. Use one of the following methods for creating parallel lines:

Drag the line; press to select the line and release to indicate the position for the parallel line. A rubberband line moves with the pointer.

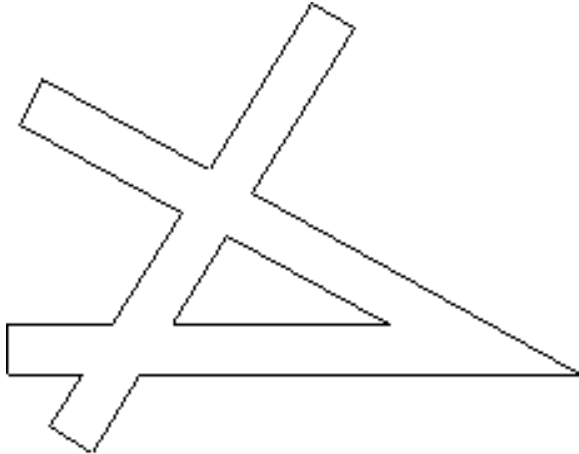
Click the line you want to duplicate and then enter a delta distance in the status line and press Enter.

The status box allows you to specify the distance between the selected line and the parallel line.

Smart Wall Tool



Draws double line walls. Where walls touch or intersect, they are automatically joined.



This tool works in much the same manner as the Single Line tool, except that it draws a double line. Each wall is defined by the side that you indicate on the drawing area and the other side is filled in. The defining side displays the points when [Show Points](#) is turned on.

1. Select the **Wall** tool.
2. In the status line, specify the **thickness** of the wall.
3. Click or drag to indicate the ends of the wall segment.

If you are dragging, you can press the **Shift** key while drawing to flip the wall to the other side.

If you are clicking, the Shift key has no effect. The automatic side of the wall is drawn in the same direction as the automatic side of the previous wall segment.

4. Continue clicking endpoints for wall segments.

The wall tool does not merge walls from different layers. If you try to use parametrics with walls, turn on the point display and dimension between the points. The points are displayed on the defining wall. The parametrics feature does not work with every wall configuration.

Caution: *Smart Walls lose their "intelligence" when they are Ungrouped and cannot regain that characteristic unless you Undo within the given eight actions.*

Arc Tools



Arc tools create an arc by any of three methods: Center-Point, 3-Point, or Tangent Point. As you create each arc, the coordinate locations, radius, angle from horizontal, and delta angle are displayed in the status line.

Center-Point Arc



Draws an arc from three points – center point, arc beginning point, and arc endpoint.

1. Click to indicate the **center** of the arc.
2. Click to indicate the **radius** and **beginning** of the arc.
3. Drag the extent of the arc, pressing at the beginning point and releasing at the endpoint of the arc.

You can simply click all three points, but the rubberband arc is not displayed when you use this method.

To construct an arc greater than 180° , you must drag, rather than click, the endpoint.

The status line allows you to specify the x,y coordinates of the center of the arc, the length of the radius, the starting angle from horizontal, and the delta angle from the start. Radius is the default status box selection.

3-Point Arc



Draws an arc through the points you select. The arc is drawn from the first position that you indicate in the direction of the next point.

To display a rubberband arc, drag the second and third points rather than clicking them.

If you click *on* an object while holding down the Ctrl key, the arc is drawn tangent to the specified object rather than through the point. You may combine the placement of points to create an arc through a specific point of one object and tangent to another object, or an arc tangent to three objects, or any other combination.

The status line indicates the x,y coordinates for each of the three points.

Tangent-Point Arc



Draws an arc beginning at the first point you specify. The second point you specify is the direction vector, and the third point you specify indicates the endpoint of the arc. Essentially the tangent-point arc tool first creates a line, then creates an arc tangent to the line, then erases the line.

1. Click the **starting point**.
2. Click the **tangent vector**. (The arc will be tangent to a line drawn from the starting point to this point).
3. Click the **endpoint**. The arc is drawn between the first and last point you indicated and tangent to the line between the first and second points.

The status line allows you to specify the x,y coordinates of the endpoints of the arc and the angle of the tangent line.

Circle Tools



Circle tools construct circles by any of four methods. Center-Point uses the center and diameter of the circle. Opposite-Point uses the diameter. 3-Point uses three points or can be tangent to existing objects. Tangent Circle draws a circle tangent to two objects, using the diameter.

Center-Point Circle



Draws a circle specified by the center point and diameter. Two methods for drawing a center-point circle are available:

[Click two locations.](#) The first click indicates the center and the second indicates the radius.

[Drag.](#) Pressing at the center and releasing at a position indicates the radius.

If you hold down the Ctrl key and click a new location, a circle is drawn with the same diameter as the last circle.

The status line shows the x,y coordinates of the center and the diameter of the circle. Diameter is the default status box selection.

Opposite-Point Circle



Draws a circle specified by the diameter. Two methods are provided for drawing an opposite point circle:

[Click two locations to indicate the diameter.](#)

[Drag. Press at the beginning and release at a position that indicates the diameter.](#)

The status line allows you to specify the x,y coordinates representing the endpoints of the diameter.

3-Point Circle



Draws a circle through the points you select. If you place any of the three points on an existing object, the circle will be drawn through that point. If you click an object while holding down the Ctrl key, the circle is drawn tangent to the object rather than through the indicated point.

You may combine the placement of these points to create a circle through a specific point of one object and tangent to another object, or a circle tangent to three objects, or any other combination.

To display a rubberband circle, drag the second and third points rather than clicking them.

The status line indicates the x,y coordinates for each of the three points.

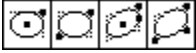
Tangent Circle



Constructs a circle tangent to the two objects you select.

You must enter a diameter for the circle in the status box before selecting the two objects.

Ellipse Tools



These tools construct an ellipse inscribed within an invisible rectangle or parallelogram. Two methods are provided for each type of ellipse. 2-Point Center uses the center point and one corner location of the rectangle. Opposite-Corner uses opposite corners of a rectangle. 3-Point Center uses the center point, the midpoint of a side and the corner of the parallelogram. 3-Corner uses three corners on a parallelogram.

2-Point Center Ellipse



Constructs an ellipse inscribed within the rectangle calculated from the two specified points – the center point and one corner of the rectangle.

Dragging creates a rubberband ellipse, so that you can see the ellipse before it is drawn. Clicking points creates an ellipse without the rubberband preview.

If the two points are on the vertical or horizontal axis, a straight line is drawn.

If you hold down the Ctrl key and click, a default ellipse or a copy of your last ellipse will be drawn, with its center at the location you click.

The status line allows you to specify the x,y coordinates of the center point and the length and angle of the axes of the ellipse.

Opposite-Corner Ellipse



Draws an ellipse inscribed in the rectangle specified by the opposite corners you indicate.

Dragging creates a rubberband ellipse, so that you can see the ellipse before it is drawn. Clicking points creates an ellipse without the rubberband preview.

If the two points are on the vertical or horizontal axis, a straight line is drawn.

If you hold down the Ctrl key and click, a default ellipse or a copy of your last ellipse will be drawn, with the lower-left point at the location you click.

The status line allows you to specify the x,y coordinates of the lower-left point and the length and angle of the axes of the ellipse.

3-Point Center Ellipse



Constructs an ellipse inscribed within the parallelogram calculated from three specified points the center point, the midpoint of a side, and a corner of the parallelogram.

If the three points are on the vertical or horizontal axis, a straight line is drawn.

If you hold down the Ctrl key and click, a default ellipse or a copy of your last ellipse will be drawn, with its center at the location you click.

The status line allows you to specify the x,y coordinates of the center point and the length and angle of the sides of the parallelogram.

3-Corner Ellipse



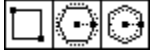
Draws an ellipse inscribed in the parallelogram calculated from the three corners you specify.

If the three points are on the vertical or horizontal axis, a straight line is drawn.

If you hold down the Ctrl key and click, a default ellipse or a copy of your last ellipse will be drawn, with the lower-left point at the location you click.

The status line allows you to specify the x,y coordinates of a corner and the length and angle of the sides of the parallelogram.

Polygon Tools



These tools draw rectangles and inscribed or circumscribed polygons.

Rectangle



Draws a horizontal or vertical rectangle, using the opposite corners you specify.

Dragging to specify the points creates a rubberband rectangle, so that you can preview the rectangle.

Clicking points creates a rectangle without the rubberband preview.

If you want to create a square, align the second point on a 45° construction line.

If you hold down the Ctrl key and click, a rectangle is drawn with the upper-left corner at the specified location and equal in size to the last rectangle you drew. A default 1-inch rectangle is drawn if you have not previously drawn a rectangle.

If the two points are on the vertical or horizontal axis, a straight line is drawn.

The status line allows you to specify the x,y coordinates of the first point, as well as the width and height of the rectangle. Width is the default status box selection.

Inscribed Polygon



Draws a regular polygon for which the radius of the circle determines the location of the vertices. The default polygon is a hexagon, but you can specify the number of sides in the status line.

(Note: The status line shows a diameter for the circle, which is the standard way of describing an inscribed polygon.)

Dragging from the center point to a vertex creates a rubberband polygon, so that you can preview the polygon. Clicking the center-point and vertex creates a polygon without the rubberband preview.

If you hold down the Ctrl key and click, an inscribed polygon is drawn with the center at the specified location, using the diameter of the last inscribed polygon you specified.

The status line allows you specify the x,y coordinates of the center, the diameter of the circle defining the polygon, and the number of sides. Diameter is the default status line selection, and the default number of sides is six.

Circumscribed Polygon



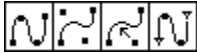
Draws a regular polygon for which the radius of the circle determines the midpoint of the sides. The default shape is a hexagon, but you can specify the number of sides in the status line.

Dragging from the center to the midpoint creates a rubberband polygon so that you can preview the polygon. Clicking the center-point and midpoint creates a polygon without the rubberband preview.

If you hold down the Ctrl key and click, a circumscribed polygon is drawn with the center at the specified location, using the diameter of the last circumscribed polygon you specified.

The status line allows you specify the x,y coordinates of the center, the diameter of the circle defining the polygon, and the number of sides. Diameter is the default status box selection, and the default number of sides is six.

Spline Tools



The spline tools create NURBS (Non-Uniform Rational B-splines) which are a superset of Bezier curves. These splines are curves created by a complex mathematical formula.

NURB splines provide designers with two interrelated functions.

First, curvature continuity remains intact even when the curve is changed, so that kinks won't develop as the spline is altered.

Second, localized control of a complex curve is provided, so that you can isolate an area for changes which will not affect the remainder of the spline.

These properties are essential in the designs created by aerodynamic engineers. Air molecules moving over a wing surface must flow smoothly to maximize aerodynamic lift. If the surface does not maintain complete curvature continuity, the air molecules will separate from the wing surface, causing a vacuum. Such a vacuum results in an eddy or swirl of air as the molecules try to fill the vacuum. This disruption of air flow increases the drag, resulting in a less effective wing design.

The automotive industry is also interested in smooth air flow to improve gas mileage. Complete curvature continuity also improves styling. The appearance of a car is one of the major sales factors. The potential buyer would not be impressed if the showroom lights' reflection on the car rippled and wavered. It is complete curvature continuity that makes a smooth reflection.

Localized control of complex curves enables minor modifications without adversely affecting the shape. For example, if a new, bigger engine would not fit under a perfectly-designed hood, the engineer using a NURB spline could raise the center of the hood without changing the aerodynamics or basic design.

NURB splines are also valuable for injection mold designs to eliminate the swirl of plastic as it is injected into the mold. Such designs provide better surface finishes and allow thinner cross sections in the die.

Through-Points Spline



The through-points spline tool draws a spline through the points that you place. Double-click to indicate the last point.

The status line shows the x,y coordinates of the last point specified.

Vector Spline



Draws a spline using vectors determined by the vertices you specify. The vector spline tool uses each point that you place as the vertex of a vector for the spline it creates.

Vellum uses these vectors to calculate the control points (two fewer than the number of vertices you specify). The spline is tangent to the first and last vectors and passes through the calculated control points.

Double-click to indicate the last point.

The status line shows the x,y coordinates of the last point specified.

Add Spline Control Point



Allows you to add another control point to an existing spline. After selecting the tool, click on the spline at the desired location for the new control point(s).

Note: In order to see the new control points, as well as the existing control points, you must first select the spline and then choose [Show Points](#) from the Layout menu (or use the Edit Objects dialog box to specify display points).

If you intend to move a point, be sure to lock the adjacent points so that you don't change the slope of the rest of the spline.

Lock Spline Control Point



Locks the specified points on an existing spline so that you can change the slope between the points without affecting the rest of the spline.

You should choose [Show Points](#) from the Layout menu before you use this tool.

Unlocking spline control points

Select the locked point and it will be unlocked. Choosing Unlock from the Arrange menu will have no effect; this command is used to prevent changes objects.

Locking more than one spline control point

Use the Shift-click method or drag a fence around the points.

Editing control points

1. Select the spline with the **Selection** tool.
2. Choose [Show Points](#) from the **Layout** menu. (If Hide Points is displayed on the menu, do not choose it.)
3. Click on the **drawing area** to deselect the spline.
4. Select **Lock Spline Control Points** from the **Spline** tool subpalette.
5. Select the points you want to lock (the points on either side of the point you will change).
6. Use the **Selection** tool to select the point you want to change.
7. Drag the point to the new location.

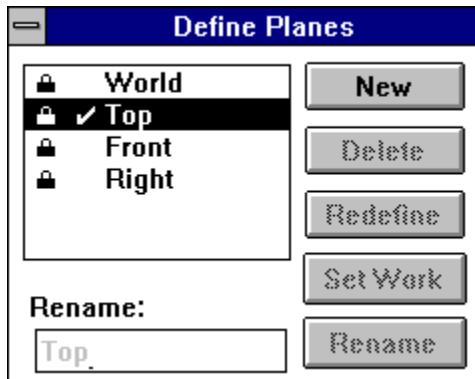
The lock spline control point tool is used to *lock* or immobilize selected control points on an existing spline. You may then adjust the curvature of the spline between the locked points without disturbing other areas of the spline.

The dotted lines that appear tangent to the end portions of the spline are **tangent control arms**. They allow you to adjust the curvature of the spline leading up to the endpoints without moving the endpoints themselves. To do this, select the endpoint of the tangent control arm and drag it to another location with the 4-way pointer that appears.

Editing the end slope

1. Select the spline with the **Selection** tool.
2. Choose [Show Points](#) from the **Layout** menu. (If Hide Points is displayed on the menu, do not choose it.)
3. Click in the **drawing area** to deselect the spline.
4. Select **Lock Spline Control Points** from the **Spline** tool subpalette.
5. Select the point you want to lock but not the endpoint.
6. Use the **Selection** tool to select the point at the end of the tangent control arm you want to move.
7. Drag the point to a new location.

Text Tool



Allows you to annotate your drawing. Begin by placing a text box on the drawing. Click the opposite corners or drag to specify the corners of the box. The size of the box determines the width of the text area. The box initially accommodates one line of text; additional lines are added or removed automatically as you type or delete.

If you hold down the Ctrl key and click in the drawing area, a text box is created with the upper-left corner at the location you specify. Its size is equal to the last text box you created.

Once the box is created, a cursor appears at the left margin. When the cursor displays, you can perform any of the following word processing activities:

Type entries.

The text automatically *wraps*, that is, words that extend beyond the right margin are *wrapped* to the next line.

Use commands on the Text menu to specify text characteristics font, size, style, alignment, spacing, and indentation.

If you change characteristics while you are typing, future entries reflect the new characteristics; previous entries do not.

If necessary, use the Delete key to erase characters to the left of the cursor.

Use the commands in the Edit menu.

The text box remains selected until you click inside the box or in the drawing area. While it is selected, you can change the entries on the status line to change the size of the text area.

When the text box is no longer selected, you can make changes according to the editing directions shown later in this section.

The status line allows you to set the x,y coordinates of the first point and the height and width of the box.

To change the width of the text box, follow these steps:

1. Drag a selection fence around a **control point** at one corner of the box.
2. Drag to a **new width**.

To edit existing text, follow these steps:

1. Click the **Text** icon.
2. Click in the text box.

You can use the arrow keys to move the cursor.

3. Drag or click to select **characters**. Double-click to select a word.
4. Use the commands on the Edit menu to cut and paste text. You can also use the Delete key to delete selected characters or to delete characters to the left of the cursor.

You can add text and the text area will enlarge to accommodate it.

To use different characteristics within the same text area, follow these steps:

1. Select the **characters** you want to change.
2. Choose the **characteristics** from the **Text** menu.

Font, size, and style can apply to individual characters within the text area; alignment, spacing, and indentation apply to the entire text area or to selected paragraphs.

To import ASCII text from another file, follow these steps:

1. Select the **Text** tool.
2. Choose **Import...** from the **File** menu.
3. Be sure that **Text** is selected, then click the **OK** button.

The text will be entered into an existing text box or into a new text box on the drawing, and you can drag the box to the location you want.

See Also: [Entering Characters from the Extended Character Set](#)

Fillet and Chamfer Tools



The Fillet and Chamfer tools construct a fillet by creating an arc of the specified radius tangent to two nonparallel lines or curves. The radius is set on the status line. The default radius is .25 inch. Chamfers are constructed across a corner the specified distance from the intersection. The default distance is .25 inch.

Fillets and chamfers are automatically trimmed. If you do not want them trimmed, hold down the Ctrl key while you are selecting the objects to fillet or chamfer.

2-Entity Fillet



Constructs a fillet tangent to the two objects you click. You can also hold down the Shift key and click once inside the corner you want filleted.

If the object to be trimmed is a corner, it is automatically trimmed away or extended. If the objects intersect, the trimming follows arbitrary rules. If you hold down the Ctrl key while you select the objects to fillet, the objects are not trimmed.

Fillets use the smallest arc between the selected geometry.

The status line allows you to specify the radius of the fillet either before or after you create the fillet.

3-Entity Fillet



Constructs a fillet tangent to the three objects (or points) you choose. Click each object individually. Be sure to click the *on* position of an object.

If you hold down the Ctrl key while you select the objects to fillet, the objects are not trimmed.

There are no entries on the status line.

2-Entity Chamfer



Creates a chamfer between two lines. Click each line making up the corner you want chamfered. You can also hold down the Shift key and click once inside the corner to be chamfered.

The lines are automatically trimmed or extended. If you hold down the Ctrl key while you select the objects to chamfer, the objects are not trimmed.

The default distance between the chamfer and the intersection of the lines is .25 inch.

The status line allows you to specify the distance from the chamfer to the intersection of the corner lines.

Angular Chamfer



Creates a chamfer at the specified angle and distance from the corner. The specified angle is the angle between the chamfer and the first line selected to designate the corner.

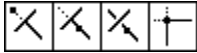
Click each line making up the corner you want chamfered. You can also hold down the Shift key and click once inside the corner to be chamfered.

The lines are automatically trimmed or extended. If you hold down the Ctrl key while you select the objects to chamfer, the objects are not trimmed.

The default distance between the chamfer and the intersection of the lines is .25 inch. The default angle is 45°.

The status line allows you to set the distance from the intersection as well as the angle.

Trim Tools



The Trim tools lengthen or shorten lines and curves as specified. Select the bounding object that limits the trim before you select the Trim tool. If you want to select different boundaries after you have selected the Trim tool, hold down the Shift key, then make selections. When you release the Shift key, Trim is in effect again.

Simple Trim



Shortens a line to the specified boundary. If you hold down the Ctrl key while using this tool, it becomes the Relimit tool.

1. Select the **object** that limits the trim.

2. Select the **Trim** tool.

Use Shift-Click to select more bounding objects.

3. Click the section of the object to be discarded.

There are no status line entries.

Rule: Point to what you want to throw away.

Relimit



Lengthens or shortens a line to a specified boundary. If you hold down the Ctrl key while using this tool, it becomes the Trim tool.

1. Select the limiting object.
2. Select the **Relimit** tool.

Use Shift-Click to select more bounding objects.

3. Click the section of the object that you want to remain.

There are no status line entries.

Rule: Point to what you want to keep.

Segment



Divides a line or curve at intersections with other lines or curves.

1. Select the intersecting objects.
2. Select the **Segment** tool.

Use Shift-Click to select more bounding objects.

3. Click the object to be segmented.

The selected object is segmented at the bounding objects. It is not necessary to see this segmentation to select parts of the segmented line.

If you hold down the Ctrl key while selecting the line to be segmented, the line will be displayed in the characteristics of the current pen style.

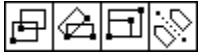
There are no status line entries.

Corner Trim



Creates a corner from the two lines you specify. Lines are extended or shortened to create the corner. Click each object or press the Shift key and click inside the about-to-be-created corner. If you want to extend a line to its theoretical intersection with another line, first click the line to be extended, then hold down the Ctrl key and click the line that will not be trimmed. There are no status line entries.

Transformation Tools



These tools move, rotate, expand or shrink, and mirror objects. Select the object you want to transform before you select the Transformation tool.

To select objects after you have selected the Transformation tool, hold down the Shift key and make selections. When you release the Shift key, Transformation is in effect again.

To make copies of the transformed objects, hold down the Ctrl key while you specify the transformation.

Move



Moves one or more objects to a new location. You can copy the selection by holding down the Ctrl key while you select the objects. If you select more than one object, they remain in the same position relative to each other.

1. Select the object.
2. Select the **Move** tool.

Use Shift-Click to select more objects.

3. Drag the selected object to a new location, pressing to select the object and releasing to indicate the new location.

You can also click a reference point, then click a destination point to move the selected object to the new location. It is not necessary for the reference and destination points to be on the object you are moving. If they are not, the move is performed relative to the specified points.

The status line allows you to specify the x,y distance that the selection is to be moved.

Rotate



Rotates one or more objects around a specified point. You can copy the selection by holding down the Ctrl key while you select the objects.

1. Select the object or objects to be rotated.
2. Select the **Rotate** tool.
Use Shift-Click to select more objects.
3. Specify the point of rotation.
4. Drag the objects or enter an angle in the status line to specify the rotation.

You can also click the reference point and then click the destination point.

The status line allows you to specify the angle of rotation. If you rotate text, fonts revert to the Vellum plotter font when they are rotated.

Expand/Shrink



Enlarges or reduces objects while maintaining the proportions. You can copy the selection by holding down the Ctrl key while you select the objects.

1. Select the object to be scaled (expanded or shrunk).
2. Select the **Expand/Shrink** tool.

Use Shift-Click to select more objects.

3. Click a point on the object that you want to remain stationary.
4. Drag a point on the opposite side of the object to a new location. Instead of dragging, you can click both the point to move and its destination point.

If you are not concerned with maintaining the aspect ratio, use the Selection tool.

The status line allows you to enter the exact scale in the status line.

Mirror



Creates the mirror image of an object or objects on the opposite side of a reference line. You can copy the selection by holding down the Ctrl key while you select the objects.

1. Select the **object** or objects to be mirrored.
2. Select the **Mirror** tool.

Use Shift-Click to select more objects.

3. Specify the **reference line** by clicking two locations or by dragging. The reference line need not be parallel to the object.

The status line has no entries.

View Control Tools



Enlarges or reduces the images on the screen and creates detail views.

Zoom In Tool



Zooms the specified factor. The default factor is 2.

When you click in the drawing area, the point you clicked is moved to the center of the screen, and the drawing is enlarged by a factor of 2.

You can also drag a box around an area, so that only that area is magnified.

The status line shows the current zoom scale. If you enter a different scale, that scale takes effect when you press Enter. Pressing the Ctrl key while using this tool causes a Zoom Out.

Zoom Out Tool



Zooms out the specified factor. The default factor is .5.

When you click in the drawing area, that position is moved to the center of the screen and the drawing is reduced by a factor of .5.

The status line shows the current zoom scale. If you enter a different scale, that scale takes effect when you press Enter. Pressing the Ctrl key while using this tool causes a Zoom In.

Detail View



Creates a detail view.

1. Select the **Detail View** tool.
2. Enter the **scale** for the detail view in the status line.
3. Use the pointer to drag a rectangle around the area of your drawing that you want to be placed in the detail view.

This rectangle becomes the detail view window frame. You may also create the rectangle by clicking to place the diagonally-opposite corners.

4. Position the pointer in the center of the detail view window frame and drag the window to a clear area of the drawing.

The status line allows you to specify an enlargement or reduction scale for the detail view.

Once a detail view is created, you can change its boundaries of a detail view by dragging a selection fence around a corner of the detail view, and then dragging the corner to a new location. The scale does not change, but the portion originally shown in the enlargement adjusts to the new size.

You can edit geometry in either the main view or the detail view and the other view will be updated. This feature is known as **associativity**. Dimensions, text, and crosshatching are view-dependent; they apply only to the view in which they are created. So, if you create a detail view of a part that is crosshatched in the main view, the detail view is not crosshatched.

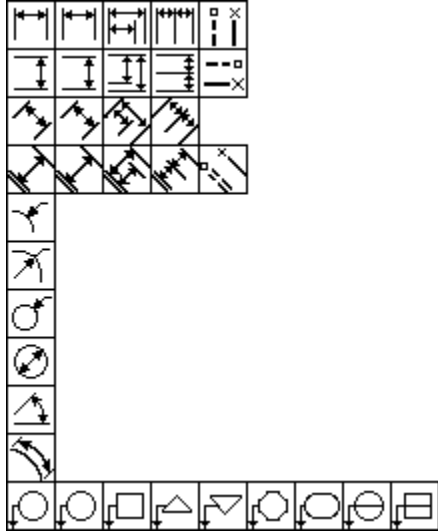
To make a view the active view, click inside it. A double-dotted line indicates that the detail view is the active view.

The zoom function works on the entire drawing rather than on individual detail views, so that the drawing scale does not change.

To zoom or scroll in a detail view, hold down the Control key and perform the same actions that you would for the draft view.

Dimension Tool Palette

The Hide/Show Palette command on the Dimension Menu displays the palette of dimensioning tools. The first four tools have subpalettes which provide baseline and continuous dimension capabilities.



Horizontal Tools

Horizontal

Horizontal Base Line

Horizontal Chain

Horizontal Ordinate Dimension

Vertical Tools

Vertical

Vertical Base Line

Vertical Chain

Vertical Ordinate Dimension

Oblique Tools

Oblique

Oblique Base Line

Oblique Chain

Perpendicular Tools

Perpendicular

Perpendicular Base Line

Perpendicular Chain

Perpendicular Ordinate Dimension

Radial Arrow Out

Radial Arrow In

Diametral Arrow Out

Diametral Arrow In

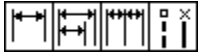
Angular Dimensioning

Arc Length Dimension

Balloons

See Also: [Entering Characters from the Extended Character Set](#)

Horizontal Tools

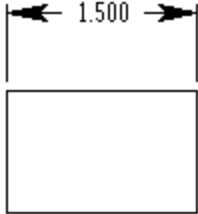


The tools on this subpalette allow you to dimension objects and space horizontally. If you are making more than one measurement, you can select either the base line style which measures each distance from a base line or the chain style which measures between each specified point.

Horizontal



Dimensions linear objects or spaces, horizontally.

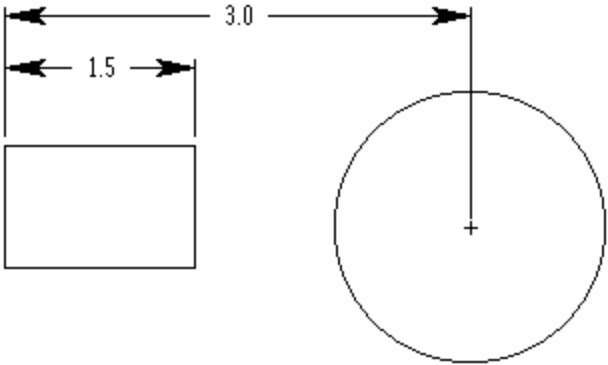


To dimension an object or space horizontally, click the left boundary first, then the right. (Click the boundaries in the opposite order if you want the dimension to display below the objects.) When the dimension text displays, you can drag it to a new location; when you release the mouse button, the dimension and extension lines are redrawn according to the position you indicate.

Horizontal Base Line



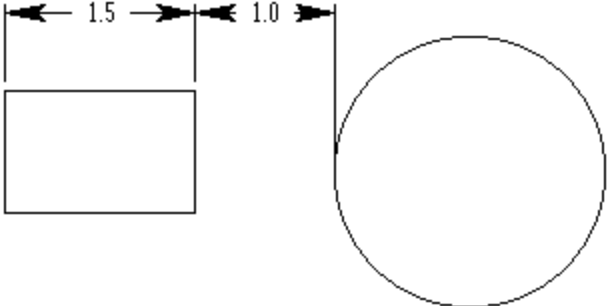
Dimensions objects or space horizontally from a base line.



Horizontal Chain



Dimensions objects or spaces end-to-end, horizontally.



Horizontal Ordinate Dimension

This tool measures the distance from one point to another horizontally and displays the dimensions above or below the measured end point. This type of dimensioning is also known as datum dimensioning,

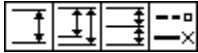
To use this tool, choose it from the Dimensions palette. (It is the last icon in the row of the Horizontal dimensioning tools.) Move the cursor to the point where you want to begin measurements and click. Then move to the point you wish to measure to and click. Any further clicks of the mouse will automatically measure from the first point.

To mark the starting point as 0,0 click twice on the starting point.

Whether the dimensions appear above or below the starting point is determined by what side the second point is measured on. If the second point measured is to the left of the starting point, the dimensions will appear below the object. If the first point measured is to the right of the starting point, the measurements will appear above the object.

Like other dimensions you can manipulate the text by changing its size or font using the "Text" menu. You can move the text by positioning the cursor over it. The cursor will change to a four-headed arrow. Then click and drag the text to the desired position. If necessary, the line pointing to the dimension will dogleg to follow the text.

Vertical Tools



The tools on this subpalette allow you to dimension objects and space vertically. If you are making more than one measurement, you can select either the base line style which measures each distance from a base line or the chain style which measures between each specified point.

Vertical



Dimensions linear objects or spaces, vertically.

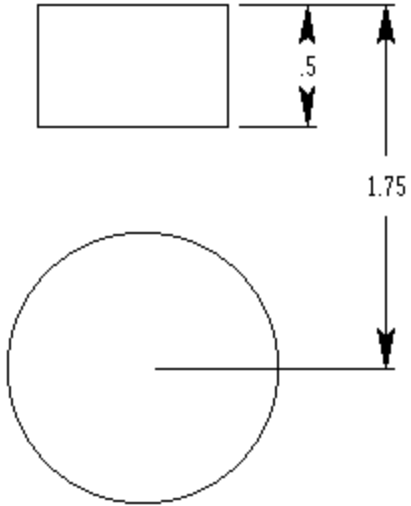


To dimension an object or space vertically, click the top boundary first, then the bottom. (Click the boundaries in the opposite order if you want the dimension to display to the left of the objects.) When the dimension text displays, you can drag it to a new location; when you release the mouse button, the dimension and extension lines are redrawn according to the position you indicate.

Vertical Base Line



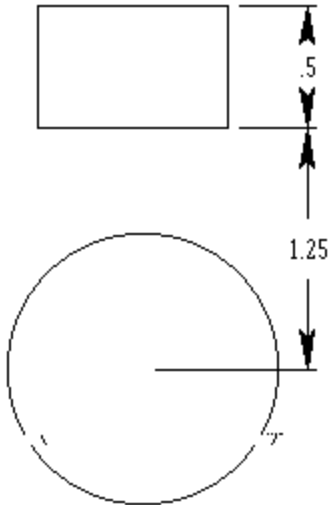
Dimensions objects or space vertically from a base line.



Vertical Chain



Dimensions objects or space end-to-end, vertically.



Vertical Ordinate Dimension

This tool measures the distance from one point to another vertically and displays the dimensions to the left or right of the measured end point. This type of dimensioning is also known as datum dimensioning,

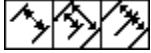
To use this tool, choose it from the Dimensions palette. (It is the last icon in the row of Vertical dimensioning tools.) Move the cursor to the point where you want to begin measurements and click. Then move to the point you wish to measure to and click. Any further clicks of the mouse will automatically measure from the first point.

To mark the starting point as 0,0 click twice on the starting point.

Whether the dimensions appear to the right or left of the starting point is determined by what side the second point is measured on. If the second point measured is above the starting point, the dimensions will appear to the left of the object. If the first point measured is below the starting point, the measurements will appear to the right of the object.

Like other dimensions you can manipulate the text by changing its size or font using the "Text" menu. You can move the text by positioning the cursor over it. The cursor will change to a four-headed arrow. Then click and drag the text to the desired position. If necessary, the line pointing to the dimension will dogleg to follow the text.

Oblique Tools

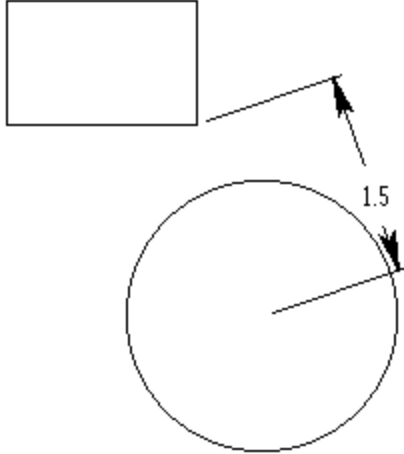


The tools on this subpalette allow you to dimension objects and space obliquely, point to point. If you are making more than one measurement, you can select either the base line style which measures each distance from a base line or the chain style which measures between each specified point.

Oblique



Dimensions objects or space, obliquely or point to point.



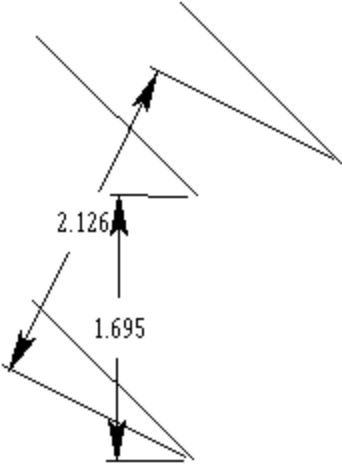
To dimension an object or space from point to point or obliquely, click the upper boundary first, then the lower. (Click the boundaries in the opposite order if you want the dimension to display below the objects.)

When the dimension text displays, you can drag it to a new location; when you release the mouse button, the dimension and extension lines are redrawn according to the position you indicate.

Oblique Base Line



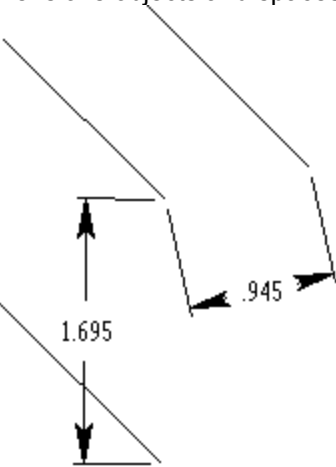
Dimensions objects and spaces point to point or obliquely from a base line.



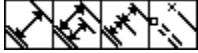
Oblique Chain



Dimensions objects and spaces point-to-point, end-to-end.



Perpendicular Tools

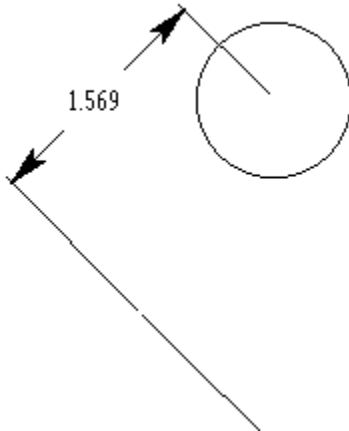


The tools on this subpalette allow you to dimension objects and space perpendicular to another object. If you are making more than one measurement, you can select either the base line style which measures each distance from a base line or the chain style which measures between each specified point.

Perpendicular



Dimensions an object or space perpendicular to another object. If most of your dimensions originate from a line edge, you can use this tool to generate both horizontal and vertical dimensions.

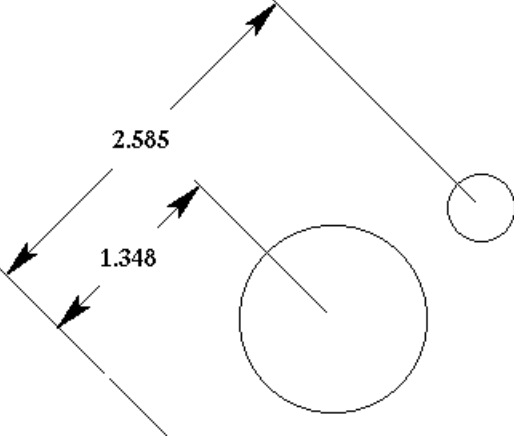


To dimension an object or space perpendicular to a line, click the base line first, then the object or location. **Note:** Be certain that you click *on* the base line, not on the endpoint of the base line. When the dimension text displays, you can drag it to a new location. When you release the mouse button, the dimension and extension lines are redrawn according to the position you indicate.

Perpendicular Base Line



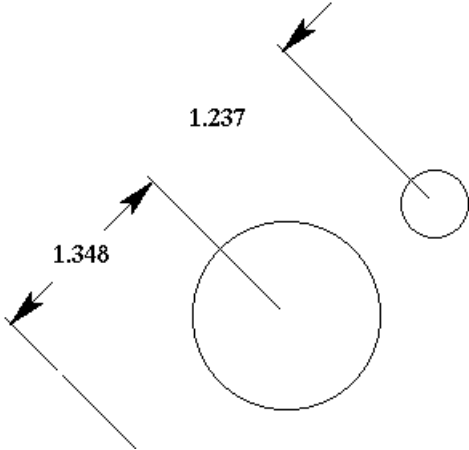
Dimensions between a point or object perpendicular to an existing object from a base line.



Perpendicular Chain



Dimensions between points or objects perpendicular to an existing object from end-to-end.



Perpendicular Ordinate Dimension

This tool measures the perpendicular distance from a line to a point. The measurement is displayed next to the point, parallel to the line originally chosen. This type of dimensioning is also known as datum dimensioning. It is useful for measuring dimensions at an angle.

To use this tool, choose it from the Dimensions palette. (It is the last icon in the row of Perpendicular dimensioning tools.) Move the cursor to the point near the end of the line you wish to measure from. Then move to the point you wish to measure to and click. Any further clicks of the mouse will automatically measure from the first line.

The ordinate dimension will appear parallel to the line you first clicked on, measuring the perpendicular distance from that line.

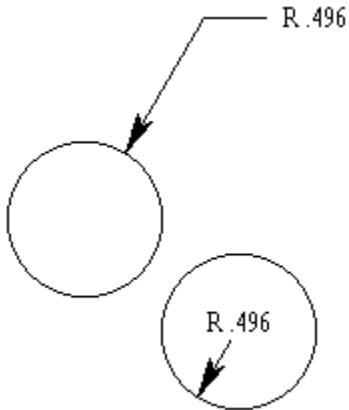
To mark the starting point as 0,0 click twice on the starting point.

Like other dimensions you can manipulate the text by changing its size or font using the "Text" menu. You can move the text by positioning the cursor over it. The cursor will change to a four-headed arrow. Then click and drag the text to the desired position. Like other dimensions you can manipulate the text by changing its size or font using the "Text" menu. You can move the text by positioning the cursor over it. The cursor will change to a four-headed arrow. Then click and drag the text to the desired position. If necessary, the line pointing to the dimension will dogleg to follow the text.

Radial Arrow Out



Dimensions the radius of a circle, arc, or fillet with the arrow on the side of the arc indicated when the arc is picked.



To dimension the radius of a circle, arc, or fillet, click near it. If you click outside it, the dimension text is displayed outside, and if you click inside, it is displayed inside.

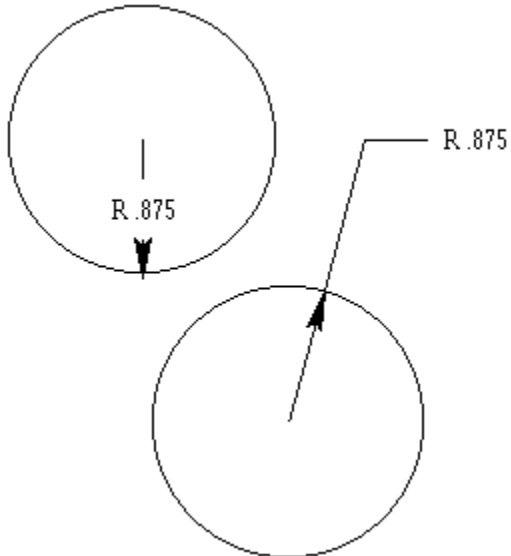
When the dimension displays, you can drag the text to a new location. When you release the mouse button, the dimension and extension line are redrawn according to the position you indicate.

You can change the position of the arrow by dragging a selection fence around the control point at the tip of the arrow, and then dragging the arrow to a new location.

Radial Arrow In



Dimensions the radius of a circle, arc, or fillet with the arrow inside the geometry.



To dimension the radius of a circle, arc, or fillet, click near it. If you click outside it, the dimension text is displayed outside, and if you click inside, it is displayed inside. Either way the arrow line starts from the arc center.

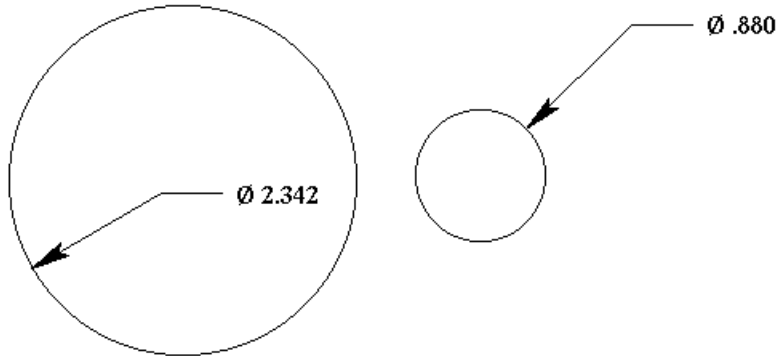
When the dimension displays, you can drag the text to a new location. When you release the mouse button, the dimension and extension line are redrawn according to the position you indicate.

You can change the position of the arrow by dragging a selection fence around the control point at the tip of the arrow, and then dragging the arrow to a new location.

Diametral Arrow Out



Dimensions the diameter of a circle with the arrow on the side of the circle indicated when it is picked.



To dimension the diameter of a circle, click near the circle. If you click outside the circle, the dimension text is displayed outside, and if you click inside, it is displayed inside. Either way, a single arrow line appears.

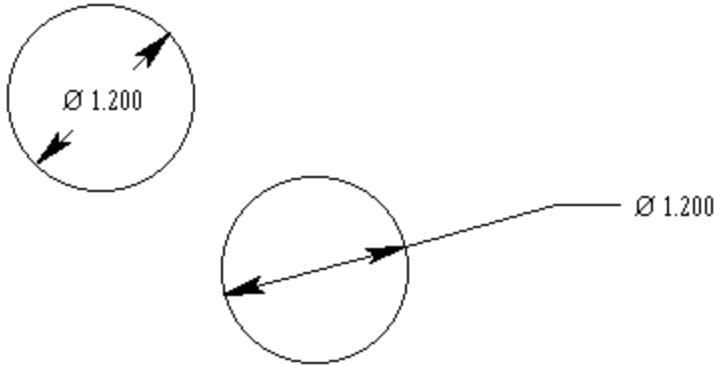
When the dimension displays, you can drag the text to a new location. When you release the mouse button, the dimension and extension line are redrawn according to the position you indicate.

You can change the position of the arrow by dragging a selection fence around the arrow's control point at the tip of the arrow, and then dragging the arrow to a new location.

Diametral Arrow In



Dimensions the diameter of a circle with the arrow on the side of the circle indicated when it is picked.



To dimension the diameter of a circle, click near the circle. If you click outside the circle, the dimension is displayed outside, and if you click inside, it is displayed inside. Either way, an arrow line is drawn through the center of the circle from one side to the other.

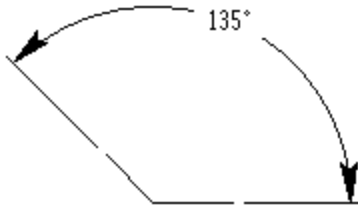
When the dimension displays, you can drag it to a new location. When you release the mouse button, the dimension and extension line are redrawn according to the position you indicate.

You can change the position of the arrow by dragging a selection fence around the control point at the tip of the arrow, and then dragging the arrow to a new location.

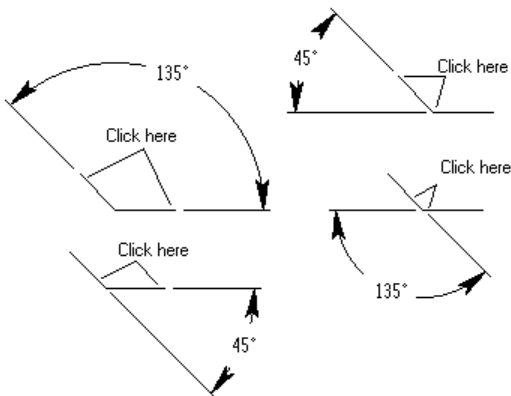
Angular Dimensioning



Dimension an angle.



To dimension an angle, click each arm, near the endpoint from which you want the angle measured. The inside angle is measured between the arms with the extension lines from the endpoints nearest the location you clicked on the arm. The smaller angle is used.



If the lines don't meet, the angle is measured from the invisible extension of the lines. The extension is displayed as part of the dimension.

When the dimension displays, you can drag the text to a new location. When you release the mouse button, the dimension and extension lines are redrawn according to the position you indicate.

Arc Length Dimension

This tool measures the length of an arc. To use it, choose the tool from the dimensions palette. Move the cursor to the arc to be measured and click on the end, middle and opposite end of the arc (as if you were creating a three-point arc). The dimension will appear on the screen.

Like other dimensions you can manipulate the text by changing its size or font using the "Text" menu. You can move the text by positioning the cursor over it. The cursor will change to a four-headed arrow. Then click and drag the text to the desired position. If necessary, the line pointing to the dimension will dogleg to follow the text.

Balloons



This dimensioning tool is used for ID positions, or adding text with a pointer to a specific point in a drawing. The balloon tool icons show the shape of the balloons they form. The icons with divided balloons have two text boxes.

To make a balloon, turn on the Dimensioning palette by choosing the "Show Palette" command from the Dimension menu. Choose the balloon tools from the bottom of the palette and select the balloon type and shape you want to use.

Follow the prompts in the help line to position the balloon. First, move the cursor to the exact point you want the balloon's arrow to point to and click. Next, move the cursor to the location you want the balloon to appear in.

The text which appears inside the balloon can be set by typing in the status line box marked "Text". You may also change the width of the balloon using the status line box marked "Width."

You can manipulate the text by changing its size or font using the "Text" menu. You can move the balloon with the cursor. Position the cursor over the balloon. The cursor will change to a four-headed arrow. Then click and drag the text to the desired position. The balloon's arrow will remain pointing at the original location, dog-legging if necessary.

