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GRAPHITE

User Guide - Graphite v7

GRAPHITE™

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User Guide Documentation

This *User Guide* is written for both Windows 98/NT/2000 and Power Macintosh platforms of Graphite. Before using this manual, however, you will need to install Graphite. Instructions are contained in the *Getting Started* section of this manual that came with the software. After installation, we encourage you to continue on with the tutorial exercises included in the *Getting Started* section of this manual. This will familiarize you with the tools, features and commands of Graphite and enable you to maximize your productivity in the shortest amount of time.

Chapter Layout

The chapters are arranged in the order of the design process starting with the basics. It then provides explanations of each tool, editing procedures, adding details and printing. Each chapter contains subheadings indicating the different sections and within those are further subdivisions as needed.

Windows and Power Mac Notations

Throughout the manual, notations are included to perform tasks using either platform. Steps are identical except for specific keyboard commands. In cases where the steps vary, each are listed.

Menus and Submenus

Choosing Commands

As you proceed through the exercises, you will be directed to choose commands contained in submenus of other menus, like the pull down menu. For example, you might be asked to select *Define* in the Color submenu of the Pen menu. That will be written as ***Pen>Color>Define***.

Tip:

Tips, referrals to other chapters, and alternative ways to accomplish a task are displayed in the margin throughout the manual.

Margin Notes

Graphite includes margin notes that provide information that may help you use Graphite. There are three types of margin notes: Tips, Tech Notes and Referrals. These notes have been given a special treatment so that you can instantly recognize their significance and locate them for future reference.

Tip

A tip provides instructions for getting the most out of Graphite. Tips may show you how to speed up an operation or how to perform some timesaving drawing technique.

Tech Note

A technical note is intended to provide additional technical information, not necessary for using Graphite but useful in understanding how it works.

Referral

A referral directs you to related information contained somewhere else in the manual for the particular topic being addressed.

Style Conventions

This manual uses various style conventions which highlight certain terms or phrases. The list below includes an explanation and an example in parentheses. The conventions are as follows:

Bold	Tool palette names (**Light** palette); Tool name (**Single Line** tool); Keyboard-entered text; Definition terms (as shown in these style conventions)
Italic	Terms used for the first time in a chapter; (*Parametrics*); Drafting Assistant notations (*midpoint*); tool and dialog box options (*Angle* data field); book references (*User Guide*); Message Line directions (*Single Line:Pick the beginning point.*); margin note headings (*Tip*); menu commands (*Extrude*); filenames (*Graphite.ini*); stand alone extensions (*.dwg*); directory names; drawing names

Bold and Italic	Command series (***Pen>ColorDefine***)
ALL CAPITALS	Key names on the keyboard (ENTER, RETURN)
Title Capitalization	Dialog box names (Edit Objects); menu names (Pen menu); special Graphite phrases (the Drafting Assistant)
all lower case	File names (Graphite.ini); stand-alone file extensions (.dwg)

User Guide Chapter Breakdown

The chapters are grouped into sections dealing with a specific area.

Sections

Referral:

Specific page information on a particular tool or command can be found in the index.

1. Overview	Contains chapters that provide you with information on the documentation layout, basic elements of Graphite and the Drafting Assistant.
2. Setting the Environment	Contains information on setting the drawing environment.
3. Creating Geometry	Contains information on creating geometry using the tools of Graphite.
4. Editing	Contains information on using the tools and commands to edit your geometry.
5. Document Detailing	Contains information on adding text and dimensions to your drawing.
6. Viewing Your Designs	Contains information on viewing your geometry.
7. Documents	Contains information on opening, saving, importing and exporting files.
8. Parametrics and Symbols	Contains information on using parametrics and placing symbols.
9. Geometric Analysis	Contains information on performing a 2D analysis on your geometry.

Appendices

Operators and Units	Describes all operators and units which are accepted by all Graphite data fields.
Special Characters	Lists all special characters not directly available from the keyboard and symbols that you can use in Graphite.
Creating Wireframe Models	Provides rules that will assist the 3D user in creating wireframe geometry.
DWG Notes	Provide notes to help the Windows user complete a successful *.dwg* translation.
Program Settings & Files	Contains all of the default settings for Graphite following installation. It also includes a list of all the folders and files that come with the program.

Other

Glossary	Defines terminology used in Computer Aided Design and Drafting (CADD) and Graphite.
Task Index	Groups various tasks according to their functions and the location in the manual.
Index	Lists features, tools and actions in Graphite and their associated page number location in the manual.

Graphics

Most of the graphics in the manuals apply to both platforms of Graphite. In those instances that require a platform and software reference, a Graphite Windows graphic is used. When necessary, both Windows and Macintosh graphics will be included.

On-Line Help

Graphite's Help (Windows only) provides a complete description of the program's many features, commands, and tools. The Help index is organized by menus and tools. In addition, context-sensitive Help is displayed when you highlight a command or tool and then press the F1 or the SHIFT+F1 key.

For information on installing Graphite, see the Readme text or Ashlar's web page, www.ashlar-vellum.com.

Graphite Overview

As mentioned in the *Getting Started* section of this manual, Graphite is a powerful program that is quick to learn and use due to its technology. Graphite will help you get your job done within your timeframe.

This chapter provides you with a brief overview of the following concepts:

- Wireframe Drafting
- The Model
- The Drafting Assistant
- The Design Process (including Drawing at Full Scale and Drawing Formats)

Wireframe Drafting

In Graphite you create wireframe models. A wireframe model consists of the geometry that makes up the edges of the object.

A wireframe model can often be used in place of a prototype, so that simulations and tests can be accomplished on the computer rather than in the laboratory. Models can be used for checking the visual specification, measuring distances between points within the model and observing the visual and real intersections of lines.

3D wireframe models are most useful for pulling off multiple views and doing 2D dimensioning and drafting on those views. This lets you do your design in 3D, and then produce 2D working drawings from that 3D model.

Referral:

The definition of models is also discussed in Chapter 13.

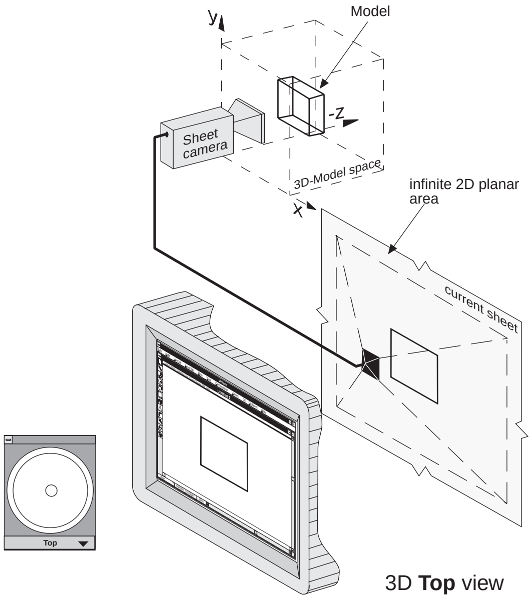
The Model

A Model exists in 3D space in the computer's memory, whether the geometry is two or three dimensional. A model is any combination of geometry, such as lines, arcs, circles, dimensions, text, etc. Models are created in an infinitely large three-dimensional area. (In the following graphics each model has its own imaginary 3D model space). Models are not created directly on the sheet. What you see on the sheet is only a projected view of the model.

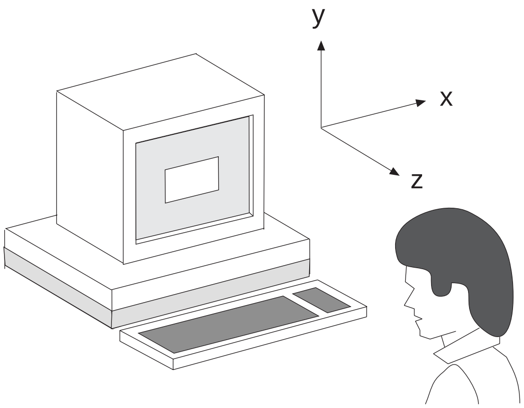
A sheet in Graphite is an infinite 2D planar area (always aligned parallel to the screen) that displays an image of one or more models. You view the model as if looking through a camera while it moves around the model. The image of a model is picked up by either a *sheet camera* or a *detail view camera*. The sheet camera projects the image directly onto the sheet (called a *Sheet View*). The detail view camera projects the image into a *view window* (called a *Detail View*) which rests on the sheet. Sheets do not actually contain any geometry, only images of geometry.

Your computer screen displays all the views on the current sheet. In the following graphic you see a Top view of the model picked up by the sheet camera and projected onto the current sheet. In the Top view the sheet camera is aligned parallel to the sheet.

When you open a document, you are looking down on top of the x, y plane and you cannot see anything in the z-direction, which is coming toward you, away from the screen.

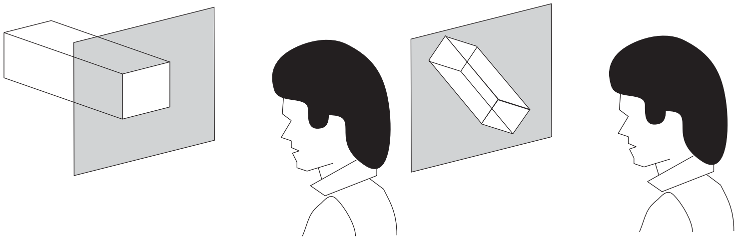


3D **Top** view

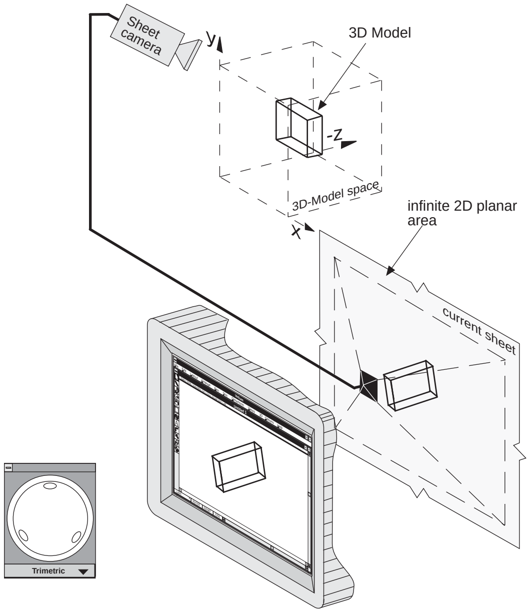


Graphite 3D Models

You must rotate the image to see what you are drawing in the z-direction.

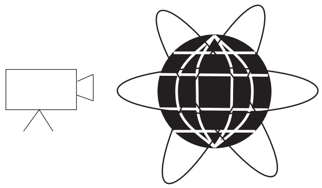


The on-screen *Trackball* lets you rotate the sheet camera around the 3D model (this changes your orientation, it does not rotate the model). The three-dimensional object geometry (the model) stays fixed, even if you get the opposite impression on the screen. In the following graphic the sheet camera is rotated to the Trimetric view.



The Sheet camera is rotated to the **Trimetric** view

When you rotate the image with the on-screen Trackball, it is as though the movement of the pointer on the Trackball corresponds to a fulcrum; the location where you press the mouse button becomes the fulcrum and the movement of the mouse rotates the model around that fulcrum point. What you see on the screen responds to the view of the sheet camera.



Graphite 3D

When you create view windows, either detail views, drafting layouts, or design layouts, each view window looks at the same model. When you make a change in any view window, the model changes; therefore, the model in every view changes.

For more information on models, see Chapter 14, “Advanced Viewing Techniques.”

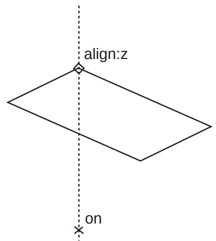
The Drafting Assistant

The Drafting Assistant is unique to Graphite and makes Graphite easy to use because it thinks like a designer. The Drafting Assistant guides a designer in the creation of geometry. It displays temporary construction lines, provides information about existing geometry, and gives notations of the relationship between new and existing geometry in all three dimensions.

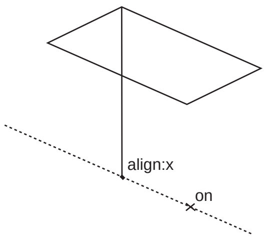
As you move the pointer near an existing geometric construction, the Drafting Assistant's *snap point* locks onto individual geometry, displaying an *on* notation. The Drafting Assistant also displays information about geometric landmarks, such as *endpoints* and *centers*, and temporary automatic construction lines, such as *alignment* and *tangents*.

The following examples illustrate the *alignment* notations for the x, y and z axes.

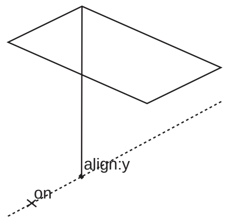
align:z The z-direction is perpendicular to the work plane.



align:x The x-direction is parallel to the work plane.



align:y The y-direction is parallel to the work plane.



For more information on the Drafting Assistant, see Chapter 4, "The Drafting Assistant."

The Design Process

The computer revolutionized the design process. Graphite has contributed to this by helping you quickly design a model that previously existed only in your mind's eye.

Graphite

For Graphite 3D, depending on your needs, you can start your design in 2D and continue on in 3D later or begin designing in 3D.

For example, you can choose to begin your design in the Trimetric view orientation so that you see all three directions at once. You can also display other view windows with the Front, Right, and Top view orientations at quarter scale to observe the construction from other angles. While you are drawing, you can zoom in and out to enlarge and reduce areas as needed.

You can also rotate, move the origin, and change the work plane to take advantage of the 3D modeling environment.

Drawing at Full Scale

In the paper world, you begin designing a model by deciding what scale to use so that the model fits on a particular size sheet of paper. With Graphite, you postpone scaling until after you have drawn the model at full scale. You can scale the geometry to fit on a standard drawing format provided by Graphite and then scale the whole drawing to fit the size of paper you need.

Constructing a 3D Model

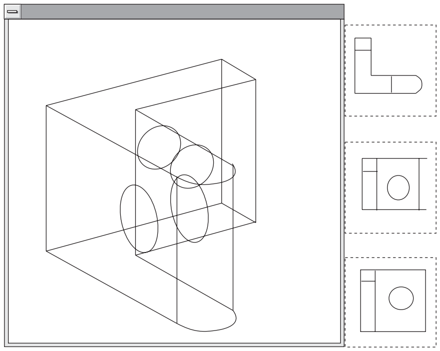
You can use several different methods to create a model. The following steps are an example of a process for a 3D model:

- Begin the construction by opening a new document.
- Set the preferences you prefer, such as the Pen Style, Grid, and selection modes.

  For more information, see Chapter 5, “Basic Environment Setting.”
- Display the Triad symbol using the *Show Triad* command. This illustrates the work plane orientation that acts as a point of reference when drawing in 3D.
- Display the Trackball.
- In the view menu of the Trackball, choose Trimetric. The sheet rotates so that your construction is seen from the Trimetric view.

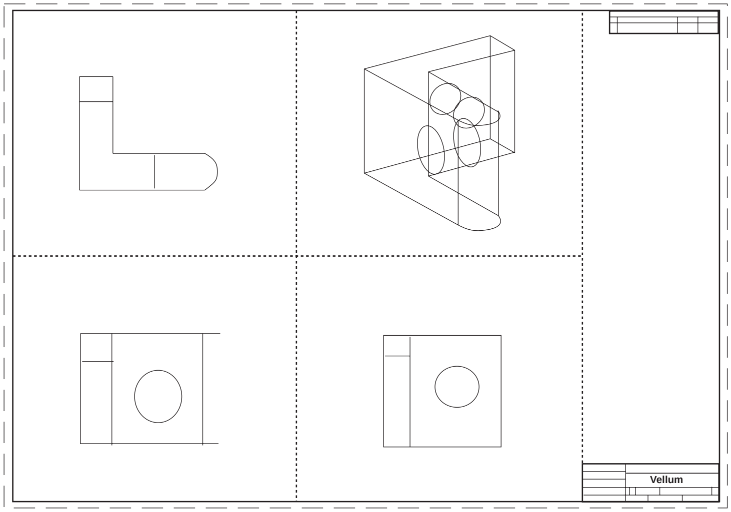
- Create the 3D model using the tool palette and Graphite commands.
- To observe multiple view orientations while you are modeling, choose *Sheet Into View* from the Views menu, and then specify the *Views Design 4* layout. View windows displaying the Top, Front, and Right view orientations are displayed at quarter scale along the right side of the screen.

  For more information on Sheet into Views, see Chapter 14, “Advanced Viewing Techniques.”



Drawing Formats

When the model is complete (or to be submitted for review), you might want a drafting version incorporating several views on a drawing format with dimensions and annotation. To create; import a drawing format and make adjustments to the views as necessary (or use a premade layout with format in *Sheet Into View*). This example shows a premade layout with four views (available only for the 3D version).



- Flatten any view that contains geometry you want to edit independently. For more information on flattening a view, see Chapter 14, “Advanced Viewing Techniques.”
- Crosshatch, dimension, annotate, and perform any view editing you need.
- Fine-tune the model if necessary.
- Specify the paper size from the *Print Setup* (Windows) or *Page Setup* (Macintosh) command in the File menu.
- Set the scale for the drawing format to fit on the plotter paper. If you don’t care about the exact scale, you can click Fit in the Drawing Size dialog box and the geometry and format will be scaled appropriately for the paper size you have specified.
- Plot the finished drawing. For more information on plotting, printing and related activities, see Chapter 15, “Graphite Documents.”

The Basics

This chapter describes the basic components of Graphite. This brief overview of useful features may be all you need to know if you are familiar with CAD software. The following topics are covered:

- Using a Mouse
- Parts of the Graphite Window
- Menu Bar, including the dialog boxes
- Preferences

For more information about standard elements such as menus, scroll bars, File menu commands, and dialog boxes, refer to the Windows or Macintosh User's Guide that came with your computer.

Using a Mouse

The mouse is your communication device; you use it to tell the computer what you want to do. Use the mouse to indicate locations, choose commands, select tools, and construct objects.

If your mouse has more than one button, you can use the right button to popup a menu that contains a variety of commands and functions. By default the right mouse button provides shortcuts to the most popularly used commands.

To set up the right mouse button options, right click in the drawing area to activate the menu. Choose ***RightMouse.*** When the dialog box appears, check the boxes to activate or deactivate items you want displayed in the popup menu.

Tips:

Windows only: If you are left-handed and your mouse has more than one button, you can change the functionality to the right mouse button. You make this change in the Control Panel of Windows.

This manual uses the following terms for mouse activities:

Pointer An arrow or any other graphic symbol that allows selection or creation of an object. Move the pointer to point to a command or an object on the screen. Depending on its location, the pointer is an arrow or looks like the current tool.

Arrow Pointer *Selection Arrow* *Center-Point Circle*

To move the pointer, move the mouse on the mouse pad. You use several different mouse actions with Graphite.

Point Move the mouse until the pointer is over the item you want.

Press Press and hold down the mouse button.

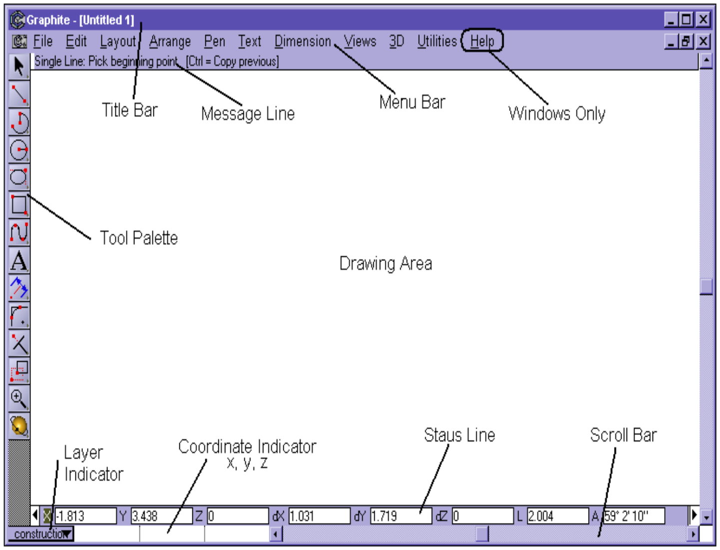
Click Quickly press and release the mouse button once.

Double-click Click the mouse button twice, quickly in succession.

Drag Press and hold down the mouse button, move the mouse, then release the mouse button.

Parts of the Graphite Window

When you start Graphite, the following window appears.



Title Bar — Includes the title of the active document and buttons for controlling the window including boxes for zooming and closing the program.

Menu Bar — Contains the Graphite menus of commands and settings. You can make choices from the menus with the mouse or by using special key combinations.

Tool Palette — Contains the drawing and editing tool icons you use for constructing, editing and annotating geometry.

Pointer — Shows the active position on the screen. If the pointer is in the drawing area, its shape represents the current tool.

Pointer Locator — Shows the x, y coordinates of the pointer location.

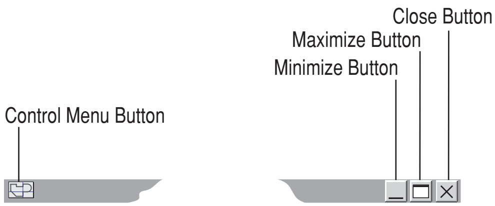
Message Line — Displays the name of the current tool and step-by-step instructions for using the tool.

Drawing Area	Consists of multiple layers where you construct and annotate geometry.
Status Line	Shows the coordinate location and other geometric parameters of the current construction.
Scroll Bars	Allow you to move around a drawing so you can see different sections of it through the Graphite window. The scroll buttons allow you to move one line at a time.
Work Layer Indicator	Displays the name of the current layer and provides a menu for changing the work layer.

Title Bar

The Title Bar includes the name of the current document, and the Control Menu, Minimize and Maximize/Restore buttons (Windows) or the Close and Zoom boxes (Macintosh).

Windows



Control Menu Button	Allows you to close, move, and change the size of the window. This button is available on all windows and many dialog boxes. Double-clicking this button closes the window without displaying the menu. If you want to choose a different option from the Control menu, click the button once to display the menu and then make your choice.
Minimize Button	Reduces the Graphite window to an icon near the lower-left corner of the screen. This action does not close or save the document, it only shrinks the window to an icon so you can perform some other Windows-related task. To redisplay the window, double-click the icon.

Maximize/Restore Displays the window, full or partial screen. Once the window appears full screen, click the button again to restore it to its previous size.

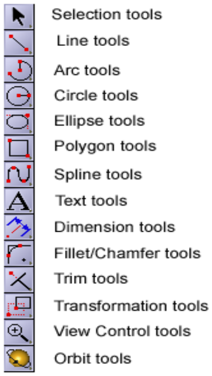
Macintosh

Close Box Closes the window when you click on it. If you attempt to close the window without saving your work, Graphite displays a message so you can decide whether to save or not.

Zoom Box Toggles the window size between the previous size and full size.

Tool Palette

A tool palette is a group of tool icons along the left side of the screen. The icons represent the tools for drawing, editing, and annotating geometry.



Selecting a Tool from the Tool Palette

1. Position the arrow pointer on the icon of the tool you want to use.
2. Click the mouse button.

The icon appears highlighted to indicate its selection. The **Single Line** tool is selected here.

Tool Subpalettes

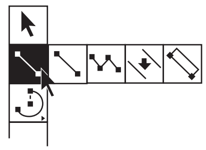
Most of the tools in the tool palette contain a subpalette of tools with related functions. The (arrow) in the lower-right corner of the tool icon represents the presence of a subpalette which contains related tools.

Viewing and selecting from a subpalette are similar to choosing a command from a menu.§

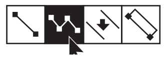
Selecting a Tool from a Subpalette

1. Position the arrow pointer on the tool.
2. Press the mouse button.

   The subpalette appears to the right of the tool.



3. Drag the pointer to highlight the desired tool.



4. Release the mouse button.

   The selected tool replaces the previous tool in the tool palette. The highlighted icon in the tool palette shows that your selection from the subpalette is the active or current tool.

The new tool is visible in the tool palette until you select another tool from the same subpalette. The tools in the subpalette remain in the same order; only the tool displayed on the tool palette changes.

Once you select a tool, additional information appears to help with your construction. The Pointer, Pointer Locator, Message Line, and Status Line all provide feedback about the active tool.

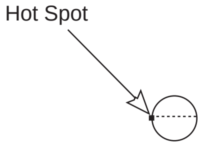
If you want to select a tool that is already displayed in the tool palette, you only need to click it; you don't need to select it from the subpalette.

Smart Pointer

When you select a tool and move the pointer into the drawing area, the pointer shape represents the selected tool.

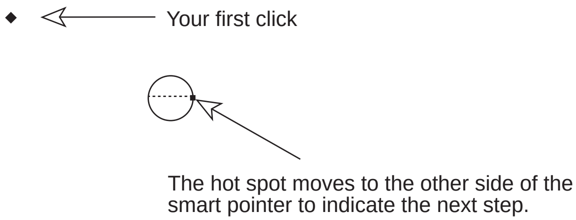
Some of the pointers, like the single line pointer, are simple cross-hairs. Others, such as the Opposite-Point Circle pointer, resemble the tool itself.

The pointer, called a smart pointer, displays indicators for multi-step procedures. Each smart pointer has a dot, the hot spot, showing the next point you should specify. The dot changes position on the pointer during each step of the construction.

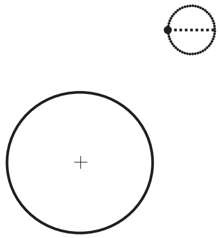


The smart pointer shows you where to click next.

For example, the Opposite-Point Circle pointer illustrated above shows that the first click of the mouse places a point on one edge of the circle you're creating. After you click a location, the hot spot moves to the other side of the pointer, showing that the next click places a point on the opposite edge of the circle. (See the graphic below).



After you click the second location, the circle appears. The hot spot moves back to its original position on the pointer so that you can create another circle.



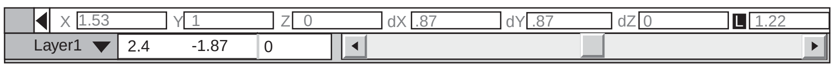
Tech Note:

The number of decimal places displayed in the locator field is determined by the *Precision* setting in the Units dialog box (choose ***Layout>Preferences>Units***).

The location indicator tracks the pointer position of any tool other than the **Selection** tool.

Pointer Locator

The Pointer Locator is two numbers to the left of the horizontal scroll buttons at the bottom of the drawing area.

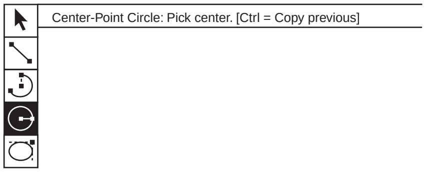


This locator continuously tracks the pointer location when the pointer is in the drawing area, displaying the X,Y coordinates of the current location relative to the origin. The origin (0,0,0) is in the center of the screen when you open a new document. When you make the grid visible by choosing ***Layout>Show Grid***, a symbol appears at the origin (0,0).

Message Line

The Message Line across the top of the drawing area provides concise instructions for the use of the selected tool.

For example, after selecting the **Center-Point Circle** tool, the Message Line appears as illustrated below:



The instructions in the Message Line for some tools also indicate optional activities. For example, if you hold down the CTRL (Windows) or OPTION (Macintosh) key while using the **Center-Point Circle** tool, the next mouse click creates a copy of the last circle with the center placed where you clicked.

Status Line

The Status Line provides measurements, angles, X,Y coordinates and delta values for the current construction. The current tool determines the number of status data fields and which of the status data fields is highlighted after the construction. For example, if you select the **Center-Point Circle** tool, the Status Line shows the X,Y coordinates for the center of the circle and the length of the diameter.

When you click the last point of the circle, the diameter (D) data field highlights in the Status Line to indicate that it is active. It shows the diameter of the circle you just created. If you type a new number, and press the ENTER (Windows) or RETURN (Macintosh) key, the diameter of the circle updates reflecting the change.

Tip:

You can also change objects with the *Edit Objects* command.

You can change any or all entries in the Status Line, but when you press ENTER (Windows) or RETURN (Macintosh), you can’t make any more changes in the Status Line.

The number of decimal places displayed in the status data fields is determined by the Precision setting in the Units dialog box (choose ***Layout>Preferences>Units***).

Moving Between Status Data Fields

Use the TAB key to move to the right, highlighting the next field. When you press ENTER (Windows) or RETURN (Macintosh), the construction redraws according to the new specifications in the Status Line. You can also use your mouse to activate a Status Line data field.

You can use the Status Line arrows to scroll if any of the status data fields are off screen.

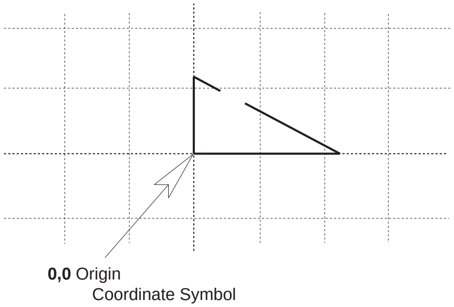
Drawing Area

You use the drawing area for all construction, editing, and annotation of geometry. Think of the drawing area as a sheet of paper of unlimited size that you use to construct full-size unscaled drawings. You use the scroll bars to move the sheet so the portion you want to work on is visible in the window.

Displaying the Grid

If you wish to work with a grid in the drawing area, choose ***Layout>Show Grid***.

When the grid is visible, constructions snap to the grid, meaning that any geometry point that you click snaps onto the closest grid point. The coordinate symbol appears at the origin when the grid is visible.

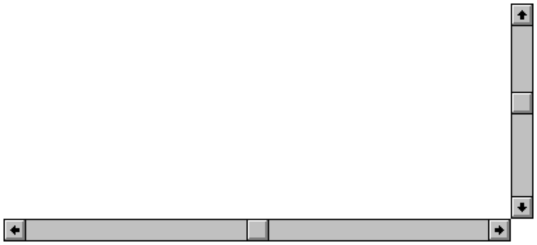


Tip:

Normally, you do not need to display the Grid, since the Drafting Assistant offers a more elegant support than the grid.

Scroll Bars

The scroll bars allow you to move the sheet up and down or right and left. You can display different parts of the drawing sheet by dragging the slider of a scroll bar to the approximate location. For example, the right, center, or left position in the horizontal scroll bar displays the right side, middle, or left side of the drawing, respectively.

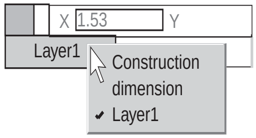


You can also click the arrows at the end of the scroll bars to move the sheet one line at a time. If you click in the scroll bar, the sheet moves one window at a time.

Work Layer Indicator

The work layer indicator in the lower-left corner of the screen shows which layer is the current work layer. New geometry goes on the work layer. If you want your construction to go on a layer, first make it the current layer.

To select a work layer, position the pointer over the work layer indicator, then press the mouse button. All available layers are then displayed in a pull-down menu from which you can select a different layer to be the current work layer.



Drag to the layer and release the mouse button. The selected layer becomes the work layer and all geometry you create will be placed onto that layer.

Menu Bar

The Graphite menus contain related commands and settings.

File	Commands that affect entire documents (files).
Edit	Commands to select and manipulate objects.
Layout	Commands and settings that specify the drawing area and provide program features and functionality, such as construction lines and 2D analysis.
Arrange	Commands for zooming (to change the area displayed in the window) and setting specifications for objects.

Pen Commands to specify pen characteristics (style, color, weight, and pattern), crosshatching, fill and arrows.

Text Commands to set the font, size, style, alignment, and indentation of text and also to create text blocks and forms (title blocks).

Dimension Commands that specify dimensions and their format and tolerance.

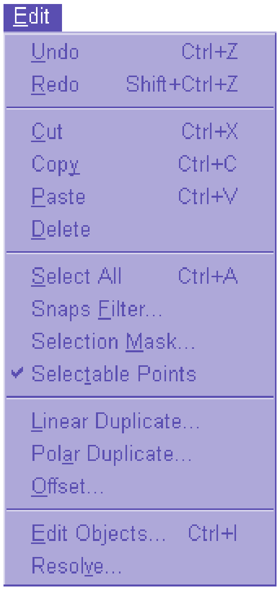
Views Commands to control multi-page documents, including Perspective on/off, Perspective Edit and View Mode. The menu also contains Define View, View the Plane, Unfold View, Flatten View and Show Trackball.

Utilities Commands to create and invoke macros.

Displaying a Graphite Menu

1. Point to the menu name.
2. Click on the name.

   The menu appears. If you want to dismiss the menu without making a choice, click outside the menu.



Choosing a Command from a Menu

1. Point to the menu name.
2. Click on the name.

   The menu appears.
3. Click on the command.

   The command executes, or the setting, such as Selectable Points, toggles on or off.

Mouse versus Keyboard

Graphite's menu items can be chosen with the mouse or with a combination of keys on the keyboard. For example, you can use various methods for displaying the Edit menu.

Windows and Macintosh:

- Click on Edit in the menu bar.

Windows only:

- Press the ALT key and then type **E**.
- Press the ALT key and then press the RIGHT ARROW key until Edit is highlighted in the menu bar; then press ENTER.

You also have various methods for choosing commands with the keyboard. For example, you can use any of the following methods to choose ***Layout>Show Grid***.

Windows:

- Press ALT and L and then type **G**.
- Press ALT and then use the RIGHT ARROW key to highlight Layout and press ENTER. Then press the DOWN ARROW key to move the highlighted area to Show Grid and press ENTER.
- Hold down the CTRL key and type **G**.

The first method is the mnemonic method. Press the ALT key with the appropriate letters for the menu and command as indicated by the underlined character in the names.

Macintosh:

- Hold down the z (command) key and type **G**.

The third method for Windows and the only one available for Macintosh is a keyboard accelerator. When available it is denoted by the key sequence listed on the menu.

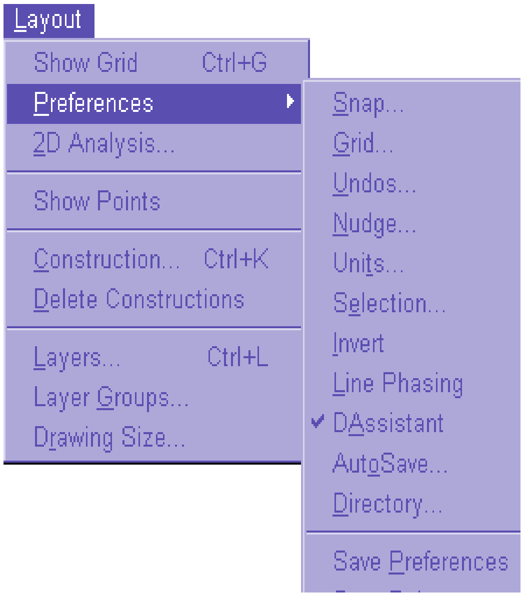
While keyboard functionality is always available, this manual generally describes making choices with the mouse.

Submenus

Commands followed by an (arrow) symbol have submenus which display when the command is highlighted.

1. Pull down the menu.
2. Click on a command followed by an (arrow) symbol.

   The submenu displays.
3. Click on the submenu.
4. Click the desired command.

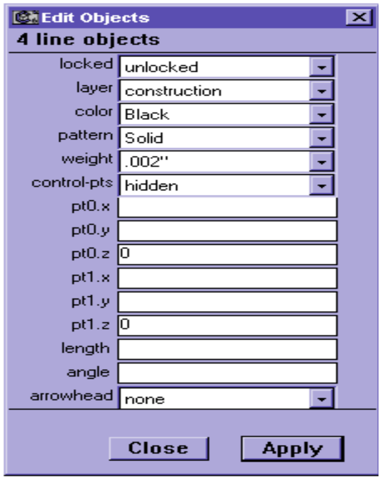


Dialog Boxes

When you choose a command followed by an ellipsis (…), such as Edit Objects in the Edit menu, a dialog box appears.

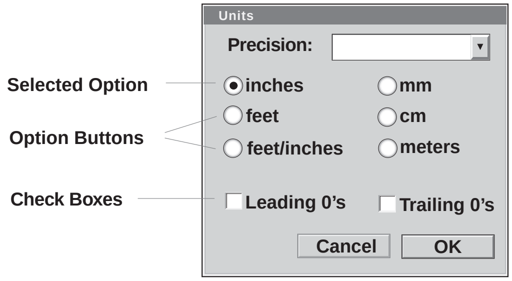
Dialog boxes allow you to qualify the command you chose by adding information. For example, in the Edit Objects dialog box above, you can change the specifications of the selected object.

If a dialog box obscures your view of the drawing area, you can move it to a new location by dragging it with the pointer on the Title bar.



Option Buttons

Option buttons indicate mutually exclusive choices; you can select only one option at a time. Click the option you want and the button turns black, as shown by the inches option below.



Check Boxes

Check boxes, as shown above, provide options you can switch on and off and which are not mutually exclusive. A (check mark) shows the options is set.

List/Entry Boxes

Tip:

Windows: You can also hold down the ALT key and type a letter to select the first item that begins with that letter, then use the arrow keys to move to the selection you want. Once your choice is highlighted, press ENTER.

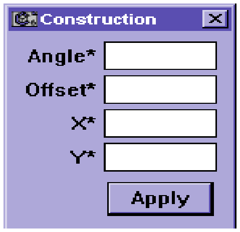
Some dialog boxes contain lists of options, displaying an arrow to provide access to the list.

If the entry includes an arrow you can display a menu which works like a submenu on the menu bar but the item you specify appears in the box once you select it.

Some list boxes also allow you to type an entry. For example, you can type a value in the Scale data field in the Drawing Size dialog box or choose from the pop-up menu. See the graphic below.

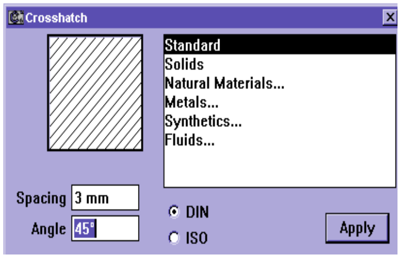
To type an entry, select the current entry (if it isn't already selected), then type a new entry. In most cases, clicking on the OK button, saves the changes.

Asterisks



When an item in the dialog box displays an asterisk (*), you can specify a value by clicking or dragging in the drawing area. This feature is particularly useful for specifying location because you don't need to know any x, y coordinates.

Apply Buttons



Some dialog boxes have an Apply button that allows you to apply the specification you just set. You can leave the dialog box open to set other specifications.

For example, once you crosshatch a part, you can leave the Crosshatch dialog box open and select other objects to be crosshatched.

Closing a Dialog Box

If a dialog box contains an OK or Cancel button or an action button such as Open, the dialog box closes when you click the button. Otherwise, you dismiss the dialog box manually by double-clicking the Control Menu (Windows) or clicking the Close Box (Macintosh) button in the upper-left corner of the box. If the dialog box has an Apply button, such as the Crosshatch dialog box above, you must dismiss it manually.

Toggling Commands

Commands that set a condition (such as Selectable Points and Arrow At Start) display a check mark (check mark) in the menu to indicate that they are active. To turn a command off, choose it and the check mark disappears.

In the case of pen styles and text characteristics, the check shows the current setting.

Other commands, Show Grid, Show Points, Show Palette, Show Trackball and Show Triad toggle to Hide (Grid, Points, Palette, Trackball or Triad) when the component is visible.

Preferences

All files are saved with their settings when you choose ***File>Save***. The characteristics used for new files (the default settings) are contained in the preferences file.

The preferences filename is prefs.vc6 (Windows) or Graphite prefs (Macintosh). You can change the default settings so that every new document opens with the settings you want. The following specifications can be set in the preferences file:

- Pen styles
- Text characteristics
- Preferences settings (snap, grid, units, selection color indicator, visualization, palette status and palette location)
- Grid display
- Layer and sheet specifications
- Work Layer
- Dimension and tolerance formats
- Arrowhead type and display
- Drawing size and scale
- Zoom scale
- Fillet radius
- Chamfer angle and length
- Resolve values

The default value of any setting that can be changed on a menu or in a dialog box can be set in the preferences file.

Changing the Default Settings Manually

1. Open the preferences file (it's in the same folder as the Graphite application).
2. Change the characteristics you want and, if needed, create the layers and sheets you want.
3. Save and close the file.

   The file must be stored in the same folder as the Graphite application.

The preferences are set for subsequent new documents.

The settings will not take effect until the next time you launch Graphite.

Changing the Default Settings with Save Preferences

1. Open a new Graphite file.

2. Set the preferences as you want them.
3. Choose ***Layout>Preferences>Save Preferences***.

The preferences are set for this file. The settings for future files take effect the next time you launch Graphite.

Important: It is advisable to use Save Preferences only before you start drawing. Otherwise, you may be saving data that you do not want in new files. For example, if you choose Save Preferences on a file with a detail view and multiple models, all new files will contain a detail view window and multiple models.

The Drafting Assistant

The patented Drafting Assistant is the feature that makes Graphite unique among design and drafting software products. The Drafting Assistant thinks like a drafter; it automatically *knows* where you typically want construction lines and displays them temporarily when you need them.

The Drafting Assistant also makes it easy to select existing points for construction by displaying information about the pointer's location in the drawing area. If a Drafting Assistant notation displays when you click, the construction snaps onto the geometry precisely, without requiring finely tuned eye-hand coordination or tedious selection of special modifiers, modes, or other specialized construction tools.

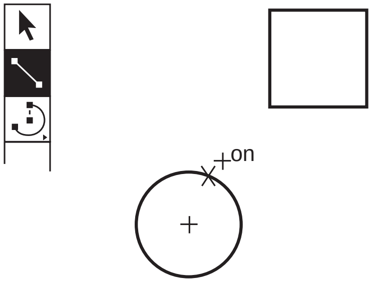
The following topics are covered in this chapter:

- Snapping onto Geometry
- Drafting Assistant Construction Lines
- Permanent Construction Lines

Snapping onto Geometry

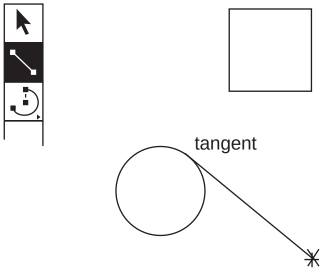
When the pointer is in the drawing area, it has a snap point function. The snap point locks onto specific points on existing objects as you move the pointer near them.

The Drafting Assistant tells you when the snap point is *on* an object.



The Drafting Assistant snapping onto a circle

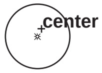
The Drafting Assistant displays information about the location of the snap point. This information appears either beside the pointer or next to the object itself.



The Drafting Assistant displaying the relationship between the circle and the line that is being constructed.

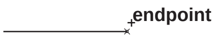
The Drafting Assistant tells you when the snap point has locked onto the points of an object.

center



The center of an arc or circle. Move the pointer across the arc or circle to display *on* for the arc or circle, then move the pointer near the center to display the *center* point notation.

endpoint

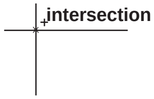


The endpoint of lines, arcs, circles, ellipses and splines.

midpoint

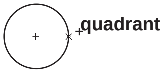
The midpoint of lines, arcs, circles, ellipses and splines.

intersection



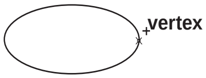
The intersection of two lines (permanent lines and the Drafting Assistant's dynamic construction lines) or curves.

quadrant



Quadrant points on an arc or circle displayed at 3 o'clock, 6 o'clock, 9 o'clock and 12 o'clock.

vertex

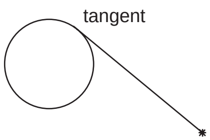


The vertices of an ellipse, spline, or dimension point.

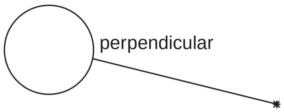
Using Tangents and Perpendiculars

If you click a point *on* an arc or circle and drag the pointer away at about a 45º angle, the Drafting Assistant locks onto the *tangent*. If you drag away at a 90º angle the Drafting Assistant locks onto a *perpendicular*.

Tangent

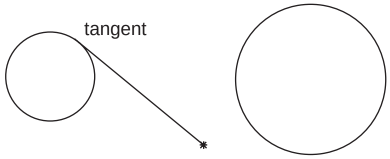


Perpendicular

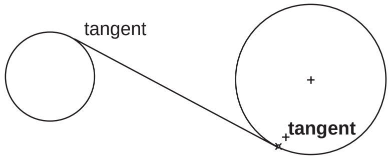


If you continue holding down the mouse button, the line remains *tangent* or *perpendicular* while you drag the ending point around the object.

This is a useful feature if, for example, you want to create a line from and tangent to an existing circle to the tangent point of another circle.



Once a line is *tangent* to the circle, it can be dragged to the tangent point on the other circle, with the tangency maintained at both ends.



The Drafting Assistant locks onto a *tangent* or *perpendicular* only when the Drafting Assistant starts from the *on* notation. You cannot begin from a specific point, such as *endpoint*, *quadrant* or *vertex*.

Tech Note:

Another way to create geometry from an exact location relative to another object is to reduce the Hit Radius of the Drafting Assistant with a lower number of pixels or to zoom in on the drawing to separate the construction points visually.

Keyboard Snap Points

You can direct the Drafting Assistant to snap onto an object. For example, you may want to start a new line from the exact center of a circle. If you hold down the mouse button and press the **c** key on your keyboard, the Drafting Assistant finds the *center* of the circle when you move the pointer near the center.

Using the Drafting Assistant for Snapping onto Geometry

The following table lists the keys for finding specific points. The desired point must be within the Hit Radius (defined later in this chapter) of the pointer. Press the mouse button first and then press one of the following keys on the keyboard.

Keyboard snap points only work when there are multiple snap points within the Hit Radius of the pointer.

Letter	Snaps To (Align)
c	center
e	endpoint
g	grid
i	**int**ersection
m	midpoint

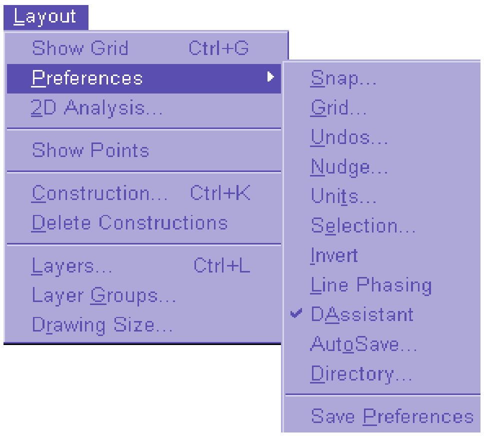
n	no point
o	on
p	perpendicular
q	quadrant
t	tangent
v	vertex
%	the percentage point set in the Snap dialog box

Pressing the SPACEBAR or clicking the mouse releases all snap restrictions.

Keyboard snap points act as a filter. For example, if you want to place the end of a line at the center of a circle but the intersection of two other objects lies near the center (within the specified Hit Radius), the Drafting Assistant will not know which point—the *center* or the *intersection*—to snap to. By pressing the **c** on the keyboard while dragging the line, the Drafting Assistant knows to snap to *center* and to disregard the *intersection* (or any other snap point that falls within the Hit Radius).

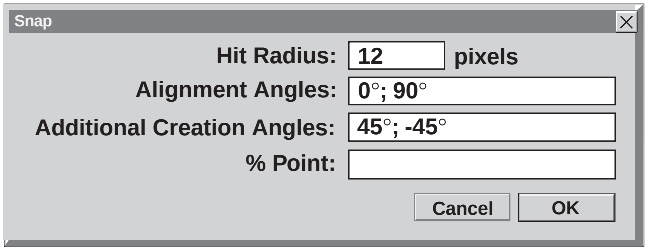
DAssistant Command

This command, found in ***Layout>Preferences***, turns the Drafting Assistant on or off. When the Drafting Assistant is not checked in the menu, the Drafting Assistant is off.



Snap Basic Command

This *Snap* command, found in ***Layout>Preferences***, sets specifications for the Drafting Assistant. This command should be distinguished from the *Snap* command, chosen through the Utilities menu, that allows you to activate snap modes of the Drafting Assistant.



Hit Radius

This setting determines the detection distance in pixels. When the pointer is within the specified Hit Radius, the Drafting Assistant notations are displayed and the object is selected when you click the mouse. The default Hit Radius is 12 pixels.

If you can't specify locations that are close together because the Drafting Assistant snaps to an existing control point, you can do any of the following:

- Decrease the Hit Radius.
- Zoom in so 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 example).

If you set the Hit Radius to zero, you disable the single click selection of the **Selection** tool (dragging a selection fence and double-clicking to select all objects will still work). Instead, you might consider using the *Selection Mask* in the Edit menu to specify that some objects cannot be selected.

Alignment Angles

These angles define the dynamic construction lines that the Drafting Assistant automatically uses. If you want to change the orientation of your drawing, you can change these specifications.
For example, you could set these angles to 30°, 90° and 150° for an isometric drawing. The defaults are

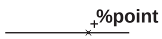
0° (horizontal) and 90° (vertical). Use a semicolon to separate the values.

To display a temporary construction line through a point, move the pointer to the point to activate it (a diamond appears); 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 deactivates.

Additional Creation Angles These lines are used by the Drafting Assistant only when you are creating geometry and they are not part of the list of lines generated from the eight active points. The defaults are 45° and -45°. Use a semicolon to separate the values.

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

For example, entering 25 instructs the Drafting Assistant to tell you when the pointer is 25% of the distance along a line as shown below.



Snap Mode Command

This *Snap* command is automatically available when you first start Graphite. This command allows you to activate and deactivate the different snaps modes of the Drafting Assistant. This command should be distinguished from the *Snap* command, chosen through the ***Layout>Preferences*** menu, that allows you to set basic specifications of the Drafting Assistant.

Choosing this item displays the following dialog box:

All snap modes are activated by default. When you want to deactivate a snap mode you click the related check box.

Drafting Assistant Construction Lines

In addition to snapping onto geometry, the Drafting Assistant also displays *dynamic construction lines*. The three types of construction lines you use most frequently—vertical, horizontal and 45° angle lines—display automatically during construction. These construction lines appear temporarily to help you align geometry. Once you've set a point, the Drafting Assistant construction line disappears—your drawing is not cluttered with extraneous lines.

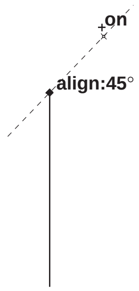
Dynamic construction lines extend automatically from the last point you created. You may want to activate other points so the Drafting Assistant displays construction lines relative to them. Simply move the pointer over the geometry to activate or "wake-up" its control points, then move away horizontally or vertically.

Of course, you also have the option of creating *permanent construction lines* and other shapes, as described later in this chapter.

Using the Drafting Assistant's Construction Lines

After you have indicated the first endpoint of a line, and you move the pointer horizontally, vertically, or in a 45° direction, the dynamic construction lines appear.

The figure here illustrates a 45° construction line relative to the endpoint of an existing line.



Displaying Dynamic Construction Lines while Constructing Geometry

1. Click a point to begin new geometry.
2. Move the pointer away from the point horizontally, vertically, or at a 45° angle.
3. While the construction line is visible and the Drafting Assistant displays ***on***, click the next point.

   The point is placed exactly on the construction line, even though the pointer wasn't exactly on that line when you clicked. The dynamic construction line disappears.

Displaying Dynamic Construction Lines with Existing Geometry

1. Without pressing the mouse button, move the pointer over an existing point.

   The point notation (*endpoint* or *midpoint*, for example) shows that the point is active.
2. Move the pointer horizontally or vertically.

   A construction line appears through the point.
3. While the construction line is visible and the Drafting Assistant displays *on*, click the desired point in the construction.

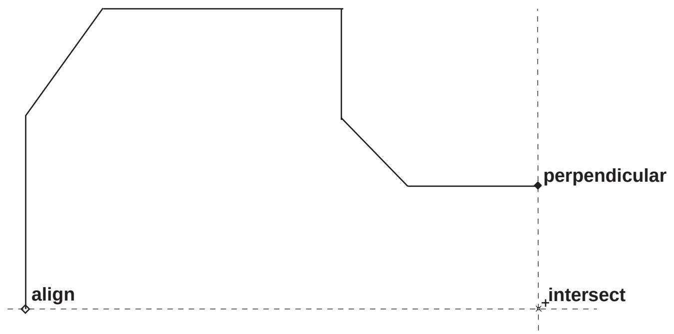
   The point is placed exactly on the construction line, even though the pointer wasn't exactly *on* that line when you clicked. The dynamic construction line disappears.

Tech Note:

You can have as many as eight active points; activating the ninth point in a series deactivates the first point.

The figure here illustrates intersecting construction lines drawn through two existing, active points.

Once you're familiar with the Drafting Assistant, you'll see how Graphite streamlines design and drafting tasks!



Setting New Drafting Assistant Construction Angles

You can add to or change the angles that the Drafting Assistant uses for dynamic construction lines by choosing ***Layout>Preferences>Snap***. Enter the construction line angles, separated by semicolons, in the appropriate data field.

Permanent Construction Lines

Referral:

More information on Layers can be found in Chapter 13, "Viewing Geometry."

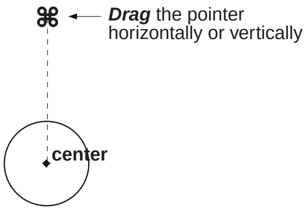
In addition to the Drafting Assistant's dynamic construction lines, you may want to create construction lines that remain displayed until you hide or remove them. There are two methods for creating permanent construction lines: strokes and the *Construction* command (in the Layout menu). Construction lines automatically appear on the construction layer, not the work layer of your drawing. You can hide the construction layer to view or print the drawing without construction lines.

When you want to get rid of all construction lines, choose ***Layout>Delete Constructions***. (Everything on the construction layer deletes, regardless of the object's pen style).

Stroke Construction Lines

Stroke construction lines are lines that you create with the mouse. Hold down the CTRL+SHIFT keys (Windows) or the ⌘ key (Macintosh) and drag the mouse horizontally or vertically. Since you are working in the drawing area, the Drafting Assistant helps you place the stroke precisely.

Holding down the CTRL+SHIFT keys (Windows) or the ⌘ key (Macintosh) changes the mouse pointer to the Stroke point (⌘).



Drag	Result
Vertically	A vertical construction line through the first point of the stroke.
Horizontally	A horizontal construction line through the first point of the stroke.

Tip:

You can use strokes to create construction lines while you are in the process of using a tool.

Using Stroke Construction Lines

Construction lines are as long as the dimensions of the viewing area of the screen or the plot region (as designated in Drawing Size dialog box in the Layout menu), whichever is larger.

For example, if you are using the **Connected Lines** tool, you can create a construction line that extends through the center of a circle:

1. Hold down the CTRL+SHIFT keys (Windows) or the ⌘ key (Macintosh). The pointer becomes the Stroke pointer (⌘).
2. Move the pointer near the *center* of the circle.

   The Drafting Assistant snaps onto the center point.
3. Drag the mouse vertically or horizontally away from the *midpoint*.

The construction line appears through the *center* and you are still in the process of creating connected lines after releasing the CTRL+SHIFT keys (Windows) or the ⌘ key (Macintosh).

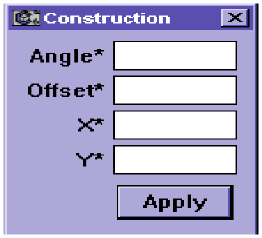
The Construction Command

Stroke construction lines are useful for creating lines through existing points. If you want to create a construction line at a location other than an existing point or at a particular angle, you can use the *Construction* command.

Tech Note:

Graphite automatically places construction lines on the construction layer. When you choose ***Layout>Delete Constructions***, everything on the construction layer is deleted, regardless of the object's pen style.

Using the Command CTRL+K (Windows) ⌘+K (Macintosh)



This command in the Layout menu creates a construction line on the construction layer of the current document.

You can specify the angle of the construction line or the offset from a reference point defined by the X, Y coordinates. The asterisk shows that the values can be specified by clicking or dragging the mouse; values can also be typed into the data fields.

The distance dragged will always be entered in the Offset data field as a positive value, regardless of the direction dragged.

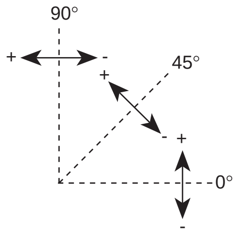
Specifying the Construction Line Angle with the Mouse

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

The angle of the vector line appears in the *Angle* data field.

Specifying the Construction Line Offset with the Mouse

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



The distance you drag appears in the *Offset* data field. Be aware that the offset is determined by the angle of the construction line as shown.

Specifying X,Y Coordinates with the Mouse

The coordinates of the last point you specified appear in the *X* and *Y* data fields, but you can change them by doing the following:

1. Click the *X* data field.
2. Enter new coordinates.

   or

   In the drawing area, click the location of the point through which the construction line should pass. The coordinates are entered automatically for both the *X* and *Y* data fields.

Tip:

Windows: You can also type a letter to select the first item that begins with that letter, then use the arrow keys to move to the selection you want. Once your choice highlights, press ENTER.

Creating Multiple Construction Lines

You can create multiple construction lines through the same point by entering the angles you want separated by semicolons.

Creating Parallel Construction Lines

You can create parallel construction lines by specifying a single angle value with different offsets separated by semicolons. You can also create parallel lines by creating one construction line using this data field, then creating new lines with the **Parallel Lines** tool.

Creating Construction Geometry

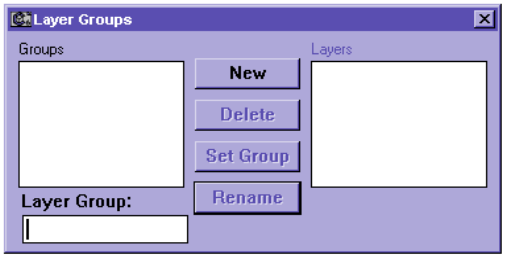
Non-construction geometry is placed on the work layer. You can create temporary construction geometry, such as arcs or circles, by making the construction layer the work layer, creating the geometry and switching to another layer to continue your work.

You can use the Construction pen style if you like, but it's not essential.

Lines made with the Construction pen do not go on the construction layer unless you make that layer the work layer.

Creating Construction Geometry

1. Choose ***Layout>Layers***.
2. Click Construction in the list box.
3. Click Set Layer.
4. Create the geometry you will use for construction. You can use the Construction pen style, but any pen style is acceptable.



5. When the construction geometry is complete, make another layer the work layer.
6. Close the dialog box.
7. Continue your work.

Once you no longer need the construction geometry, choose ***Layout>Delete Constructions*** to remove all geometry on the construction layer.

Tip:

You can select construction lines in the usual manner from any layer; the construction layer doesn't have to be the work layer.

Removing Construction Lines

If you have many construction lines and you want to delete only one or two of them, select the lines you want to remove, and then choose ***Edit>Delete*** or press the BACKSPACE (Windows) or DELETE (Macintosh) key. You can remove all the construction lines that you've created by choosing ***Layout>Delete Constructions***.

Delete Constructions

This command in the Layout menu deletes all construction lines and any geometry on the construction layer. The Drafting Assistant's dynamic construction lines appear only temporarily and are not affected by this command. Any geometry on the construction layer (regardless of the pen style used) is deleted by this command.

You can retrieve deleted construction geometry within the eight (8) level limit of the *Undo* command.

Basic Environment Settings

This chapter describes the options you have for adapting Graphite to your needs. The following topics are covered:

- Pen Styles
- Pen Characteristics, including the color palette
- Setting Units
- The Grid
- Drawing at Full Scale
- The Drafting Process
- OS Settings
- Saving Preferences
- Saving Palettes

Pen Styles

The pen style determines the appearance of lines on the screen and during plotting. If you are using a monochrome monitor or printer, all lines will be black but the weight and pattern will be visible. Any line thickness of less than .016 inch appears one pixel wide on the screen. When you print or plot such lines, you can see the different weights.

The default pen style is Outline—solid, black lines, .01 inch wide. You can change to a different pen style or change an individual characteristic of the current pen.

You can choose from nine different pen styles in the Pen menu. The characteristics are listed below.

Pen	**Pattern**	**Weight**	**Color**
Outline	Solid	0.010	Black
Visible	Solid	0.020	Black
Hidden	Hidden	0.016	Red
Dash	Dashed	0.016	Yellow
Center	Center	0.010	Green
Pen	**Pattern**	**Weight**	**Color**
Phantom	Phantom	0.010	Cyan
Dimension	Solid	0.002	Blue
Balloon	Solid	0.005	Green
Construction	Dotted	0.002	Magenta

When you want to change pens, you have three options: choose a different pen style from the Style submenu in the Pen menu; choose different characteristics for the current pen from the pen characteristics listed in the Pen menu, (Color, Weight and Pattern); or use the *Edit Style* command in the Style submenu to permanently alter the characteristic of the pen. The current pen style is checked in the Style submenu.

Changing the Pen Characteristics of an Object

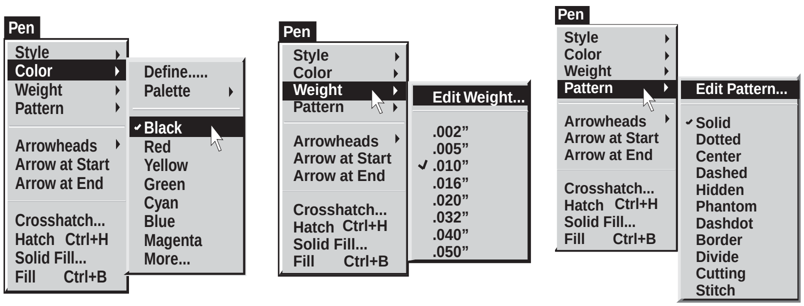
1. Select the object.
2. Choose ***Pen>Style*** and select the pen style you want to change the characteristics of the selected object. If you only want to change one characteristic rather than the entire pen style, choose the characteristic from the pen characteristics listed in the Pen menu.

   When you make this change, the current pen also changes, thus affecting future constructions.

If you want to change the pen characteristics of an object without changing the current pen, select the object, then choose ***Edit>Edit Objects*** and change the characteristics in the dialog box.

Pen Characteristics

The following characteristics are available from the submenus in the Pen menu.



Color

There are 234 color options including 170 user definable colors and 64 predefined colors. This **Color** palette displays the 64 predefined colors when you select ***Pen>Color>Palette***.

Each definable color can be independently assigned from the 16.7 million colors available.

This *Color* command specifies the color of the current pen and any other selected geometry without changing any other pen characteristics.

Specifying the Color of the Current Pen

1. Choose ***Pen>Color***. The submenu appears.
2. Choose the color you want.

   The pen takes on the new color as selected in the submenu.

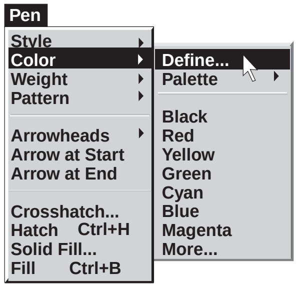
Defining Colors

You can define as many as 234 colors. If you choose ***Layout>Preferences>Save Preferences*** following their definition, they will be available each time you open a new file.

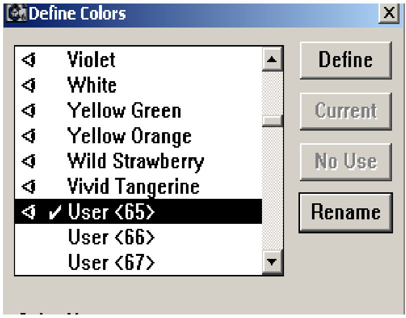
Because the color displays are different for Windows and Macintosh machines, the process for defining a color vary slightly. Step #5 and higher take this into account.

Windows (Macintosh users continue on page 5-7)

1. To define a new color, choose ***Pen> Color>Define***.

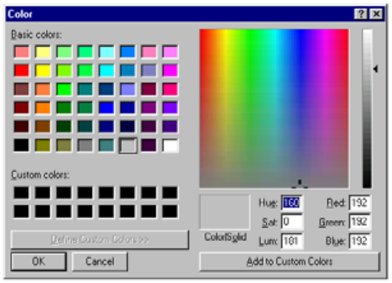


2. The Edit Colors dialog box appears.
3. In the dialog box window, scroll down and select one of the undefined color numbers, like <User65> (the first definable color available).
4. Enter the color name (limited to 15 characters) in the *Color Name* data field and click Rename. The name entered replaces User <65> in the colors list, as displayed here.



5. Click Define. The color display appears on the screen.

6. Click Define Custom Colors to display the full color display.



The color display contains the following elements:

Basic Colors	This section displays the 64 default colors available in the palette.
Custom Colors	You can display 16 more user defined colors in this area for a particular file. These save with the particular file but do not save as a default setting when using ***Layout>Preferences>Save Preferences***.
Add to Custom Colors	Use this button to place a color in the Custom colors section.
Color Palette	You select your color by using this palette.
Color/Solid	The selected appears in this swatch.
Hue, Saturation, Luminance	These fields are automatically filled in when you select a color. You can also enter values into these fields.
Define Custom Colors	Click this when you see a partial color display to show the full color display.
Red, Green, Blue	These fields are automatically filled in when you select a color. You can also enter values into these fields.
Cancel, OK	Click Cancel to close the display without saving the color selection. Click OK to accept the color selection.

7. In the palette on the right, place the pointer within the target cursor and drag to the desired color. The color displays in the **Color/Solid** area below the palette.

   You can also adjust the *Hue*, *Saturation*, *Luminance*, and the percentage of Red, Green and Blue by entering your own values. The *% of black* can be adjusted by moving the cursor over the arrow in the scroll bar to the right of the color palette and drag up or down to the desired location. All the values in the *Luminance*, *Red*, *Green* and *Blue* data fields change as well. Hue and Saturation do not change.

8. Once you have chosen the color, click OK, the Color display closes and you are returned to the Edit Colors dialog box.

   The colors that you rename, remain at the end of the color list in the Edit Colors window in the order that you placed them. They are not alphabetized.

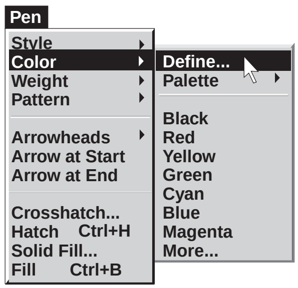
9. To make the added colors appear in the palette when you choose ***Pen>Color>Palette***, click Use. An icon of an eye appears. The color is now visible in the palette.

   You can also place the cursor to the left of the color name and click to display the eye icon.

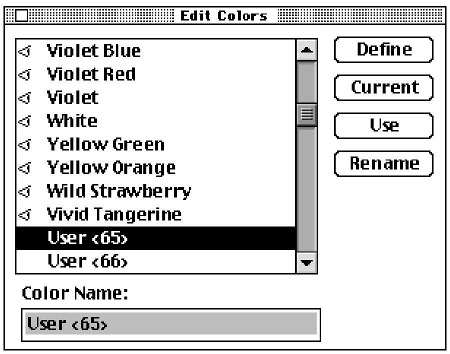
10. Close the dialog box.
11. Repeat this process to define up to 170 colors.
12. You can save these colors as default color settings for all new files by choosing ***Layout>Preferences>Save Preferences***.

Macintosh

1. To define a new color, choose ***Pen> Color>Define***.



2. The Edit Colors dialog box appears.
3. In the dialog box window, scroll down and select one of the undefined color numbers, like <User65> (the first definable color available).
4. Enter the color name (limited to 15 characters) in the *Color Name* data field and click Rename. The name entered replaces User <65> in the colors list, as displayed above.

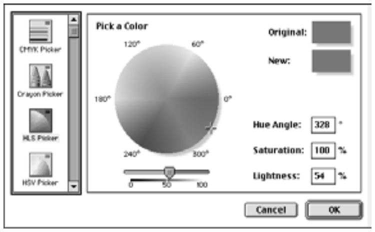


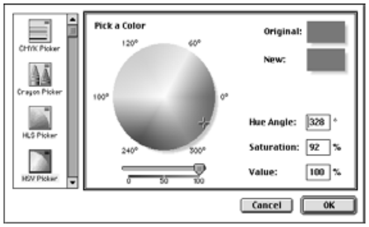
5. Click Define. If you click a definable color, one of four color displays appears: *HSL Picker*, *HSV Picker*, *CYMK Picker*, or the *Crayon Picker*.

Tech Note:

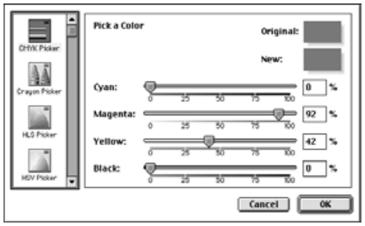
Mac Users with Operating Systems previous to 8.0 have two color display options rather than four.

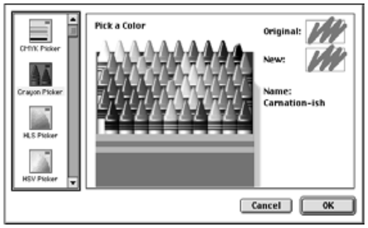
HSL Picker *HSV Picker*





CYMK Picker *Crayon Picker*





The *HSL Picker*, *HSV Picker*, *CYMK Picker* and *Crayon Picker* contain the following elements:

Color Wheel (HSL)	Contains the color spectrum from which you choose a color. Click or drag within the wheel to select the color. The color values appear in the **Hue** and **Saturation** data fields.
Lightness Slide Bar (HSL)	Allows you to set the lightness for a particular color. Place the cursor over the slide and drag to the desired lightness. The lightness value appears in the data field.
Hue Angle, Saturation, Lightness Data Fields (HSL)	Values can be entered into these fields to select a color.
Color Slide Bar (CYMK)	There are three bars; one each for Red, Green and Blue. Place the cursor over the slide and drag it to adjust the color value for each bar. Values are entered in the data field.
Red, Green and Blue Data Fields (CYMK)	Values can be entered into these fields to select a color.

Name (Crayon)	The color name displays when you choose a crayon color.
Crayon Box (Crayon)	Displays the colors available for selecting.
More Choices/Fewer - Choices button	This toggles between the two options. Choosing More Choices displays two icons for the *HSL Picker, HSV Picker, CYMK Picker* or *Crayon Picker* displays. Choosing Few Choices turns off the icon display.
Original/New Fields	*Original* displays the current color setting for the color you are defining. When you define a new color, the swatch is gray. New displays the new color you have selected.
Cancel and OK buttons	Clicking Cancel closes the color display without saving the color selection. Clicking OK closes the dialog box and saves the settings you have made.

6. Choose a color from the color display.
7. Click OK and the color will be defined as chosen.
8. Repeat this process to define up to 170 colors.
9. You save these colors as default color settings for all new files by choosing ***Layout>Preferences>Save Preferences***.

Editing User-defined Colors

You can edit the colors you have previously defined using a procedure similar to that for defining colors.

1. Choose ***Pen>Color>Define***.
2. The Edit Colors dialog box appears.
3. In the dialog box window, scroll down and select one of your defined colors.
4. Click Define. The color display appears.

   Edit the color. (Windows: Click Define Custom Colors to display the full palette.)
5. Click OK to save the changes.

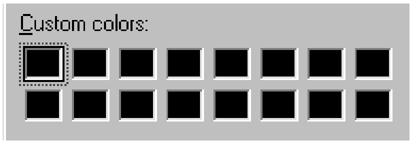
Tech Note:

The 64 predefined colors that came with Graphite cannot be redefined or edited.

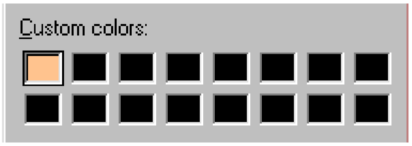
Add to Custom Colors (Windows only)

This feature is standard for the Windows platform. It allows the you to save 16 user defined colors for a particular file and display in the *Custom colors* area of the Color display when the *Define* command is chosen.

1. Repeat steps 1 through 5 in the previous section, “Defining a Color,” and choose *Define* to show the partial color display.
2. Place the cursor over one of the available boxes in the *Custom colors* area and click. A dotted line appears around that box to show that it is selected.



3. Click Define Custom Colors and the full color display appears.
4. Move the cursor to the color palette and drag it to the color you desire. You can also adjust the % of black in the color by dragging the arrow, in the scroll bar to the right, to the desired location.
5. Click Add to Custom Colors. The color appears in the C*ustom colors* patch.
6. To fill in the others boxes, go through the same process.



If a name has not been assigned to one of these colors previously, the program will choose the next available undefined color number to assign to the color. You will have to later rename it for your own purposes.

Saving Colors

Choose ***Layout>Preferences>Save Preferences*** to save the colors you have defined. Colors created for a specific file are saved with the file if they have not been saved using *Save Preferences*.

(Windows only) Custom Color only saves with a specific file. See the previous section.

Customizing the Color Palette

This feature allows you to choose which colors display in the **Palette** when you choose ***Pen>Color>Palette***.

Displaying a Color in the Palette

1. Choose ***Pen>Color>Define***.

2. Select the color you want to display.
3. Click Use. An eye icon appears next to the color name.

   You can also click in the blank space to the left of the name to display the icon.

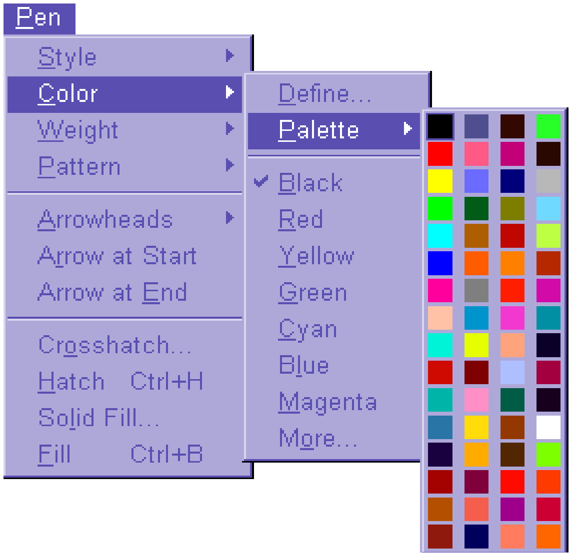
   When you choose ***Pen>Color>Palette***, the color patch has been added to the palette.

Removing a Color from the Palette

1. Choose ***Pen>Color>Define***.
2. Select the color you want to remove.
3. Click No Use. The eye icon next to the color name is removed.

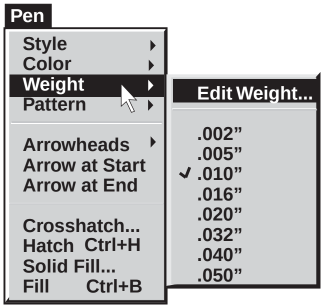
   You can also click on the eye icon to remove it.

   When you choose ***Pen>Color>Palette***, the color patch has been replaced by a patch with an X in it.



Weight

This command in the Pen menu sets the pen width of the current pen and selected lines without changing any other pen characteristics.



The line weights shown in this submenu depend on the units selected in the Preferences submenu. If you have selected metric units, the line weight will be shown in millimeters instead of inches.

Note: Any line thickness of less than .016 inch appears one pixel wide on the screen. When you print or plot such lines, you can see the different weights.

Specifying a New Weight for the Current Pen

1. Choose ***Pen>Weight***. The submenu appears.
2. Drag to the weight you want.

   The pen takes on the new weight, as selected in the submenu.

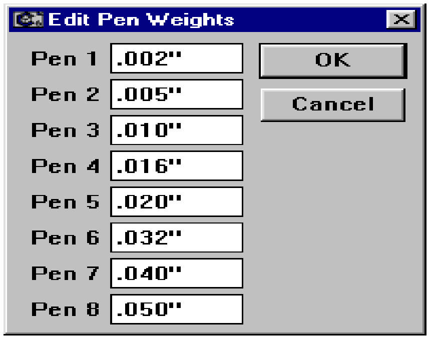
Edit Weight Command

This command from the Weight submenu in the Pen menu sets the line weights. Graphite comes with eight default line weights ranging from .002” (.05mm) to .050” (1.27mm). Objects drawn in Graphite are drawn in one of the eight available pen weights. Choose ***Pen>Edit Weight*** to change the weight of one of the pens to any value between .001” (.0254mm) to .050” (1.27mm).

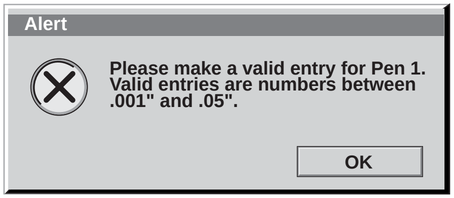
There cannot be more than eight different pen weights in a drawing, so changing a pen weight in the Edit Pen Weights dialog box changes the weight of every object drawn with that pen’s previous weight.

Editing Pen Weights

1. Choose ***Pen>Weight>Edit Weight***. The Edit Pen Weight dialog box appears.
2. Select the pen weight you want to change by clicking on the pen number. That pen’s data field is selected.
3. Change the pen weight by typing a new value into the data field. If no units are given with the entry, Graphite applies the unit specified in the Units dialog box from Preferences.



If the entry does not fall with in the valid range from 0.001” (.0254mm) to .05” (1.27mm), Graphite prompts you with an Alert box similar to the one below.



Clicking OK in the Alert box returns you to the Edit Pen Weights dialog box to make the necessary changes.

4. Click OK in the Edit Pen Weights dialog box. All existing lines drawn in the pen weight you edited change to reflect the new weight, as will all future lines drawn in that pen weight.

   Clicking Cancel ignores all changes made to any pen weights and closes the Edit Pen Weights dialog box.

Undoing a Pen Weight Edit

You cannot undo editing a pen weight with the *Undo* command. To return a pen back to Graphite’s default, you must enter the original value in the Edit Pen Weight dialog box, following the steps described above.

The default weights are as follows; Pen 1 - .002", Pen 2 - .005", Pen 3 - .010", Pen 4 - .016", Pen 5 - .020", Pen 6 - .032", Pen 7 - .040", Pen 8 - .050".

Edited pen weights affect only the pen weights in the current file. To save the edited pen weights so that they are available in all future files, choose ***Layout>Preferences>Save Preferences***.

Tech Note:

Remember that pen weights thinner than .015’’ appear on the screen as one pixel thick. However, you should be able to see the difference in pen weights on your prints and plots.

Pattern

Graphite provides you with eleven standard line patterns and the ability to create nineteen additional line patterns. Using the *Pattern* command, you can set the pattern of the current pen and any selected lines without changing any other pen characteristics. You can also define new patterns and edit all line patterns currently available.

Specifying a New Pattern for the Current Pen

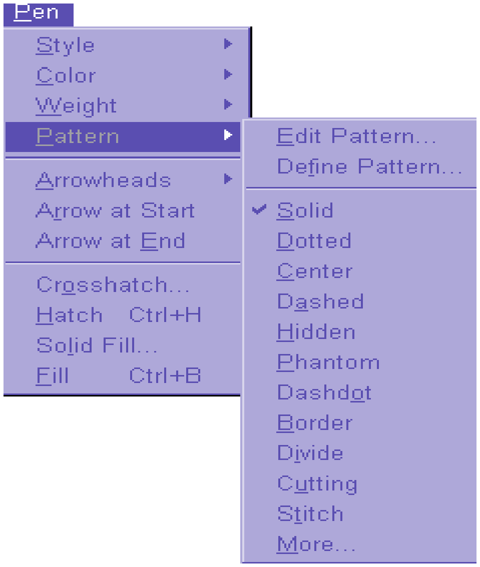
You can choose a standard line pattern or a user-defined pattern. When you select a pattern, regardless of whether an object is selected, the pattern becomes the current pen. If you had previously chosen a particular pen style, you will notice that style is no longer checked in the pen style submenu.

Setting a Standard Pattern

1. Choose ***Pen>Pattern***. The submenu appears.

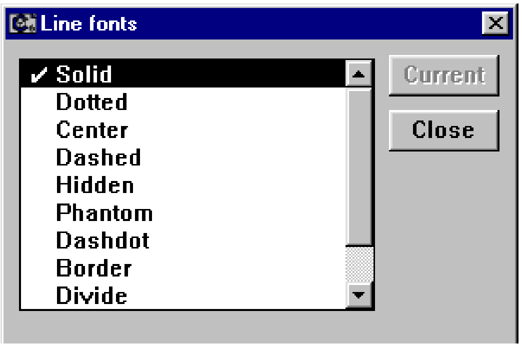
2. Select a pattern.

   The pen takes on the new pattern.



Setting a User-defined Pattern

1. Choose ***Pen>Pattern>More***. The Line fonts dialog box appears.



2. Scroll down to the desired pattern and select it.

3. Click Current.

   The pen takes on the new pattern.

Defining and Editing Pen Patterns

Graphite allows you to edit the standard patterns included with the program or create and edit up to nineteen of your own patterns.

Define Pattern Command

This command, found by choosing ***Pen>Pattern>Define Pattern***, allows you to define up to nineteen line patterns.

Graphite considers each line segment and each space between line segments separate dashes. Each dash has a handle, represented by a box attached to the vertical line extending from the end of each dash.

One complete element of the line pattern appears in the pattern window. The default visible length is two inches. If you want to create patterns with dashes or elements larger than two inches, change the value in the *Visible Length* data field to an appropriate number; the Pattern Window scales the pattern element accordingly.

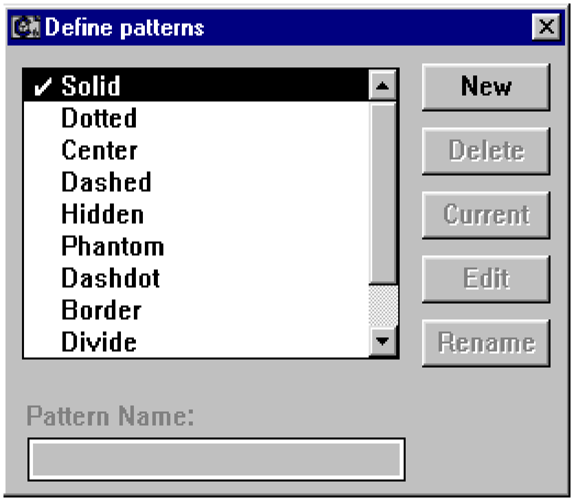
Graphite automatically scales line patterns by the inverse of the scale factor set in Drawing Size dialog box or Sheet Into View dialog box so that the pattern spacing is appropriate for the viewing and drawing scale of the geometry. Sometimes, you might want to set the pattern spacing independently of the viewing or drawing scale, or change the pattern altogether. The Edit Pattern dialog box lets you control the pattern spacing for nine of Graphite's eleven different line patterns. Solid and Dotted patterns cannot be edited.

These patterns can be saved as default patterns by choosing ***Layout>Preferences>Save Preferences***. If you want these patterns to be saved with the file only and not as default patterns, just save the file.

Defining a Pen Pattern

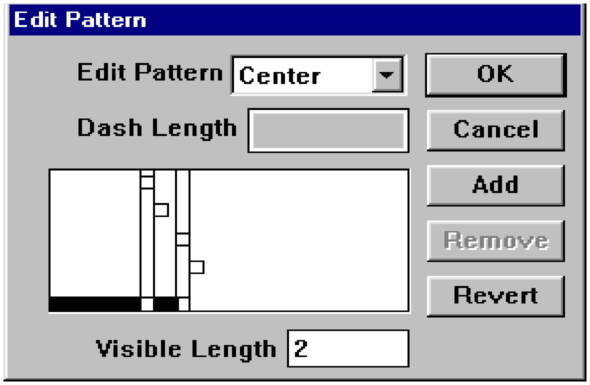
1. Choose ***Pen>Pattern>Define Pattern***. The Define patterns dialog box appears.
2. Click New.

   The User1 default name is added to the bottom of the patterns window and appears in the Pattern Name field.



3. Click Edit. The Edit Pattern dialog box appears with the default name, User1, in the Edit Pattern field.

   By default there are two dashes, a solid line segment and a blank segment.



4. Create you own pattern by modifying the length of these segments and/or adding dashes.

   Clicking the dash handle (to the right of the dash) activates the handle for the pattern segment—the handle is now filled. The dash's current length appears in the Dash Length data field. You have two ways to change the length of a dash.

   - Enter a new value in the Dash Length data field and changes the length of the selected dash.
   - Drag the handle. As you drag, the dash's length updates in the **Dash Length** data field. Release the mouse button when the dash is the length you want.

   To add a new dash, click Add. A solid and blank dash are added with a length of zero. The solid dash handle is activated. You can change the length of the new dashes by dragging their handles or by typing in a value in the Dash Length data field. If you select a handle before clicking Add, the new dashes are placed in front of the selected dash.

   To remove a dash, select its handle and click Remove. The dash is removed from the pattern element.

   Note: Pattern elements start with solid dashes, and then switch to blank dashes, and then, if necessary, back to solid, and so on. Adding or removing dashes may change existing dashes from solid to blank or from blank to solid.

5. When the pattern element is properly defined, click OK. The new pattern is created and you are returned to the Define patterns dialog box.

   Click Cancel to close the Edit Pattern dialog box without retaining the pattern settings.

6. Select the Pattern Name field and enter a new name for your pattern.
7. Click Rename.
8. Click Current if you want to set the pen to the new pattern.

9. Click the Close button in the title bar to close the dialog box.

Edit Pattern Command

This command, found by choosing ***Pen>Pattern>Edit Pattern***, allows you to edit all line patterns but Solid and Dotted.

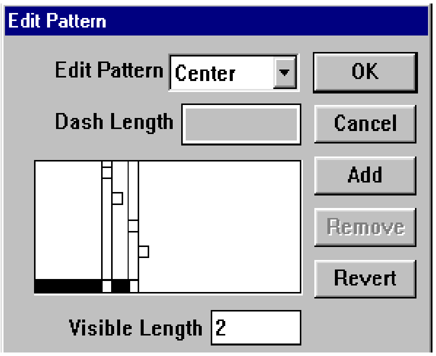
If you edit a pattern already used in geometry within your drawing, the geometry automatically updates to the revised pattern. For example, if you edit the Phantom line pattern, every line in the existing document drawn in the Phantom line pattern changes to reflect the revision.

Edited line patterns also scale by the inverse of the viewing and drawing scales, but editing the pattern lets you control how the patterns appear on the screen and on paper.

Editing a Pen Pattern

1. Choose ***Pen>Pattern>Edit Pattern***.

   The Edit Pattern dialog box appears.

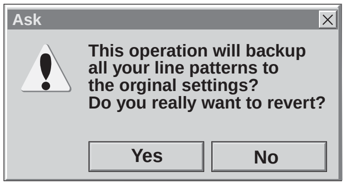


2. Select the line pattern you want to change from the *Edit Pattern* list.
3. Modify the line pattern by changing the lengths of existing dashes or by adding or removing dashes to or from the element.
4. When the pattern element is properly defined, click OK and the dialog box closes. All existing lines drawn in the line pattern you edited change to reflect the new pattern element, as will all future lines drawn in the line pattern.

   Click Cancel to ignores all the changes made to any pattern elements and close the Edit Pattern dialog box.

Undoing a Pattern Edit

You cannot undo a pattern edit with the *Undo* command. For the standard patterns that came with the program, you return them back to the default by clicking **Revert** in the Edit Pattern dialog box. Revert changes the line patterns back to their



defaults, not just the pattern selected from the Edit Pattern list. A warning message is presented when you click Revert.

Clicking Yes changes the patterns back to Graphite's original settings. Clicking No returns you to the Edit Pattern dialog box without returning any patterns back to their original settings.

To undo changes made to user-defined patterns, previously saved as a preference, save the file (if necessary), close and relaunch the program, and then open the file. All user-defined patterns return to their previously defined settings. Edited Patterns affect only the patterns in the current file. To save the edited pattern so that it is available in all future files, choose ***Layout>Preferences>Save Preferences***.

Pen Styles

Pen styles help simplify your design process by defining the color, weight and pattern type for each style. Then rather than having to set each pen characteristic each time you want a specific type of line, you can choose one of the predefined styles. You are provided with nine pen styles.

The following are the default settings for the styles in the Pen menu:

Outline

Solid, .01 inch, black lines - the default pen setting.

Visible

Solid, .02 inch, black lines.

Hidden

Hidden pattern, .016 inch, red lines. Hidden lines are actually visible on the screen and in your drawing; they represent lines that would be hidden in a solid object.

Dash

Dashed, .016 inch, yellow lines.

Center

Center, .01 inch, green lines.

Phantom

Phantom, .01 inch, cyan lines.

Dimension

Solid, .002 inch, blue lines. (These lines are .002 inch thick, regardless of the weight you specify.)

Balloon

Solid, .005 inch, green lines.

Construction

Dotted, .002 inch, magenta lines. This does not create construction lines but uses construction line characteristics. (True construction lines are placed on the construction layer and are deleted when you choose ***Layout>Delete Constructions***.

Modifying Pen Styles

You may want to modify a pen style to suit your needs. You can change one characteristic of the style by choosing it in the Pen menu or you can change the style for the duration of the work session by choosing ***Pen>Style>Edit Style***.

Changing One Characteristic of a Pen Style

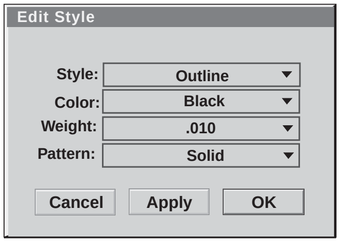
1. Choose the pen you want to modify.
2. In the Pen menu, choose a characteristic—Color, Weight, or Pattern.

To return a standard pen to its original specifications, choose a different pen style from the menu and then choose the modified pen again. You may also change all of the specifications for a standard pen by redefining it with the *Edit Style* command by choosing ***Pen>Style>Edit Style.***

Edit Style Command

This command from the Style submenu of the Pen menu sets the characteristics for the pen styles.

Redefining the Specification of a Pen Style



1. Choose ***Pen>Style>Edit Style***.

   The dialog box appears.

2. Select the pen style you want to change by pressing the *Style* data field.

   When the list of pen styles appears, choose the style you want to change.

3. Specify the characteristics (*Color*, *Weight*, and *Pattern*) you want for that style.

4. Click OK to set the specifications and close the dialog box, or click Apply to put the specifications into effect for a single pen style and leave the dialog box open to make changes to another pen style.

When you edit a style, you set new specifications for future uses of that pen. The specifications remain in effect for the document in which they are set until you change it again with the *Edit Style* command. You can change the default setting of any pen style by saving changes in the preferences file. See the “Saving Preferences” section at the end of this chapter.

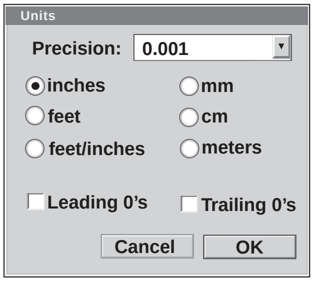
Although style characteristics can be changed, style names cannot. Keep this in mind if you change the pen pattern. Otherwise, the style name may not reflect the pattern and cause confusion if the changes are saved as a preference.

Setting Units

Graphite is set to measure geometry in inches. When you open a new drawing you should set the precision, units, and fractional or decimal specifications, to suit your needs.

Units Command

This command is found by choosing ***Layout>Preferences>Units***. A dialog box displays.



Precision

Sets the number of decimal places in all data fields within Graphite (in Edit Objects, the Status Line and all dialog boxes). The default is 0.001.

When you choose a fractional precision, measurements appear as fractions rather than decimals. You can set the format for fractions by choosing ***Dimension>Linear***.

When Graphite rounds off values it stores the number, accurate to 16 decimal places; therefore, you can change the precision displayed at any time and maintain the desired accuracy.

Altering the decimal or fractional precision does not affect any dimensions. The precision of dimensions is selected by going to the Dimension menu and choosing the setting in the Linear and Angular submenus. However, a fractional precision is required if you want fractional dimensions. The same is true for decimal dimensions.

Inches, feet

Displays measurements in English units. Measurements less than 1 foot appear in inches and those greater than 1 foot appear in feet and inches.

mm, cm, meters

Displays measurements in metric units.

Leading 0's

Determines whether or not the zero to the left of the decimal point appears.

Trailing 0's — Determines whether or not zeros to the right of the decimal point are displayed. If set, the precision determines the number of trailing zeros.

All geometry reflects the new unit measurement, converting automatically between English and metric. When you change the units, existing dimensions update to reflect the change.

Using the Status Line to Specify the Unit of Measure

You can use the Status Line to specify a unit of measure other than the one in effect for the document. If the units are set to inches and you draw a line, you can use different units of measure such as inches ("), feet ('), feet and inches (x'y”), millimeters (mm), centimeters (cm) and meters (m). You can also mix the units in the mathematical expression as long as you label the units properly—for example, 10" + 25.4 cm.

In the following example, if the units are set to inches, you can still enter **23'** in the Length data field as shown below.

L 23'

When you press ENTER (Windows) or RETURN (Macintosh) a line 23 feet long is drawn. Graphite converts the length to inches and Status Line displays the length in inches because of the Units setting.

In addition, you can use mathematical, exponential, and trigonometric expressions in the Status Line. Appendix A provides examples of the valid operators.

L cos(30)

The Grid

When the Grid is visible, objects snap to the grid spacing. If you want to place an object between the grid marks, you must either turn off the grid, change the grid spacing, or zoom in so that the spacing is larger than the Hit Radius. The grid spacing automatically reflects the units you set in the Units dialog box. When you change the units, the appearance of the grid spacing remains the same, and the values listed in this dialog box change automatically.

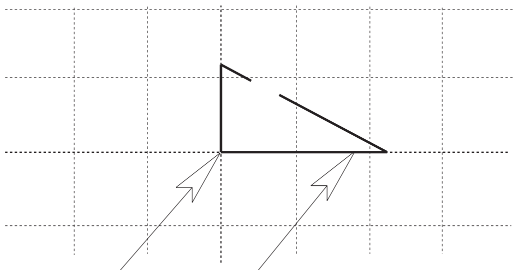
Grid Command

This command, found in ***Layout>Preferences***, allows you to set the spacing of the grid lines. You can specify the number of ticks (subdivisions) per unit (spacing). You can change the default setting by saving changes in the preferences file.

Show Grid - CTRL+G (Windows);⌘+G (Macintosh)

This command in the Layout menu controls the display of the grid. The grid is made up of horizontal and vertical lines of dots. The spacing of the grid lines and the number of tick marks between the intersections are set in the Grid dialog box by choosing ***Layout>Preferences>Grid***. When the grid is visible, the Drafting Assistant snaps to the grid spacing. In other words, if the grid is set to .25 inch spacing, you can't construct an object closer than .25 inch to another object.

The coordinate symbol appears at the origin location (X=0, Y=0) when the grid is visible.



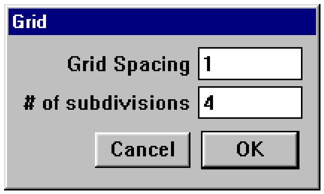
Tech Note:

To change the Hit Radius, choose ***Layout>Preferences>Snap***.

Changing the Grid Spacing and Subdivision

1. Choose ***Layout>Preferences>Grid***.

   The Grid dialog box displays.



2. Enter the desired values for *Grid Spacing* and/or *# of subdivisions*.
3. Click OK and the changes are saved.

   Click Cancel if you don't want to save the changes.

Drawing at Full Scale

Whether you are designing or drafting a highly detailed blueprint, you should create the geometry at its actual size. Graphite allows you to construct the part using full-scale specifications and then set the visual scale of the drawing. In this way, the part dimensions to its true-to-life measurements. Drawing at full scale has the following advantages:

- Scaling mistakes are eliminated.
- Dimensions are automatic (you must dimension manually if you do not draw at full scale).
- Associative dimensions update when the object is edited (manual dimensions do not).
- The size relationship of imported parts is compatible.
- Calculations for 2D analysis are accurate.

Once your project is drawn, you can dimension it, scale it visually and size it to fit into a standard drawing format, if you wish. The actual size of the geometry remains constant unless you edit it.

When you open a new Graphite document, the drawing area is a sheet that is infinitely large so that you can design anything at full size. For a simplistic example, here's how to draw and view a line 83 feet long:

1. Draw a line, specifying **83**' for the length.

   The line extends off the screen.

2. Choose ***Arrange>Zoom All***.

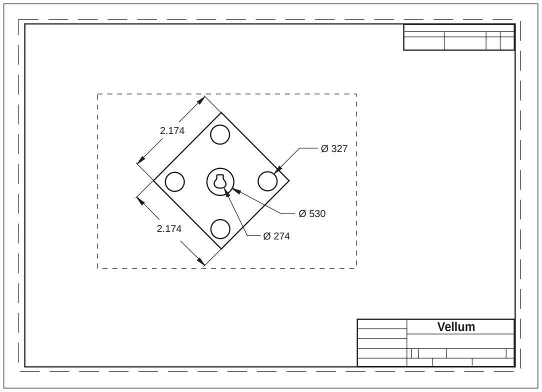
   The entire 83 foot line is visible on the screen.

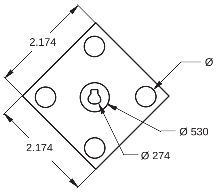
Using the draw to scale/Zoom All method, you can create accurate full-scale drawings which are displayed at the magnification you choose. The actual size of an object is not affected by zoom magnification or reduction.

Zoom All magnifies or reduces all objects on your drawing to fill the screen—regardless of the size of the objects.

The Drafting Process

The drafting process may or may not include using a standard drawing format.





Since you create the geometry at full scale, the basic difference in the two drafting processes is how the visual scale of the geometry is set so that it fits on the paper or in a drawing format. Most steps are discussed in the following two drafting procedures. They are also discussed in greater detail in later chapters.

Drafting without a Drawing Format

1. Start Graphite and display an Untitled document.
2. If necessary, set the Units of measure you want to use by choosing ***Layout>Preferences>Units***.
3. Construct the geometry at full size, saving the construction as you work. Use the Zoom commands and tools to display the part at an appropriate magnification so that you can see what you are doing.
4. Choose ***File>Print Setup*** (Windows) or ***Page Set up*** (Macintosh) command to set the paper size you will use for plotting or printing.
5. Choose ***Layout>Drawing Size*** to set the drafting scale for plotting or printing.

Specify the drafting scale you want to use for your drawing, such as *1:120* or *1":10'* for *1* inch to represent *10* feet. If you are not concerned about scale and you only want the final geometry to fit on the selected paper size, you can specify Fit to adjust the geometry automatically to the paper size.

6. Dimension the part and add text.

   When you specify the visual scale for your drawing with the *Drawing Size* command, Graphite scales text, dimensions, hatches and line patterns inversely with the same scale factor so that all text and dimensions you add after scaling will have exactly the size you specify in the **Size** submenu in the Text menu. That means if you set the text size for text and dimensions to 0.25 inches it will be plotted at a size of 0.25 inches on the paper.

7. Make final adjustments.
8. Choose ***File>Print***.

Drafting with a Drawing Format

1. Start Graphite and display an Untitled document.
2. If necessary, set the Units of measure you want by choosing ***Layout>Preferences>Units***.
3. Construct the geometry at full size, saving the construction as you work. Use the Zoom commands and tools to display the part at an appropriate magnification so that you can see what you are doing.
4. Use the *Sheet Into View* command from the Views menu to set the visual scale so that the part fits into the size of the drawing format you select in the pop-up menu in the Sheet Into View dialog box.
5. Dimension the part and add text.

   When you specify the visual scale for your drawing with the *Sheet Into View* command, Graphite scales text, dimensions, hatches and line patterns inversely with the same scale factor so that all text and dimensions you add after scaling will have exactly the size you specify in the Size submenu of the Text menu. That means, if you set the text size for text and dimensions to 0.25 inches, it plots at a size of 0.25 inches on the paper.

6. Make any final adjustments.
7. Choose ***File>Print*** to print or plot your drawing.

Tech Note:

If you add text and dimensions before scaling the drawing with the *Drawing Size* command, you have to select all text and dimensions with the *Selection Mask* and set the text size again to the value you want, to have it plotted on the paper. See Scaling Text and Dimensions in Chapter 5, "Graphite Documents."

OS Settings

Windows Settings

Graphite.ini

The *Graphite.ini* file is an ASCII text file that can be opened and edited in any text editor, including *NotePad*. This file contains setting for the number of fonts that appear in the Font submenu, the size of the screen when Graphite is launched, and the number of files that appears in the *Recent File List*.

To make changes to the *Graphite.ini* file, open it in any text editor and make the desired changes. Save the file as a text file. The next time Graphite is launched, it will account for the new setting in the *Graphite.ini* file.

MaxFont

This option lets you pick the number of fonts that appear in the **Font** submenu from the Text and Dimension menus. Graphite takes the fonts in alphabetical order, up to the number that you specify in the *Graphite.ini* file. Fonts that do not show up in the menu can be accessed by choosing ***Text>Font>Other***. By default this value is left blank, so Graphite displays all the fonts it can.

Screen

This option lets you set the size of the screen that Graphite opens to. There are two options. *Screen=FULL* opens to a maximized window. *Screen=Standard* opens to the standard size, which is almost maximized but with room for icons at the bottom. If no value is specified, the screen opens to the standard size.

Recent File List

This option determines the number of files that appear in the *Recent File List* from the **File** menu. Supply a positive integer after *Files*= to specify the number of files that are to appear in the File menu. The names and paths of recently opened files are saved in the list that follows the *Files*= line. *File1*= is always the most recent file opened. As new files are opened, the file list gets shifted down, so the old File1 becomes File2, and so on. The default number of files is four (4). The maximum number is eight (8).

Referral:

For more information about using the *Recent File List*, consult the discussion of the *Recent File List* in Chapter 15, "Graphite Documents."

Macintosh Settings

Fonts

All of your system fonts are available when choosing ***Text>Fonts***.

Recent File List

You can specify the number of files that should appear in the *Recent File List* of the File menu. A file named recent.lst can be found in the Scripts folder. This file is an ASCII text file that can be opened and edited in any text editor. The following is a sample of the contents of the *recent.lst* file:

```
/aRecentFiles
4           % Number of recent files remembered (8 maximum)
store
/aRecentFileList [
(MacHD:Graphite 3D 3.0:Samples 3D:YF-22)
(MacHD:Graphite 3D 3.0:Samples 3D:Chair 3D)
(MacHD:Graphite 3D 3.0:Samples 3D:Question Mark)
()
] store
```

Supply a positive integer before *% Number of recent files* to specify the number of files that are to appear in the File menu. The names and paths of recently opened files are saved in the list that follows the */aRecentFileList [* line. The first file in the list—YF-22 in the example above—is always the most recent file opened. As new files are opened, the file list gets shifted down. The default number of files is four (4). The maximum number is eight (8).

Saving Preferences

All files save with the settings set for the file. The characteristics used for new files (the default settings) appear in the preferences file.

Preferences Command

This command in the Layout menu displays a submenu for setting various specifications. Descriptions of the items in the submenu appear individually in this reference section. You can change the default setting by saving changes in the preferences file, *prefs.vc6* (Windows) or *Graphite prefs* (Macintosh).

Snap	This box sets the alignment angles, hit radius and other Drafting Assistant settings. This is different from the *Snap* command explained in Chapter 4.

Grid	This dialog box allows you to specify the spacing of the grid lines. Choose ***Layout>Show/Hide Grid*** to turn the display of the grid on and off.
Units	This dialog box specifies the units and precision for the current Graphite document.
Selection	Use this dialog box to set the color or blinking which shows that an object is selected.
Invert	(Windows only) This option allows you to invert your screen from white to black.
DAssistant	This is a toggle switch for turning the Drafting Assistant on or off. A check mark in front marks the *on* status.
AutoSave	This dialog box allows the user to set a time interval for the program to save files.
Save Preferences	If you have set the characteristics you like, such as units, pen style and dimension format, you can make those settings the default settings. Choose ***Layout>Preferences>Save Preferences.*** You cannot save geometry in this way; you would have to open the preferences file, create the geometry and save the file.
Save Palettes	This allows you to save the status (visible/hidden) and the location of the Dimension, Symbol and-Function palettes.

Saving Preferences Information and Characteristics

The preferences filename is *prefs.vc6* (Windows) or *Graphite prefs* (Macintosh). The file must be stored in the same folder as the Graphite application.

If you want to use different settings for your work, you can change the default settings, and then every new document opens with the settings you have selected. The following specifications can be set in the preferences file:

- Pen styles
- Text characteristics

- Preferences settings (snap, grid, units, selection color indicator, and Visualization settings)
- Grid display
- Layer and sheet specifications
- Work layer
- Dimension and tolerance formats
- Arrowhead type and display
- Drawing size and scale
- Zoom scale
- Fillet radius
- Chamfer angle and length
- Resolve values
- User defined colors

Changing the Default Settings

1. Create a file which has the preferences you want.
2. Choose ***Layout>Preferences>Save Preferences***.

   The preferences are set for subsequent new documents after restarting Graphite.

Be careful choosing *Save Preferences* when working in a file with multiple sheets, models, details views, or layers. Even though your geometry will not be saved, all other data will be. All new files opened subsequently will contain items that may not be desired. On a regular basis, check the Preferences file for the items mentioned. The original default setting for the file includes one sheet, one model, three layers (Construction, Dimension, and Layer 1) and no detail views.

Save Palettes

In the Preferences submenu you find the *Save Palettes* command. When you select this command the Status (visible/hidden) and the Location of all palettes are saved to preferences. When you restart Graphite all palettes that were displayed when you closed Graphite are automatically reopened.

Location and Status for the following palettes are saved with this command:

- Dimension palette
- Symbol palette
- Function palette

Neither the **Floating** tool palette nor the Trackball save with this command.

For Windows, the Location and Status of these palettes are saved in the *Graphite.ini* file, under the section *Palettes*.

Advanced Environment Settings

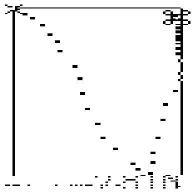
This chapter describes the various advanced options that you have for adapting Graphite to your needs. The following topics are covered:

- Customization
- File Organization
- 3D Viewing

Floating Tool Palette

Graphite's standard tool palette is fixed at the left side of the Graphite window. But you may display an additional **Floating** tool palette which you can drag and place anywhere on the screen.

Displaying the Floating Tool Palette



1. Press and hold the CTRL+SHIFT keys (Windows) or the ⌘ key (Macintosh).
2. Drag the mouse pointer first down and then up.
3. Release the CTRL+SHIFT keys (Windows) or the ⌘ key (Macintosh).

   The **Floating** tool palette displays.

You can close the **Floating** tool palette by double-clicking the Control menu (Windows) or the Close box (Macintosh) in the upper left corner of the **Floating** tool palette.

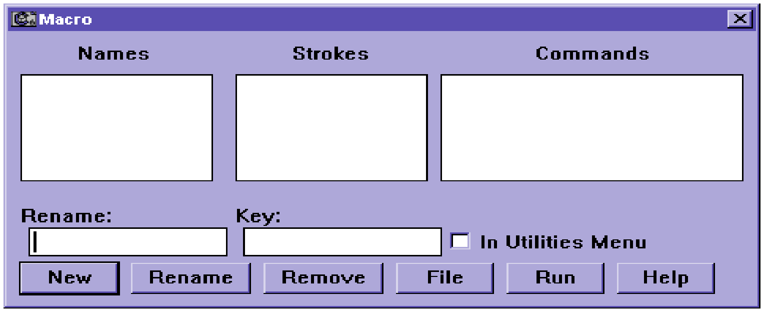
Macros

The Macro capability of Graphite allows you to create macros and access them through menu commands, stroke commands, or key combinations providing an alternative way to invoke Graphite commands.

A macro automates a repetitive task by executing a group of simple commands. If you can point and click, you can define a macro. That's how easy it is!

Macro Command

If you choose ***Utilities>Macro***, the following dialog box displays:



The Macro dialog box is modeless in the sense that there are no *Start Recording* or *Stop Recording* buttons. Whatever action you perform (selecting menu items or defining strokes) records automatically in the relating list boxes as long as the Macro dialog box is open.

The Macro dialog box contains the following buttons:

New — Clicking New adds a macro named *Macro1* to the *Names* list box. You can rename the macro in the Rename data field by overwriting **Macro1** with a new name and then clicking the **Rename** button.

Rename — Clicking Rename overwrites the selected macro name in the *Names* list box with the name you entered in the Rename data field. (For Windows, macro names may only contain alpha-numeric characters. No spaces or other characters are allowed.)

Remove — The Remove button removes any selected entry from one of the three list boxes.

File

The File button writes all currently defined macros to files in the Scripts folder in the Graphite installation folder. The file names are generated directly from the macro names. The files are output in such a way that when Graphite is subsequently started, the macros activate automatically. Macro files can be moved to other locations, or to other machines for use in other copies of Graphite.

Run

The Run button runs the macro currently selected in the *Names* list box. Macros can call other macros if they are in the Utilities menu. This is done by clicking on the relevant entry in the Utilities menu, like any other menu item. This feature allows a macro to call another macro, which in turn could call the original macro. Such Infinite Loops are checked when an attempt is made to include a macro, and if detected, an alert box is posted resulting in the macro not being included.

Names list box

In this list box all names of the macros defined are displayed alphabetically. When one of the items is selected with the cursor, the *Strokes* and the *Commands* list boxes, as well as the *Key* and *In Utilities Menu* fields are automatically filled in with the relevant data that defines the selected macro.

Strokes list box

The *Strokes* list box shows a group of stroke commands, if any, that can run the selected macro. The names in this list box relate to the encoding of the shape of the stroke and you need not be concerned about them. It is optional to associate a stroke to define a macro.

Commands list box

The *Commands* list box shows the set of menu commands that define this macro. They are shown in this form: *MenuName; EntryName*

Key

You can assign your own key combinations for a macro by clicking in the *Key* edit box and entering the key combination or typing it directly in the *Key* entry field. For example, enter CTRL+SHIFT (by pressing the CTRL key first, keeping it depressed and pressing the SHIFT key and then the key **P**). The *Key* edit box displays CTRL+SHIFT P.

Windows users: if you try to create a CTRL key combination and the key combination is already in use, you cannot execute the macro. Macintosh users: if you try to use a ⌘ key combination already in use, Graphite prompts you to assign a different key combination.

In Utilities If you mark this option the macro name will be included as a menu item in the Utilities menu to access the macro directly.

Creating a Macro

1. Choose ***Utilities>Macro***. The Macro dialog box displays.
2. Click New.

   The name *Macro1* displays in the *Rename* field.
3. Overwrite the name if desired. (For Windows, you can only use alpha-numeric characters.)
4. Select menu commands in the order you want the new macro to execute.

   The selected commands are automatically filled in the *Commands* list box.

   If you do not desire a stroke command for your macro, go to step 6.
5. Define a *Stroke* command on the drawing area.

   Depress the CTRL+SHIFT keys (Windows) or the ⌘ key (Macintosh) and drag a symbol on the drawing area you want to be associated with the new macro.

   They can be letters or a figure like a circle. You can not use Graphite's standard stroke commands like *Zoom* or *Construction* strokes, since they would be executed immediately. A letter and number combination describing the executed **Stroke** displays in the *Stroke* list box automatically. When you define a stroke, you must repeat it several times to define all variations of the stroke, so that all possible variations are covered and can be recognized by Graphite.
6. Mark the *In Utilities Menu* option if you want the new macro included in the Utilities menu.
7. Enter a Key short cut in the *Key* entry field.

Click in the *Key* data field and enter the key combination or type it directly in the *Key* data field.

8. Close the Macro dialog box by double-clicking the Control menu (Windows) or the Close box (Macintosh) in the upper left corner of the dialog box.

Editing a Macro

1. Display the Macro dialog box.
2. Select a Macro in the *Names* list box.
3. Rename the Macro in the *Rename* data field and click the **Rename** button.
4. Redefine the Macro by executing menu items and assigning a new stroke equivalent to the macro.
5. Alter the Macro key combination in the *Key* data field.
6. Close the Macro dialog box by double-clicking the Control menu (Windows) or the Close box (Macintosh) in the upper left corner of the dialog box.

Removing a Macro

1. Display the Macro dialog box.
2. Select a Macro in the *Names* list box.
3. Click Remove.

Running a Macro

Each Macro has a unique name and can be run in one of four ways:

1. It can be invoked with a key combination. Subsequent typing of that key combination will cause the associated macro to run.
2. A macro can be invoked by a family of stroke commands you defined in the Macro dialog box.
3. You can select the macro in the Utilities menu if you marked the option *In Utilities Menu* in the Macro dialog box.
4. You can display the Macro dialog box, select a macro in the *Names* list box and click Run.

Executing a macro with the Run button is useful for testing a macro. Should it not work properly, you can highlight any dialog box entry and remove it by pressing the Remove button. Modify your macro as needed then test it again by pressing the Run button. Continue this process until your macro runs properly.

Macro Limitations

Macros are not general programs. There are limitations as to what can be encoded in a macro. The following rules apply to macros.

- Only menu items can be included in a macro.
- A menu item might cause a dialog box to appear, but the macro cannot fill in the required entries.
- There are no conditionals (like *if* or *ifelse*) or loops (like *while*).
- Subroutines are allowed in that one macro can call another macro, provided the called macro is in the Utilities menu and therefore a menu item. If a macro calls a nested macro, which would end in an infinite loop, an alert box appears and the macro aborts.
- Macro names must be limited to alpha-numeric characters with no spaces.

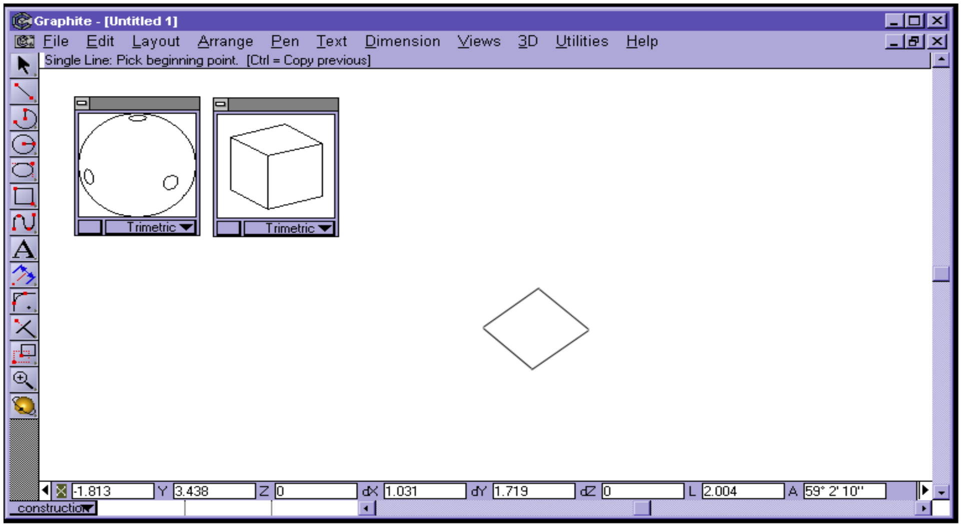
File Organization

Graphite provides you with the ability to organize your files. The *Directory* command in Preferences allows you to specify the file location for opening, saving, importing and exporting. See Chapter 15 for more information.

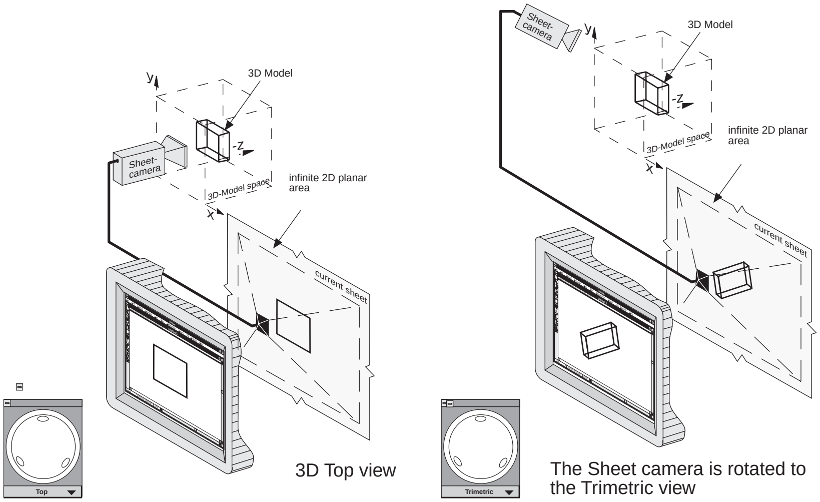
3D Viewing

Although the transition between 2D and 3D drawing in Graphite is seamless, there are a few settings and features that are unique to 3D. When you open a new document an empty work area displays. This work area, a sheet, is an infinitely-large drawing space where you can create a model of any size. The newly-opened sheet displays the Top view by default on the x, y plane of the model space picked up by the sheet camera. Since you are going to construct geometry in 3D, you need to rotate the sheet camera to a 3D view.

You can also start drafting in 2D and then continue drafting in 3D simply by changing the view.



Once you have rotated the sheet camera, it is as though the model rotates so that you can see all the x-, y-, and z-directions while you are drawing.



You have several options for this rotation:

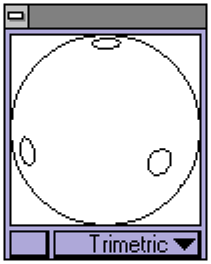
- Manually rotate with the on-screen Trackball.

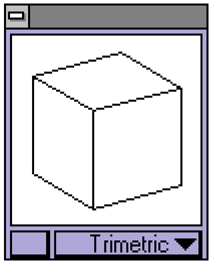
- Select a view from the pop-up menu in the Trackball window.
- Choose a view from the Views submenu in the Views menu.

Show/Hide Trackball/TrackCube

This command toggles the display of the Trackball/TrackCube window. You can use the Trackball/TrackCube to rotate the view orientation in the active detail view window or the sheet view, as long as no view windows have been created. You can drag the Trackball window around the screen. Fore more information on detail views and sheet views, see Chapter 13, “Viewing Geometry.” To rotate between the TrackBall and the TrackCube, click on the box in the lower left corner.

Rotating the Sheet View or Active View





1. Choose ***Views>Show Trackball***.
2. Drag the pointer on the Trackball to rotate the view.

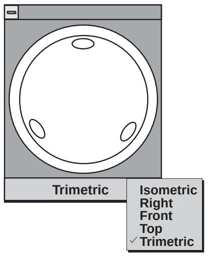
   The model rotates as you drag. The model rotates around the center of the active view or sheet.

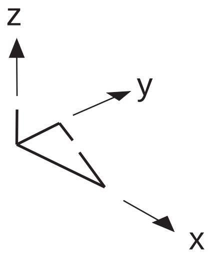
Using the Trackball View Menu

The Trackball/TrackCube has a pull-down menu for specifying the view of the current view window or the sheet view, if there are no view windows.

1. Move the pointer to the current view name displayed at the bottom of the Trackball/TrackCube window.
2. Press the mouse button. The Views menu displays.
3. Choose the view orientation you want to display in the current window.

   The view orientation changes to your specification in the view window.



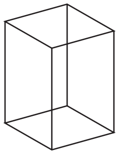


Tip:

If you want to see the Triad at the origin (0,0,0), set the grid spacing to 0 in the Preferences submenu in the Layout menu.

Trackball Locking

This feature enables you to lock the trackball /TrackCube along two screen axes so you can rotate the view of your geometry around the third screen axis. The screen axes are different from the axes of your geometry unless you set your view to top.

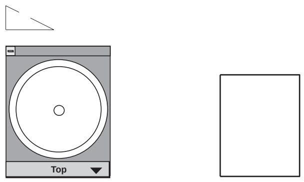


***Example*:**

You can rotate the view of this block around the x-,y- or z-axes of the geometry or the screen.

Rotate View around an Axis of the Geometry

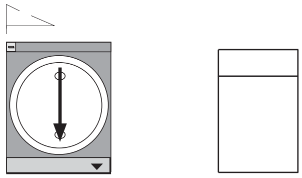
To rotate you view around an axis of your geometry, change the view to Top, either from the Trackball menu or choosing ***Views>Views>Top***. This makes the screen axes the same as the axes of the block.



Rotate View around the X-axis

To rotate around the x-axis, place the cursor on the Trackball, hold down the SHIFT key and drag the cursor up or down. Your geometry rotates around the x-axis.

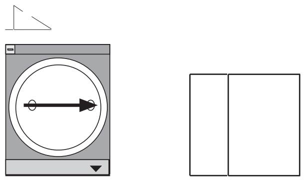
Holding down the SHIFT key locks both the y- and z-axes.



Rotate View around the Y-axis

To rotate around the y-axis, place the cursor on the Trackball, hold down the CTRL key (Windows) or the OPTION key (Macintosh) and drag the cursor to the left or right. Your geometry rotates around the y-axis.

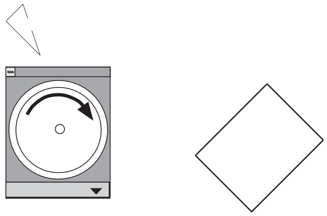
Holding down the CTRL key (Windows) or the OPTION key (Macintosh) locks both the x- and z-axes.



Rotate View around the Z-axis

To rotate around the z-axis, place the cursor on the Trackball, hold down the CTRL+SHIFT keys (Windows) or the ⌘ key (Macintosh) and drag the cursor in a circular motion. Your geometry rotates around the z-axis.

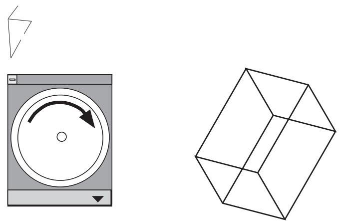
Holding down the CTRL+SHIFT keys (Windows) or the ⌘ key (Macintosh) locks both the x- and y-axes.



Rotate View around the Screen Axis

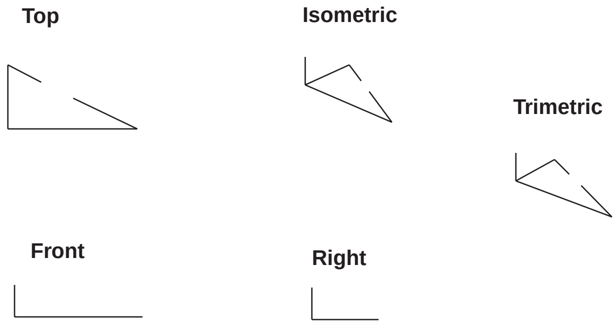
With the same block, displayed in any view, simply use the key combinations listed previously to rotate around the desired axis.

If your geometry is displayed in the trimetric view and you want to rotate it around the screen's z-axis, place the cursor on the Trackball, hold down the CTRL+SHIFT keys (Windows) or the ⌘ key (Macintosh) and drag the cursor in a circular motion. Your geometry rotates around the screen's z-axis.



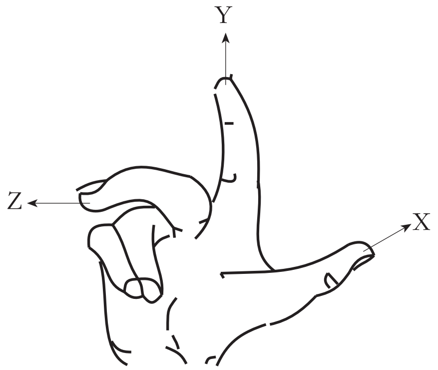
Show/Hide Triad

This command toggles the display of the Triad symbol in the upper-left corner of the view windows. The Triad illustrates the orientation of the x, y, z axis and the work plane. This symbol also temporarily displays at the origin and rotates as you rotate the view orientation manually with the Trackball.



If you have not created a detail view, the Triad symbol displays in the upper-left corner of the sheet. Once a view window is created, the sheet camera cannot be rotated any more; it remains stationary at the world orientation. For more information on detail views and sheet views, see Chapter 13.)

The Triad represents the principle of the right-hand rule—a memory aid for the relative directions of the positive axes. With your right palm upturned, the thumb (x) points right, the index finger (y) points straight ahead, and the middle finger (z) points up. If you move your hand to indicate the x and y-axes, you can easily see the direction of the z-axis.



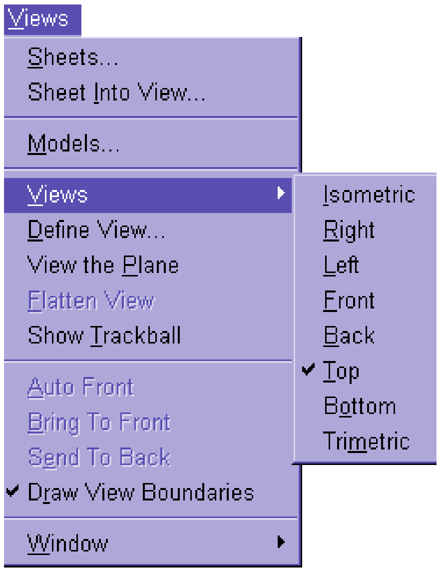
The Right Hand Rule

Views Command

This command displays the view orientation submenu so that you can set the orientation of the sheet view or active view window.

The standard views, Front, Isometric, Right, Top and Trimetric, are always displayed in this menu. You can create new views with the *Define View* command and those views will be listed in this menu. Views can also be selected from the Trackball menu. (See Chapter 14 for more information on *Define View*.)

The selected view orientation affects the active area—the sheet view or the active view window.



Drawing Tools

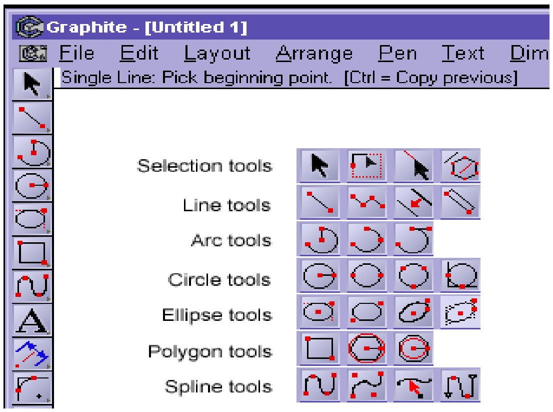
Referral:

For more information about the Drafting Assistant, see Chapter 4.

This chapter describes how to create geometric objects that are the building blocks for your designs through the use of Graphite's drawing tools. It also describes how the Drafting Assistant helps you design faster and easier. The following topics are contained in this chapter:

- Drawing Techniques
- The Status Line
- The Message Line
- Drawing Tools - Description and Use

Drawing begins with the tool palette, located at the left of the Ashlar Graphite drawing area shown below:



The tool palette has seven subpalettes of tools that you can use to create and select geometric objects. These seven tools are the drawing tools.

Drawing Techniques

Most drawing tools allow two methods for creating an object: clicking and dragging.

Clicking This method allows you to create objects by clicking points with the mouse. The benefit of this technique is that you can perform unrelated activities, such as zooming, while you are creating the geometry.

Dragging The drag method allows you to indicate two points by dragging from one to the other. Press and hold down the mouse button to set the first point. Next, drag the mouse to the location for the next point and release the mouse button. The benefit of this technique is that a rubberband image appears during the construction to show you how the object will appear once you release the mouse button.

You can use either or both methods to construct most objects.

To make object creation easier, Graphite's tools contain a feature we call Smart Pointers. As you work with a tool, the tool icons show you the points you must indicate to create an object with the tool.

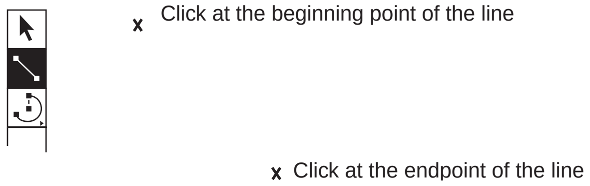
Each dot on the icon (the **Single Line** tool icon is shown here) represents a point you must place either by clicking or by dragging. The smart pointer indicates the order for designating points as you are drawing the geometry.

Creating a Line by Clicking

1. Select the **Single Line** tool from the tool palette.

2. Move the pointer to the drawing area and click to set the starting point of the line.

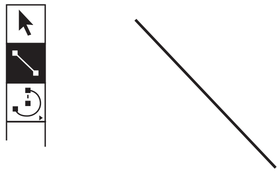
3. Move the pointer to the desired location for the endpoint of the line and click.



Tech Note:

Notice that no rubberband line guides you when you click to draw a line.

The line is drawn between the two points.



Immediately after construction, you can make changes in the Status Line at the bottom of your drawing area to alter the length or position of the line. If the geometry isn't satisfactory, just press the BACKSPACE (Windows) or the DELETE (Macintosh) key.

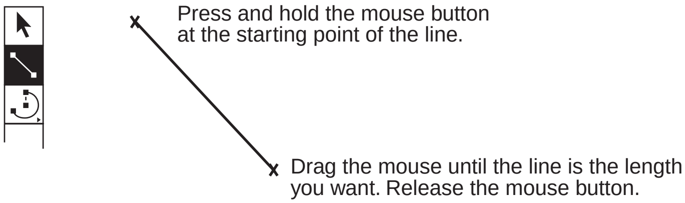
Creating a Line by Dragging

Tip:

Notice that when you drag between points, a rubber-band line guides you. If you use the clicking method, you can perform unrelated activities, such as zooming, between clicks.

1. Click the **Single Line** tool in the tool palette.

2. Move the pointer into the drawing area.
3. Press and hold down the mouse button at the desired location for the starting point of the line.
4. With the mouse button still held down, move the mouse to the desired location for the endpoint of the line.



5. Release the mouse button to set the endpoint.

   Immediately after construction, you can make changes in the Status Line at the bottom of the drawing area to alter the length or position of the line. If the geometry is not satisfactory, just press the BACKSPACE (Windows) or the DELETE (Macintosh) key.

The Status Line

Whenever you select a tool from the tool palette, the Status Line appears along the bottom of the drawing area. It contains fields that provide information about the current construction. For example, when the **Single Line** tool or **Connected Lines** tool is the current tool, the Status Line contains fields for the X and Y coordinates of the beginning point and the change of X and Y values for the endpoint of the line. It also contains the value of the length and angle of the line.

You can use the Status Line in three ways:

- To create an object with keyboard entries only.
- To edit an object that was just created and is still selected.
- To create an additional object using the current tool.

Whenever you construct an object, the status field containing the specification that you are most likely to change is selected (the field is highlighted). For example, when you draw a line with the **Single Line** tool, the Length data field is highlighted so you can enter a value for the length. Just type a value. When you press ENTER (Windows) or RETURN (Macintosh), the line is redrawn at the new length.

You can set the number of decimal places for the Status data field entries or fractional round off by going to ***Layout>Preferences>Units***. In the Units dialog box you set the precision.

Tip:

Notice how each method affects the selection of the contents of the status field. Clicking once inside the field places the cursor where you clicked. If you double-click inside the field, the entire entry is selected so you can alter it.

Using the Status Line Fields

If you want to make an entry in a different status field, you can use one of four selection methods:

- Press the Tab key to cycle the selection highlight through the status fields from left to right.
- Click inside the status field and the pointer becomes an *I-beam* text cursor.

- Double-click inside the status field to select the entire contents of the field.
- Click the field label to select the entire contents of the field.

Using the Status Line with Drawing Tools

Once you have drawn an object, you can adjust the specifications —the length, angle and location—in the Status Line. You must make these changes immediately, before you construct another object, select a different tool, or choose a command.

You can make only one series of changes in the Status Line (in as many fields as necessary); after you press ENTER (Windows) or RETURN (Macintosh) Graphite redraws the object to your specifications. Once you have pressed ENTER (Windows) or RETURN (Macintosh), you must make subsequent changes with the *Edit Objects* command in the Edit menu.

Note: When working in English units and you want to enter values in the Status Line that combine feet and inches, enter them as **x'y"** and not **x'-y"**. The dash reads as a minus sign and **x'-y"** is calculated as an equation. See Appendix A for operators and units.

Try the following exercises to create and change a single line with the Status Line.

Altering Geometry in Progress with Status Fields

1. Select the **Single Line** tool.
2. Click two locations in the drawing area. The length (**L**) field automatically highlights in the Status Line.
3. Type **3**. The 3 is entered directly in the L field.
4. Press the Tab key to select the next status field. The angle (**A**) field now highlights.
5. Enter **15**.
6. Press ENTER (Windows) or RETURN (Macintosh).

   Pressing ENTER (Windows) or RETURN (Macintosh) completes the data entry for this object. The line redraws 3 units long and at a 15° angle.

You must remember that when you press ENTER (Windows) or RETURN (Macintosh), Graphite constructs the object based on the specifications in the status fields. Pressing ENTER (Windows) or RETURN (Macintosh) a second time in the above example would create a second line with the same values. Since the lines have the same values, the second line overlays the first line and you won't be able to see it.

Creating Additional Geometry with Status Fields

1. With the **Single Line** tool still selected from the previous example, click 2 more points.
2. Type **4**.
3. Press the Tab key and type **25**.
4. Press ENTER (Windows) or RETURN (Macintosh). Another line is drawn.

Creating New Geometry with Status Fields

1. Choose the **Selection** tool so the Single Line status field clears.
2. Click the **Single Line** tool again.

   The X status field is active, showing that you can enter a value for the X coordinate of the beginning point of the line.
3. Type **0**.

   Note: Do not press ENTER (Windows) or RETURN (Macintosh) until you reach step #10.
4. Press the Tab key. The Y field highlights.
5. Type **0**.
6. Press the Tab key.

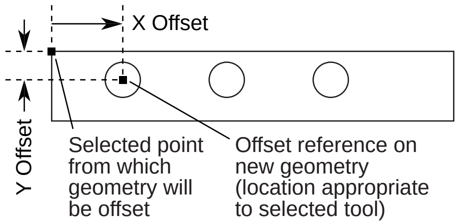
   The dX field highlights. The dX value is the delta X, the numeric difference between the beginning and ending X coordinate.
7. Type **2**.
8. Press the Tab key. The dY field highlights.
9. Type **2**.
10. Press ENTER (Windows) or RETURN (Macintosh). The line is drawn.

Creating Geometry Offset from a Point

If you want to create some geometry that is offset from an existing point, you can use the Status Line to specify the offset.

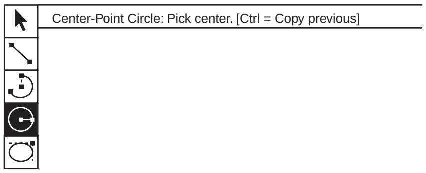
1. Select the tool you want to use.
2. Move the pointer over the control point from which you want the offset.
3. Click once to lock onto that point.

4. Click in the appropriate X or Y field in the Status Line, placing the text cursor at the end of the entry.
5. Type the offset (such as **+ 3"**) and press ENTER (Windows) or RETURN (Macintosh).
6. Continue with your construction.



The Message Line

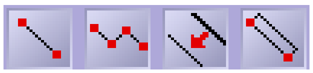
The Message Line is an important feature when drawing. Upon selection of a drawing tool, the line displays what tool you have selected and the first step in its use. As you follow each step, the line displays the next one until the steps are completed. The Message Line may also display additional commands that can be used with the tool. See the example below.



Drawing Tools - Description and Use

This section describes the drawing tools found in the tool palette.

Line Tools



The Line tools in the **Line** tool palette create line segments, connected lines, lines parallel to existing lines and smart walls. As you create a line, the coordinate locations, line length and angle from horizontal appear in the Status Line. The line is also drawn with the current pen specifications for color, weight and pattern.

Single Line Tool

This tool, in the second subpalette of the main tool palette, draws a line between two points. You can click or drag to draw a line.

Using the Single Line Tool

1. Select the tool. The Message Line reads, *Single Line: Pick the beginning point. [Ctrl* = *Copy previous* (Windows) or *Option* = *Copy previous* (Macintosh)*]*.
2. Click two endpoints of the line,

   or

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

To copy the line just created, as the Message Line indicates, hold down the CTRL (Windows) or OPTION (Macintosh) key and click once in the drawing area, a line appears that is identical to the one just drawn. The location of the click designates the location of the first endpoint.

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, the line Length is the selected status field.

Drawing a Line Perpendicular to Another Object

1. Construct the object.
2. Move the pointer to the object until a Drafting Assistant ***on*** notation appears (*endpoint*, *midpoint*, *quadrant,* etc., will not work, only "*on*").
3. Drag straight away from the object in a perpendicular direction. A *perpendicular* line appears attached to the object. Keep holding the mouse down and move the cursor. Notice that the line stays perpendicular but slides along the object.
4. Drag to the desired length for the perpendicular 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*, *quadrant*, or *midpoint*.
4. Drag in the appropriate direction (straight for perpendicular or at an angle for tangent) until the Drafting Assistant *perpendicular* or *tangent* 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.

Creating a Point

You can create a point by creating a line and setting its length to zero. Set the starting point of the line with the mouse, then type **0** in the Length entry field of the Status Line and press ENTER (Windows) or RETURN (Macintosh). The line ("point") displays a + to indicate its location. You can also click the **Single Line** tool twice in the same spot.

Connected Lines Tool

This tool, in the second subpalette of the main tool palette, draws lines in which the endpoint of one line segment is the beginning point of the next.

Using the Connected Lines Tool

1. Select the tool. The Message Line reads, *Connected Lines: Pick the beginning point*.
2. Click or drag to indicate the endpoints of the line segments.

   If you click a point and then change your mind, press the ESC key, or choose *Undo* to remove the last line. Pressing the BACKSPACE (Windows) or DELETE (Macintosh) key removes all connected lines in the current construction.

   The Message Line changes after the beginning point is chosen, asking you to pick the endpoint and also reminding you to double-click when you want to complete the series of segments.
3. Indicate the last point by double-clicking or by choosing another tool.

After completing at least one segment with the **Connected Lines** tool, you can create a tangent arc off of the last line by holding down the CTRL (Windows) or the OPTION (Macintosh) key (the pointer temporarily changes to an "arc" icon) and clicking or dragging to the next point. The Message Line notes this added feature. Several tangent arcs can be strung together by continuing to hold down the CTRL (Windows) or OPTION (Macintosh) key. (Note: The radius of these arcs cannot be edited with this method.)

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 selected status field. By entering successive sets of data and hitting ENTER (Windows) or RETURN (Macintosh) after each set, a continuous string of connected lines can be entered from the keyboard.

Parallel Lines Tool

This tool, in the second subpalette of the main tool palette, constructs lines parallel to existing lines. Use one of the following methods for creating parallel lines:

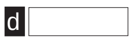
Using the Parallel Lines Tool

1. Select the tool. The Message Line reads, *Parallel Lines: Drag new line off existing line.*

2. Select the **Parallel Lines** tool, drag off the desired line and release to indicate the position for the new parallel line. A rubberband line moves with the pointer. The Status Line shows the distance you dragged. If you type in your own value and hit ENTER (Windows) or RETURN (Macintosh), the parallel copy moves to the offset defined by your value.

   or

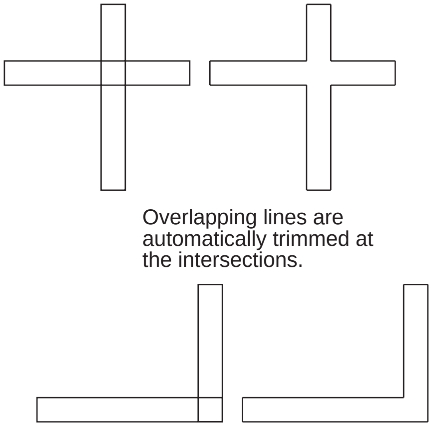
   Select the **Parallel Lines** tool, click the line you want to duplicate, and then enter a delta distance in the Status Line and then press ENTER (Windows) or RETURN (Macintosh). (This method does not allow you to control on which side of the original line the new line is drawn.)

The status field allows you to specify the distance between the original line and the new parallel line.

Smart Wall Tool

The **Smart Wall** tool, in the second subpalette of the main tool palette, constructs a double line like those used in architectural drawings. Please note: this tool will function correctly only in the **Top** view. Results will not be accurate if you draw in any other view.



Tip:

If you want to use the Smart Wall feature for single lines, you have to create the lines with the **Smart Wall** tool and enter **0** for the thickness of the wall.

This tool works in much the same way as the **Single Line** tool except that it draws double lines or walls. Walls created on the same layer automatically trim to where they touch or intersect. Automatic trimming of smart walls occurs only when the walls are on the same layer.

As you drag the mouse, the defining side of the wall falls on the construction line you drag along. The other side of the wall automatically fills in at the thickness you specify in the Status Line. The position of the automatic wall depends on your last activity. If the second wall is on the wrong side of the construction line, press the SHIFT key to flip it. Notice the Message Line reads "Shift = Flip" as a reminder of this feature.

If you want to fillet or chamfer or otherwise alter a Smart Wall layout, you have to ungroup them with the *Ungroup* command in the Arrange menu. The result is that the selected Smart Walls turn into separate line segments and the "smarts" are stripped out. This cannot be undone. Regrouping the pieces will not make them Smart Walls again. It is recommended that you place all symbols, before you ungroup and fillet/chamfer the walls.

Using the Smart Wall Tool

1. Select the tool. The Message Line reads, *Double Line: Pick start point. [Shift = Flip].*
2. In the Status Line, specify the thickness (T) of the wall.
3. Click or drag to indicate the ends of the wall segment.

You can press the SHIFT key to flip the wall to the opposite side.

4. Continue specifying both endpoints for wall segments, as needed.

If you delete a merged segment, the remaining walls redraw.

The Status Line allows you to specify thickness of the wall, the X,Y coordinates of the beginning point, the wall length and the angle from horizontal. Once the wall segment is created, wall Length is the selected status field.

Wall Symbols

Parametric symbols (like door and window symbols) can be constructed in a way that automatically cuts an opening in the Smart Wall they get placed on. Examples of this are included with Graphite in the Architect library within the Symbols folder. When you move the symbol along the wall to another position the wall automatically closes at the old position and the opening appears at the new position.

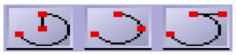
1. The symbol must contain a Smart Wall piece that is thicker than the wall segment it will be placed on.

2. Set the pen pattern of the smart wall piece to *dotted*. (Any geometry in a symbol that is of the dotted pen pattern will be placed on the drawing, but will be invisible.) This results in an invisible Smart Wall piece that cuts the Smart Walls on the drawing.

Parametrics works with Smart Wall configurations, but it can be tricky. If you use parametrics, turn on the point display (***Layout>Show Points***) and hook dimension objects right to the displayed points rather than to the lines. Keep in mind that double walls obscure the fact that smart walls are actually single lines.

For more information on parametrics, see Chapter 16.

Arc Tools



You can use the **Arc** tools to create an arc by any of three methods:

- Center-Point
- 3-Point
- Tangent Point

As you create each arc, the Status Line displays entries such as coordinate locations, radius, angle from horizontal and delta angle.

The arc is drawn with the current pen specifications for color, weight and pattern.

Center-Point Arc Tool

This tool, in the third subpalette of the main tool palette, draws an arc from three points: the center point, arc beginning point and arc endpoint.

Using the Center-Point Arc Tool

1. Select the tool. The Message Line reads, *Center-Point Arc: Pick center. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*]*.
2. Click the *center* point of the arc.

   The Message Line now tells you the next step in using the tool.
3. Click the radius and beginning of the arc.
4. Drag the extent of the arc, pressing at the beginning point of the arc and releasing at the endpoint.

You can simply click all three points, but the rubberhand arc won't appear with this method. To construct an arc greater than 180°, 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 selected status field.

X Y R A dA

3-Point Arc Tool

This tool, in the third subpalette of the main tool palette, draws an arc through the points you select.

Using the 3-Point Arc Tool

1. Select the tool. The Message Line reads, *3-Point Arc: Pick first point. [Ctrl = Tangent to object* (Windows) or *Option = Tangent to object* (Macintosh)*]*.
2. Click the first endpoint of the arc. Notice that the Message Line tells you the next step for using the tool.

3. Drag or click the second and third positions. (Dragging displays the rubberband arc.)

   The arc is drawn from the first position that you indicate, through the second position, and ends at the third position.

If you click *on* a separate object while holding down the CTRL (Windows) or OPTION (Macintosh) key, the arc is drawn tangent to the specified object rather than through the exact click-point. For each of the three clicks that define the arc, if the CTRL (Windows) or the OPTION (Macintosh) key is held down and the cursor is placed on some other object, Graphite will define the arc to be *tangent* to that object at the nearest tangency point. If the CTRL (Windows) or OPTION (Macintosh) key is not held down, then the arc will be created directly through that exact point. So, each of the three points can act separately.

The Status Line indicates the X,Y coordinates for each of the three points.

Tangent-Point Arc Tool

This tool, in the third subpalette of the main tool palette, 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.

Using the Tangent-Point Arc Tool

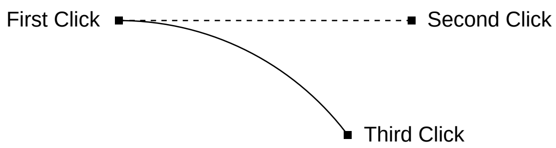
1. Select the tool. The Message Line reads, *Tangent-Point Arc: Pick beginning point of arc (tangent line).*

2. Click the starting point. This is both the starting point of the arc and the starting point of the temporary tangent line.

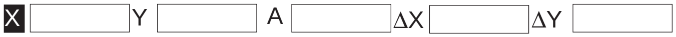
   The Message Line displays the next step for using the tool.

3. Click the endpoint of the tangent line.

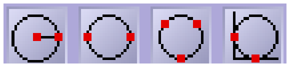
4. Click the endpoint of the arc. The arc is drawn between the first and last point you click and tangent to the line between the first and second points.



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



The **Circle** tools on the tool palette construct circles by any of four methods:

- Center-Point
- Opposite-Point
- 3-Point
- Tangent

Tech Note:

A **Circle Center Line** tool is available for creating axis dimensions. For more information, see Chapter 12.

Center-Point uses the center and diameter of the circle. Opposite-Point uses the diameter. 3-Point uses three points and can be tangent to existing objects. Tangent-Point draws a circle tangent to two objects, using the diameter specified by the user. The circle is drawn with the current pen specifications for Color, Weight and Pattern.

Center-Point Circle Tool

This tool, in the fourth subpalette of the main tool palette, draws a circle specified by the center point and diameter.

Using the Center-Point Circle Tool

1. Select the tool. The Message Line reads, *Center-Point Circle: Pick center. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*].*
2. Click two locations; the first click places the center and the second determines the radius, or Drag, pressing at the center and releasing at a position indicating the radius. If you drag the points, a rubberband image guides your con-

struction. You can create a copy of the last circle by holding down the CTRL (Windows) or OPTION (Macintosh) key and clicking once where you want the center for the new circle.

The Status Line shows the X,Y coordinates of the center and the diameter of the circle. Diameter is the selected status field.

X 1 Y 1 D 1

Opposite-Point Circle Tool

This tool, in the fourth subpalette of the main tool palette, draws a circle specified by the diameter.

Using the Opposite-Point Circle Tool

1. Select the tool. The Message Line reads, *Opposite-Point Circle: Pick first point on circle. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*]*.

2. Click two locations to indicate the diameter,

   or

   Drag, pressing the mouse button at the beginning and releasing it at a position that indicates the diameter. If you click the points, after the first point is chosen, the Message Line will read, *Pick opposite point on circle*. If you drag the points, a rubberband image guides your construction.

Create a copy of the last circle drawn by holding down the CTRL (Windows) or the OPTION (Macintosh) key and clicking once to place first point of the new circle.

The Status Line allows you to specify the X,Y coordinates representing the endpoints of the diameter.

X 0 Y 0 dX 1 dY 1

3-Point Circle Tool

This tool, in the fourth subpalette of the main tool palette, draws a circle through the points you select.

Using the 3-Point Circle Tool

1. Select the tool. The Message Line reads, *3-Point Circle: Pick first point. [Ctrl = Tangent to object* (Windows) or *Option = Tangent to object* (Macintosh)*]*.

2. Click the first point on the circle.

   Notice that the Message Line displays the next step for using the tool.

3. Drag or click the second and third points. (Dragging displays the rubberband circle.)

If you place any of the three points on an existing object, the circle is drawn through that point. If you click an object while holding down the CTRL (Windows) or OPTION (Macintosh) 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.

If you drag the first two points, a rubberband image guides your construction.

The Status Line indicates the X,Y coordinates for each of the three points.

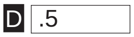
Tangent-Point Circle Tool

This tool, in the fourth subpalette of the main tool palette, draws a circle tangent to the two objects you indicate.

Using the Tangent Circle Tool

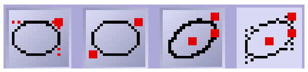
1. Select the tool. The Message Line reads, *Tangent Circle: Enter diameter then pick first tangent object.*

2. You can enter a diameter for the circle in the status field.

3. Click the objects to which the circle is to be tangent.

Ellipse Tools



The **Ellipse** tools on the tool palette construct ellipses inscribed within an invisible rectangle or parallelogram by any of four methods:

- 2-Point Center
- Opposite-Corner

- 3-Point Center
- 3-Corner

The 2-Point Center Ellipse uses the center point and one corner location of the rectangle. The Opposite-Corner Ellipse uses opposite corners of a rectangle. The 3-Point Center Ellipse uses the center point, the midpoint of a side and the corner of the parallelogram. The 3-Corner Ellipse uses three corners on a parallelogram. You can click or drag to create the ellipse. Dragging creates a rubberband ellipse, so you can see the ellipse before it is drawn.

The ellipse is drawn with the current pen specifications for Color, Weight and Pattern.

Ellipses do not have *center* points by design. If you draw lines between opposite *vertex* points on the ellipse, the *intersection* of the two lines will be the center of the ellipse. For a 2-Point Center Ellipse (inscribed in a rectangle) the Drafting Assistant's horizontal and vertical temporary construction lines will cross at the center point once you have touched two *vertex* points.

2-Point Center Ellipse Tool

This tool, in the fifth subpalette of the main tool palette, constructs an ellipse inscribed within the rectangle calculated from the two specified points: the center point and one corner of the rectangle.

Using the 2-Point Center Ellipse Tool

1. Select the tool. The Message Line reads, *2-Point Center Ellipse: Pick center of ellipse. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*]*.
2. Indicate the center of the ellipse.

   The Message Line has now changed, telling you to, *pick corner of control rectangle.*
3. Indicate the corner of the rectangle defining the ellipse.

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

You can create a copy of the last ellipse by holding down the CTRL (Windows) or the OPTION (Macintosh) key and clicking where you want the center.

The Status Line allows you to specify the X,Y coordinates of the center point and the length and angle of the semi-major and semi-minor axes of the ellipse.

X		Y		L1		A1		L2		A2	90°

Opposite-Corner Ellipse Tool

This tool, in the fifth subpalette of the main tool palette, draws an ellipse inscribed in the rectangle specified by the opposite corners you indicate.

Using the Opposite-Corner Ellipse Tool

1. Select the tool. The Message Line reads, *Opposite-Corner Ellipse: Pick first corner of rectangle. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*]*.
2. Indicate one corner of the rectangle defining the ellipse.

   The Message Line now reads, *Opposite-Corner Ellipse: Pick opposite corner of control rectangle*.
3. Indicate the opposite corner of the defining rectangle.

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

You can create a copy of the last ellipse by holding down the CTRL (Windows) or the OPTION (Macintosh) key and clicking where you want the lower-left point.

The Status Line allows you to specify the X,Y coordinates of the lower-left point and the length and angle of the major and minor axes of the ellipse.

X		Y		L1		A1		L2		A2	90°

3-Point Center Ellipse Tool

This tool, in the fifth subpalette of the main tool palette, 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.

Using the 3-Point Center Ellipse Tool

1. Select the tool. The Message Line reads, *3-Point Center Ellipse: Pick center of the ellipse. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*]*. The Message Line will guide you through each successive step.

2. Indicate the center of the ellipse.
3. Indicate the midpoint of the side of the rectangle defining the ellipse.
4. Indicate the corner of the rectangle defining the ellipse.

If the three points are on the vertical or horizontal axis, a straight line is drawn. You can create a copy of the last ellipse by holding down the CTRL (Windows) or the OPTION (Macintosh) key and clicking where you want the center.

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.

X Y L1 A1 L2 A2 90°

3-Corner Ellipse Tool

This tool, in the fifth subpalette of the main tool palette, draws an ellipse inscribed in the parallelogram calculated from the three corners you specify.

Using the 3-Corner Ellipse Tool

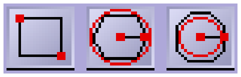
1. Select the tool. The Message Line reads, *3-Corner Ellipse: Pick first corner of control parallelogram. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*]*. The Message Line will guide you through each successive step.
2. Indicate one corner of the parallelogram defining the ellipse.
3. Indicate another corner of the defining parallelogram.
4. Indicate the final corner of the defining parallelogram.

If the three points are on the vertical or horizontal axis, a straight line is drawn. You can create a copy of the last ellipse by holding down the CTRL (Windows) or the OPTION (Macintosh) key and clicking where you want the lower-left corner of the parallelogram.

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.

X Y L1 A1 L2 A2 90°

Polygon Tools



These tools on the tool palette draw rectangles, inscribed polygons or circumscribed polygons. They create lines which are individual objects. Dragging to indicate the points of the polygon creates a rubberband image.

If you want to treat a polygon as a single object, select all lines of the polygon and choose the *Group* command in the Arrange menu.

The polygon is drawn with the current pen specifications for Color, Weight and Pattern.

Rectangle Tool

This tool, in the sixth subpalette of the main tool palette, draws a horizontal or vertical rectangle, using the opposite corners you specify.

Using the Rectangle Tool

1. Select the tool. The Message Line reads, *Rectangle: Pick first corner of rectangle. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*]*.
2. Click the opposite corners of the rectangle,

   or

   Drag the opposite corners of the rectangle.

You can create a square by aligning the second point on the 45° construction line. If the two points are on the vertical or horizontal axis, a straight line is drawn. If you want to draw a square from the center rather than opposite corners, use one of the other polygon tools, specifying four sides. You can't create a rectangle from the center.

You cannot change the Width and the Height of a rectangle in Edit Objects dialog box. You must edit each line individually, because rectangles become four line objects after you deselect the rectangle.

You can create a copy of the last rectangle by holding down the CTRL (Windows) or the OPTION (Macintosh) key and clicking where you want the upper-left corner.

A default 1-inch rectangle is drawn if you have not previously drawn a rectangle.

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 selected status field.

Inscribed Polygon Tool

This tool, in the sixth subpalette of the main tool palette, draws a regular polygon where the radius of the circumscribing circle determines the location of the polygon's vertices.

The default polygon is a hexagon, but you can specify the number of sides in the Status Line. The Status Line shows a diameter for the circle, the standard way of describing a polygon inscribed in a circle.

Using the Inscribed Polygon Tool

1. Select the tool. The Message Line reads, *Inscribed Polygon: Pick center of polygon. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*].*

2. Two options: Click the center of the polygon. (The Message Line reads, *Inscribed Polygon: Pick vertex of polygon.*) Then pick a point on the circumference of the circumscribing circle, or

   Drag from the center of the polygon to a point on the circumference of the circumscribing circle.

You can create a copy of the last inscribed polygon by holding down the CTRL (Windows) or OPTION (Macintosh) key and clicking where you want the center.

The Status Line allows you to 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 Tool

This tool, in the sixth subpalette of the main tool palette, 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. (Note: The Status Line shows a diameter for the circle, the standard way of describing a polygon circumscribed around a circle.)

Using the Circumscribed Polygon Tool

1. Select the tool. The Message Line reads, *Circumscribed Polygon: Pick center of polygon. [Ctrl = Copy previous* (Windows) or *Option = Copy previous* (Macintosh)*]*.

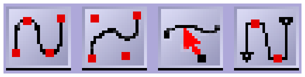
2. Click the center of the polygon and the midpoint of one of the sides (as directed by the Message Line) or

   Drag from the *center* of the polygon to the *midpoint* of one of the sides.

You can create a copy of the last circumscribed polygon by holding down the CTRL (Windows) or OPTION (Macintosh) key and clicking where you want the center.

The Status Line allows you to specify the X,Y coordinates of the center, the diameter of the circle defining the polygon, and the number of sides. The default number of sides is six. Diameter is the default Status Line selection.

Spline Tools



The spline tools on the tool palette 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 when the curve is changed. Kinks don't develop as the spline is altered. Second, NURB splines provide localized control of a complex curve. You can isolate an area and make changes without affecting the remainder of the spline.

These properties are essential in aerodynamic designs. Air molecules moving over a wing surface must flow smoothly for maximum aero-dynamic lift. If the surface does not maintain curvature continuity, the air molecules separate from the wing surface and cause a vacuum. Such a vacuum causes an eddy as the molecules try to fill it. This disruption of air flow increases the drag, which is not a part of an effective design.

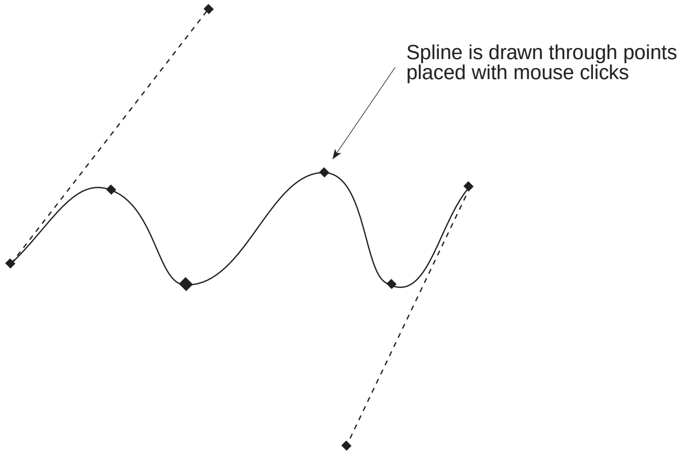
The automotive industry needs smooth air flow to improve gas mileage. Complete curvature continuity also improves styling. The appearance of a car is one of the major sales factors. The potential buyer would not be impressed if the showroom lights' reflection on the car rippled and wavered. It is complete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

Localized control of complex curves allows you to make minor modifications without adversely affecting the shape. For example, if a new, bigger engine wouldn't fit under a perfectly-designed hood, you could use a NURB spline to raise the center of the hood without changing the 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 Tool

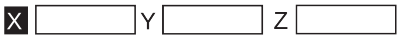
The **Through-Points Spline** tool, in the seventh subpalette of the main tool palette, draws a spline through the points that you place.



Using the Through-Points Spline Tool

1. Select the tool. The Message Line reads, *Through-Points Spline: Pick control point (Double-click last point)*.
2. Click the points for the vectors of the spline.
3. Double-click the last point.

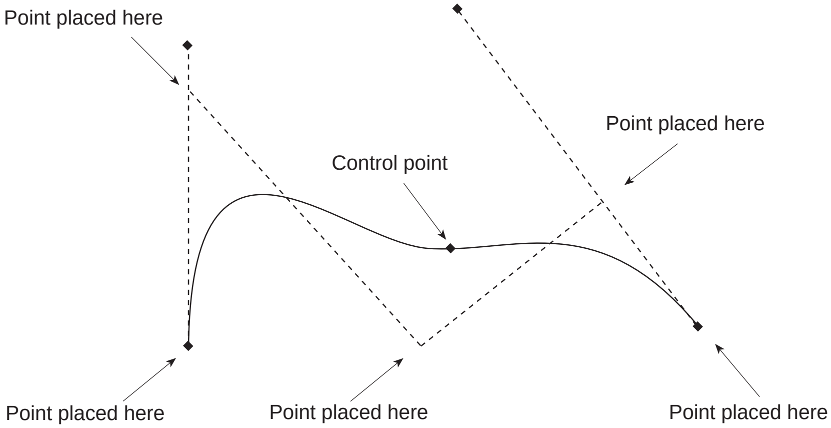
The Status Line shows the X,Y, and Z coordinates of each point as you go along. (Graphite shows only X and Y coordinates.)



If you create a spline that crosses over itself, the Drafting Assistant will not be able to find that intersection. This is by design.

Vector Spline Tool

This tool, in the seventh subpalette of the main tool palette, draws a spline using vectors determined by the points you specify. The **Vector Spline** tool uses each point that you place as the vertex of a vector for the spline it creates.



Using the Vector Spline Tool

1. Select the tool. The Message Line reads, *Vector Spline: Pick control point (Double-click last point).*
2. Click the points for the vectors of the spline.
3. Double-click the last point.

Graphite 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.

The Status Line shows the X,Y, and Z coordinates of each point as you go along.

Add Spline Control Point Tool

This tool, in the seventh subpalette of the main tool palette, allows you to add another control point to an existing spline.

Using the Add Spline Control Point Tool

1. Select the tool. The Message Line reads, *Add Spline Control Point: Pick location on spline.*
2. Click on the spline at the desired location(s) for the new control point(s).

Note: To see the new control point(s), as well as the existing control points, you must first select the spline and then choose ***Layout>Show Points*** (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 you don't change the slope of the rest of the spline.

Lock Spline Control Point Tool

This tool, in the seventh subpalette of the main tool palette, allows you to lock specified points on an existing spline so you can change the slope between the points without affecting the rest of the spline.

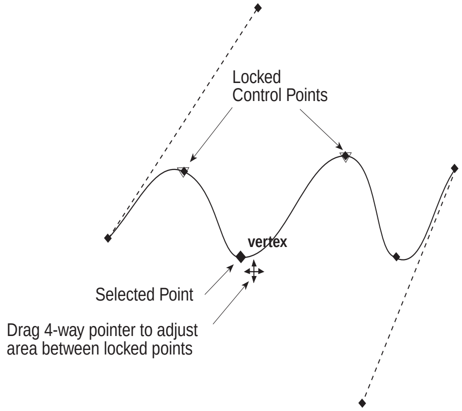
Tech Note:

The *Lock* command in the Arrange menu is used to prevent changes to objects and doesn't affect spline control points.

Using the Lock Spline Control Point Tool

1. Select the spline.
2. Choose ***Layout>Show Points***.
3. Select the tool. The Message Line reads, *Lock Spline Control Point: Pick control point.*
4. Click the *vertices* you want to lock.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.



Unlocking Spline Control Points

Use this tool and click the pointer to unlock it.

Locking More than One Spline Control Point

You can lock more than one point by simply clicking each point.

Editing Control Points of a Spline

1. Select the spline with the **Selection** tool.
2. Choose ***Layout>Show Points***. (If the menu says *Hide Points*, you don't need to choose it.)
3. Click on the drawing area to deselect the spline.
4. If necessary, select the **Lock Spline Control Points** tool from the **Spline** tool subpalette.
5. If necessary, click 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 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 of a Spline

1. Select the spline with the **Selection** tool.
2. Choose ***Layout>Show Points***. (If the menu says *Hide Points*, you don't need to choose it.)
3. Click in the drawing area to deselect the spline.
4. If necessary, select the **Lock Spline Control Points** tool from the **Spline** tool subpalette.
5. If necessary, 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.

You cannot move both *endpoints* of a spline at the same time. You must select and move the *endpoints* individually.

3D Drawing Tools

The process of moving from 2D to 3D geometry creation is relatively seamless. All of the tools you use to create 2D geometry in Graphite are also used to create 3D geometry. However, there are some tools, features and concepts that are specific to 3D. The following topics are covered in this chapter:

- Geometry in 3D
- 3D Features & Tools
- Work Planes

Geometry in 3D

The Graphite tools behave in predictable ways in 3-dimensional space. Once you know the rules, you can manipulate the tools to construct the geometry you want.

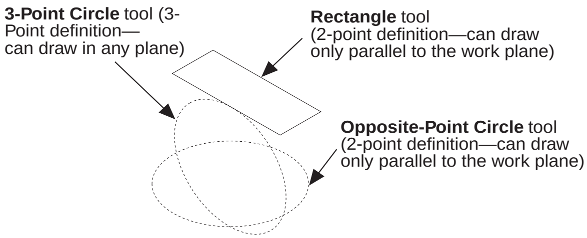
In traditional 3D computer-aided design, you work as though you have propped a piece of glass on one of the faces of the model and draw on that. Every time you want to work on a different plane, you have to pick up your glass and carry it to the new position. With Graphite, you can move around inside the model and draw wherever you want. Generally, all you have to worry about is the design, not moving the piece of glass.

The Drafting Assistant keeps tabs on the angles and intersections for you in all three dimensions. If geometry exists, you can draw relative to it. When it is simpler to use a work plane, or no geometry exists which is relative to the plane you want to use, you can move the work plane with the *Planes*, *3-Point Plane* and *Define Plane* commands in the 3D menu.

Referral:

Work planes are discussed later in this chapter

For example, if you want a circle in the work plane or parallel to the work plane, you can use tools which require only two specification points, such as the **Center-Point Circle** tool or the **Opposite-Point Circle** tool. If you want a circle on a non-parallel plane, you must use three-point specification tools, such as the **3-Point Circle** tool or **Tangent-Point Circle** tool.



When a tool creates geometry from two points, the geometry lies on a plane parallel to the work plane. If the tool uses three points to define the geometry, the geometry will lie on the plane defined by the three points.

If your geometry does not allow you to specify three points (or two entities) in the plane you want to use, you must move the work plane.

Reminders:

To see geometry in 3D you have two choices.

- In the Views menu, choose *Views* and select a view. The default views are Isometric, Right, Front, Top and Trimetric.
- In the Views menu, choose *Show Trackball*. When the Trackball appears you can select a view from its own menu.

As you create or view 3D geometry, displaying the Triad will show you the orientation of the work plane on which you are drawing. The *Show Triad* command is located in the 3D menu. See Chapter 6, "Advanced Environment Settings," for more information on the Trackball and the Triad.

Construction Rules for Tools Used in 3D

Lines

Single, connected and parallel lines have no 3D restrictions.

Tangent lines pulled off a curve, circle or spline will be tangent if created in the same plane as the curve, circle or spline.

Tangent lines that are tangent to only one object can be created only with objects on the work plane at the origin. Tangent lines tangent to two objects can be created on any work plane as long as both objects are on the same work plane.

Arcs A center-point arc is drawn parallel to the work plane.

A 3-point arc is drawn in the plane designated by the three points defining the arc.

A tangent-point arc is drawn in the plane of the three defining points

Circles A center-point circle and opposite-point Circle are drawn parallel to the work plane.

A 3-point circle is drawn in the plane designated by the three points defining the circle.

A tangent circle is drawn in the plane of the two defining entities.

Ellipses A 2-point center ellipse and opposite-point ellipse are drawn parallel to the work plane.

A 3-point center ellipse and 3-corner ellipse are drawn in the plane designated by the three points defining the ellipse.

Rectangles All rectangles and polygons are created parallel to the work plane.

Splines Splines have no 3D restrictions.

Fillets/ Fillets and chamfers are constructed in the plane defined by the selected entities.

Tracer This tool does not work with 3D objects.

3D Features and Tools

Graphite provides you with tools and features specific to the 3D environment.

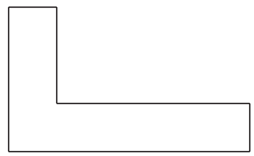
Extrude Command

This command, in the 3D menu, allows you to take 2D geometry and make it 3D through a simple extrusion process. You also have the option to extrude as a surface.

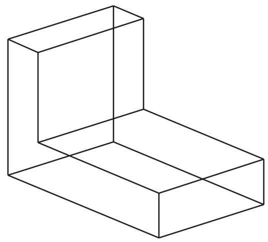
When you begin thinking about using this command, consider the following points:

- Which face shows the most detail?
- Will the extrusion be uniform or do some portions of the part have different dimensions?

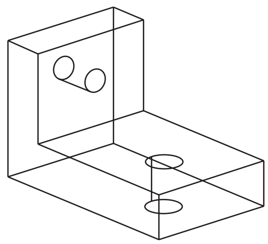
For example, the top view of a bracket provides the most detail. Draw it with the **Connected Lines** tool on the tool palette.



Once you have the basic shape drawn, you can use the *Extrude* command from the **3D** menu to create the depth of the bracket.

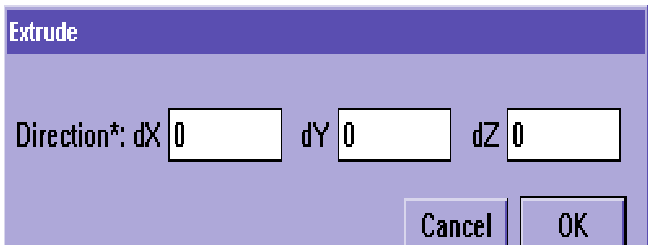


Once extruded, you can add some circles and extrude them to create holes in the bracket.



Using the Extrude Command

1. Draw the geometry to be extruded.
2. Select the geometry if it is not already selected.
3. Choose ***3D>Extrude***.
4. Enter the distance and direction or drag to specify the extrusion. Positive or negative values indicate the direction.



The Tab key can be used to move between the x, y, and z coordinate data fields.

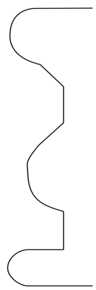
5. Click OK and the geometry is extruded.

Notes:

- X and Y are on the current work plane and Z is perpendicular to that plane. This is not necessarily relative to the original x,y,z axes that were used when the geometry was constructed.
- If the object was drawn in the x,y plane, the extrusion should be in the z-direction. Therefore you should enter a z-value in the dialog box. You can think of the screen as the 0 coordinate with positive values extending toward you and negative values away from you.
- You can also manipulate the work plane to determine the direction of an extrusion.

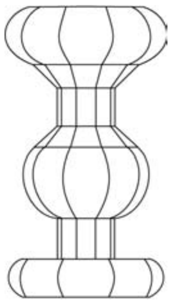
Revolve Command

The *Revolve* command, in the 3D menu, lets you create a model as though it were turned on a lathe. It copies and revolves a 2D object into a 3D object. You also have the option to surface your geometry at the same time it is revolved.

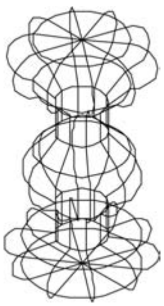


You begin by drawing half of the model on an axis for revolving.

Then use the *Revolve* command to revolve the image a specified number of steps and angle.

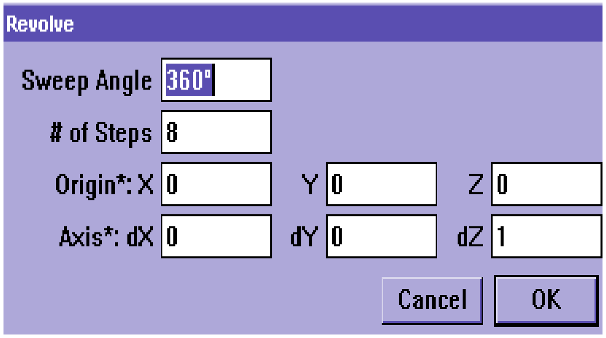


Finally, you can show the Trimetric view to see more detail of the wireframe.



Using the Revolve Command

1. Draw the geometry to be rotated.
2. Select the geometry if it is not already selected.
3. Choose ***3D>Revolve***. The Revolve dialog box displays.



4. Specify the number of degrees (*Sweep Angle*) for the revolution.
5. Specify the *# of Steps* (copies or divisions).
6. Tab to the *Origin* data field in the dialog box.
7. Place the target cursor at the startpoint of the axis of revolution.
8. Drag along the axis from the startpoint to the endpoint. The six data fields are automatically filled in.
9. Click OK and the selected geometry is now revolved around the axis as specified, according to the right hand rule of revolution.

Tip:

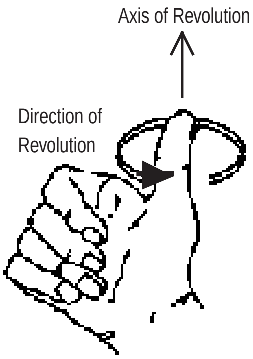
You can use the Tab key to move through the data fields.

Tip:

To establish the axis of revolution, you can also click the startpoint and the endpoint.

Right-hand Rule of Revolution

The direction of revolution is determined by the right-hand rule of revolution which states that if the thumb is pointed toward the positive axis of revolution, the revolution will be in the same direction in which the fingers are curled.



Note: When revolving a circle that will eventually be surfaced, the Axis of Revolution should be on an *endpoint* not a *quadrant*.

Work Plane

The Work Plane is the plane on which you create your geometry. It is an x, y plane with an origin of 0, 0, 0 for all data input. You can move the work plane as desired by creating your own or choosing one of the standard work planes available in Graphite.

As you learn to use Graphite, you should attempt to use the Z-Drafting Assistant without moving the work plane. Even though the Z-Drafting Assistant does a lot of the work for you, the work plane is still an essential element of 3D modeling.

The Z-Drafting Assistant assumes that geometry is being created in the current work plane unless it snaps to an *align:z* or to a logical snap point that is not in the current work plane.

If you are creating geometry that is not parallel to the current work plane and that does not snap to one of the Drafting Assistant's constraints like *endpoint* or *midpoint*, and if you are using a tool that requires only two points for creation, such as the **Center-Point Circle**, **Rectangle**, or **2-Point Ellipse**, you need to use a work plane to place the geometry properly. Also, tools which use three points for specification can be drawn non-parallel to the work plane.

Tech Note:

If a work plane is named Temp Plane, it was created with the *3-Point Plane* command and will exist only for the duration of the current Graphite session. You can rename the TempPlane work plane with the *Define Plane* command to make it a permanent plane for that file.

Also, there can only be one plane called Temp Plane in a document. If you create another plane using the *3-Point Plane* command, without changing the name of the first, the new Temp Plane replaces the old one.

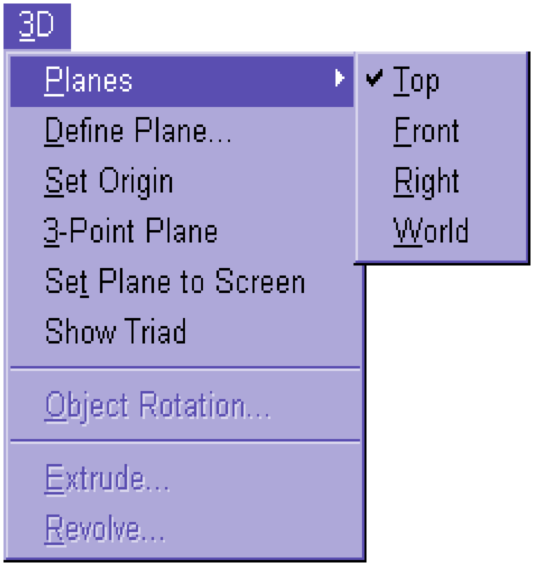
Planes

This command, in the 3D menu, displays the Planes submenu so that you can set the current work plane. The standard work planes—Front, Right, Top and World—are always displayed, as well as any temporary plane (TempPlane) you have generated and any planes you have created with the *Define Plane* command. See margin notes.

Setting the Work Plane

1. Choose ***3D>Planes***.

The Planes submenu displays.



2. Drag down the submenu until the plane you want highlights.
3. Release the mouse button.

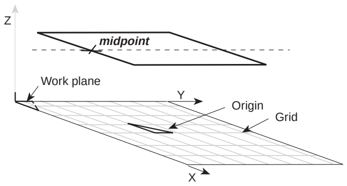
   The work plane in the sheet view or active view window rotates as specified, and the work plane name is checked in the submenu.

Since you have only changed the plane, the appearance of your geometry, its view orientation, remains unchanged. Therefore, you may also want to set the view to match the current work plane.

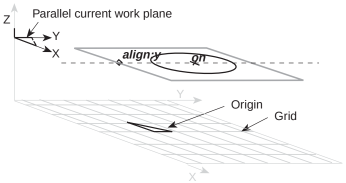
If you want to move both the origin and change the orientation of the work plane, use either the *3-Point Plane* command or the *Define Plane* command.

Specifying the Position of the Work Plane

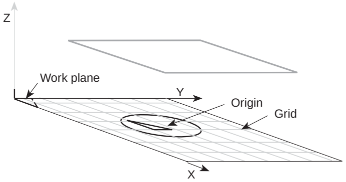
In conventional CAD-programs you have to specify both the orientation of the work plane and its exact location along the z-axis. In Graphite you need only to specify the orientation of the work plane. Once that is done, then all planes parallel to that orientation act equally as the current work plane, e.g. the Drafting Assistant identifies the location of the work plane automatically.



When you identify one point of an object with the Drafting Assistant...



...the new object geometry is created on the plane of the identified object.



If you do not identify an object the new object geometry is placed onto the work plane at the origin.

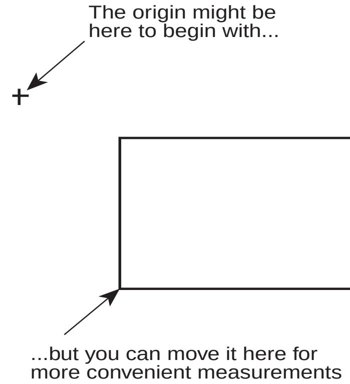
Another way to explain the relationship presented in the graphics above, is to remember the role of the Drafting Assistant. For new geometry to be placed on the same plane as another piece of geometry, you need to brush over that “old” geometry to wake up one of its control points like midpoint or endpoint. Graphite will then retain this plane information as you create the new geometry.

If no object exists or no geometry is referenced for plane information, Graphite places the new object geometry onto the work plane at the origin.

The Origin and the Work Plane

Occasionally, when you are working on a model, you may simply want to move the origin of the work plane. This is especially useful for measuring distances. For example, you can open a document and start drawing without regard to the location of the origin, then move the origin to a convenient location for future reference.

Choose ***Layout>Show Grid*** to display the origin.



Set Origin Command

This command, in the 3D menu, sets a new origin in the current work plane.

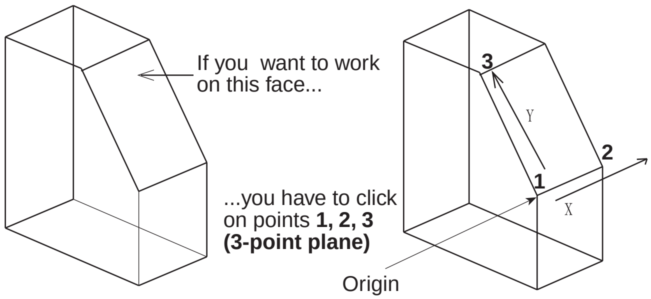
1. Choose ***3D>Set Origin***.
2. Click in the drawing area to indicate the location for the new origin.

   The origin of the current work plane is moved while the orientation of x,y,z remains the same.

If you want to move the origin and change the orientation of the work plane, use either the *3-Point Plane* command or the *Define Plane* command.

Setting a New Work Plane

You may want to define a work plane other than the standard planes. For example, if you want to work on an angled face, you can reorient the work plane.



You can define a new plane with two commands on the 3D menu, *3-Point Plane* and *Define Plane*. Defining a new work plane is accomplished by simply specifying the origin and indicating the horizontal and vertical directions. You can use *3-Point Plane t*o set the work plane, and then use *Define Plane* to name and save the plane.

3-Point Plane Command

This command, in the 3D menu, sets a temporary work plane from the three points you specify. Follow the directions in the Message Line.

1. Choose ***3D>3-Point Plane***.
2. Click in the drawing area to indicate the point you want for the origin of the new work plane.
3. Click a point to define the positive x-axis.
4. Click a point to define the positive y-axis.

   A temporary work plane is created and named TempPlane. The work plane moves, but the view does not change.

You can rename this plane by going to the 3D menu and choosing *Define Plane*. There is an option to *Rename* in the dialog box. See a later section for more information. If you do not rename the new plane, it will be replaced by the next *3-Point Plane* you specify and will be removed from the list when you exit from Graphite.

Define Plane Command

This command, in the 3D menu, allows you to define a new work plane, name a temporary plane or change a standard plane. The standard planes cannot be changed unless you click the lock icon off. Their default configurations are defined as follows:

Front	The x,z plane.
Right	The y,z plane.
Top	The x,y plane.
World	The origin and plane of the original geometry when it was created. This plane cannot be changed.

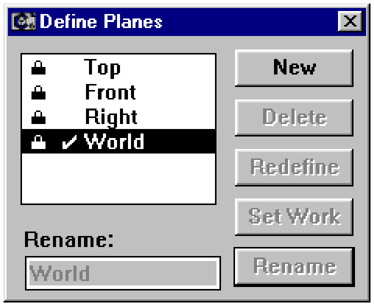
If you change any of the standard planes, you can always revert to the world plane to return to the original orientation.

If you have used other computer applications, you may be familiar with the terms *World Coordinate System* and *User Coordinate System*. The World Coordinate System is equivalent to Graphite's *world work plane* and the User Coordinate System is equivalent to the *current work plane*.

Specifying a New Work Plane

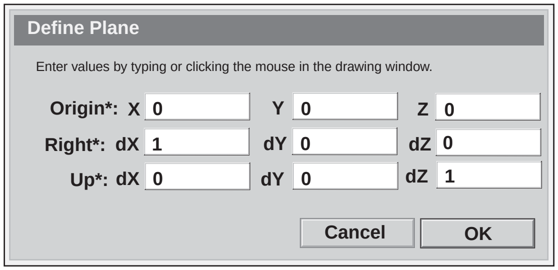
1. Choose ***3D>Define Plane***.

   The Define Plane dialog box displays.



2. Click New.

   The Redefine Plane dialog box displays and the *Origin* data field highlights



   The new work plane is named Plane 1 by default.

3. On the drawing area, click the location for the new origin.

   The *Right* data field is selected.

4. Click a location on the positive x-axis.

   The *Up* data field is selected.

5. Click a location on the positive y-axis.
6. Click OK.

   The definition box goes away and the new plane is defined. The new work plane is named Plane 1 by default. If this plane is not renamed, the next plane created will be named Plane 2.

7. Select Plane 1, the new plane just created.
8. Click Set Work.

The current plane changes to the new specification.

9. Rename this work plane if you want.
10. Dismiss the dialog box if you have no other need to define work planes.

    Be aware that simply changing the work plane does not mean that work you do will be on the visible face. The view orientation must be adjusted accordingly.

Renaming a Plane

1. Choose ***3D>Define Plane***.
2. If necessary, click the lock icon to unlock the plane, then click the plane to be renamed in the list box.
3. Type a new name.
4. Click Rename.

   The plane is renamed. If the renamed plane was a temporary plane, it becomes permanent. The name is added to the Planes submenu and the list in the Define Plane dialog box.

The World plane cannot be renamed.

Deleting a Plane

1. Choose ***3D>Define Plane***.
2. If necessary, click the lock icon to unlock the plane and then click the plane to be deleted from the plane list.
3. Click Delete.

   You cannot delete a plane if it is set as the current work plane. First make another plane the current work plane and then it can be deleted.

If you want to delete a standard plane (Front, Right, or Top), you must unlock it. You can never delete the World plane. If you do choose to delete a standard plane, do not choose *Save Preferences*. Otherwise, you will have to recreate the plane if you want it for future files.

Redefining a Plane

You can use *Define Plane* to change the origin or orientation of the axes of any plane, except the World plane.

1. Choose ***3D>Define Plane***.

2. Click the lock icon for the plane you want to change.
3. Click the name of the plane you want to change.
4. Click Redefine.

   The Redefine Plane dialog box displays the settings as they appear on the screen with the origin selected.
5. On the drawing area, click the location for the new origin.

   The *Right* data field becomes selected.
6. Click a location on the positive x-axis.

   The *Up* data field becomes selected.
7. Click a location for the positive y-axis.
8. Click OK.

   The plane is redefined.

You cannot redefine the standard work planes—Front, Right, and Top—without unlocking them. You can never redefine the World plane, which was established from the original orientation of the model.

Set Plane to Screen

This command, in the 3D menu, sets the work plane to be coincident with the screen, where the origin is in the center of the screen; the x-axis is coincident with the width of the screen and the y-axis is coincident with the height of the screen.

This sets the plane to match the screen in all views which is particularly useful when you are moving from view to view while drafting.

If you want to name this work plane, use the *Define Plane* command.

Selecting Objects

Once you have constructed objects, you may want to make changes. One of the basic rules of Graphite says that you must first select an object before you can edit it. Just as you first have to select a drawing tool from the tool palette before you can start drawing, an object must first be selected before a function can be applied.

Modifying an object is always a two-step process:

1. Select the object.
2. Specify the action for the selected object.

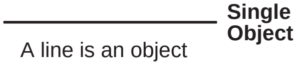
For example, you could select a circle and then change the pen style to *Center* to indicate a bolt-hole circle.

The following topics are covered:

- Objects
- Indicating Selection
- Selection Process

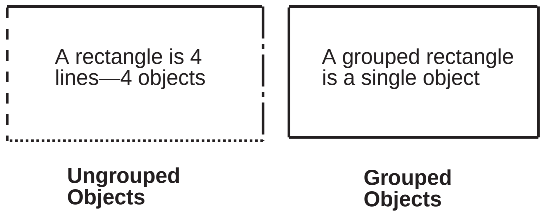
Objects

A single piece of geometry is an object.



Several objects that have been grouped with the *Group* command are also an object.

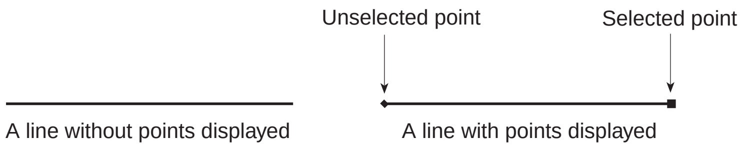
For example, the four lines of a rectangle are four objects. If you group them, Graphite treats them as a single object.



A point is an object, too. Every type of geometry contains two or more points, sometimes called *control points*. A line has two control points, one at the beginning and one at the end of the line. If you select an object and choose ***Layout>Show Points***, the points are visible on the selected object.

Referral:

Selecting points is described in a later section of this chapter.



Keep in mind that you must select an object first and then the command to execute a function.

If you select a point without selecting the geometry it defines, Graphite treats the point as an object.

Indicating Selection

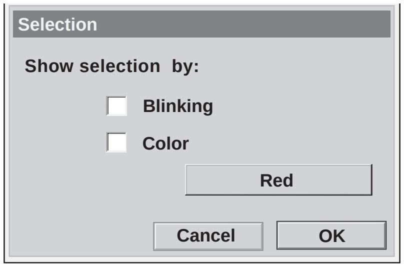
When you select an object, its appearance on the screen shows that it is selected. It is a specific color (such as red), it blinks, or both. Graphite usually displays selections according to your monitor type, red for a color monitor and blinking for a monochrome monitor. You can change the color or turn blinking off and on for the selection indicator. If you wish, you can choose both color and blinking.

Selection Command

This command, found under ***Layout>Preferences***, determines the appearance of selected objects. If you have a color monitor, selected objects appear in a color (which you can change with this setting). If you have a monochrome monitor, selected objects blink.

If you have a color monitor, you can also specify blinking with or without color.

To choose a color, move the pointer to the color box and click on the color name to display the color list. Drag to the color you want and release or click the color. Once the selection color is changed, all current and future selected items appear in the new color.



You shouldn't use black for indicating selection because many of the standard pens use black lines.

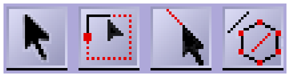
Selection Process

Selecting an object does not affect the properties of the object. A selected object highlights, but this highlighting goes away once the object is deselected. While points and objects are selected in a similar way, point selection is controlled by the *Selectable Points* setting in the Edit menu. For this reason, selecting objects and selecting points are discussed separately.

Graphite provides you with **Selection** tools and commands for selecting your objects.

Selecting Tools

You have four selection tools, **Selection**, **Tracer**, **Select by Line** and **Select by Polygon**.



Selection Tools

The **Selection** tool, in the first subpalette of the main tool palette, selects one or more objects or points in the drawing area.

Selecting a Single Object

1. Click the **Selection** tool in the tool palette. The Message Line reads, *Select: Select [Shift = Extend, Ctrl = Copy* (Windows) or *Option = Copy* (Macintosh)*]*.

2. Move the pointer to an object and click. The object is selected, and any previously selected objects are deselected.

Selecting Multiple Objects using the SHIFT key

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

   The objects you clicked are selected. If you then click one of the selected objects, it is 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 lie completely inside the selection fence are selected. If a portion of an object is outside the region you dragged, that geometry is not selected but the control points of the geometry which lie inside the selection fence are selected.

Tip:

You can specify that only certain objects, layers, or colors are selected by setting a selection mask with the *Selection Mask* command. *Select All* is useful if you want to make a global change in a drawing, such as changing the width of all lines.

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

Selecting All Objects

You can also select all objects by double-clicking on the **Selection** tool.

Tracer Tool

This **Tracer** tool, in the first subpalette of the main tool palette, selects boundaries and uses coincident endpoints and line intersections to trace the perimeter of the geometry.

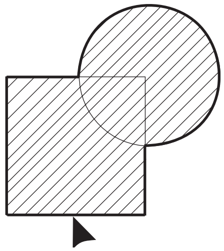
Tip:

The **Tracer** tool is most useful for selecting boundaries for Crosshatching, Fills and for 2D Analysis because it eliminates the need to segment geometry.

1. In the **Selection** tool subpalette, select the **Tracer** tool. The Message Line reads, *Tracer: Pick curve to start trace. [Shift = Extend].*
2. Click on a boundary line.

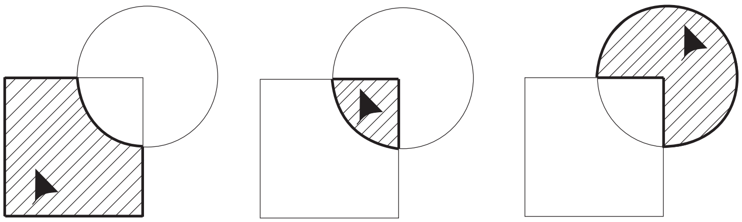
   The Tracer moves from one line to another making its selection.

If you have two overlapping objects as shown below, the **Tracer** tool selects the perimeter of the objects, depending on where you click.



If you click on the outside of the perimeter line, the **Tracer** tool selects the outer most perimeter.

The crosshatching of the figures below shows the boundaries selected when you click the **Tracer** tool at the cursor locations shown.



If you click on the inside of the line, the tracer selects the inner most perimeter.

The **Tracer** tool does not select actual geometry, but rather creates temporary geometry that Graphite uses for operations like Fills, Crosshatching and 2D Analysis.

The **Tracer** tool does not work on a 3D model.

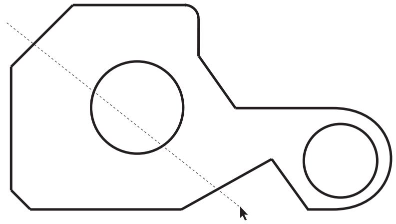
Select by Line Tool

This tool, in the first subpalette of the main tool palette, selects one or more objects along a straight line path when you drag a line over the desired objects.

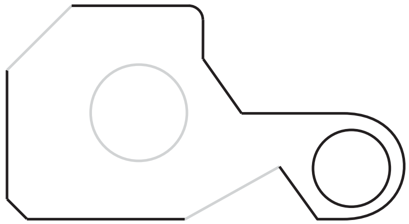
1. Choose the **Select by Line** tool in the Selection tool palette. The Message Line reads, *Select By Line: draw line [Shift = Extend, Ctrl = Copy* (Windows) or *Option = Copy* (Macintosh)*]*.

2. In the drawing area, drag across the objects you want to select (as shown in the graphic below).

3. The objects intersected by the line are selected (as shown below in gray) and any previously selected objects are deselected.

Select by Polygon Tool

This tool, in the first subpalette of the main tool palette, selects one or more objects when a polygon is drawn around them. Each object must be completely enclosed by the selection polygon for the object to be selected. This tool functions similarly to the **Connected Lines** tool.

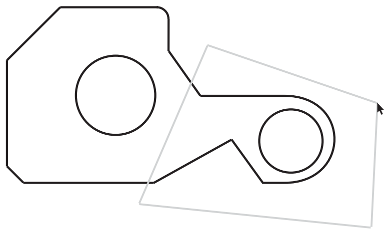
1. Choose the **Select by Polygon** tool in the **Selection** tool palette. The Message Line reads, *Select By Polygon: draw polyline [Shift = Extend]*.

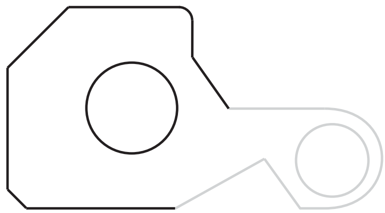
2. Click or drag to indicate the endpoints of the line segments that will make up this selection polygon. The selection polygon appears as line segments and are selected (the default is red).

If you click a point and then change your mind, press the ESC key or choose *Undo* to remove the last line. Pressing the BACKSPACE key (Windows) or the DELETE key (Macintosh) removes the entire selection polygon.

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



You must completely enclose the object or objects you want to select. Objects not completely within the selection polygon will not be selected. The graphic below shows the selected objects in gray.



Deselecting

To deselect an object, click anywhere in the drawing area where there is no object, or click any of the creation tools 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.

Selection Commands

Graphite provides you with various selection commands for selecting objects.

Select All - CTRL+A (Windows); ⌘+A (Macintosh)

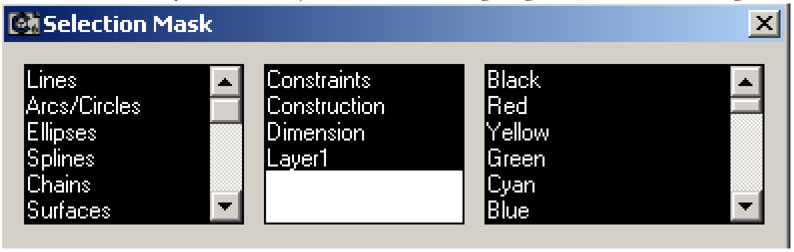
This command in the Edit menu selects all objects except those on a hidden layer or excluded by the Selection Mask. You can also double-click the **Selection** tool to select all objects. If you choose *Select All* while using the **Text** tool, all characters in the current text area are selected.

Tip:

The *Selection Mask* 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 16, "Parametrics").

Selection Mask

This command in the Edit menu allows you to limit selection by object type, layer and color. You can only select objects that are highlighted in the dialog box.



For example, if circles are not highlighted, when you choose *Select All* from the Edit menu, everything but the circles is selected. In this way, you can select such combinations as only blue splines or only red objects on a particular layer.

Using the Selection Mask

1. Choose ***Edit>Selection Mask***.

   The Selection Mask dialog box appears.

   The highlighted items 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 they are highlighted.

While the dialog box is visible, you can select, create 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.

Selecting or Deselecting Listed Items

- To select one item, simply click on the item and the rest of the list will be deselected.
- To deselect a list quickly, click one item in the list.
- To select a contiguous group of items, click on the item at the top or bottom of the desired group list, then hold down the SHIFT key and click or drag up or down to select the other items in the group.

- To select or deselect non-contiguous items, hold down the CTRL (Windows) or the ⌘ key (Macintosh) and click on the items.

The *Selection Mask* includes objects like lines, arcs/circles, text, etc. Most of the objects are self-explanatory except for those mentioned below.

Chain	This object is created when you use *Flatten View* in the Views menu. For more information see Chapter 14, "Advanced Viewing Techniques."

Selecting Points

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

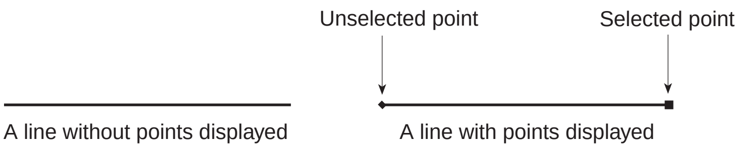
Being able to select points is useful in two situations: stretching selected geometry (described later in this chapter) and control point selection for transformations.

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.

Show Points

This command in the Layout menu toggles the display of the *control points* (endpoints, midpoints, center points, and knot points) for selected objects. When points are displayed, you can select a point by clicking it. If points are not displayed (but *Selectable Points* is set in 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, in the Edit menu, use the *Edit Objects* command.



If you select and drag a line, the line and the endpoints move. If you select and drag only an endpoint of the line, the endpoint moves and the line length changes while the other endpoint of the line remains fixed.

Displaying Points

1. Select the geometry.

2. Choose ***Layout>Show/Hide Points*** to toggle the display of points on and off.

To turn off the point display once the points of an object are showing, you must select the object again and choose either ***Layout>Hide Points*** or the *Control Points* option in the Edit Objects dialog box.

Showing and Hiding Points with Stroke

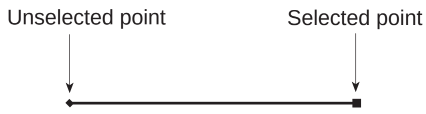
When you hold down the CTRL+SHIFT keys (Windows) or the Command (⌘) key (Macintosh) and click an object, the display of the object's points toggles on or off. If the points are hidden when you click the object, the points will be displayed.

Selecting Points

You can select a control point whether the points are visible or not; however, note that *Selectable Points* in the Edit menu must be on (displaying the ✔ symbol in the menu). To select points if points are not visible:

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

   The selected point displays as a square.



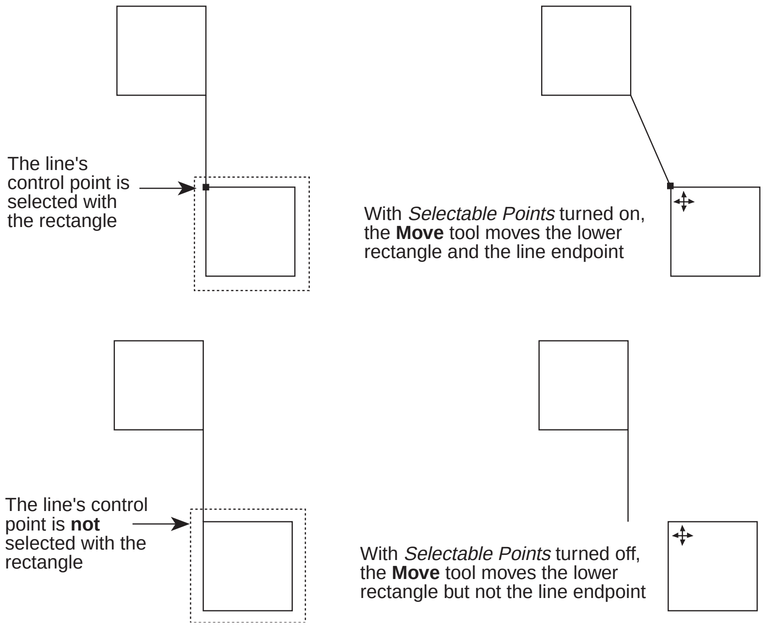
To select points if they are visible:

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

   The selected point displays as a square.

Selectable Points

This command in the Edit menu gives you the ability to select points that aren't displayed. When *Selectable Points* is not set, you cannot select points by dragging a fence. 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. The following example illustrates the use of *Selectable Points*.



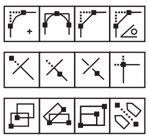
Editing Objects

Once you create an object, you can select and alter it by choosing a command or selecting a tool to edit it. This chapter describes the common editing activities you can perform on a selected object and compares different methods for performing the same action.

The specific topics covered are:

- Editing Tools
- Moving Objects with Tools
- Copying Objects with Tools
- Sizing Objects with Tools
- Editing Commands
- Duplicating Objects
- Changing the Characteristics of Objects
- Arranging Geometry

Editing Tools



Fillet and Chamfer tools

Trim tools

Transformation tools

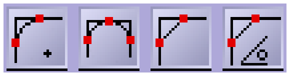
These tools allow you to change an object physically, either altering the geometry, changing the size, changing the location, or orientation of the selected object. Three subpalettes on the tool palette provide these editing capabilities.

Fillet and Chamfer — The **Fillet** and **Chamfer** tools round the corners of non-parallel lines or cut them at straight angles, respectively.

Trim — The **Trim** tools cut off or extend a line to the boundary limit you specify. You can also segment a line at an intersection, or create corners from intersecting lines.

Transformation — The **Transformation** tools move, rotate, expand or shrink, and mirror objects.

Fillet and Chamfer Tools



These tools on the tool palette construct fillets and chamfers from corners formed by nonparallel lines or curves. The fillets and chamfers are automatically trimmed. If you do not want them trimmed, hold down the CTRL (Windows) or the OPTION (Macintosh) key while you are selecting the objects to fillet or chamfer.

2-Entity Fillet Tool

This tool, in the ninth subpalette of the main tool palette, constructs an arc tangent to the two objects you click. Fillets use the smallest arc between the selected geometry.

Using the 2-Entity Fillet Tool

1. Click the **2-Entity Fillet** tool. The Message Line reads, *2-Entity Fillet: Pick first entity [Shift = Corner, Ctrl = No trim* (Windows) or *Option = No trim* (Macintosh)*]*.
2. Enter the arc radius you want in the Status Line. The default radius is .25 inch.
3. Select the first entity. The Message Line now reads, *2-Entity Fillet: Pick second entity to fillet [Ctrl = No trim* (Windows) or *Option = No trim* (Macintosh)*]*.
4. Select the second entity. The fillet is created.

You can also hold down the SHIFT key and click once inside the corner you want to fillet. (See the Message Line at the top of your drawing area for a reference to this feature.) If you hold down the CTRL (Windows) or OPTION (Macintosh) key while you select the objects to fillet, the objects are not trimmed. (See the Message Line).

The Status Line allows you to specify the Radius of the fillet either before or after you create the fillet.

R .25

3-Entity Fillet Tool

This tool, in the ninth subpalette of the main tool palette, constructs a fillet tangent to the three objects you choose.

Using the 3-Entity Fillet Tool

1. Click the **3-Entity Fillet** tool. The Message Line reads, *3-Entity Fillet: Pick first entity [Ctrl = No trim* (Windows) or *Option = No trim* (Macintosh)*]*.
2. Click the three objects you want to fillet as directed by the Message Line.

If you hold down the CTRL (Windows) or OPTION (Macintosh) key while you select the objects to fillet, the objects are not trimmed (See the Message Line).

There are no Status Line entries.

2-Entity Chamfer Tool

This tool, in the ninth subpalette of the main tool palette, creates a chamfer across a corner at the specified distance from the intersection of two lines.

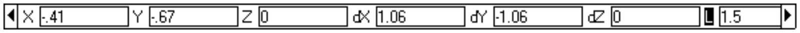
Using the 2-Entity Chamfer Tool

1. Click the **2-Entity Chamfer** tool. The Message Line reads, *2-Entity Chamfer: Pick first entity chamfer [Shift = Corner, Ctrl = No trim* (Windows) or *Option = No trim* (Macintosh)*]*.
2. In the Status Line, enter the distance you want the chamfer from the corner. (The default distance is .25 inch.)
3. Click each line making up the corner you want chamfered as directed by the Message Line. You can also hold down the SHIFT key and click once inside the corner you want to chamfer. (See the Message Line).

The lines are automatically trimmed or extended.

If you hold down the CTRL (Windows) or the OPTION (Macintosh) key while you select the objects to chamfer, the objects are not trimmed. (See the Message Line).

The Status Line allows you to specify the distance (Length) from the chamfer to the intersection of the corner lines.



Angular Chamfer

This tool, in the ninth subpalette of the main tool palette, creates a chamfer at the specified angle and distance from the corner. The specified angle is that between the chamfer and the second line of the corner. The specified length is the distance between the *corner and the intersection of the chamfer* and the *second line of the corner*. (The *second line of the corner* refers to the second line chosen when creating the chamfer).

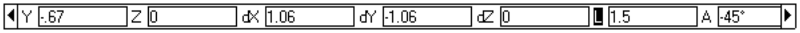
Using the Angular Chamfer

1. Click the **Angular Chamfer** tool. The Message Line reads, *Angular Chamfer: Pick first entity to chamfer [Ctrl = No trim* (Windows) or *Option = No trim* (Macintosh)*]*.
2. In the Status Line, enter the length you want the intersection of the chamfer and the second line of the corner from the corner. The default Length is .25 inch.
3. Enter the angle between the chamfer and the second side. The default is 45°.
4. Click each line making up the corner you want chamfered as directed by the Message Line.

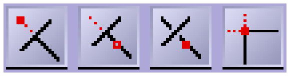
   The lines are automatically trimmed or extended.

If you hold down the CTRL (Windows) or the OPTION (Macintosh) key while you select the objects to chamfer, the objects are not trimmed. (See the Message Line).

The Status Line allows you to set the distance (Length) from the intersection as well as the Angle.



Trim Tools



The **Trim** tools on the tool palette lengthen or shorten lines and curves. In general, you select the limiting object(s) before you select the tool. However, if you don't select everything you need, you can hold down the SHIFT key (the tool becomes a temporary **Selection** tool) and click additional objects after the **Trim** tool is selected (see the Message Line).

Simple Trim Tool

This tool, in the tenth subpalette of the main tool palette, shortens a line to the specified boundary. If you hold down the CTRL (Windows) or the OPTION (Macintosh) key while using this tool, it becomes the **Relimit** tool.

Rule: Point to what you want to remove.

Using the Simple Trim Tool

1. Select the object that limits the trim.
2. Select the **Trim** tool. The Message Line reads, *Simple Trim: Pick section to trim [Shift = Select boundary, Ctrl = Relimit* (Windows) or *Option = Relimit* (Macintosh)*]*.

   If necessary, use SHIFT-Click to select more boundary objects.
3. Click the section of the object to be discarded.

There are no Status Line entries.

Relimit Tool

This tool, in the tenth subpalette of the main tool palette, lengthens or shortens a line to the specified boundary. If you hold down the CTRL (Windows) or the OPTION (Macintosh) key while using this tool, it becomes the **Trim** tool.

Rule: Point to what you want to keep.

Using the Relimit Tool

1. Select the object that limits the change.

2. Select the **Relimit** tool. The Message Line reads, *Relimit: Pick section to retain [Shift = Select boundary, Ctrl = Trim* (Windows) or *Option = Trim* (Macintosh)*]*.

   If necessary, use SHIFT-Click to select more boundary objects.

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

There are no Status Line entries.

Segment Tool

This tool, in the tenth subpalette of the main tool palette, divides a line or curve at intersections with other lines or curves.

Using the Segment Tool

1. Select the objects that limit the segmentation.

2. Select the **Segment** tool. The Message Line reads, *Segment: Pick entity [Shift = Select boundary, Ctrl = Current pen* (Windows) or *Option = Current pen* (Macintosh)*]*.

   If necessary, use SHIFT-Click to select more boundary objects.

3. Click the object to be segmented.

   The selected object segments at the boundary objects. Even though you cannot see the segmentation on the screen, you can select parts of the segmented line by clicking.

If you hold down the CTRL (Windows) or OPTION (Macintosh) key while selecting the line to be segmented, the new segment appears in the characteristics of the current pen style (see the Message Line).

There are no Status Line entries.

In order to see the segmentation on the screen, first select the boundaries and the object you want to segment and then choose ***Layout>Show Points***. After the segmentation, the endpoints of the segmented parts are displayed.

Corner Trim Tool

This tool, in the tenth subpalette of the main tool palette, creates a corner from the two objects you specify. Lines are extended or shortened to create the corner.

Using the Corner Trim Tool

1. Click the **Corner Trim** tool. The Message Line reads, *Corner Trim: Pick first entity to trim [Shift = Corner, Ctrl = No Trim* (Windows) or *Option = No Trim* (Macintosh)*]*.
2. Click each object as directed by the Message Line.

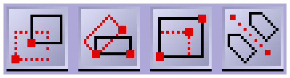
   You can also press and hold the SHIFT key and click inside the about-to-be-created corner.

There are no Status Line entries.

Extending Lines to a Theoretical Intersection

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 (Windows) or OPTION (Macintosh) key and click the line that is not to be trimmed (see the Message Line).

Transformation Tools



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

You can copy at the same time you transform objects by holding down the CTRL (Windows) or the OPTION (Macintosh) key while you specify the transformation.

When you press the SHIFT key you can select additional objects after you have selected a Transformation tool. (See the Message Line). As soon as you release the SHIFT key the Transformation tool is active again.

Move Tool

This tool, in the eleventh subpalette of the main tool palette, moves the selected objects to a new location. You can copy the selection by holding down the CTRL (Windows) or OPTION (Macintosh) key while you select the objects. If you select more than one object, they remain in the same position relative to each other.

Referral:

Enlarging or shrinking an object with the **Move** tool is described under “Sizing objects with Tools” in this chapter.

Using the Move Tool

1. Select the object(s) to be moved.

2. Select the **Move** tool. The Message Line reads, *Move: Pick beginning reference point [Shift = Select, Ctrl = Copy* (Windows) or *Option = Copy* (Macintosh)*]*.

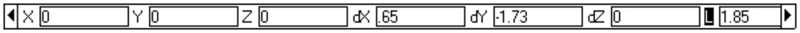
   If necessary, use SHIFT-Click to select more objects.

3. Drag the selected object to a new location, pressing to set a reference point 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.

The move is performed relative to the specified points.

The Status Line allows you to specify the distance that the selection is to be moved in a particular direction.



A positive or negative value entered in a data field of the Status Line determines the direction along the axis. A negative value moves the object to the left or down on the screen. A positive value to the right or up.

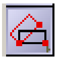
Nudge Tool

The nudge tool is used to move items a specified distance using the arrow keys.

Using the Nudge Tool

1. Set the nudge distance . Go to **Layout>Preferences>Nudge** or Right Mouse Click and choose **Nudge**.
2. Select an item or items to move or Nudge.
3. Use an arrow key to move the selected items in the desired direction. By holding down the *Control* key you can double the nudge distance.

Rotate Tool



This tool, in the eleventh subpalette of the main tool palette, rotates one or more objects around a specified point. You can copy the selection by holding down the CTRL (Windows) or OPTION (Macintosh) key while you select the objects. If you select more than one object, they remain in the same position relative to each other.

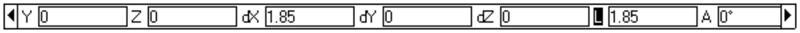
Keep in mind that you always rotate about the Z-axis with this tool.

Using the Rotate Tool

1. Select the object(s) to be rotated.
2. Select the **Rotate** tool. The Message Line reads, *Rotate: Pick center of rotation [Shift = Select, Ctrl = Copy* (Windows) or *Option = Copy* (Macintosh)*]*.

   If necessary, use SHIFT-Click to select more objects.
3. Select the center of rotation.
4. Drag the objects or enter an Angle in the Status Line to specify the rotation.

The Status Line allows you to specify the angle of rotation.

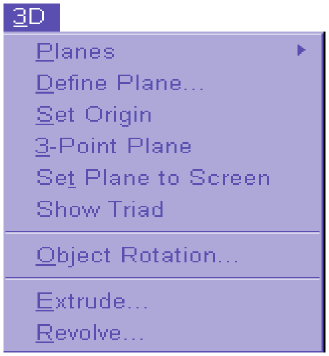


You can also click the reference point and then click the destination point. It is not necessary for the reference and destination points to be on the object you are moving. If they are not, the rotation is performed relative to the specified points.

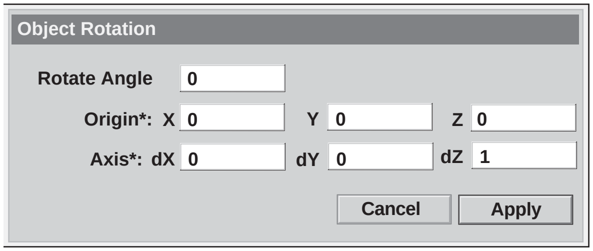
With this tool you can rotate the selected geometry around an axis through the origin you specify and parallel to the z-axis of the work plane. The *Object Rotation* command allows you to rotate around any axis specified.

Object Rotation Command

This command, in the 3D menu, allows you to rotate geometry around any axis that you specify.



1. Select the geometry that you want to rotate.
2. Choose ***3D>Object Rotation***.
3. In the *Rotate Angle* field, enter the angle you want to rotate the geometry.



4. Specify the *Axis of Rotation* around which you want to rotate the geometry. Tab to the *Origin* data field in the dialog box.
5. Place the target cursor along the desired axis.
6. Drag along the desired axis. Don't worry about whether the z-axis is pointed in the right direction. The six data fields are automatically filled in.
7. Click OK. The object rotates.

Expand/Shrink Tool

Tip:

Using the beginning and ending reference point for the **Expand/Shrink** tool is particularly useful if you want to change the size of an object relative to the size of another object.

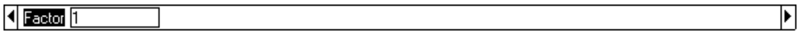
This tool, in the eleventh subpalette of the main tool palette, enlarges or reduces objects while maintaining the proportions. You can copy the selection by holding down the CTRL (Windows) or OPTION (Macintosh) key while you select the objects. If you select more than one object, they remain in the same position relative to each other.

Using the Expand/Shrink Tool

1. Select the object(s) to be expanded or shrunk.
2. Select the **Expand/Shrink** tool. The Message Line reads, *Expand/Shrink: Pick anchor point [Shift = Select, Ctrl = Copy* (Windows) or *Option = Copy* (Macintosh)*]*.

   If necessary, use SHIFT-Click to select more objects.
3. Click a point on the object that you want to remain stationary.
4. Drag the selected object to a new location, clicking to set a reference point and releasing to indicate the new location.

The Status Line allows you to enter the exact scale (Factor).



To expand or shrink geometry in one direction only, use the **Stretch** tool. See a later section.

Mirror Tool

This tool, in the eleventh subpalette of the main tool palette, 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 (Windows) or OPTION (Macintosh) key while you select the objects. If you select more than one object, they remain in the same position relative to each other.

Using the Mirror Tool

1. Select the object(s) to be mirrored.
2. Select the **Mirror** tool. The Message Line reads, *Mirror: Pick beginning of reference line [Shift = Select, Ctrl = Copy* (Windows) or *Option = Copy* (Macintosh)*]*.

   If necessary, 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.

   Hold down the CTRL (Windows) or OPTION (Macintosh) key, before specifying the reference line to create a mirrored copy.

The Status Line has no entries.

Note: When working in 3D, you can mirror selected geometry through the axis you specify and parallel to the work plane.

Additional Transformation Tool

The **Stretch** tool is not a part of the **Transformation** tool palette but is transformational in nature. It can be added into your program using the *Add Function* command and is accessed through the customized tool palette which appears when you choose ***Utilities>Show Palette***. See Chapter 6 for more information.

Stretch Tool

This tool allows you to scale objects along one axis and a specified angle.

Using the Stretch Tool

1. Select the object.
2. Choose ***Utilities>Show Palette***.
3. Select the **Stretch** tool. The Message Line reads, *Stretch, Vector-defined: Click point to remain fixed.*
4. Click on one point of the selected rectangle.
5. Enter the desired *Scale factor* and *Angle* in the Status Line.
6. Press the ENTER (Windows) or RETURN (Macintosh) key.

The rectangle is scaled at the specified factor and along the specified angle.

You can also click the reference points as directed in the Message Line to set the scale and the stretch angle.

The Status Line allows you to enter the exact scale and angle for the stretch.

Scale 1 Angle 0°

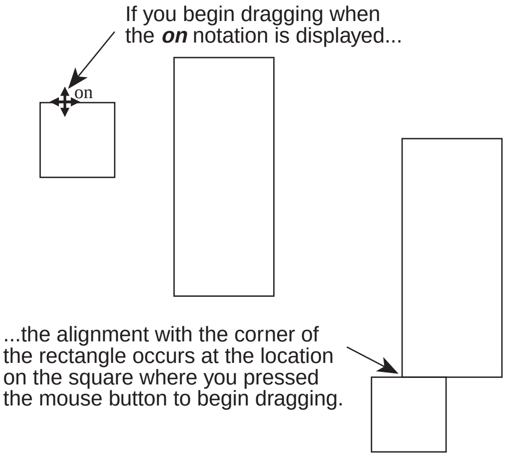
Moving Objects with Tools

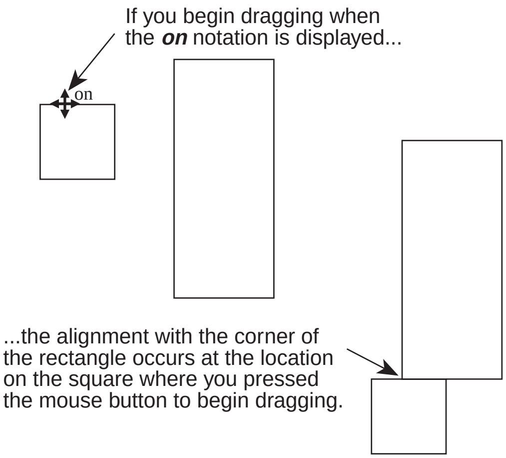
When an object is selected, move the pointer near the object until the 4-way Move symbol appears as shown below and drag the object to a new location.

Note: If you have problems displaying the 4-way Move symbol, it might be because you have just used the *Paste* command. The 4-way Move symbol does not appear when you move the pointer to the selected object because the drawing tool is still in effect if you copy and paste a selected object immediately after it is drawn. You have to click the **Selection** tool before you can move the selected geometry.

Using the Drafting Assistant for Moving

When the pointer becomes the 4-way Move symbol, you can drag the object around and see the Drafting Assistant's notations relative to the object's location when you press the mouse button. In this way, if you move the pointer over a control point so the Drafting Assistant activates it, you can align the object you are moving with that point.



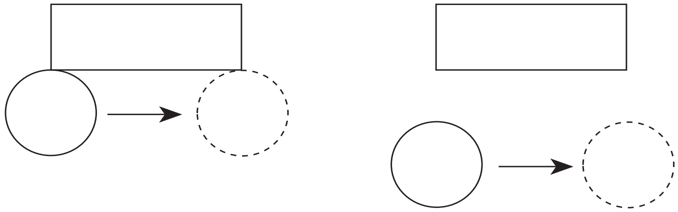


Selected Move versus Move Tool

You can move objects either with the **Selection** tool or with the **Move** tool.

Moving with the Selection Tool

The move function of the **Selection** tool allows free movement of the selected object.



To move a circle from one location to another as shown above, use the **Selection** tool and the Drafting Assistant to align the 90° *quadrant* with the corner of the rectangle. The circles do not need to touch the rectangle to be aligned with it.

To move a circle or an arc from its center point, you must use the **Move** tool rather than the **Selection** tool.

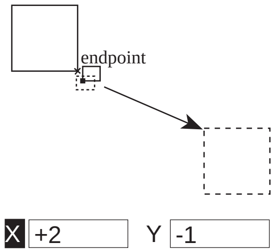
Moving with the Nudge Tool

This tool is accessed through the right mouse button. To set the nudge distance go to ***Layout>Preferences>Nudge***. When you have selected to object to move use the arrow keys and the object will move the distance you specified.

Moving with the Move Tool

The **Move** tool (on the Transformation subpalette) allows you to move the selection with reference to other geometry.

Move a 1-inch square 2 inches in the X direction and 1 inch in the Y direction. See the following graphic.



1. Select the square to be moved.
2. Select the **Move** tool.
3. Click one corner of the square when the *endpoint* notation displays.
4. Enter **2** in the X data field on the Status Line and **-1** in the Y data field.

   A positive or negative value entered in the Status Line determines the direction along the X or Y axis. A negative value moves the object to the left or down on the screen, and a positive value moves the object to the right or up.

Tip:

The **Move** tool can be used to move a circle or an arc by its center point. The **Selection** tool cannot do this.

5. Press the ENTER (Windows) or RETURN (Macintosh) key.

   The square moves.

Moving Objects to Another Layer

1. Choose ***Edit>Edit Objects***.
2. In the Layer field, scroll down to the layer to which you want to move the object.
3. Click OK. The object is now located on the new layer.

Copying Objects with Tools

You can copy selections with the *Copy* command as discussed earlier or by holding down the CTRL (Windows) or OPTION (Macintosh) key while using the following tools:

- **Selection** tool
- **Single Line** tool
- **Center-Point** and **Opposite-Point Circle** tools
- Ellipse tools
- Polygon tools
- **Text** tool
- Transformation tools

Copying with the Selection Tool

You can hold down the CTRL (Windows) or OPTION (Macintosh) key and drag a copy of the selection to a new location.

1. Click the **Selection** tool.
2. Select the object(s) you want copied.
3. Hold down the CTRL (Windows) or OPTION (Macintosh) key.
4. Drag a copy of the selection to a new location.
5. Release the CTRL (Windows) or OPTION (Macintosh) key.

Copying with the Geometry Tools

You can make copies with the following drawing tools:

- **Single Line** tool
- **Center-Point** and **Opposite-Point Circle** tools
- **Ellipse** tools
- **Polygon** tools
- **Text** tool

To create a copy with the drawing tools, proceed as follows:

1. Construct the geometry you want to copy.
2. Hold down the CTRL (Windows) or OPTION (Macintosh) key.
3. Click a new location. The click determines the location of the first point specified during the construction of the original geometry (the *center* of a center-point circle, for example).
4. Release the CTRL (Windows) or OPTION (Macintosh) key.

Copying with the Transformation Tools

You can hold down the CTRL (Windows) or OPTION (Macintosh) key while using a Transformation tool to make a copy of the selected geometry.

1. Select the object you want to copy and transform.
2. Select one of the four **Transformation** tools.
3. Hold down the CTRL (Windows) or OPTION (Macintosh) key.
4. Perform the transformation according to the directions in the Message Line.
5. Release the CTRL (Windows) or OPTION (Macintosh) key.

   A copy transforms and the original remains unchanged.

Copy CTRL (Windows) or Copy OPTION (Macintosh) versus Copy Command

Usually, using the copy option of a geometry tool is faster than using the *Copy* and *Paste* commands in the Edit menu. The *Copy* command is very useful for copying to a different document or application.

Sizing Objects with Tools

Normally you size an object with the **Selection** tool or the **Expand/Shrink** tool. In some cases it is useful to use the **Move** tool for sizing objects. You can stretch objects by selecting a point and dragging it to a new location.

Referral:

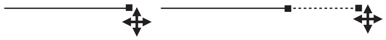
Sizing with the **Expand/Shrink** tool is described in an earlier section.

Sizing an Object with the Selection Tool

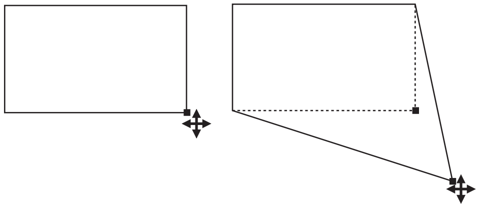
You can stretch objects by selecting a point and dragging it to a new location.

1. In the Edit menu, be certain that *Selectable Points* is set.
2. Click the **Selection** tool.
3. Drag a selection fence around the control points that represent the area you want to stretch.
4. Drag the points to a new location.

A line can be dragged to a new length.



Intersecting lines can be dragged to new lengths.



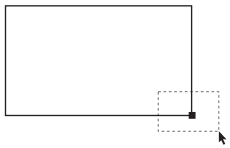
Sizing an Object with the Move Tool

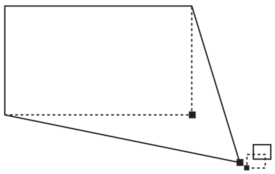
You can also size an object with the **Move** tool. Using the **Move** tool allows you to specify the distance the selected point(s) should be moved by specific values along the X and Y direction in the Status Line.

To move the corner of the rectangle in the next graphic with the **Move** tool, proceed as follows:

1. In the Edit menu, be certain that *Selectable Points* is set.
2. Click the **Selection** tool.
3. Drag a selection fence around the lower right corner of the rectangle.
4. Click the **Move** tool.
5. Enter **+1** in the X data field and **-1** in the Y data field in the Status Line.
6. Press ENTER (Windows) or RETURN (Macintosh).

   The corner of the rectangle moves the distance you specified in the Status Line.

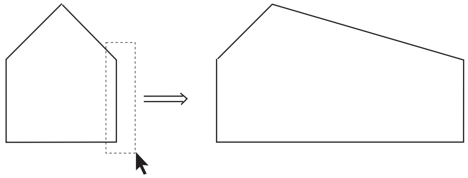




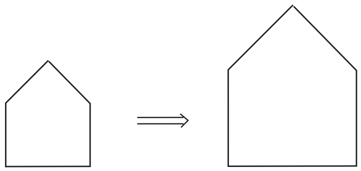
Use the **Move** tool for sizing objects only once since the next values you enter in the Status Line do not refer to the new position of the moved point(s), but always to the original one. That means if you enter (in our example) **–1** in the X data field and **+1** in the Y data field, the point is not moved back to its original position, but in the opposite direction referring to its original position.

Selected Sizing versus Expand/Shrink Tool

Dragging a control point of a selected object changes the size of the object, but it also distorts the object, changing the proportion between height and width.



The **Expand/Shrink** tool on the Transformation subpalette enables you to resize geometry while maintaining its proportions.



Use the **Expand/Shrink** tool to enlarge or shrink a shape proportionally

In addition, you can specify proportions by clicking points on other geometry. For example, you can resize an object to fit within another object by clicking the boundary into which the resized object must fit.

Selected Sizing versus Stretch Tool

You can drag one control point of an object for selected sizing as shown in the previous section. Using the **Stretch** tool, however, you can resize along one axis and at a specified angle. See an earlier section on this tool.

Editing Commands

Four commands in the Edit menu allow you to change objects without changing the physical geometry. You can copy or move objects within the same document—on the same sheet or to other sheets. You can also copy or move objects to other documents, even to other applications.

The *Cut*, *Copy* and *Paste* commands do not alter the object's attributes—the layer and pen style (color, weight, and pattern). If you want to change the attributes of an object, choose ***Edit>Edit Objects***.

Tech Note:

The difference between the *Cut* and *Copy* commands is that cutting a selection removes it from the drawing area while a copied selection remains in the drawing area.

Cut - CTRL+X (Windows); ⌘+X (Macintosh)

This command in the Edit menu removes the selected objects and places them on the Clipboard. Each selection you cut or copy to the Clipboard replaces the previous Clipboard contents.

Cutting Objects

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

Once you cut a selection, you can paste it. 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 onto a document in a different application.

Tip:

If you want to move an object from one layer to another, choose the ***Edit>Edit Objects.***

Moving Geometry with the Cut Command

1. Select the objects you want to move.
2. Choose ***Edit>Cut***.
3. Display the location where you want the selection to appear in the drawing area, scrolling if necessary.
4. Choose ***Edit>Paste***.

The object appears in the center of the drawing area on the original layer on which it was created or onto the work layer if it's from a different application. The object is selected so you can move it, if you want.

Tip:

You can also create a copy of an object or text box with the **Selection** tool. First select the object, then hold down the CTRL (Windows) or the OPTION (Macintosh) key and drag a copy of the object to a new location.

You can copy an object using any of the **Transformation** tools by holding down the CTRL (Windows) or the OPTION (Macintosh) key while using the tool.

Copy - CTRL+C (Windows); ⌘+C (Macintosh)

This command in the Edit menu places a copy of the selection onto the Clipboard without deleting the original selection. 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.

Copying Objects

1. Select the objects to be copied.
2. Choose ***Edit>Copy***.

   The selection goes on the Clipboard, and it remains in the current document.

Paste - CTRL+V (Windows); ⌘+V (Macintosh)

This command in the Edit menu pastes a copy of the Clipboard contents onto the center of the drawing area. The Clipboard contents are not changed when you use the *Paste* command.

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, in the Edit menu, you can *Linear Duplicate* or *Polar Duplicate*.

Rules:

1. If the object was cut or copied from a Graphite document, when it is pasted into another Graphite document it goes on the layer with the same name from which it was cut or copied. A pasted selection retains its original attributes.

   Any geometry on a layer that is not in the new Graphite document is placed on the current work layer. So, make sure that the new document has all the layers of the pasted geometry by creating those corresponding layers in the new document before you paste the geometry.
2. Objects cut or copied from non-Graphite documents are pasted onto the work layer.

The Clipboard

The *Cut* and *Copy* commands place a copy of the selected object on the Clipboard. The Clipboard is a buffer—a temporary storage place that holds the last cut or copied selection. The contents of the Clipboard are objects, not bitmaps or picts.

Erasing Geometry

You can use several methods to erase objects:

- You can select the object(s), then press the BACKSPACE (Windows) or DELETE (Macintosh) key. (You can retrieve what you deleted with the *Undo* command).

- You can select the object(s), then choose the *Delete* command from the Edit menu. (You can retrieve what you deleted with the **Undo** command).
- You can select the object(s), then choose ***Edit>Cut***. (You can retrieve what you cut by using the *Paste* command, as long as you have not cut or copied anything else).

Delete Command

This command in the Edit menu 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 (Windows and Macintosh) or BACKSPACE (Windows only) key to delete a selection. (You can retrieve what you deleted with the *Undo* command).

Cut Command

See an earlier section for information on how to use this command.

Retrieving with Undo and Redo

Once an object is deleted or cut, you can retrieve it with the *Undo* command in the Edit menu. The number of *Undo* commands can be specified in the ***Layout>Preferences>Undos***. You can use CTRL+Z (Windows) or ⌘+Z (Macintosh) to issue the *Undo* command from the keyboard.

Undo Command - CTRL+Z (Windows); ⌘+Z (Macintosh)

This command in the Edit menu reverses the last action. 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 resizing the window or quitting.

If you are using a tool which involves a multi-step process, such as constructing with **Connected Lines** or creating a **3-Entity Fillet**, choosing *Undo* reverts to the beginning of the process. Some other activities can be stopped by pressing the ESC key.

Redo Command - CTRL+SHIFT+Z (Windows); No Key Command (Macintosh)

This command in the Edit menu reinstates the last action reversed by *Undo*. You can *Undo* and *Redo* an unlimited number depending upon what you specified in your Preferences.

Remove Dots Command (Antidot)

The *Remove Dots* command is automatically available to you in Graphite. If it is not, you must add it to the program, using *Add Command* in the Utilities menu. See Chapter 6 for more information.

This command removes all lines with zero length (a.k.a) points from the entire drawing.

To use the command, choose ***Utilities>Remove Dots***. The geometry does not have to be selected to use this command.

Duplicating Objects

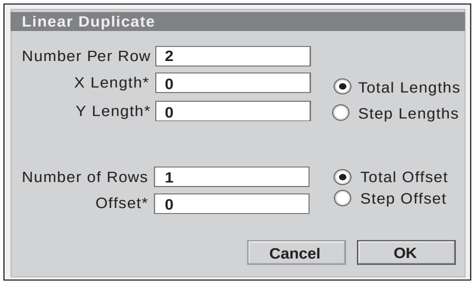
Graphite allows you to create multiple copies of an object and arrange them in an array along a straight line or in a circular pattern.

If you begin a duplication and discover that it is not what you want, you can stop the process by pressing the ESC key, and then choosing *Undo* to remove the copies from your document.

Linear Duplicate Command

This command in the Edit menu allows you to create an array of copies along a straight line.

When you select the command, the dialog box appears:



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.

If you don't know the number: You can also use math operators to determine the number of copies.

	For example, if you want to place studs 16 inches apart on a 17 foot wall, you would enter (17*12)/16 and Graphite will truncate the result to create 12 copies.
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. The X Length and Y Length can be set by entering values, or by clicking the mouse in the drawing area.
Total Lengths	The Total Lengths is the distance from the selected object to the last copy.
Step Lengths	The Step Lengths 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 offset is the perpendicular distance between the row containing the selected object and the last row.
Step Offset	The offset is 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. You can specify the Offset by entering a value or by pressing and dragging the mouse in the drawing area.

Using Linear Duplicate

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

   A dialog box appears.

   An asterisk (*) appearing beside an entry field indicates that you can fill in the field by dragging the mouse in the drawing area to indicate the length. 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 button.
5. Click the *X Length** data field.
6. Move the pointer to the drawing area and drag to indicate the length of the row of copies.

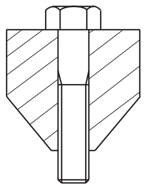
   Both the X and Y values appear in the data fields.

7. If you want more than one row, enter the *Number of Rows*, and click a button to specify the type of offset.
8. Enter the offset, if any. You can use a negative number.

   Dragging to indicate the offset is particularly useful if you want the offset to equal the distance between two objects.

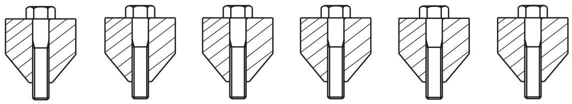
9. Click OK.

An Example



Suppose you want to illustrate six machine screws. You could draw geometry, then use *Linear Duplicate* to copy the geometry in an array.

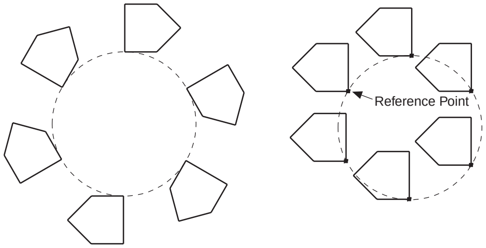
1. Create and select the geometry you want to duplicate.
2. Choose ***Edit>Linear Duplicate***.
3. Enter **6** in the *Number Per Row* data field.
4. Click the *Step Length* option.
5. Click in the *X Length* data field.
6. Move the pointer into the drawing area and drag across the widest point of the step.
7. Click at the end of the entry in the *X Length* data field.
8. Type ***2** to multiply the width of the geometry by two.
9. Click OK.



The geometry duplicates in a linear array.

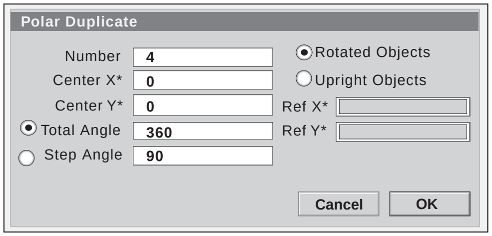
Polar Duplicate Command

This command in the Edit menu allows you to copy and rotate the selected geometry. When you duplicate an object in a circular array, you can specify the number of duplications, the center of the array and whether the objects are copied in a rotated orientation or an upright orientation.



Rotated Objects   Upright Objects

When you select this command, the dialog box appears.



The Polar Duplicate 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.

The Center X and Center Y can be set by entering values, or by clicking the mouse in the drawing area.

Total Angle
The total number of degrees between the center of the selected object and the center of the last copy.

Step Angle
The number of degrees 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. Each single arbitrary point of all copies have the same distance to the center of the circular array. That means that any point of the object can be the reference point.

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 duplicated around the center, as specified. The selected objects are reproduced in the same position relative to each imaginary point that is duplicated. With the option Rotated Objects the reference point has a constant distance to the center of the circular array.

You can specify the Ref X and Ref Y by entering a value or by clicking the mouse in the drawing area.

Using Polar Duplicate

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

   A dialog box appears.

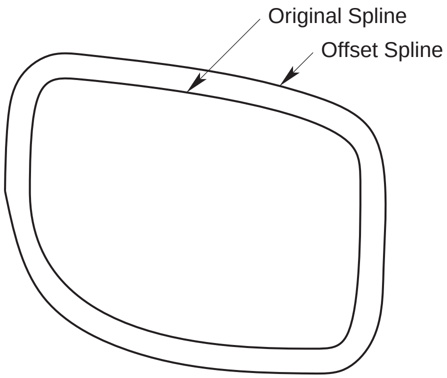
   An asterisk **(***) appearing beside an entry field indicates that you can fill in the data field by dragging the mouse in the drawing area to indicate the length. This is a very handy and accurate way to specify this information.
3. Enter the *Number* of objects in the circular array.
4. Click the *Center X** data field.
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* appear in the data fields.
6. Specify *Upright* or *Rotated* objects by clicking a button. See the next section for the definition of these terms.
7. If you specified *Upright*, click a location for the reference point in the drawing area.
8. If you do not want the copies in a complete circle, click the *Total* or *Step Angle* option.
9. Enter a value for the specified angle type.
10. Click OK.

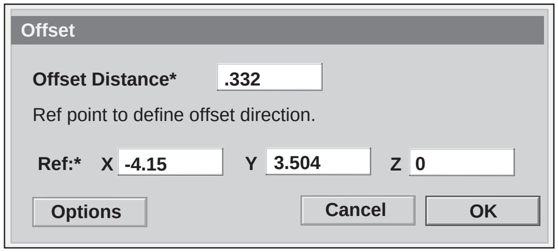
If you want to place a selected object three times in a semi-circle, you have to enter 3 for the Number and 90° for the Step Angle.

Offset Command

This command in the Edit menu allows you to create offsets from lines, arcs, circles, ellipses and splines.



When you select this command, the Offset dialog box appears.



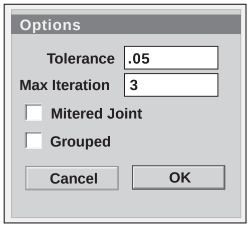
The dialog box contains the following settings:

Offset Distance*

The offset distance is the distance away from the original geometry that the new geometry is created. The asterisk indicates that you may use the cursor to input an offset distance by dragging between any two points on the drawing area. The cursor must be in the *Offset Distance* field while dragging. For multiple offsets additional values may be entered manually, separated by semicolons.

The X, Y, or Z direction values can be filled in manually or indicated by a mouse click on the drawing area offset from the object. The point clicked determines which side of the object the offset should be performed. The offset direction (positive or negative value) is also filled in the respective entry fields when you drag the offset distance since the vector you drag determines the Offset direction automatically.

Clicking the Options button displays this dialog box.



The Options dialog box contains the following settings:

Tolerance For Spline and Ellipse objects the offset distance is approximated by a spline. By setting a *Tolerance* for this type of object you can determine how accurate the offset is relative to its theoretical position.

Max Iteration Graphite iterates when calculating the offset of *Spline* and *Ellipse* objects to improve the approximation the number of times indicated in the *Max Iteration* field. The default value is 3. Setting a high value (100, 1000, etc.) decreases performance.

Mitered Joint If you mark this option, the offset object will have square external corners instead of filleted corners. This option is, by default, not selected. The radius of the filleted corners is automatically set to the offset distance. For instance: a box that is offset by .25 has a fillet at each corner with a radius of .25.

Grouped If this option is marked, the new offset geometry will be grouped when created. This option is, by default, not selected to produce ungrouped geometry.

Creating an Offset

1. Select the object you want to offset.

2. Choose ***Edit>Offset***.

   The Offset dialog box displays.

3. Click into the *Offset Distance** field.
4. Input an offset distance by dragging between any two points on the drawing area.
5. Click in one of the *Ref** entry fields.
6. Enter an offset direction by dragging between any two points on the drawing area.
7. Click the Options button.

   In the Options dialog box, specify a *Tolerance*, if needed, and whether or not the object will be offset with mitered corners.

   The new geometry ungroups, and the original geometry remains selected.

For Graphite, if an offset is created with the work plane not set to match the plane of the object being offset, the results may be unpredictable. The *Z** value is ignored in the Offset dialog box, and the distance is applied in the plane of the object being offset.

Changing the Characteristics of Objects

You can edit selected objects either by changing individual characteristics, such as weight or color, from the menu, or by changing the specifications in the Edit Objects dialog box.

The Edit Objects dialog box also provides information about the selected object in addition to allowing you to make changes.

Edit Objects Command - CTRL+I (Windows); ⌘+I (Macintosh)

This command in the Edit menu allows you to edit selected objects by changing individual characteristics, such as weight, layer, or pen style, or other specifications. Changes made through this dialog box can be reversed with the *Undo* and *Redo* commands.

Editing Objects

Tip:

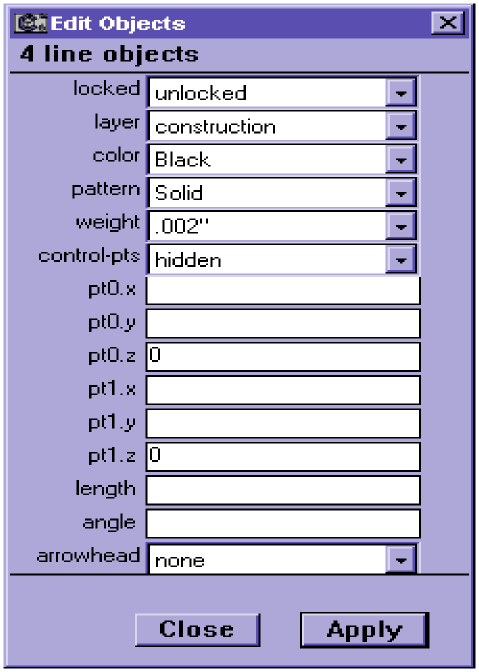
You can change a single characteristic faster by simply selecting the object and changing the characteristic from the menu. But if you are changing several characteristics or any measurements or positions, you should use the *Edit Objects* command.

1. Select the object to be edited.
2. Choose ***Edit>Edit Objects***.

   The dialog box appears.
3. Change the information in the data fields.

   Double-click the data field and type a new entry (do not press ENTER (Windows) or RETURN (Macintosh).

   Many data fields have pop-up menus for selection. Press the down arrow and then drag to the selection you want.
4. Make any other changes you want.
5. Click Apply.
6. Double-click the Control button (Windows) or single click the Close button (Macintosh) to close the Edit Objects dialog box.



You can use *Undo* and *Redo* to reverse changes made through this dialog box.

Tip:

You can use the **Expand/Shrink** Transformation tool as well as the *Edit Objects* command to edit the measurements of existing objects. This dialog box is a fast, easy way to make several changes at once.

The specifications 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
- Absolute coordinates for the starting point and ending point of the object

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.

Tech Note:

In the Edit Objects dialog box you can use mathematical operators. A list of all allowed operators is in Appendix A.

If you are 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.

If more than one object is selected when you choose *Edit Objects*, only the common information is displayed. Entries are blank when the information isn't common. For example, if you select two concentric circles, the center will be displayed in the Edit Objects box but the diameter box will be blank.

Tech Note:

You can use different units in each data field like inch (''), feet('), feet/inches (x'y''), centimeter (cm) millimeter (mm) or meter (m). You are also allowed to mix English and metric units as long as they are labeled correctly like 10'' + 25.4 cm.

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 change the text entry in the Edit Objects dialog box to **2**, all the dimensions will display a 2.

Moving a Object to a Different Layer

1. Select the object(s) to be moved.
2. Choose ***Edit>Edit Objects***.
3. Click the arrow for the Layer data field.
4. Click the layer from the list.
5. Choose Apply.

Arranging Geometry

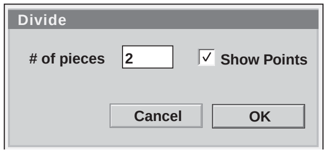
The Arrange menu contains some commands that enable you to subdivide geometry, group geometry so it behaves like a unit and lock geometry to prevent changes. This menu also contains a command to refresh the screen after you have changed geometry.

Divide Command

This command in the Arrange menu subdivides the selected geometry into the specified number of equal parts.

1. Select the object to be divided.
2. Choose ***Arrange>Divide***.

   The dialog box appears.
3. Enter the number of equal parts that you want to divide the selection.
4. Click OK.



You can see the divisions when you display the points by clicking **Show Points** in this dialog box or choosing **Layout>Show Points**.

Group Command - CTRL+Y (Windows); ⌘+Y (Macintosh)

This command in the Arrange menu combines selected objects to function as a single object.

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

Once you group geometry, you can't 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.

You can also combine groups. For hierarchical groupings, Graphite ungroups each group in the order in which they were combined.

To create temporary groups, which is useful when moving multiple objects, you can drag a selection fence around several objects to treat them as a single unit while they are selected.

Changing the Members of a Group without Adding a Hierarchical Level

When you want to add new geometry to an existing group, and you select the group and the new geometry and choose the **Group** command, you end up with a group within a group. If you want to make a single group, follow these directions:

1. Select the group.
2. Choose ***Arrange>Ungroup***. The geometry ungroups with the individual objects selected.
3. Hold down the SHIFT key and select the geometry you want to add to the group.

4. Choose ***Arrange>Group***.

You can follow a similar procedure to remove members of a group.

Ungroup Command

This command in the Arrange menu separates selected grouped objects into their individual components.

1. Select the group.
2. Choose ***Arrange>Ungroup.***

The individual objects in a group are selected.

Lock Command

This command in the Arrange menu prohibits editing or movement of selected geometry.

Preventing Accidental Changes to One or More Objects

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

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

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

Preventing Changes to an Entire Document

1. Choose ***Edit>Select All***.
2. Choose ***Arrange>Lock***.

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

Unlock Command

This command in the Arrange menu removes the lock in the selection. You can change and move the unlocked objects.

1. Select the locked object(s).

2. Choose ***Arrange>Unlock***.

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

Redraw Screen Command- CTRL+R (Windows); ⌘+R (Macintosh)

This command in the Arrange menu refreshes the screen. When you make changes to your constructions, the geometry may not be redrawn cleanly in the drawing area.

Redrawing the Screen

To redraw all of the geometry and remove extraneous geometry, choose the *Redraw Screen* command from the Arrange menu.

Stopping a Screen Refresh

Windows: You can press the ESC or BREAK key to stop the redrawing of the screen. For interrupting long operations such as redraw or linear or polar duplicate, it is recommended that you use the BREAK key. If the operation was initiated by a control key command (such as CRTL+R for redraw) the ESC key is read by MS-Windows and it brings up a task list at the end of the operation.

Macintosh: You can press ESC or the Command (⌘) key to stop the redrawing of the screen.

Adding Details

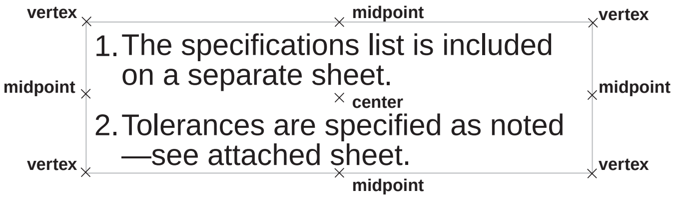
Once you have created your basic geometry, you may want to include notations or highlight certain portions of your geometry. This chapter discusses a few ways to do so. The topics covered include:

- Text
- Autotext, Text Blocks and Forms (title blocks)
- Crosshatching and Solid Fills
- Arrows

Text

When you want to annotate your drawing, use the **Text** tool to create a text box for entering characters from the keyboard. You can create, import and edit text on screen. You can use the Text menu to set the characteristics, such as font, text size, style, alignment, line spacing and indent.

You have nine text handles for each text object. These will help you move and/or align your text as you want.



Text Tool

This tool, the eight tool of the main tool palette, allows you to create and edit text on screen. The text you enter has the characteristics set in the Text menu.

Using the Text Tool

1. Select the **Text** tool from the tool palette. The Message Line reads, *Text: Pick opposite corners of text entry box [Ctrl = Copy Previous* (Windows) or *Option = Copy Previous* (Macintosh)*]*.
2. Create a text box by dragging or by clicking two locations to place the opposite corners of the text area.

   The text box is as tall as a single line of text and as wide as you indicate. The text cursor appears in the box, ready for your typing.

3. Enter the desired text from the keyboard.

   Pressing the BASKSPACE (Windows) or the DELETE (Macintosh) key erases characters to the left of the cursor.

   The text you enter automatically *word wraps* to the next line when it reaches the right side of the text box you created in step #2. If you resize the text box, the words rewrap automatically.

The Status Line allows you to enter the X and Y location of the first text box corner, the text box width and height.

X [ ] Y [ ] W [ ] H [ ]

Importing a Text File

Tech Note:

An ASCII text file is usually created by word processing software and saved as plain text. It usually contains only standard keyboard characters—letters, numbers, punctuation, and spaces—without the formatting typically found in word processing documents.

You can import an ASCII file that contains text you want to use in your drawing. For example, you can place specifications you've written with a word processor on your drawing without retyping it. The entire file imports, so if you want to use only part of the document, create a new file that contains only the information you want entered on the drawing.

1. Select the **Text** tool from the tool palette.
2. Create a text box by dragging or clicking twice to place the opposite corners of the text area.
3. Choose ***File>Import***.

   A dialog box appears.

4. Select the text file you want to import.

The Import dialog box appears and the Text file type is specified.

5. Click OK.

   The text file appears in the text box you created.

Editing Text

1. Select the **Text** tool from the tool palette.
2. Move the pointer directly over the text to be edited. The pointer turns into an *I-beam* text cursor when it is over text. (This only occurs when the **Text** tool is active.)
3. Use the text cursor to select text.

   You can drag either direction to select characters and double-click on an individual word to select the word.

   You may also simply click to place the text cursor within the existing text if you want to add more to it.
4. Modify the selected text by choosing the appropriate commands from the Text menu.

Tip:

If you want to change characteristics for the entire text entry, use the **Selection** tool to select the text block.

Changing the Characteristics of a Block of Text

You can change the font, text size, style, spacing and indentation of a block of text.

1. Use the **Selection** tool to select the block of text to be changed.
2. Select the new characteristic from the Text menu.

   The selection shows the new characteristic.

Tip:

If you want to change the characteristics of a word or portion of a text block, use the **Text** tool to select what you want to change.

Changing the Size of the Text Box

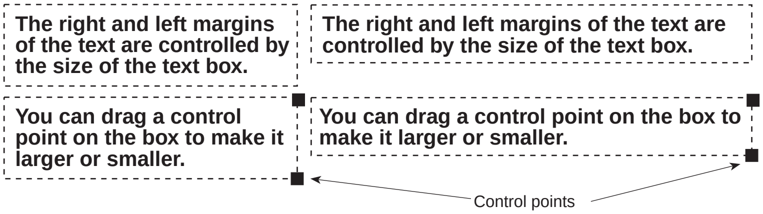
1. Click the **Selection** tool.
2. Drag a selection fence around the right side of the text box.

   The dotted lines shown below do not appear on your screen.

   Note: For single lines of text, if you are unable to select the control points on the right side of the text box, it may be that your original text box extends further than the edge of the text. With the **Selection** tool, first select the text by clicking on it. When the text box appears, drag a selection fence around the right side.

   The control points mark the boundaries of the text box when you first created it.
3. Drag the control points to the right as shown below or to the left if you want to make the text box smaller.

The area changes size and the text rewraps automatically.



In the Edit menu, *Selectable Points* must be active for selecting the control points of the text box.

Moving Text

1. Choose the **Selection** tool.

2. Select the text object you want to move.
3. Move the cursor over the text object. The cursor becomes a 4-way Move symbol.

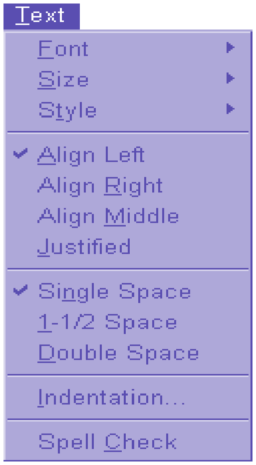
4. Place the 4-way Move symbol at the desired handle location using the Drafting Assistant notations as a guide.
5. Drag the text object to the new location.

Spell Check

Spell Check any text item in a drawing by using this command. To use the spell check feature you must first select the text to be checked, then go to ***Text>Spell Check***. The spell check dialog box operates the same as any other spelling checker. You must have the required files in the directory named "Spell" of your Graphite application folder. The user requested additions are compiled in the userdic.tlx file.

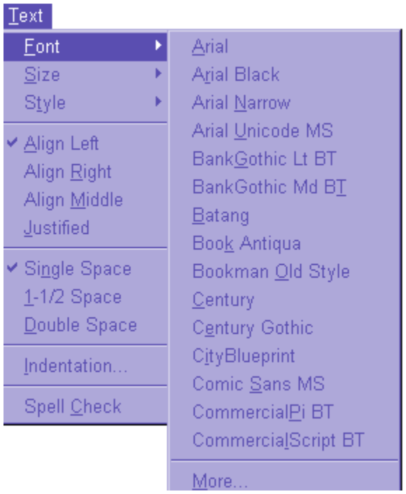
Text Menu

The Text menu contains commands for changing the text font, size, and style, as well as the justification, spacing and indentation.



Font

This command in the Text menu changes the font for selected text or future text entries.



The fonts listed in the menu are those installed on your computer and include Ashlar's Plotter fonts. Use a Plotter font whenever you are creating text for a drawing that you intend to send to a plotter, since plotting Postscript or TrueType fonts take much more time to plot or may be substituted.

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

Note: The font used for dimensions are the same set of fonts. However, they are set in the Dimension menu.

Special characters and accents are available as described in Appendix B.

Extended Font Selection (Windows)

The Font submenu can display up to 20 fonts. If you have installed more than 20 fonts select *More* in the Font submenu of the Text menu for more available fonts.

Choosing *More* brings up a dialog box that allows you to specify the font, text size and text style. The text size is measured as point size. If you want to specify

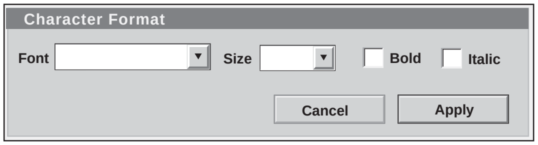
Tech Note:

Macintosh users: All fonts available in your system are displayed in the fonts list so there is no Extended font selection process.

the size in the units you have set for your drawing, you have to type the exact unit such as " for inch or mm for millimeter after the value.

1. Choose ***Text>Font>More***.

   The Character Format dialog box displays:



2. Select the character format. The text size is always measured as a point size (pt). If you want to specify the size in another unit like in millimeters, include mm with your numeric entry.
3. Click Apply.

You can leave this dialog box open to assign other character formats. If you want to close this dialog box, double click the Control pull-down menu in the upper left corner of the dialog box.

The character format you specify stays in effect until you choose another character format.

Rotating Text

Text, including Postscript and TrueType fonts, can be rotated with the **Rotate** tool like any other object in your drawing. Text that is neither horizontal nor vertical may appear jagged on the screen, but it should print out at your system's highest quality.

Font Sizes and ANSI Standards

A typical system font specified for .156 inches may produce a character that is only .125 inches. The variation depends on the proportional spacing of the font. If you want a font set to exact specifications, use a Plotter font, which conforms to ANSI standards.

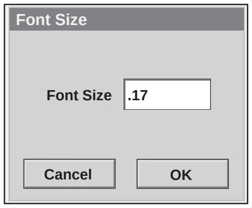
Size

This command in the Text menu sets the font size for the selected text or for future text entries in the current document. This can be specified as either points (12 pt) or units (.156 inches) as defined in the Units dialog box within ***Layout> Preferences>Units***. If no unit is specified, Graphite picks the default unit.

Specifying a Non-standard Text Size

Choosing *Other* from the Size submenu brings up a dialog box allowing you to specify the exact text size you desire in whatever units you've set for your drawing. You can specify point size by entering pt after the value in the entry box; for example, if you enter **11 pt** the font size will be 11 point.

1. Choose ***Text>Size>Other***.
2. Enter the size you want. The size is measured in the current units as set in the Preferences submenu. If you want to specify a point size, include pt with your numeric entry.



3. Click OK.

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

Text Size and Scaling

If you have made or will make changes to the scale in the Drawing Size dialog box (in the Layout menu), you must change the text size inversely. For example, if you scaled your drawing 1:4, you must multiply the text size by 4 for it to display properly on your drawing. You could also use the *Keep Text Size* and *Keep Dimension Text Size* options from the Drawing Size dialog box when changing the Drawing Scale to change the size of the text relative to the size of the other objects.

If it looks right on the screen, the scaling is appropriate. See Chapter 15, "Graphite Documents," for more information on scaling.

Style

This command in the Text menu sets the style (such as **Bold** or Underline) for selected text or future text entries.

The style you specify stays in effect until you choose another style. A check (✔) indicates the current style. You can change the default setting by saving the changes in the preferences file (*prefs.vc6* - Windows or *Graphite prefs* - Macintosh). Choose ***Layout>Preferences>Save Preferences***.

Text Alignment

The four commands in the second section of the Text menu align your text within the text box. The text can be aligned on the left, the right, centered in the middle, or it can be fully justified so that it is aligned with both left and right margins of the text box.

If the **Text** tool is active, both the selected text and future text are aligned as specified, but existing text is not aligned. If you choose this command when a tool other than the **Text** tool is active, only future text entries are aligned. If a text entry is selected when you choose this command, the selection and future text entries are aligned as specified.

Align Left

This command in the Text menu aligns the selected and future text at the left margin of the text area.

Text alignment is controlled from the *Text* menu. You can align text with the left or the right margins, with both margins, or centered between the margins.

Align Middle

This command in the Text menu centers the selected and future text in the text area.

Text alignment is controlled from the *Text* menu. You can align text with the left or the right margins, with both margins, or centered between the margins.

Align Right

This command in the Text menu aligns the selected and future text at the right margin of the text area.

> **Text alignment is controlled from the *Text* menu. You can align text with the left or the right margins, with both margins, or centered between the margins.**

Justified

This command in the Text menu aligns both the left and right margins of the selected and future text in the text area.

> **Text alignment is controlled from the *Text* menu. You can align text with the left or the right margins, with both margins, or centered between the margins.**

Text Spacing

The Text menu has commands for single spacing, double spacing, or using a space and a half between the lines of your text.

The check (✔) in the Text menu indicates the current line spacing. You can change the default setting by saving changes in the Preferences file.

> **Text spacing is controlled from the *Text* menu.**
>
> **Text can be single-spaced, as in the first paragraph, or double-spaced like in this paragraph.**
>
> **Text can also be spaced at one and a half spaces, as in this paragraph.**

Single Space

This command in the Text menu sets the spacing of selected and future lines of text so each line occupies one space in the text area.

1-1/2 Space

This command in the Text menu changes the spacing of selected and future text so each line occupies one and a half spaces in the text area.

Double-Space

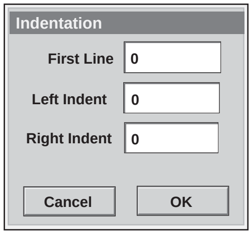
This command in the Text menu sets the spacing of selected and future text so each line occupies two spaces in the text area.

Indenting Text

If you wish to indent the text on the right or left side of the text box, or if you wish to have the first line indented, choose ***Text>Indentation***.

Indentation Command

This command in the Text menu sets the indentation of paragraphs for the selected text area as defined by the units set in the Preferences submenu.



1. Choose ***Text>Indentation***.

   The dialog box appears.

2. Specify the indentation you want. You can use point values if you include pt in your entry.
3. Click OK.

You can specify the number of units for any or all the choices:

First Line Sets the number of units for the indentation of the first line of each paragraph.

This is an example of a paragraph with the first line indented.

Left Indent Sets the number of units for the indentation of the left margin of each paragraph.

2. This is an example of a paragraph with a left indent.

Right Indent Sets the number of units for the indentation of the right margin of each paragraph.

This is an example of a paragraph with a right indentation.

Plotter Fonts

You can use all *TrueType* and *PostScript* fonts that are installed on your computer. (SHX fonts are not supported on the Macintosh platform.)

In addition, Graphite offers the following Plotter fonts.

Plotter

ABCDEFGHIJKLMNOPQRSTUVWXYZ
abcdefghijklmnopqrstuvwxyz
~!@#$%^&*()_+-=\<>?,.?/:;'"{}[]
123456789

Plotter Extended

ABCDEFGHIJKLMNOPQRSTUVWXYZ
abcdefghijklmnopqrstuvwxyz
~!@#$%↑&*()_+−=\<>?,.?/:;'"{}[]
123456789

Plotter Greek

ΑΒΧΔΕΦΓΗΙϑΚΛΜΝΟΠΘΡΣΤϒ∇ΩΞΨΖ
αβχδεφγηιϑκλμνοπϑρστυϵωξψζ
~!@#$%~&*()_+−=\<>?,.?/:;'"{}[]
123456789

Plotter Roman

ABCDEFGHIJKLMNOPQRSTUVWXYZ
abcdefghijklmnopqrstuvwxyz
~!@#$%^&*()_+−=\<>?,.?/:;'"{}[]
123456789

Plotter Italic

ABCDEFGHIJKLMNOPQRSTUVWXYZ
abcdefghijklmnopqrstuvwxyz
~!@#$%^&*()_+−=\<>?,.?/:;'"{}[]
123456789

Plotter Symbol

ℵ'|||±∓× ÷=≠≡<>≦≧∝~√⊂∪⊃∩∈→↑
←↓∂∇√∫∮∞§†‡∃∏∑()[]{}{}√∫≈≃
~!@#$%~&*()_+−=\<>?,.?/:;'"{}[]
123456789

Plotter ISO

ABCDEFGHIJKLMNOPQRSTUVWXYZ
abcdefghijklmnopqrstuvwxyz
~!@#$%^&*()_+-=\<>?,.?/:;'"{}[]
123456789

Crosshatching and Solid Fills

Tech Note:

Graphite: Crosshatching appears parallel to the work plane and only in the view in which it was created. In other words, it is drawn correctly when viewed along the z-axis of the work plane.

Graphite can crosshatch and fill any enclosed area in your drawing and then automatically update the crosshatching or fill when you change the dimensions of the enclosed area. You can specify a closed area that has a hole or other cutout in it, and Graphite accurately excludes the hole area from the crosshatching.

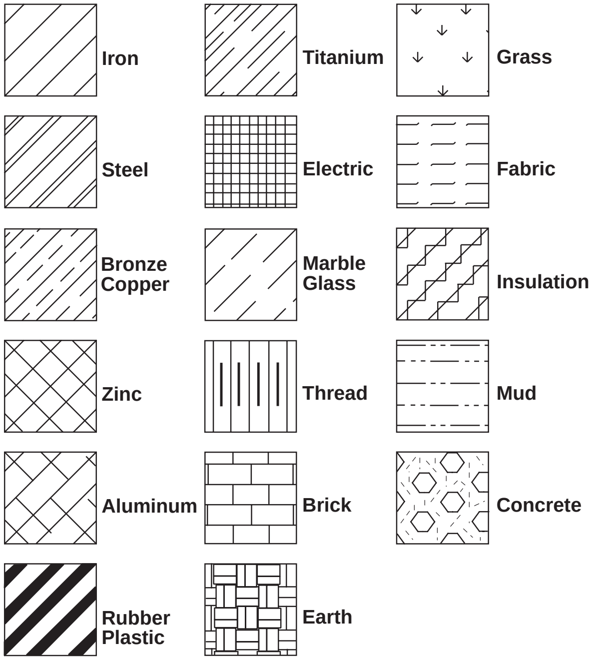
In the Pen menu you will find two fill commands—*Solid Fill* and *Fill* — and two crosshatch commands—*Crosshatch* and *Hatch*. *Fill* creates a solid fill for a selection with the current pen color and *Hatch* crosshatches the selection with the default pattern or the last pattern you had selected. *Solid Fill* lets you set the tolerance for the fill. If you want to change the hatch patterns, choose *Crosshatch* and specify the pattern you want to use. If you want to change the default hatch pattern, make changes to the preferences file as described in Chapter 5, Basic Environment Settings.

You can choose the *Fill* command from the keyboard with CTRL+B (Windows) or ⌘+B (Macintosh) and the *Hatch* command from the keyboard with CTRL+H (Windows) or ⌘+H (Macintosh).

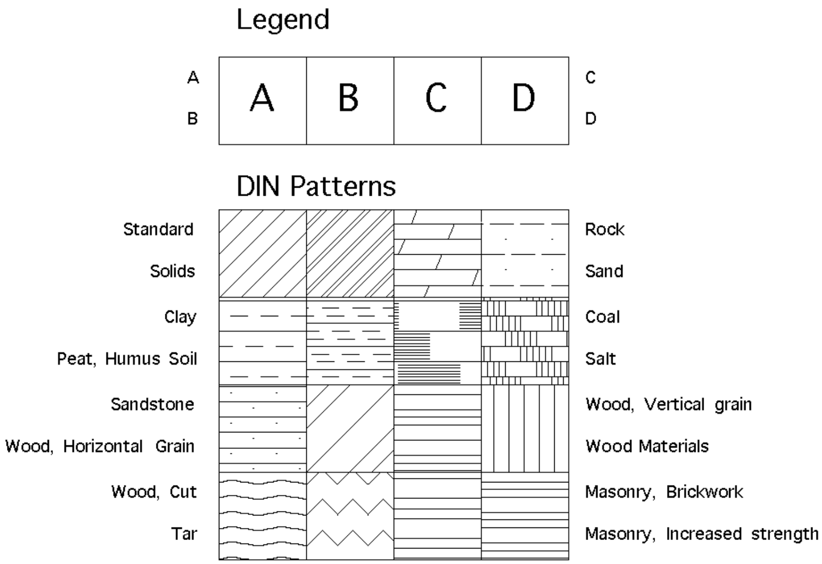
Crosshatch Patterns

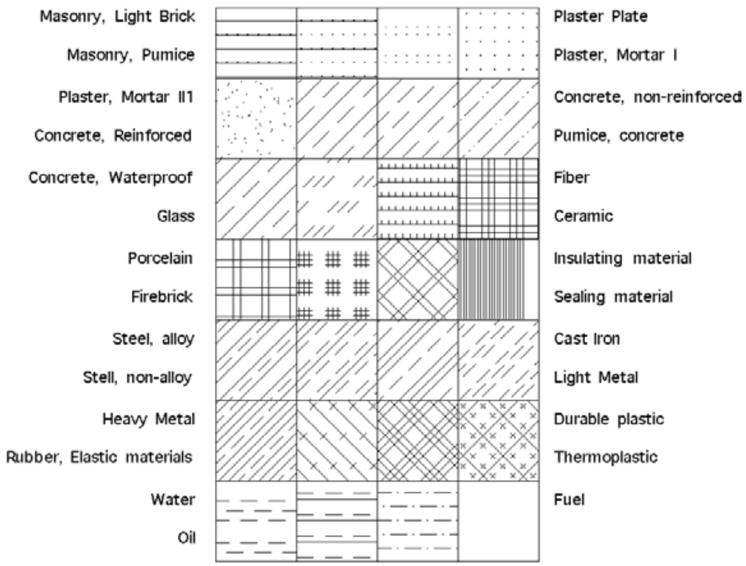
A wide variety of Crosshatch patterns are available in both ISO or DIN styles. Crosshatching appears parallel to the work plane, and only in the view in which it was created. In other words, it is drawn correctly when viewed along the z-axis of the work plane.

The following ISO crosshatching options are available.



The following DIN crosshatching options are available.



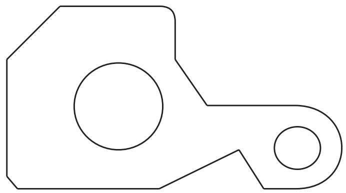


Tech Note:

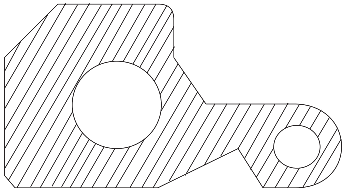
Crosshatching and Smart Walls: If you want to crosshatch smart walls containing smart symbols, you have to segment the walls first to create hatch boundaries. Since smart symbols cover only wall segments, they can't be used as hatch boundaries.

Specifying the Fill or Crosshatch Area

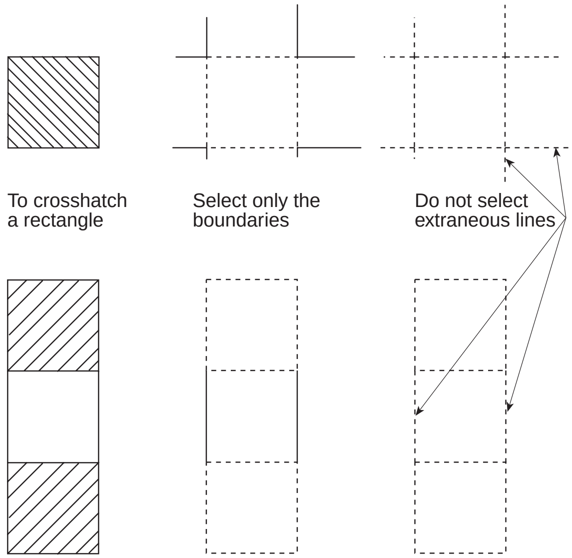
The selected objects must completely enclose the area you want to crosshatch or fill. For example, to crosshatch or fill the object below; you would select all of the lines making up the part, including the holes.



The exterior lines define the boundary for the crosshatching or solid fill. The two holes tell Graphite that those enclosed areas within the original area should not be crosshatched or filled.Once you have selected the lines and holes, in the Pen menu, you can choose the *Hatch*, *Fill*, *Crosshatch*, or the *Solid Fill* command and specify the pattern you want to use.



It is important that you do not select extra objects when you specify the crosshatch or fill area because Graphite does not know how to treat the extra objects.



Tip;

To crosshatch a figure that is not a closed figure, you must add lines or segment lines to make a closed figure. In this way, you can crosshatch individual sections of the geometry. See "Crosshatching" in the "Advanced Features" section of the *Getting Started* section of this manual for a step-by-step example.

Tip:

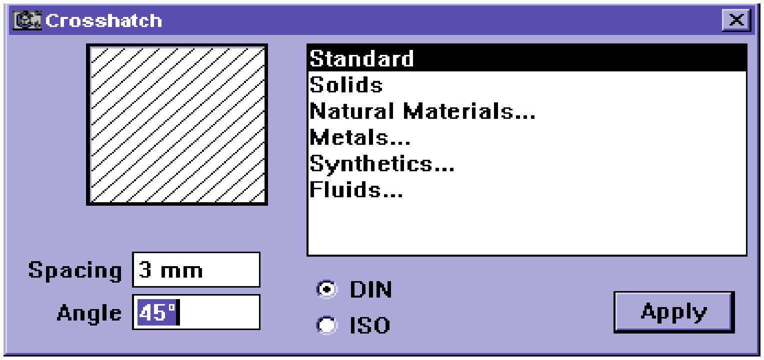
You may want to use the **Tracer** tool to select the boundary.

Crosshatch Command

This command in the Pen menu crosshatches the selection with one of the industry standard crosshatching patterns. You can change the default pattern by saving changes in the preferences file. See Chapter 5, "Basic Environment Settings."

Adding Crosshatching

1. Select the objects which represent a closed boundary. The closed boundary may include cutout areas such as a hole. Select both the outside boundary and the hole.



2. Choose ***Pen>Crosshatch***.

   The dialog box appears, displaying DIN crosshatching options.

3. To display ISO crosshatching options, click the ISO radio button in the dialog box.

Using ISO Crosshatching

1. Click the ISO radio button to display the ISO patterns. The dialog box displays.

2. Select the pattern you want from the menu.

   The pattern you select appears in the Pattern display area as it will appear on the drawing.

3. Change the spacing and angle, if desired.

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

Click Apply.

Using DIN Crosshatching

1. Select the pattern you want from the menu. Those patterns that have an ellipsis (...) following their name have their own submenu. Double-click on one of those names to display the submenu.

   To return to the main menu, double-click on the ellipsis at the top of the submenu.

The pattern you select appears in the Pattern display area as it will appear on the drawing.

2. Change the spacing and angle, if desired.

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

3. Click Apply.

Editing an Existing Crosshatch Pattern

1. Select the existing crosshatching by clicking it with the Selection tool.
2. Choose ***Pen>Crosshatch***.
3. Make the changes you want in the Crosshatch dialog box.
4. Click Apply.

The crosshatching changes as you specified.

Note: You cannot edit the line style of any specific crosshatch pattern, only adjust the spacing or angle.

Hatch Command - CTRL+H (Windows); ⌘+H (Macintosh)

This command, located in the Pen menu, 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.

The best way to select a hatch boundary is with the **Tracer** tool.

Crosshatching with the Current Pattern

1. Select the object you want to crosshatch.
2. Choose ***Pen>Hatch***.

   The selected area is crosshatched with the current pattern.

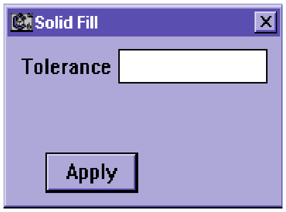
Solid Fill Command

This command in the Pen menu fills the selection with the current pen color.

A solid filled object will cover its background only if it was the last entity drawn. If you want a solid filled object to cover its background you have to select it again before printing or saving to bring it on top.

Creating a Solid Fill

1. Select the objects which represent a closed boundary. The closed boundary may include cutout areas like a hole. Select both the outside boundary and the hole.
2. Choose ***Pen>Solid Fill***. The Fill dialog box appears.
3. Enter a *Tolerance* for the fill.



   The tolerance controls how smooth the linearization is for fills in circles, arcs, ellipses and closed splines. Increasing the value of *Tolerance* makes the fill smoother. The default value is 0.01 (A value of 0.001 produces smoother edges than .01. This is also affected by Drawing Scale.)
4. Click Apply.

   The selected object fills with the current pen color.

Changing the Color of an Existing Solid Fill

1. Select the existing solid fill by clicking it with the **Selection** tool.
2. In the Pen menu, choose a new color from the Color submenu.

   The Fill color changes as you specified and will be visible as soon as you deselect the solid fill.

Changing the Tolerance of an Existing Solid Fill

1. Select the existing solid fill by clicking it with the **Selection** tool.
2. Choose ***Pen>Solid Fill***.
3. Change the *Tolerance* and press Apply.

You can also change the *Tolerance* in the Edit Objects dialog box.

Fill Command - CTRL+B (Windows); ⌘+B (Macintosh)

This command is located in the Pen menu and fills the selected objects using the current pen color. If you want to change the fill color, in the Pen menu, change the color from the Color submenu.

Fill with the Current Pen Color

1. Select the object you want to fill.
2. Choose ***Pen>Fill***.

The selected area fills with the current pen color.

Arrowheads

When you want to use arrowheads on lines or circular arcs that are not a part of dimensions, specify the placement of arrowheads in the Pen menu. You can have an arrowhead at the beginning or end of a line or circular arc, or at both the beginning and end.

Tech Note:

Arrowheads for dimensions are controlled with commands from the Dimensions menu.

Width of Arrowhead

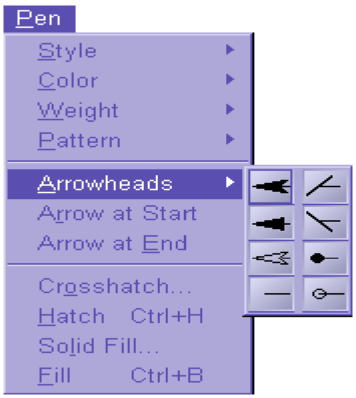
The width of the arrowhead is based on the text size.

Arrowheads Types

The Arrowheads submenu allows you to choose one of eight arrowheads for arrow lines.

Tech Note:

The size of dimension arrowheads is set using the *Arrow Size* command from the Dimensions menu.



Arrow at Start Command

This command in the Pen menu places an arrowhead at the beginning of selected and subsequent lines and circular arcs. You can choose the type of arrowhead from the Pen menu. A check (✔) indicates the current arrowhead setting. You can change the default setting by saving changes in the preferences file.

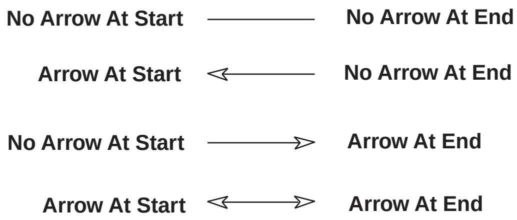
Tip:

You can also add or remove arrows to or from lines and arcs in Edit Objects.

Arrow at End Command

This command in the Pen menu places an arrowhead at the end of selected and subsequent lines and circular arcs. You can choose the type of arrowhead from the

Pen menu. A check (✔) indicates the current arrowhead setting. You can change the default setting by saving changes in the preferences file.



The start and end of a line and an arc is determined by the point that was created first.

Dimensions

This chapter explains how to use the dimension tools in Graphite. The topics are:

- Dimension Menu
- Associative Dimensions
- Using the Dimension Tools
- Dimension Appearance
- Parametric Dimensions

Dimension Menu

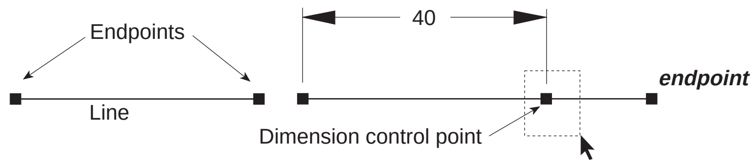
The Dimension menu contains commands for displaying the palette of dimensioning tools and for setting the format and tolerance limits for dimensions. When you are ready to start dimensioning, choose ***Dimension>Show Palette*** to display the dimension palette beside the tool palette. The dimension palette can be moved so that it doesn't obscure your work. If you want to dismiss the palette, choose ***Dimension>Hide Palette***. You select dimension tools in the same way you select tools from the tool palette. The first four tools have subpalettes which also work like the subpalettes of the tool palette.

Associative Dimensions

Graphite's geometric dimensions are associative—when you make a change to the geometry, the dimension changes also. This is not true if you have entered a manual dimension (entered a value for the **#** in the dimension text data field) in order to use parametrics.

This associativity is a tremendous time saver because dimensions automatically update whenever you make a change in the geometry. You can even change the units from English to metric (one setting in the Preferences submenu in the Layout menu) and every dimension on your drawing will reflect the change.

When you extend a line by selecting the *endpoint* of the line and dragging it to a new position the dimension changes also, because the dimension has a control point at the same position of the line *endpoint*. So when you select the *endpoint* of the line you also select the dimension control point.



Dimensions are associative relative to the points they measure. If you change the length of a line using the Edit Objects dialog box, the dimension will not update because the point was not changed. To correct the dimension, select the dimension's *vertex* point and drag it to the new *endpoint* of the line.

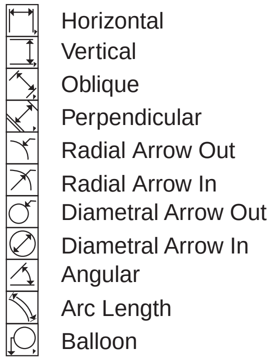
Using the Dimension Tools

Graphite's dimensioning tools are on a separate palette which you can drag around the drawing area.

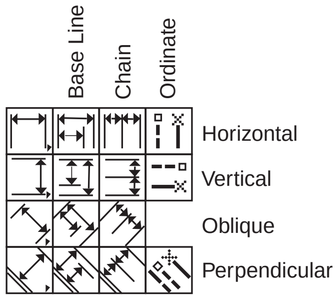
Show Palette

This command in the Dimension menu displays the dimension palette on the screen.

The dimensioning tools can be used to measure either an object or the area between objects. Once you have dimensioned an object (or space), the dimensions update when you make changes to the geometry.



The first four tools on the dimension palette have subpalettes for base line, chain, and ordinate style dimensions.



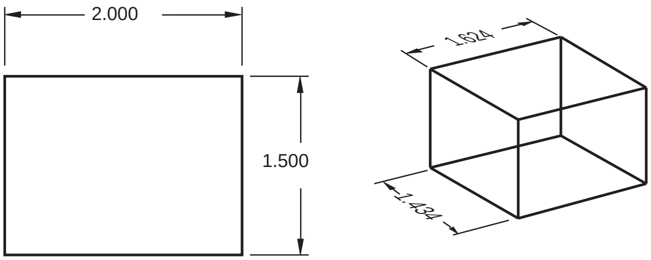
Some of the dimensioning tools, such as the **Horizontal** and **Vertical** tools, require that two points be selected; others, such as the **Radial** and **Diametral**, require only one.

You can save the location and the status (on/off) of the **Dimension** palette by choosing ***Layout>Preferences>Save Palettes***.

Dimensioning and the Work Plane

Dimensions appear in the current work plane of the active view. You should set the work plane to correspond to the view before you begin adding text or dimensions in a view.

For example, if the current work plane is the top plane, then the dimensions will appear parallel to the work plane regardless of the view. See the graphic on the next page.



All point to point dimensions are also placed parallel to the work plane. If you were working in the front plane and you were using the horizontal dimension tool, the dimension would be placed parallel to the front plane.

Dimensioning Objects and Placement

1. If necessary, display the **Dimension** palette.
2. Select the appropriate dimension tool.

   The dimension pointer is a smart pointer with a hot spot—the dot shows which side of the object to select first. If you select as indicated by the position of the dot, the text appears above or to the right of the leader. If you select in the opposite order, the text appears below or to the left of the leader.
3. Click the points you want to measure.

   Dimensions automatically use the dimension pen style and current dimension text characteristics. If you want to change the dimension pen color or weight, you can do so in the dimension menu. The dimension pattern can only be changed in the Edit Object dialog box for selected dimensions.
4. Move the dimension to a new location if its placement is not where you want it. (See the margin note.)
5. If you are planning to use the dimension with parametrics, change the entry in the text data field on the Status Line.

Tech Note:

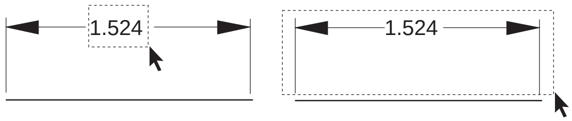
Steps #4 and #5 must be done in order. If you change the entry in the text box and then press ENTER (Windows) or RETURN (Macintosh), you can no longer move the dimension, because it will no longer be selected. If you have this problem, select the dimension and drag it to the location you want.

Moving the Dimension

When the dimension text appears, it is selected so that you can move it to a new location. Move the pointer to the dimension text. When the pointer changes to the 4-way Move symbol (as pictured below), drag the dimension to its new location.

If you want to move a dimension later, use the **Selection** tool. Click the dimension once, then drag it to the new location.

You can also drag to select the dimension text or to select the entire dimension or several dimensions at once.



Using the Dimension Status Line Fields

The status fields shown below appear when a dimensioning tool is selected.

Text

The # symbol in the text data field indicates that the dimension is the actual value of the object's measurement. If you delete this symbol, the dimension will no longer be associative (that is, it won't update if you change the geometry it measures).

If you are going to use parametrics, you must delete the # symbol and enter values, algebraic expressions, or variables in the text data field. Parametrics are described in Chapter 16.

Some of the dimensioning tools, such as the Radial tools, add a letter in the text status data field which appears in the dimension itself. You can add text before or after the # symbol. You can enter parentheses to enclose the resulting dimension in parentheses (#), or enter **# 2 Plcs** to display the dimension followed by the text 2 Plcs. When you include the # symbol, Graphite uses the measurement of the geometry as well as the text you add.

Tolerance — The Upper and Lower fields appear if you have specified a tolerance format (in the Linear or Angular submenu of the Dimension menu). You enter the limits for the tolerance in these fields.

Automatically Placing Dimensions on a Separate Layer

Graphite automatically creates a Dimension Layer unless you have removed it from the *prefs.vc6* (Windows) or the *Graphite prefs* (Macintosh) file. To create a Dimension layer, do the following:

1. Create a layer named **Dimension**. (This name must be spelled correctly.) You don't have to make the dimension layer the work layer.
2. Dimension as usual. Subsequent dimensions automatically go on the Dimension layer rather than the work layer.

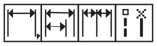
Adding Tolerance and Limits

1. From the Dimension menu, choose the desired style, either Linear or Angular.
2. Create the dimension.
3. Enter the values in the Status Line for upper and lower tolerance limits and press ENTER (Windows) or RETURN (Macintosh).

Using Fixed Dimensions

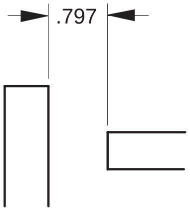
If you delete the # symbol in the text data field (on the Status Line for a newly-created dimension or in the Edit Objects dialog box) and replace it with other text, the entry in the text data field is fixed—it does not update if you change the units or the size of the geometry.

Horizontal Dimension Tools



These tools measure horizontal spaces or the distance between linear objects.

Horizontal Dimension Tool



This tool dimensions an object or space horizontally.

Using the Horizontal Dimension Tool

1. Select the **Horizontal** Dimension tool. The Message Line reads, *Horizontal: Pick first dimension point*.

2. Click the left point of the geometry.

   The Message Line now reads, *Horizontal: Pick second dimension point*.

3. Click the right point.

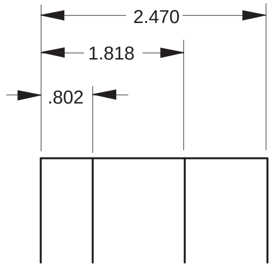
   The dimension appears. You can drag it to a new location. Dimension and extension lines automatically redraw.

   Click the points in the opposite order if you want the dimension to display below the objects.

When the dimension text appears, you can drag it to a new location; when you release the mouse button, the dimension and extension lines are redrawn.

Horizontal Base Line Dimension Tool

This tool dimensions objects or spaces horizontally from a base point.



Using the Horizontal Base Line Dimension Tool

1. Select the **Horizontal Base Line** Dimension tool. The Message Line reads, *Horizontal Base Line: Pick first dimension point*.

2. Click the base point of the geometry.

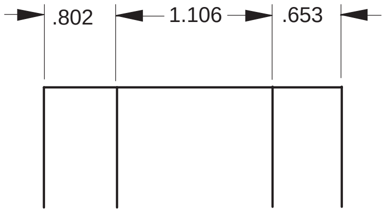
   The Message Line now reads, *Horizontal Base Line: Pick second dimension point*.

3. Click the point. The dimension appears.

4. Click the next place for the dimension. This dimension appears above the first, measured from the base point. Continue clicking the points you want dimensioned. You can drag each dimension to a new location. Dimension and extension lines automatically redraw.

   Click the points in the opposite order if you want the dimension to display below the objects.

Horizontal Chain Dimension Tool



This tool dimensions objects or spaces from end-to-end, horizontally.

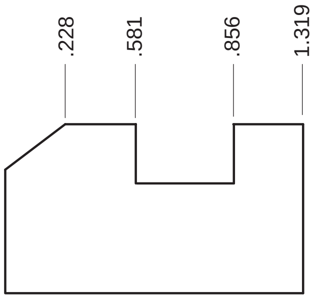
Using the Horizontal Chain Dimension Tool

1. Select the **Horizontal Chain** Dimension tool. The Message Line reads, *Horizontal Chain: Pick first dimension point*.
2. Click the first point on the geometry.

   The Message Line now reads, *Horizontal Chain: Pick second dimension point*.
3. Click the point. The dimension appears.
4. Click the next place for the dimension. This dimension appears, measured from the last point clicked. Continue clicking all the points you want dimensioned. You can drag each dimension to a new location. Dimension and extension lines automatically redraw.

   Click the points in the opposite order if you want the dimension to display below the objects.

Horizontal Ordinate Dimension Tool



This tool dimensions objects or spaces horizontally from a base point.

Using the Horizontal Ordinate Dimension Tool

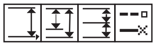
1. Select the **Horizontal Ordinate** Dimension tool. The Message Line reads, *Horizontal Ordinate: Pick first dimension point*.
2. Click the base point of the geometry.

   The Message Line now reads, *Horizontal Ordinate: Pick second dimension point*.
3. Click the next point. The dimension appears, measured from the base point.
4. Click the next point for the dimension. This dimension appears measured from the base point. Continue clicking all the points you want dimensioned. If you want to display a dimension at the base point, click the base point after you have dimensioned all other points.

You can drag each dimension to a new location. Dimension and extension lines automatically redraw.

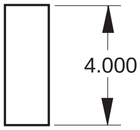
Reminder: To display the dimension number of the base point you have to click it twice: at the first and the last dimension points.

Vertical Dimension Tools



These tools measure vertical space or the distance between vertical objects.

Vertical Dimension Tool



This tool dimensions an object or space vertically. Click the top point first, then click the bottom.

Using the Vertical Dimension Tool

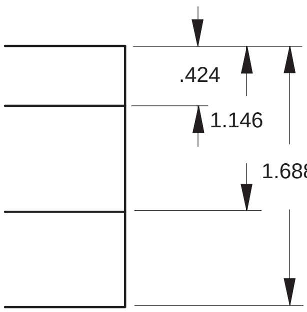
1. Select the **Vertical** Dimension tool. The Message Line reads, *Vertical: Pick first dimension point.*
2. Click the top point of the geometry first.

   The Message Line now reads, *Vertical: Pick second dimension point.*
3. Click the bottom point.

   The dimension appears. You can drag it to a new location. Dimension and extension lines automatically redraw.

   Click the points in the opposite order if you want the dimension to display below the objects.

Vertical Base Line Dimension Tool



This tool dimensions objects or space vertically from a base point.

Using the Vertical Base Line Dimension Tool

1. Select the **Vertical Base Line** Dimension tool. The Message Line reads, *Vertical Base Line: Pick first dimension point.*

2. Click the base point of the geometry.

   The Message Line now reads, *Vertical Base Line: Pick second dimension point*.

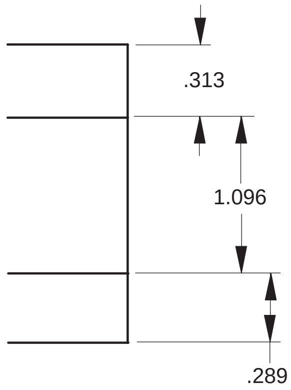
3. Click the point. The dimension appears.

4. Click the next place for the dimension. This dimension appears to the right of the first, measured from the base point. Continue clicking all the points you want dimensioned. You can drag each dimension to a new location. Dimension and extension lines automatically redraw.

   Click the points in the opposite order to display the dimension below the objects.

Vertical Chain Dimension Tool

This tool dimensions objects or space from end to end, vertically.



Using Vertical Chain Dimension Tool

1. Select the **Vertical Chain** Dimension tool. The Message Line reads, *Vertical Chain: Pick first dimension point*.

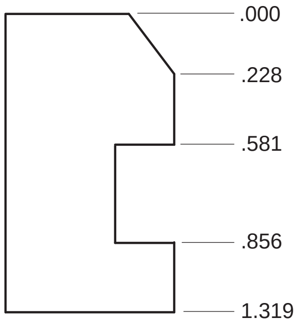
2. Click the first point on the geometry. The Message Line now reads, *Vertical Chain: Pick second dimension point*.

3. Click the point. The dimension appears.

4. Click the next place for the dimension. This dimension appears, measured from the last point clicked. Continue clicking all the points you want dimensioned. You can drag each dimension to a new location. Dimension and extension lines automatically redraw.

   Click the points in the opposite order if you want the dimension to display below the objects.

Vertical Ordinate Dimension Tool



This tool dimensions objects or space vertically from a base point.

Using the Vertical Ordinate Dimension Tool

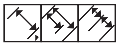
1. Select the **Horizontal Ordinate** Dimension tool. The Message Line reads, *Vertical Ordinate: Pick first dimension point*.
2. Click the base point of the geometry.

   The Message Line now reads, *Vertical Ordinate: Pick second dimension point*.
3. Click the next point. The dimension appears, measured from the base point.
4. Click the next point for the dimension. This dimension appears measured from the base point. Continue clicking all the points you want dimensioned. If you want to display a dimension at the base point, click the base point after you have dimensioned all other points.

   You can drag each dimension to a new location. Dimension and extension lines automatically redraw.

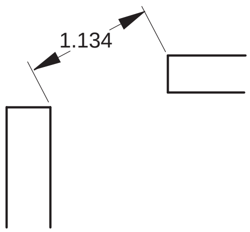
Reminder: To display the dimension number of the base point you have to click it twice: at the first and the last dimension points.

Oblique Dimension Tools



These tools measure space or objects, obliquely or point to point.

Oblique Dimension Tool



This tool dimensions an object or space from point to point or obliquely.

Using the Oblique Dimension Tool

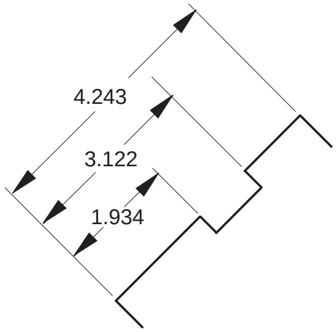
1. Select the **Oblique** Dimension tool. The Message Line reads, *Oblique: Pick first dimension point*.

2. Click the left point first. The Message Line now reads, *Oblique: Pick second dimension point.*
3. Click the right point. The dimension appears. You can drag it to a new location. Dimension and extension lines automatically redraw.

   Click the points in the opposite order to display the dimension below the objects.

Oblique Base Line Dimension Tool

This tool dimensions objects and spaces point to point or obliquely from a base point. The results shown reflect colinear dimension points.



Using the Oblique Base Line Dimension Tool

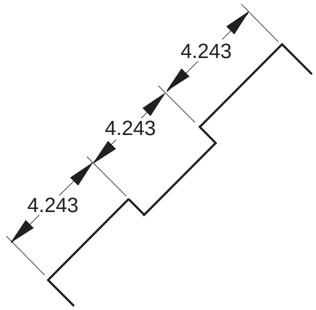
1. Select the **Oblique Base Line** Dimension tool. The Message Line reads, *Oblique Base Line: Pick first dimension point.*
2. Click the base point of the geometry.

   The Message Line now reads, *Vertical Base Line: Pick second dimension point.*
3. Click the point. The dimension appears.
4. Click the next place for the dimension. This dimension appears to the right of the first, measured from the base point. Continue clicking all the points you want dimensioned. You can drag each dimension to a new location. Dimension and extension lines automatically redraw.

   Click the points in the opposite order if you want the dimension to display below the objects.

Oblique Chain Dimension Tool

This tool dimensions objects and spaces point-to-point, end-to-end.   The results shown reflect colinear dimension points.



Using Oblique Chain Dimension Tool

1. Select the **Oblique Chain** Dimension tool. The Message Line reads, *Oblique Chain: Pick first dimension point.*

2. Click the first point on the geometry.

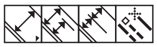
   The Message Line now reads, *Oblique Chain: Pick second dimension point*.

3. Click the point. The dimension appears.

4. Click the next place for the dimension. This dimension appears, measured from the last point clicked. Continue clicking all the points you want dimensioned. You can drag each dimension to a new location. Dimension and extension lines automatically redraw.

   Click the points in the opposite order if you want the dimension to display below the objects.

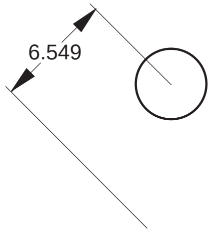
Perpendicular Dimension Tools



These tools measure an object or space perpendicular to another line. If most of your dimensions start from a line edge, you can use this tool to generate both horizontal and vertical dimensions.

For 3D geometry, perpendicular dimensions are created in the plane of the baseline.

Perpendicular Dimension Tool



This tool dimensions an object or space perpendicular to a line.

Using the Perpendicular Dimension Tool

1. Select the **Perpendicular** Dimension tool. The Message Line reads, *Perpendicular: Pick first dimension point*.

2. Click the Click the base line. Be certain that you click *on* the base line, not on the *endpoint* of the base line.

   The Message Line now reads, *Perpendicular: Pick second dimension point*.

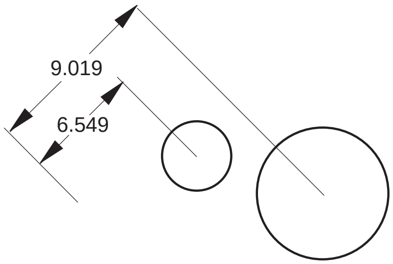
3. Click the object or location.

   The dimension appears. You can drag it to a new location. Dimension and extension lines automatically redraw.

   Click the points in the opposite order if you want the dimension to display below the objects.

Perpendicular Base Line Dimension Tool

This tool dimensions between a point or object perpendicular to an existing base line.



Using the Perpendicular Base Line Dimension Tool

1. Select the **Perpendicular Base Line** Dimension tool. The Message Line reads, *Perpendicular Base Line: Pick first dimension point*.

2. Click the base point of the geometry.

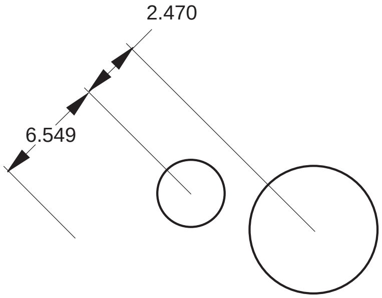
   The Message Line now reads, *Perpendicular Base Line: Pick second dimension point*.

3. Click the point. The dimension appears.
4. Click the next place for the dimension. This dimension appears above the first, measured from the base point. Continue clicking the points you want dimensioned. You can drag each dimension to a new location. Dimension and extension lines automatically redraw.

   Click the points in the opposite order if you want the dimension to display below the objects.

Perpendicular Chain Dimension Tool

This tool dimensions between points or objects perpendicular to an existing baseline.



Using the Perpendicular Chain Dimension Tool

1. Select the **Perpendicular Chain** Dimension tool. The Message Line reads, *Perpendicular Chain: Pick first dimension point*.

2. Click the first point on the geometry.

   The Message Line now reads, *Perpendicular Chain: Pick second dimension point*.

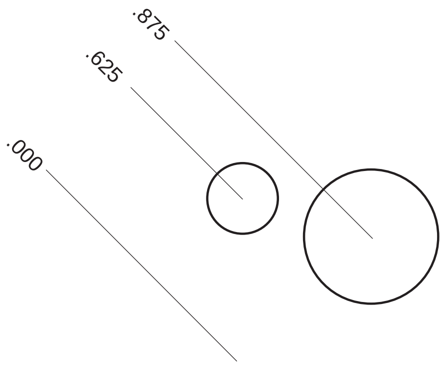
3. Click the point. The dimension appears.

4. Click the next place for the dimension. This dimension appears, measured from the last point clicked. Continue clicking all the points you want dimensioned. You can drag each dimension to a new location. Dimension and extension lines automatically redraw.

   Click the points in the opposite order if you want the dimension to display below the objects.

Perpendicular Ordinate Dimension Tool

This tool dimensions between a point or object perpendicular to an existing base line.



Using the Perpendicular Ordinate Dimension Tool

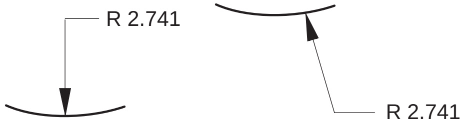
1. Select the **Perpendicular Ordinate** Dimension tool. The Message Line reads, *Perpendicular Ordinate: Pick first dimension point*.
2. Click the base point of the geometry.

   The Message Line now reads, *Perpendicular Ordinate: Pick second dimension point*.
3. Click the next point. The dimension appears, measured from the base point.
4. Click the next point for the dimension. This dimension appears measured from the base point. Continue clicking all the points you want dimensioned. If you want to display a dimension at the base point, click the base point after you have dimensioned all other points.

   You can drag each dimension to a new location. Dimension and extension lines automatically redraw.

Reminder: To display the dimension number of the base point you have to click it twice: at the first and the last dimension points.

Radial Arrow Out Dimension Tool



This tool measures the radius of a circle, arc, or fillet.

Using the Radial Arrow Out Dimension Tool

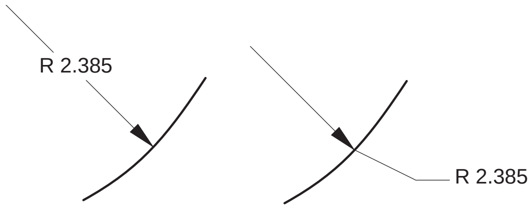
1. Select the **Radial Arrow Out** Dimension tool. The Message Line reads, *Radial Arrow Out: Select arc/circle*.

2. Click near the circle, arc or fillet you want to dimension. If you click outside it, the dimension text appears outside, and if you click inside, it appears inside.

   The dimension is placed on the outside of the selected object. When the dimension appears the leader line is placed at the nearest 15° increment from the location you clicked. You can drag the text to a new location.

For 3D geometry, radial dimensions are created in the plane of the arc or fillet.

Radial Arrow In Dimension Tool



This tool measures the radius of a circle, arc, or fillet with the arrow inside the geometry.

Using the Radial Arrow In Dimension Tool

1. Select the **Radial Arrow In** Dimension tool. The Message Line reads, *Radial Arrow In: Select arc/circle*.

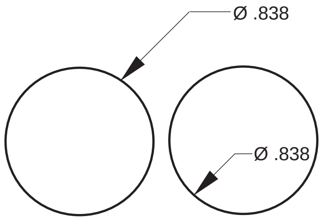
2. Click near the circle, arc or fillet you want to dimension. If you click outside it, the dimension text appears outside, and if you click inside, it appears inside. Either way the arrow line starts from the arc center.

   The dimension text appears inside the object. The arrow line starts from the arc center. When the dimension appears the leader line is placed at the nearest 15° increment from the location you clicked. You can drag the text to a new location.

For 3D geometry, radial dimensions are created in the plane of the arc or fillet.

Diametral Arrow Out Dimension Tool

This tool measures the diameter of a circle.



Using the Diametral Arrow Out Dimension Tool

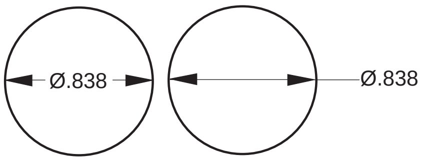
1. Select the **Diametral Arrow Out** Dimension tool. The Message Line reads, *Diametral Arrow Out: Select arc/circle.*
2. Click near the circle or arc you want to dimension. If you click outside the circle, the dimension text appears outside, and if you click inside, it appears inside (depending on the circle size and the font size).

   The dimension appears outside the object. When the dimension appears the leader line is placed at the nearest 15° increment from the location you clicked. You can drag the text to a new location.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.

For 3D geometry, diametral dimensions are created in the plane of the circle.

Diametral Arrow In Dimension Tool

This tool measures the diameter of a circle.



Using the Diameter Arrow In Dimension Tool

1. Select the **Diametral Arrow In** Dimension tool. The Message Line reads, *Diametral Arrow In: Select arc/circle.*
2. Click near the circle or arc you want to dimension. If you click outside the circle, the dimension appears outside, and if you click inside, it appears inside (depending on the circle size and the font size).

   The dimension appears inside the object (depending on the circle and font size). When the dimension leader line is placed at the nearest 15° increment from the location you clicked. You can drag the text to a new location.

For 3D geometry, diametral dimensions are created in the plane of the circle.

Angular Dimension Tool

This tool measures the angle between two lines.

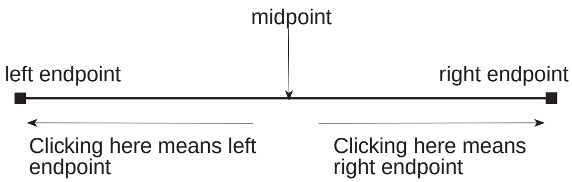
Using the Angular Dimension Tool

1. Select the **Angular** Dimension tool. The Message Line reads, *Angular: Pick first line*.
2. Click on the first line near the endpoint from which you want the angle measured.

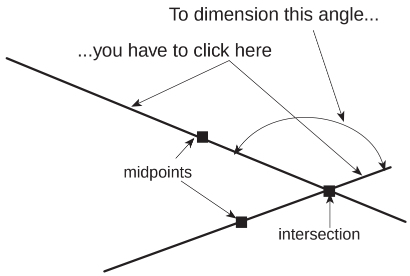
   The Message Line now reads, *Angular: Pick second line*.
3. Click near the endpoint of the second line.

   The inside angle is measured between the lines. The smaller angle is used.

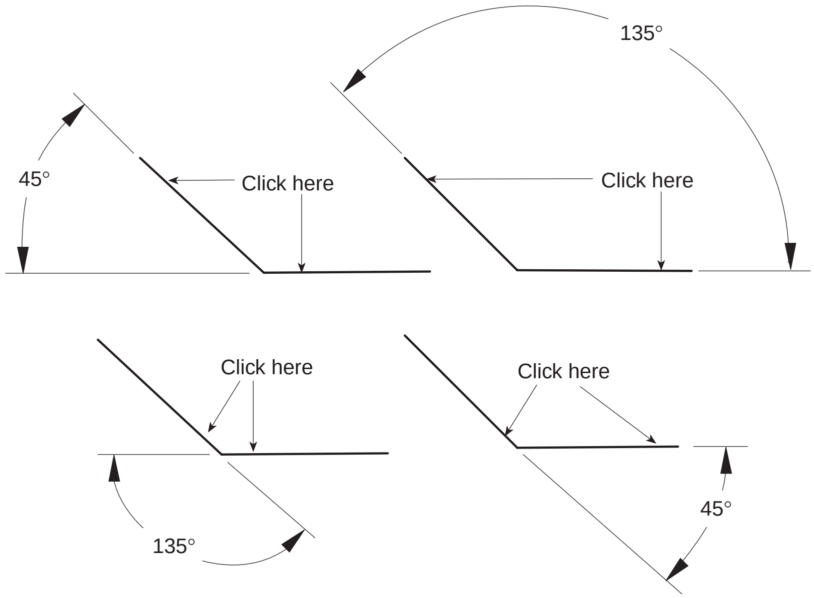
The angle between the endpoints of the lines is measured so you do not have to click these endpoints, just near them. Graphite interprets all mouse clicks left of the line midpoint as the left endpoint and all mouse clicks right of the line midpoint as the right endpoint.



If you dimension intersecting lines don't mix up the intersection with the midpoint of the lines.



If the lines don't meet, the angle is measured from the invisible extension of the lines. The extension appears as part of the dimension.



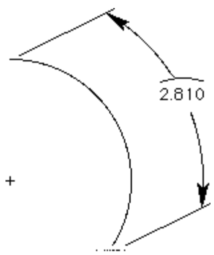
When the dimension text appears, you can drag it to a new location; when you release the mouse button, the dimension and extension lines are redrawn.

Angular dimension will display all angles properly if you keep the angles larger than 3 degrees. For smaller than that, you should build the dimensions manually using lines and text objects.

The **Angular** Dimension tool will not dimension angles over 180°.

For 3D geometry, Angular dimensions are created in the plane of the two lines.

Arc Length Dimension Tool



This tool measures the length of an arc.

Using the Arc Length Dimension Tool

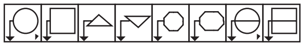
1. Select the **Angular** Dimension tool. The Message Line reads, *Arc Length: Pick start point on the arc*.
2. Select the start point. The Message Line reads, *Arc Length: Pick point on the arc*.

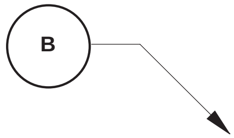
3. Select a point on the arc. The Message Line reads, *Arc Length: Pick end point on the arc*.
4. Select the opposite end of the arc (as if you were creating a three-point arc).

Like other dimensions you can manipulate the text by changing its size or font using the Text menu in the Dimension menu. You can move the text by positioning the cursor over it. The cursor changes to a four-headed arrow. Then drag the text to the desired position. If necessary, the line pointing to the dimension will dogleg.

The **Arc Length** Dimension tool does not put witness lines perpendicular to the arc for angles less than 180 degrees.

Balloon Dimension Tool





This tool attaches a Balloon symbol to geometry, placed in the location you click.

Using a Balloon Dimension Tool

1. Select the **Balloon** Dimension tool. The Message Line reads, *Balloon: Pick geometry to point to*.
2. Click the point on the geometry for the dimension arrow. The Message Line reads, *Balloon: Pick balloon location*.
3. Click the second point to specify the location of the symbol.

   The balloon symbol appears.
4. Enter the text and frame width in the Status Line data fields. Press ENTER (Windows) or RETURN (Macintosh) to update the dimension.

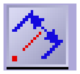
   You can move the dimension by dragging the text to a new location.

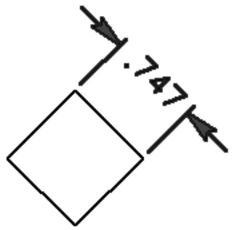
Balloon dimension text does not increment as you place additional balloons. If you want specific text in the balloon dimension, you must enter it yourself.

The Status Line contains Text and Width data fields.

Additional Tools for Dimensioning

Automatic Dimensioning Tool





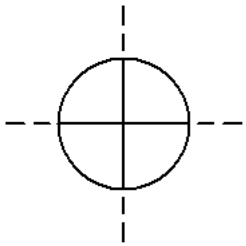
This tool automatically dimensions lines (independently from their angle), circles, ellipses and arcs.

Using the Automatic Dimensioning Tool

1. Choose ***Utilities>Show Palette***.
2. Choose the **Automatic Dimensioning** tool. The Message Line reads, *Automatic Dimensioning: Select object for Auto dimensioning*.
3. Click at each object with the **Automatic Dimensioning** tool.

   All lines are dimensioned correctly. All circles, arcs and ellipses are dimensioned with a radial dimension.

Circle Center Line Dimension Tool



This tool creates a center line for any circle you have created. It also gives you the option to display a thread symbol with the center line.

Using the Circle Center Line Dimension Tool

1. Choose ***Utilities>Show Palette***.
2. Choose the **Circle Center Line** tool. The Message Line reads, *Circle Center Line: Select circle*.
3. Click on the circle with the tool.

   If you press the CTRL (Windows) or the OPTION (Macintosh) key while clicking on the circle, a thread symbol is added.
4. In the Axis overlap data field of the Status Line, enter the desired axis overlap.
5. Press the ENTER (Windows) or the RETURN (Macintosh) key.

   A circle center line with an optional thread symbol is added to the circle with the specified overlap.

The Status Line allows you to enter the Axis overlap.

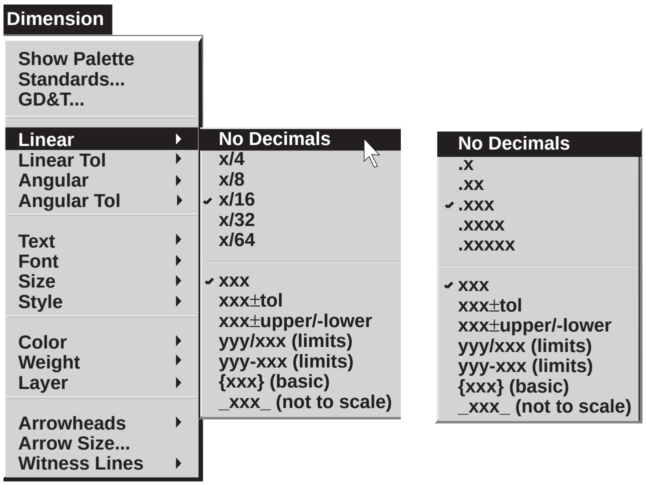
Dimension Appearance

Graphite automatically creates dimensions according to ANSI Y14.5, DIN, ISO or JIS standards. However, many companies and individuals have developed their own standards. The commands in the Dimension menu allow you to control virtually every aspect of the dimension appearance, without having to construct dimensions manually and while retaining the associativity of Graphite's *smart dimensions*.

Settings made in the Dimension menu affect the currently selected dimension and all future dimensions.

Linear

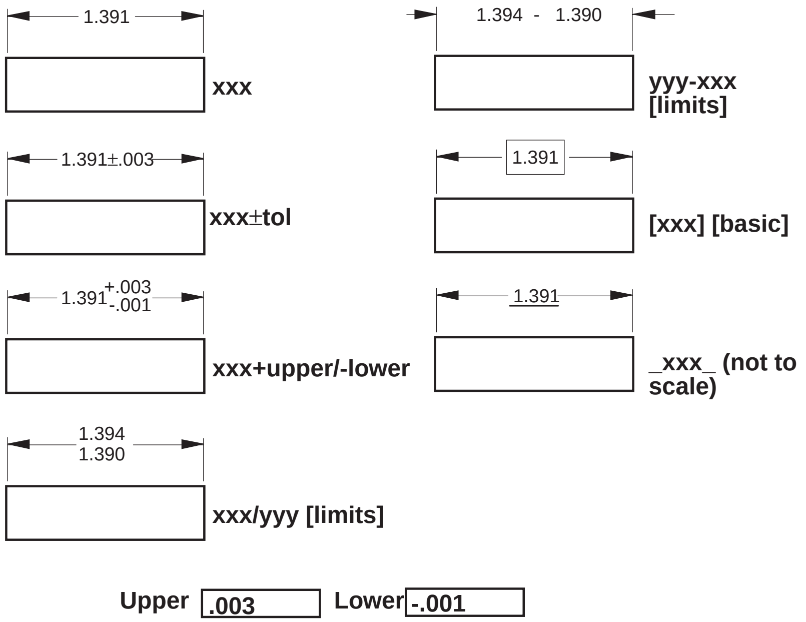
This command in the Dimension menu allows you to set the precision—either fractional or decimal—of the dimension's nominal value and the format of tolerance and limits for linear dimensions. The default dimension appearance is decimals.



The default is three decimal places (or x/16 for fractional precision) with no tolerance or limits.

Dimension Tolerance

The following graphic shows the appearance of each tolerance format if you entered the limits shown in the Status Line.



Tech Note:

Use the [xxx] (basic) format to create dimensions for GD&T symbols.

Dual Dimensioning

You can also choose to display your dimensions in both inches and millimeters (dual dimensioning).

Fractional Display or Decimal Display

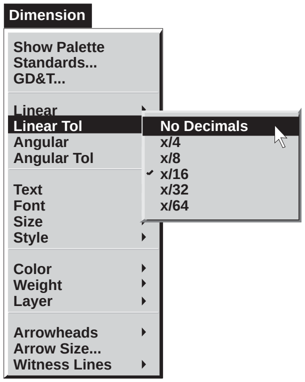
To change the current linear appearance of your dimensions, you must first change the precision setting in the Units dialog box.

Choose ***Layout>Preferences>Units***. In the Units dialog box, change the Precision to a decimal or fractional option by holding down the mouse button on the arrow to the right of the data field and scrolling to the desired precision. That precision will now be displayed in the Status Line data fields.

Graphite will still calculate out to sixteen decimal places. However, the Status Line will display the nearest fraction or decimal to the calculated value based on the precision selected.

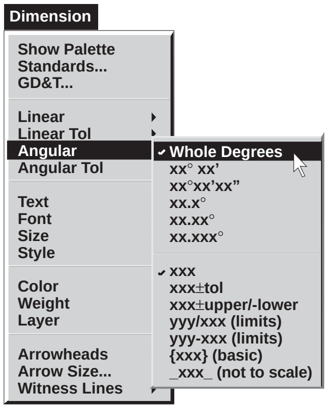
Linear Tolerance

This command in the Dimension menu allows you to set the precision of a linear dimension's tolerance values.



Angular

This command in the Dimension menu allows you to set the format for Angular dimensions.

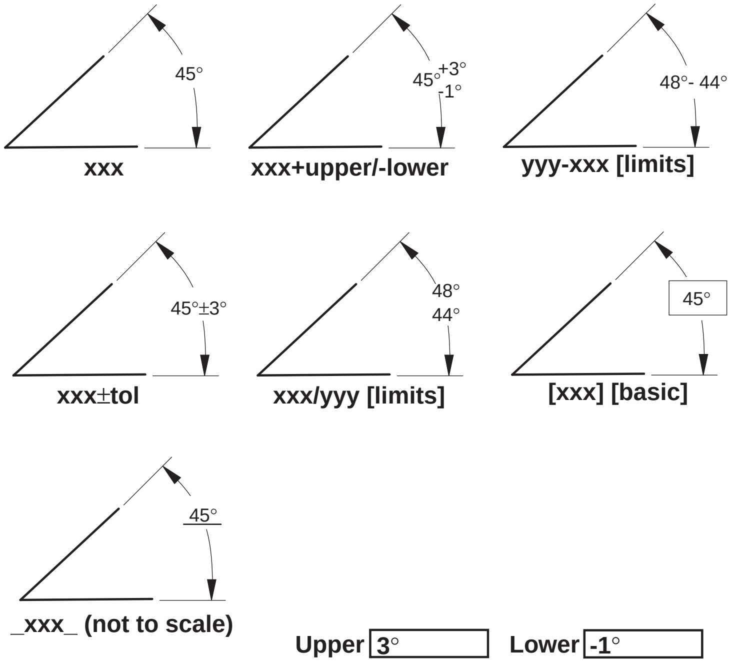


You can set degrees, minutes and seconds as well as the tolerance and limits for these dimensions.

The default format is degrees with no minutes or seconds and no tolerance or limits. The graphic here shows the appearance of each tolerance format if you entered the limits shown in the Status Line.

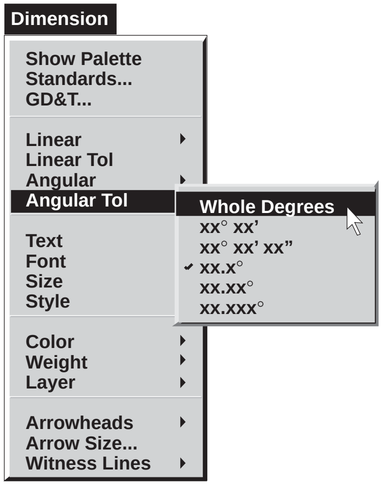
Tech Note:

Use the [xxx] (basic) format to create dimensions for GD&T symbols.



Angular Tolerance

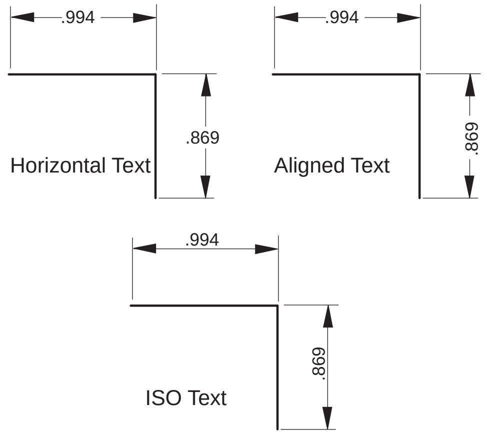
This command in the Dimension menu allows you to set the precision of an angular dimension's tolerance values.



Text

This command in the Dimension menu allows you to specify the position and orientation for dimension text.

The default is *Horizontal* text. The *Break-in*, *Over* and *Under* options produce text that is aligned with dimension leader lines. *Over* and *Under* display text above or below leader lines, while *Break-in* places the text between leader lines.



Tech Note:

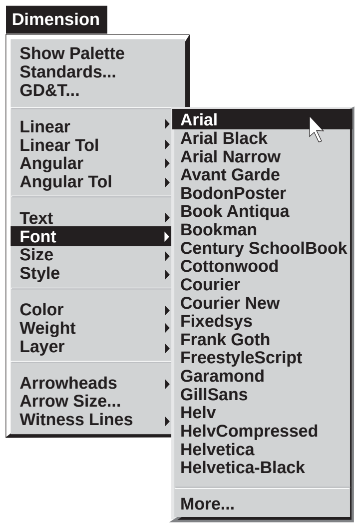
The *Font* command only affects dimensions. To change the font of other text objects, choose ***Text>Font.***

The Font submenu in the Text menu has no effect on dimensions. You must use the *Font* command from the Dimension menu to specify the font for dimension text.

Font

This command in the Dimension menu lets you pick the font for dimension text. The fonts in this list are the same as those available in the Font submenu from the Text menu.

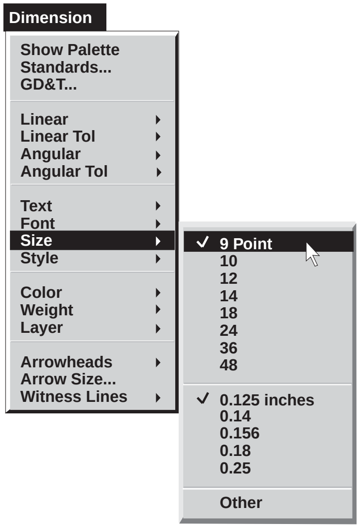
Windows: If you have more fonts in than appear in the Font submenu, you can access them by choosing the *More* command.



Size

This command in the Dimension menu lets you pick the size of dimension text only. The sizes in this list are the same as those available in the Size submenu from the Text menu.

Selecting the *Other* option allows you to specify a non-standard font size for dimension text.



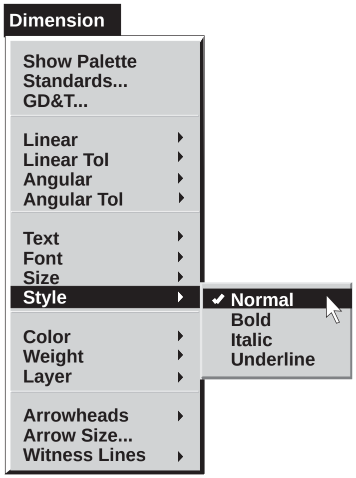
Tech Note:

The *Size* command only affects dimensions. To change the size of other text objects, use *Edit Objects* or the Size submenu from the Text menu.

The size submenu from the Text menu has no effect on dimensions.

Style

This command in the Dimension menu lets you pick the style of dimension text. The styles in this list are the same as those available in the Style submenu from the Text menu. However, these settings here, do not affect those in the Text menu.



Tech Note:

The *Style* command only affects dimensions. To change the style of other text objects, use the Style sub-menu from the Text menu.

The Style submenu from the Text menu has no effect on dimensions.

Tech Note:

The *Color* command only affects dimensions. To change the color of other objects, use *Edit Objects* or the Color submenu from the Pen menu.

The Color submenu from the Pen menu has no effect on dimensions.

Color

This command in the Dimension menu allows you to specify the color of the dimension.

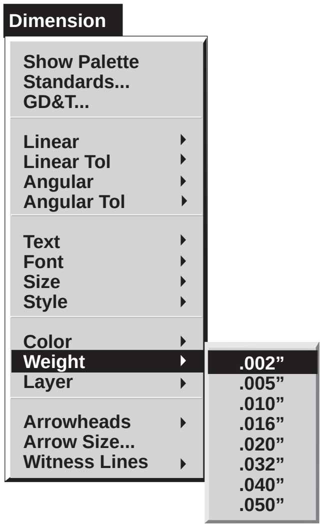
The default color is blue.

Tech Note:

The *Weight* command only affects dimensions. To change the weight of other objects, use *Edit Objects* or the Weight submenu from the Pen menu.

The Weight submenu from the Pen menu has no effect on dimensions.

The values shown in the Weight submenu may be different if the available line weights have been changed using the *Pen Weight Editor.*



Weight

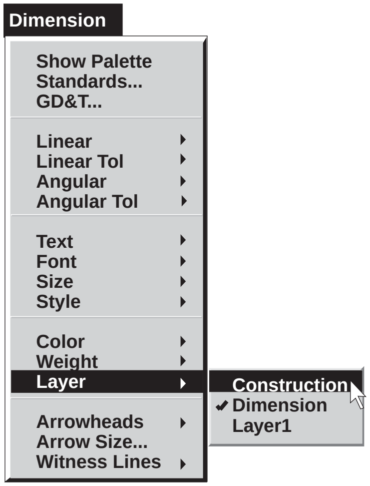
This command in the Dimension menu allows you to specify the weight of dimension lines. Dimension lines are typically drawn in the thinnest weight available. For Graphite, that is 0.002" or 0.05mm.

The default weight is 0.002" or 0.05mm.

Layer

This command in the Dimension menu allows you to specify the layer on which dimensions are placed. Dimensions can be placed on any visible layer.

The default layer is the Dimension layer. If the layer on which dimensions are to be placed is hidden or deleted, future dimensions will be placed on the current work layer.

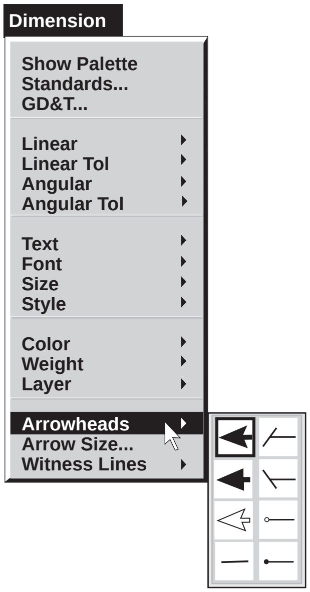


Tech Note:

The layers in your list will be different than the ones shown here if you have added layers of your own or deleted any of the default layers.

Arrowheads

This command in the Dimension menu allows you to specify the type of arrowhead used for dimensions.



Tech Note:

The *Arrowheads* command only affects dimensions. To select the arrowhead type for lines or arcs, use the Arrowheads submenu from the Pen menu.

The Arrowheads submenu from the Pen menu has no effect on dimensions.

Arrow Size Command

This command in the Dimension menu lets you specify the size of the arrowhead that you have selected.

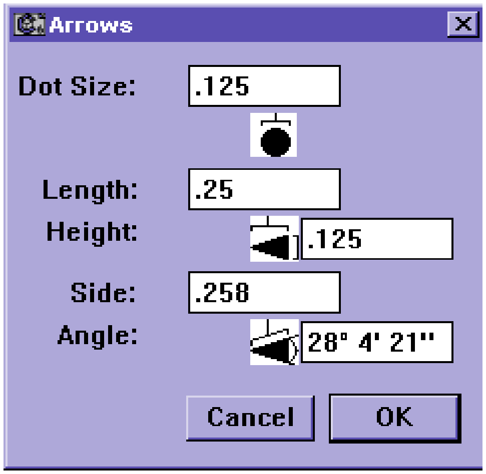
Dot Size: This value describes the diameter of circular and slash arrowheads. The value displayed is in the current units specified in the Units dialog box from the *Preferences* command.

Length: This value describes the length of the arrowhead as the horizontal distance from its tip to the furthest extension of its base.

Height: This value describes the height of the arrowhead as the vertical distance of its base.

Side: This value describes the length of the edge of the arrowhead.

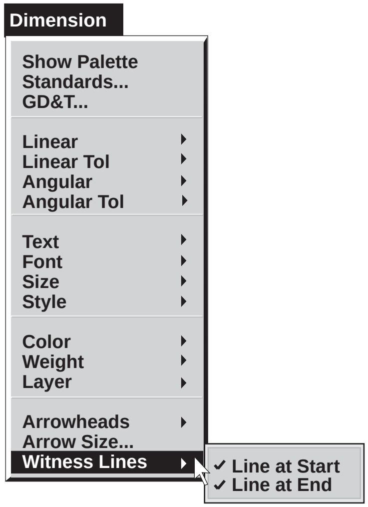
Angle: This value describes the angle of the tip of the arrowhead.



If you change any value in the *Length*, *Height*, *Side*, or *Angle* entry data fields, Graphite will change the values in the other entry data fields accordingly.

Witness Lines

This command in the Dimension menu lets you specify which sides of the linear dimension should have witness lines. A check mark indicates that a witness line will be placed at the specified location. This option is useful if you are going to plot a drawing that contains baseline or chain dimensions. By turning off one or more of the overlapping witness lines, you can prevent the plotter from drawing multiple witness lines when only one needs to be drawn.



Tip:

Witness Lines for existing linear dimensions can also be toggled on or off in *Edit Objects*.

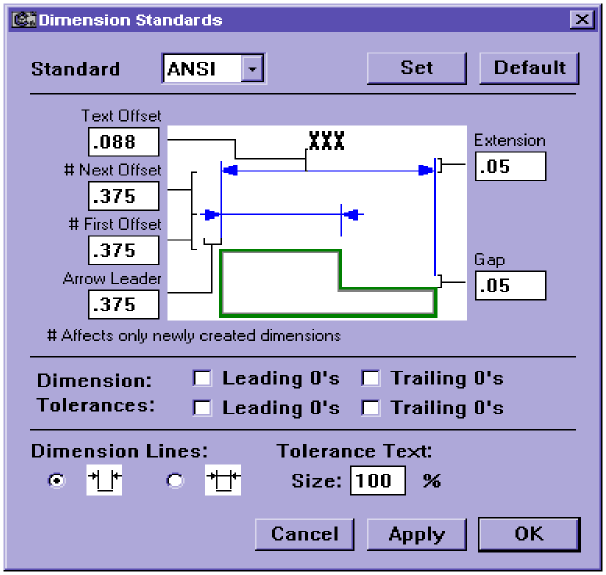
The *Start* of a dimension is the first point clicked. The *End* of a dimension is the last point clicked. The default settings have witness lines at both the *start* and *end* of a dimension.

This command affects only linear dimensions. Radial and diametral dimensions are not affected by the settings made in the *Witness Lines* command.

Dimension Standards

The Dimension Preferences dialog box can be opened by choosing ***Dimension>Standards***. This dialog box lets you pick the standard your dimensions should follow, as well as gives you the ability to define the characteristics for your own standard. Most changes made in the Dimensions Standards dialog box affect currently selected dimensions and newly created ones; changes made to the *First Offset* and the *Next Offset* affect only newly created dimensions.

Changing the standard will also change the Dimension menu settings. Setting dimension standards saves the values in the Dimension Standards dialog box as well as the settings in the Dimension menu, except for the *Witness Lines* command.



Standard

This pull-down lets you pick the standard your dimensions should follow. Graphite is supplied with settings for ANSI, DIN, ISO and JIS standards. There is an additional setting in the pull-down list, Other, where you can specify your own settings, or you can change the settings for one of the supplied standards.

The Set button maps the current settings to the standard that appears in the pull-down list. It also saves all of the settings from the Dimension menu.

The Default button reverts to the factory settings of the selected standard.

Text Offset

This entry data field provides the distance from the base of the dimension text to the leader lines. A positive entry places the text above the leader lines, and a negative value places it below the leader lines. A zero value places the text on the leader lines.

Gap

This entry data field provides the distance between the dimension point and the witness line.

Extension

This entry data field provides the distance the witness lines extend beyond the leader lines.

Arrow Leader

This entry data field provides the distance the leader lines extend beyond the witness lines when the arrow lines appear outside of the witness lines.

First Offset

This entry data field provides the perpendicular distance between object being dimensioned and the leader line and dimension text.

Changes to this value only affect newly created dimensions.

Next Offset

This entry data field provides the perpendicular distance between subsequent leader lines created in a base line dimensioning.

Changes to this value only affect newly created dimensions.

Leading 0's &Trailing 0's

Selecting these check boxes determines whether Dimension or Tolerance values display leading or trailing zeros. Neither option is selected by default.

Selecting Leading 0's places a 0 before all dimensions that are less than 1.

Selecting Trailing 0's places 0's after the decimal point. This option is affected by the precision set in ***Layout>Preferences>Units***. Example: If your precision is set at .001, a measurement of three inches will be displayed as 3.000.

These options do not apply to dimensions displayed in fractions.

Solid Leader	These radio buttons allow you to determine whether the leader lines that appear outside of the witness lines should be connected with another line. The default value does not place this line.
Tolerance Text Size %	This entry data field lets you specify the size of the tolerance text as a percentage of the nominal value. The default value is 100%—the same size as the nominal value.

Clicking OK accepts the changes made in the Dimension Standards dialog box. Clicking Apply accepts any changes made in the Dimension Standards dialog box to any currently selected dimensions without closing the dialog box. Clicking Cancel ignores any changes and closes the Dimension Standards dialog box.

Parametric Dimensions

You must dimension objects to use the parametrics feature; you must specify values, variables, or expressions for each dimension. Parametrics is discussed in detail in Chapter 16.

Creating a Parametric Dimension

1. Create the dimension as usual.

   The Status Line displays a # symbol in the text data field.

2. Enter a value (variable or algebraic expression) in the text data field.

   The dimension reflects the value or variable you entered, regardless of the actual value of the geometry you just dimensioned.

3. Continue dimensioning as needed to define the part completely.
4. Use the parametric feature as described in Chapter 16.

Converting Parametric Dimensions to Associative Dimensions

When you bring in a parameterized part with the *Import* command, the geometry is drawn as you specify, but the dimensions display the variables and expressions which defined them as symbols. You can change the dimensions to be associative so that they reflect the actual measurements of the geometry, even if you change the geometry.

1. Select the dimensions.
2. Choose ***Edit>Edit Objects***.

3. Enter a # in the text data field.
4. Click Apply and close the dialog box.

   The parametric variable changes to reflect the geometry's actual measurement.

Viewing Geometry

This chapter describes several Graphite features that allow you to view the geometry in your drawing in different ways. The following topics are covered:

- Zooming
- Layers
- View Displays and Detail Views
- Light Settings
- Perspective
- Views Menu

Zooming

Graphite gives you several ways to change the magnification of your drawing by zooming in and out using commands, zoom tools, and strokes.

Zoom Commands

Zoom In, *Zoom Out*, or *Zoom All* from the Arrange menu changes the view magnification of your geometry depending upon your selection. The *Zoom Previous* command takes you to the last magnification. If you want to zoom a particular area, use the Stroke feature or the **Zoom In** tool for zooming. Strokes are described later in this chapter.

Zoom All - CTRL+F (Windows); ⌘+F (Macintosh)

This command in the Arrange menu zooms in or out to make all objects on your drawing fill the screen, regardless of how big or small the objects are.

Tip:

Use the **Zoom** tools or Stroke zoom to zoom at a particular location.

Zoom In - CTRL+] (Windows); ⌘+] (Macintosh)

This command in the Arrange menu zooms in on the center of the screen by a factor of two. You cannot specify a particular area for enlargement.

Zoom Out - CTRL+[(Windows); ⌘+[(Macintosh)

This command in the Arrange menu zooms out from the center of the screen by a factor of two. You cannot specify the area of reduction.

Zoom Previous

This command in the Arrange menu zooms to the previous magnification, as many as five times.

Zoom Tools

Select the appropriate zoom tool from the **View Control** tool palette.

Using a Magnifying Glass Tool

With the **Zoom** tools you drag a box around an area on the screen, so only that area magnifies on the screen.

1. Select a **Zoom** tool from the tool palette.
2. With the mouse button pressed, drag a box around an area on the screen you want to magnify or reduce.
3. Release the mouse button.

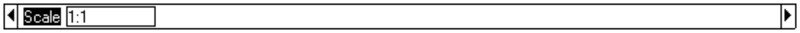
   The content of the dragged box is made visible on the screen.

Zoom In Tool

This tool, in the last subpalette of the main palette, zooms in by the specified factor. The default factor is two. This is a visual rather than a physical change.

When you click in the drawing area, that position redisplays in the center of the screen, and the drawing enlarges by a factor of two.

You can also drag a box around an area, so only that area magnifies.



The Status Line shows the current zoom scale. If you enter a different scale, that scale takes effect when you press ENTER (Windows) or RETURN (Macintosh).

Pressing the CTRL (Windows) or the OPTION (Macintosh) key while using this tool causes it to change to, and behave like, the **Zoom Out** tool.

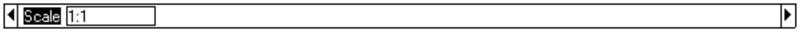
Zoom Out Tool

This tool, in the last subpalette of the main palette, zooms out by the specified factor. The default factor is one half. This is a visual rather than a physical change.

Tip:

Using the CTRL (Windows) or the OPTION (Macintosh) key, you can toggle between the **Zoom In** and the **Zoom Out** tool.

When you click in the drawing area, that position display in the center of the screen and the drawing reduces in size by a factor of one half.



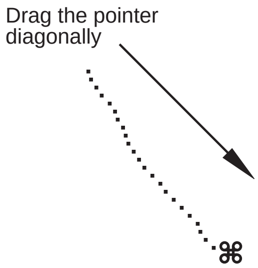
The Status Line shows the current zoom scale. If you enter a different scale, that scale takes effect when you press ENTER (Windows) or RETURN (Macintosh).

Pressing the CTRL (Windows) or the OPTION (Macintosh) key while using this tool causes it to change to, and behave like, the **Zoom In** tool.

Stroke Zoom

You can use stroke commands to zoom—magnify or reduce your drawing. Stroke commands are useful because they don’t require you to get out of the tool you’re using in order to zoom. Hold down the SHIFT+CTRL keys (Windows) or the ⌘ key (Macintosh) and drag the pointer that appears diagonally across the screen as described below. The pointer takes on the ⌘ shape when you hold down the SHIFT+CTRL (Windows) or the ⌘ key (Macintosh) keys.

Using Stroke Zoom



Drag Diagonally	**Result**
Upper left to lower right	Zoom-in enlargement centered over the stroked area.
Lower right to upper left	Reverses Zoom In stroke to the previous magnification.
Upper right to lower left	Zoom-out reduction—the current screen reduces to the size of the area defined by the stroke.
Lower left to upper right	Reverses Zoom Out stroke to previous magnification.

Note: For Zoom In and Zoom Out, the size and location of the stroke rectangle is important for determining the result of the Zoom operation. For *Zoom Previous*, the size and location of the stroke rectangle is irrelevant. All cases just give the previous magnification.

Orbit and Pan tools

The **Orbit** tool is used to rotate the sheet camera around the 3D model. The 3D objects is fixed even if you have the opposite impression. The **Pan** tool is used to move the sheet camera in the x and y directions. Panning passed the document edges will automatically scroll the view in that direction.

Layers

Layers are like pages, some transparent and some invisible. You can use layers to show and hide various components of your drawing. They are particularly useful in helping you view and plot complex drawings. For example, when you dimension a part, the dimensions can be placed on a separate layer which you can display or not, as your needs require. Layers allow you to plot different versions of the same document for quick and easy specialized blueprints.

- You can hide the dimension layer to exhibit the idea of a design to a planning team and show the dimensions when presenting the drawing to engineers.
- You can hide some drawing components when you print or plot. For example, you can hide the construction layer so that construction lines and geometry don't print or plot, but they remain in the drawing ready for use when you want to alter them. 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. Nor can those objects be deleted, using *Select All* unless the layer is visible.
- You can construct different layouts using one layer as the basis. For example, you can use a floor plan and then construct the electrical and plumbing plans on different layers. In that way, you can turn off one layer (the electrical plan, for instance) and print the other (the plumbing plan) with the floor plan.

Graphite allows you to have up to 255 visible or hidden layers in your drawing.

Using Colors with Layers

Many CAD drawings show different components in different colors. When you specify a different color for a layer, the layer remembers the color so that if you add geometry to a layer later, the new geometry appears in the same color as the last geometry constructed on that layer. In this way, you can use the Visible pen in red on one layer, green on another layer and so on.

Tip:

Layers facilitate color specific wiring diagrams or electrical and plumbing blueprints.

1. Choose ***Layout>Layers***.
2. Create a new layer and rename it **Outline green**, indicating you always want to use the pen style Outline with a green color on this layer.
3. Make the layer, Outline green, current.
4. Select ***Pen>Style>Outline***.
5. Change the pen color from Black to Green by selecting ***Pen>Color>Green***.
6. Draw a circle. The circle is created with the green color.
7. Select the **Single Line** tool.
8. Make Layer 1 current.

9. Draw a line. The line is created in black (the default color for the Outline pen style).
10. Select the **Rectangle** tool.
11. Make the layer, Outline green, current and draw a rectangle.

    The rectangle is created in green.

Notes:

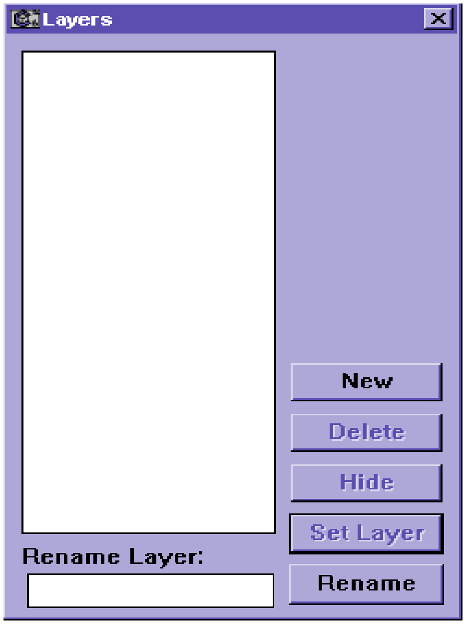
- Layers can be saved as Preferences with specified colors.
- Layers do not have an orientation or origin in Graphite.

Layers Command - CTRL+L (Windows); ⌘+L (Macintosh)

This command in the Layout menu allows you to create, delete, hide, show and rename layers, as well as set layer specifications. The work layer is the active layer—the layer on which the current construction is created.

You cannot hide or delete a layer that is the current (active) work layer.

The visible layers are indicated in the list box by an *Eye Symbol*, and a check mark indicates the work layer.



When you open a new drawing, the default layers include Construction, Dimension and Layer 1.

Construction	This layer automatically accepts all Construction lines created with stroke commands or with the Construction dialog box. All lines of the "construction" line type go on the current work layer, not this construction layer.
Dimension	Dimensions are placed on the layer you select from the Layer list of the Dimension menu. The default layer is the Dimension layer. Generally, this layer should be reserved for dimensions.
Layer 1	This layer is the current work layer when you open a new file. If the file only contains the default layers, all geometry is normally placed on Layer 1.

You can edit any geometry or text that is visible, regardless of its layer. If you want to make some geometry unselectable but still visible, use the *Selection Mask*. You can also specify the selectability of layers with the *Selection Mask*.

Creating a New Layer

Click New to create another layer. You can name the layer by typing the name in the Rename data field and clicking Rename. You can have as many as 255 layers.

Tip:

You can also type a name, then click New.

Renaming a Layer

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

Deleting a Layer

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

   Note: The layer and everything on that layer deletes; however, a warning appears before this happens.

Hiding a Layer

1. Click the layer(s) you want to hide in the list.

2. Click Hide. The visible mark beside the layer name disappears.

Notes:

- You can't hide the work layer
- If a layer is hidden and you choose *Select All* and delete, the objects on the hidden layer are not deleted.

Displaying a Hidden Layer

1. Click the layer(s) you want to make visible in the list.
2. Click Show.

   A visible mark appears next to the layer's name in the list.

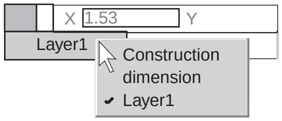
Making a Layer the Work Layer in the Layers Dialog Box

1. Select the name from the list.
2. Click Set Layer.

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

Changing the Work Layer with the Work Layer Indicator Box

You can use the Work Layer Indicator pop-up box at the lower left of the drawing area to specify a different layer as the work layer.



1. Press the mouse button on the box and the menu displays.
2. Drag to the layer you want to make the work layer and release the mouse button.

   The layer you selected is now the work layer.

Tech Note:

You can select a contiguous group of items in the Selection Mask by clicking on the first or last item in the desired group, then holding down the SHIFT key and either clicking on the items or dragging up or down to the other end of the desired list.

If you want to select or deselect noncontiguous items, hold down the CTRL (Windows) or the ⌘ (Macintosh) key and click the items.

Determining How Many Objects are on a Layer

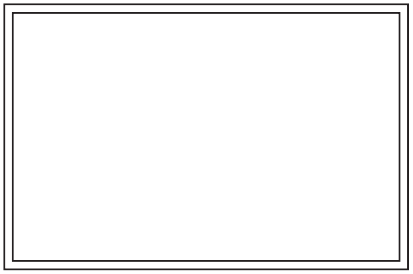
1. Choose ***Edit>Selection Mask***.
2. Select the desired layer from those listed in the center box.
3. Choose ***Edit>Select All***.
4. Choose ***Edit>Edit Objects***. The number of objects is listed at the top of the Edit Objects dialog box.
5. Close the Selection Mask and Edit Objects dialog boxes.

Layer Examples

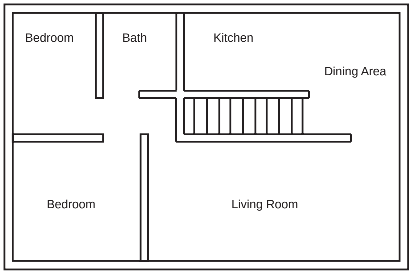
Layers are like transparent sheets that you can turn on and off. This feature is particularly useful for designs with several variations of a component or to reveal as much or as little detail as needed for the design. Here are two such uses.

Floor Plans

A common use of layers is for different floor, plumbing and wiring plans for a building. The following examples show simplified drawings using layers.

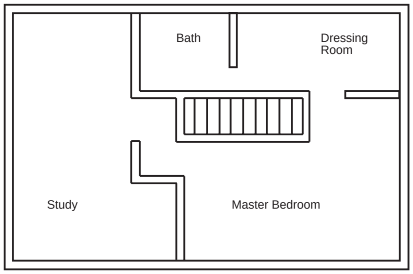


Layer 1: The foundation



Layer 2: First Floor

Layers can be used to display different floor plans of the same building.



Layer 3: Second Floor

In order to print or plot the layers, simply use the Layers dialog box to turn on the layers you want to print and turn off all other layers using the Hide or Show button. When you can see what you want to print on the screen, choose ***File>Print***.

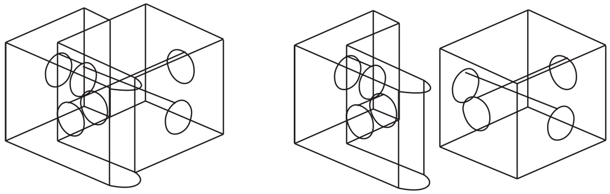
Using Smart Walls with Layers

You can also observe how smart walls function in these examples. The interior walls on Layers 2 and 3 overlay the exterior walls of the foundation.

Smart walls merge only on the same layer; therefore, the interior walls are not merged with the exterior walls and walls on Layer 2 are not merged with walls on Layer 3.

Assembly

Layers can also be used to create interlocking parts of an assembly. You create one part on Layer 1 and copy the common geometry onto Layer 2. Then you create the remainder of the interlocking part on Layer 2.



In this way, you can turn the display of the layers on and off to present the parts appropriately. You are already familiar with these parts from a tutorial in the *Getting Started* section of the manual.

Layer Groups

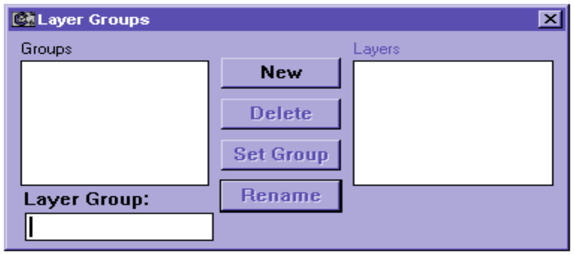
Layer Groups are particularly useful for viewing and plotting different layers of a complex drawing. For example, the document described in the Layers section that shows the floor plan of a house might have several layers illustrating the plumbing layout and several layers illustrating the electrical plan. The plumbing layers can be shown in the Plumbing Layer Group but hidden in the Electrical Layer Group. Similarly, the electrical layers are shown in the Electrical Layer Group but hidden in the Plumbing Layer group. To create a print to go to the electrical contractor, you can simply make the Electrical Layer Group be the work group. This will show the layout of the house with the electrical plan contained on various electrical layers but not the plumbing information which is on the hidden plumbing layers. Rather than having to Show or Hide numerous individual layers, you only have to select a single Layer Group to get the format that is desired.

Layer Groups Command

The *Layer Groups* command in the Layout menu extends the functionality of layers by creating Layer Groups. A Layer Group records, stores and displays layer settings—which layers are shown and which layers are hidden, as well as which layer is the work layer. This lets you show and hide different layers quickly by selecting a single Layer Group rather than setting the attributes of numerous individual layers.

The Layer Groups dialog box lists the layer groups on the left and the individual layers on the right. Visible layers for the selected (highlighted) Layer Group are indicated in the Layer list by the Eye Symbol, just as they are in the Layers dialog box. Visibility can be toggled by clicking the Eye Symbol, or at the location where the Eye Symbol should be for hidden layers. The work layer of the selected group is set by double-clicking on the desired layer from the Layer list. Layers that are created after a Layer Group has been defined are included in the group but are not visible.

The current set of visible layers and the work layer need not correspond to any layer group; thus, it is not necessary that a Layer Group be designated as the "work" group. Setting a work group simply changes the model's current layer settings to those of the specified Layer Group. Also, if a Layer Group is designated the work group and layer attributes are subsequently modified—either by the Layer dialog box or by changing the work layer in the Layer Indicator pull-down list—the Layer Group loses its "work" status.



There must be at least one Layer Group created before you can use any of the buttons other than New or change any settings for any layers.

New — This button creates a new Layer Group, called Group1, Group2, Group3, etc., by default. Use the Rename button to rename the Layer Group with an appropriate name. You can have up to 255 Layer Groups in a single file.

Delete — This button deletes the selected (highlighted) Layer Group. This does not delete the layers in the group.

Set Group — This button sets the currently selected (highlighted) group as the work group. This changes the layer settings for the current model to those specified by the layer group.

Rename — This button renames the currently selected (highlighted) group with the name in the Layer Group data field.

A check mark by a Layer Group indicates that the model's current layer settings match those of the indicated Layer Group.

Creating a Layer Group

1. Click New. A new Layer Group appears in the group list and is selected. It records the current layer settings and displays these settings in the Layers list.

   The Layers list contains all the layers present in a file.

2. Choose the layer settings. Visible layers are indicated by the Eye Symbol and can be hidden by clicking on the symbol. Hidden layers can be made visible by clicking the location where the Eye Symbol would be. The work layer is indicated by the check mark and can be set by double-clicking on the desired layer.
3. Close the dialog box and the settings are retained. If you want more layer groups, repeat steps #1 and #2 before closing the dialog box.

Renaming a Layer Group

1. Select the desired Layer Group from the group list.
2. Enter a new name in the Layer Group data field.
3. Click Rename.

Deleting a Layer Group

1. Select the desired Layer Group from the group list.
2. Click Delete.

Deleting the work group, which is indicated by a check mark, deletes the group from the list, but the layer settings remain in effect until they are changed.

Making a Layer Group the Work Group

1. Select the desired Layer Group from the group list.
2. Click Set Group. This changes the layer settings for the current model to those specified by the layer group. These settings are shown in the layers list.

A check mark by a Layer Group indicates that the model's current layer settings match those of the indicated Layer Group.

Changing the Layer Setting of a Layer Group

1. Select the desired Layer Group from the group list.
2. Change the layer settings as desired. Visible layers are indicated by the Eye Symbol and can be hidden by clicking on the symbol. Hidden layers can be displayed in the layer group by clicking in the location where the Eye Symbol

should be. The work layer is indicated by the check mark and can be set by double-clicking on the desired layer. The new settings automatically replace the old settings.

Layers, Sheets, and Models

All Models and Sheets share the same set of layers in a file. For more information about Models and Sheets, see Chapter 14.

View Displays

This section explains various ways to enlarge or reduce all or part of the geometry for both viewing and printing. This visual change of the view does not affect the actual measurement of the geometry (the model). This is different from the view orientation of your geometry that you would set using the *Trackball* or the *View* commands in the Views menu. (See Chapter 6, "Advanced Environment Settings.")

There are two types of views in Graphite: the *Sheet View* and the *Detail View*.

Sheet Views

The Sheet view shows all of your geometric construction on the sheet outside of any existing view windows, at the scale you specify. The Sheet view is the default view when you start Graphite.

The sheet view is picked up by the *Sheet camera* and projected on the current sheet. The sheet view is infinite in size, having no boundaries. In other words, when drawing on a sheet, you can create geometry as large as you desire.

Referral:

You will find more about Sheets and Views in the Chapter 14, "Advanced Views Techniques," in this manual.

Note: When you select ***File>New***, you are looking at Sheet 1 of your untitled file. It is the only sheet available until you add more. See Chapter 14 for more information about sheets and the Sheet camera.

Detail Views

In drafting, details views are used to provide more information about a specific area of the drawing. They are typically shown at a larger scale than the original geometry. Graphite provides a tool that automatically creates detail views so that you do not have to redraw the geometry.

A detail view is like a window which looks at some geometry in your drawing from a particular camera angle and displays it at a scale you specify. A detail view can be created in Graphite in two different ways: with the **Detail View** tool or the *Sheet*

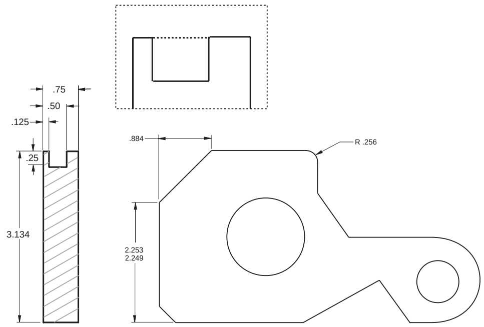
Referral:

See the next section for information on associativity.

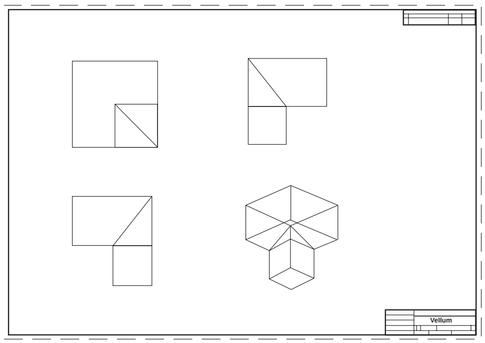
Into View command in the Views menu. The *Sheet Into View* command is an automated way of creating arrangements of detail views in specific layouts. The detail views in each case behave in the same way.

Detail Views are always placed in a view window and are associative so that when you modify geometry in any view, all views of that geometry reflect the change. There are three deliberate exceptions to this associativity, in line with standard drafting practice: dimensions, crosshatching, and text appear only in the view in which they were created.

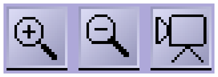
Graphite allows you to create detail views of your drawing in which some part of the geometry is scaled and placed in a view window. In the illustration here, the detail view shows a 2 : 1 enlargement of the side notch.



You will probably use more than one view window with different view orientations to show the various faces of your geometry. Detail view windows are useful for observing the geometry as you work. They are also essential for creating finished drawings. The following graphic displays another model in four different detail views.



Detail View Tool



The **Detail View** tool in the last subpalette of the main tool palette creates a detail view of the designated area of your drawing.

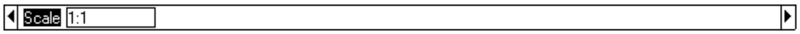
Creating an Associative Detail View

An associative detail view is one in which a change will appear in the view if a change is made to geometry in another view.

1. In the View Control subpalette, select the **Detail View** tool. The Message Line reads, *Detail View: Enter view scale then pick first corner of viewing frame.*

2. Enter a scale for your detail view in the Status Line.

3. Use the pointer to drag a rectangle around the area of your drawing that you want placed in the detail view. This rectangle becomes the window frame.

   You can also click the first corner and the opposite corner to define the view.

4. Position the pointer in the center of the detail view window frame and drag the window to a clear area on your drawing.

Tip:

You may also create the view window rectangle by clicking to place the diagonally-opposite corners.

Creating a Non-Associative Detail View

You may want a detail view that is not associative; that is, when you make changes to the original geometry, the geometry in the view does not change.

1. Create a detail view on an open space.
2. Use the **Selection** tool to click inside the view and make it active.
3. Drag a selection fence around all of the geometry inside the original view.

   The geometry inside the view is selected.

4. Choose ***Edit>Copy***.
5. Choose ***Views>Models***.
6. Click New to create a new model (by default, model 2).
7. Click Current to make the new model the current model in the new view. The geometry in the view disappears.

Referral:

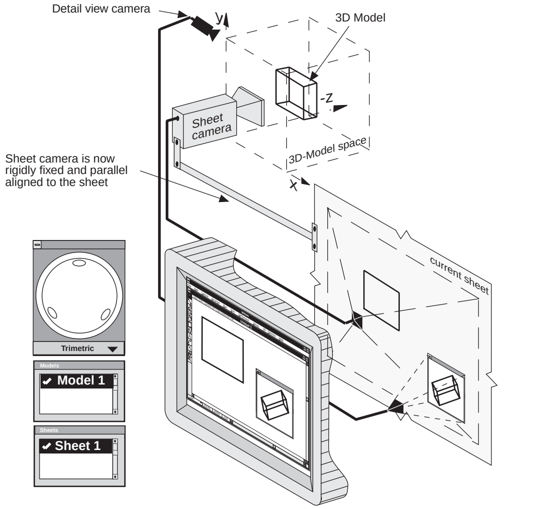
You will find more about Models and Views in the Chapter 14, "Advanced Viewing Techniques."

8. Choose ***Edit>Paste***.

   The geometry is pasted into the new detail view, but it is no longer associative, since it is part of a different model.

Rotating the Viewing Area and Detail Views

You can change the exist, you can only change the view orientation within a view window. The sheet view is picked up by the sheet camera and is rigidly fixed to the sheet itself and aligned parallel to the sheet in the Top view.



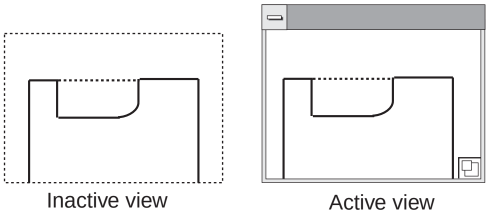
1 Model with 1 additional active Trimetric view window

Tech Note:

When you click the detail view, the view becomes active and the view window with a title bar appears.

Active View

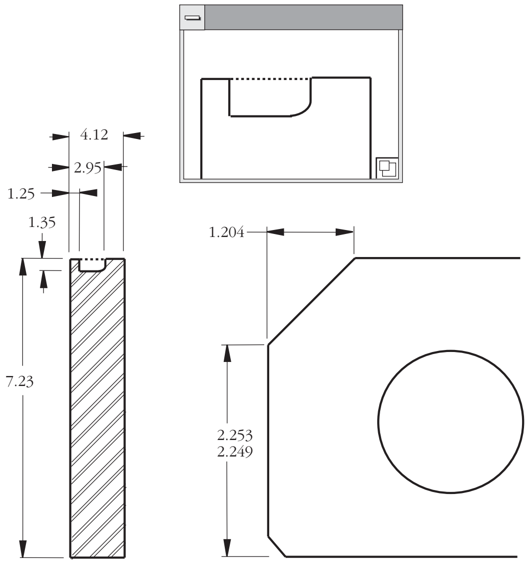
The detail view window displays a title bar only when it is the current or active view.



Click inside the detail view window when the view is inactive, as in the left graphic above. The detail view window becomes active, displaying a title bar and control button for its pull-down menu, as in the right graphic above.

The sheet view is made active by clicking in the drawing area away from all detail views. (In other words, all the detail views look inactive.) You must activate the sheet to construct geometry that is outside of a view.

A view must be active for you to work in it. Either a detail view is active or the sheet view is active, but only one can be active at a time.

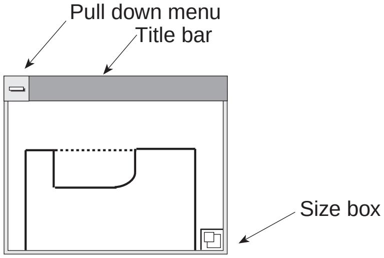


Dimensions and hatch pattern only on the sheet

View Window

The view window includes a title bar, size box and Control pull-down menu only when the detail view is active.

You can drag the view around by the title bar and resize the window by dragging the size box.



Tech Note:

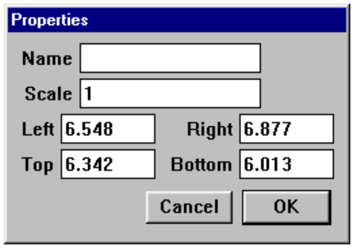
In the Graphite View window, the Control Menu in the upper left corner does not contain the standard control commands but special view window commands.

Control Pull-down Menu

The Control pull down menu provides options for manipulating the view.

Properties

The *Properties* option allows you to name the detail view, change the view scale (entering 2 would zoom into the area by a factor of 2), and define the locations of its corners relative to the origin.



Tech Note:

Using *Cut* or *Copy* within the Detail View does not cut or copy the geometry inside, but rather cuts or copies the detail view itself. To cut or copy just the geometry, use these commands from the Edit menu.

Cut

The *Cut* option allows you to remove the Detail View from the sheet and place it on the Clipboard. This is useful for placing the view on a different sheet. Choose the ***Edit>Paste*** to paste the cut view window. This command does not cut the geometry. To cut the geometry, choose ***Edit>Cut***.

Copy

The *Copy* option allows you to place a copy of the view window on the Clipboard. This is useful for pasting multiple identical views onto one or more sheets. Choose the ***Edit>Paste*** to paste a copy of the view window. This command does not copy the geometry. To copy the geometry, choose ***Edit>Copy***.

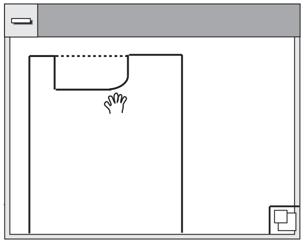
Delete

This option deletes the active window and makes the sheet active. It does not delete geometry.

Pan

This command allows you to move the geometry inside the view window. A hand icon appears which you can use to drag the contents of the window.

When you release the mouse button, the *Pan* function ends.



Resize

This *Resize* option allows you to drag a new view window. This function differs from dragging the window borders. With the *Resize* command, you do not have to change each border individually since you define a new window that is substituted for the original one.

If you cannot see the content of the view window after you have applied the *Resize* command, choose ***Arrange>Zoom All*** with the SHIFT (Windows) or CONTROL (Macintosh) key pressed. This ”fits“ all the geometry of that model into that view window at the necessary view scale.

Changing the View Scale

When you are trying to fit a view or group of views in a drawing format, you may have to scale a view.

1. In the Detail View window's Control menu, choose *Properties*.
2. Specify the scale you want and click OK.

Tip:

You can also scroll within a view by holding down the SHIFT (Windows) or the CONTROL (Macintosh) key while moving the scrollbars.

Copying a View Window

You can use the view window menu to make another view window in case you want to add to or change the arrangement you selected with *Sheet Into View*.

1. Click inside the window to make it active.
2. Choose *Copy* from the view window control menu.
3. Choose ***Edit>Paste***.

   The copy of the window appears slightly offset from the original window. You can drag it to a new location.

Panning the Geometry in a View Window

1. Click inside the window to make it active.
2. Choose *Pan* from the view window menu.

   The pointer changes to a mover hand icon.
3. Drag the geometry to the location you want.

Zooming within a View

1. Click inside the view to make it active.
2. Hold down the SHIFT (Windows) or the CONTROL (Macintosh) key.
3. Choose ***Arrange>Zoom*** (***In***, ***Out***, ***Previous***, or ***All***)

   or

   Choose one of the **Zoom** tools in the tool palette.

   The geometry inside the view is zoomed.

All commands from the Arrange menu work inside the view when you hold down the SHIFT (Windows) or the CONTROL (Macintosh) key while choosing from the menu.

The keyboard Zoom commands are not designed to work within a view window.

Views Menu

The view control commands in the Views menu allow you to control the active view. When you have overlapping views, you can think of them as a stack where only the top view can be active. If the *Auto Front* command is turned on and you click a view it comes to the top of the stack and you can work in it. This may cause some problems when you click an object in the top view to select it, but an underlying view activates instead.

To remedy this situation, turn off the *Auto Front* command. Select the view you want to work in and choose ***Views>Bring To Front***. The objects in the view you brought to the front remain on top, even when you click a different view visible in the top view's space.

Auto Front Command

The *Auto Front* command in the Views menu overrides the *Bring To Front* or *Send To Back* settings. When this setting is in effect, you can simply click on a view to bring it to the front and make it active. When this setting is not active, you can't inadvertently bring a view to the front when you are working on another view. This command only works for views.

Bring To Front Command

This command in the Views menu brings the specified view to the front of the stack of views.

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

This command counteracts the *Auto Front* command, so when you click, you select objects in the view rather than activating another view.

Send To Back Command

This command in the Views menu sends the specified view to the back of the stack of views.

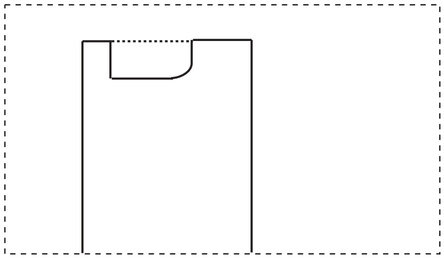
1. Select the view.
2. Choose ***Views>Send To Back***.

This counteracts the *Auto Front* command, so when you click, you activate another view rather than selecting objects.

Draw View Boundaries Command

This command in the Views menu shows the boundaries of all inactive views. This boundary will be printed if you leave it displayed when you choose the *Print* command.

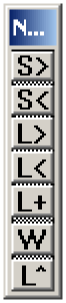
If you want to print the view without boundaries, in the Views menu, deactivate Draw View Boundaries.



Windows

This submenu on the Views menu shows the names of all the open Graphite documents. To bring a different document to the top window, choose it from the submenu.

Navigator Palette



The Navigator Palette allows you to easily move through sheets and layers in a circular file fashion. It can also display the active dovuments and isolate layers.

Sheet Navigation

Click on the S> button to move forward throught the visible sheets. Click on the S< button to move backward. Only one sheet at a time can be shown.

Layer Navigation

Click on the L> button to move forward through the visible layers. Click on the L< button to move backwards.

Click on the L+ to show all layers. Clicking L^ will pop up a list of available layers. Choosing one of the layers will make it become the active layer and all other layers will be visibly turned off. You can restore full visiblity to all layers by clicking on the L+ button or choosing **Layout>Layers** and setting the visibility individually.

Window Navigation

Click on the W button to display a popup menu of available documents. Choosing one of the documents will make it become the active document.

Advanced Viewing Techniques

Sheets, Views and Models are features in Graphite that add a great amount of flexibility to the design process. This chapter explains each of them and how they interact with each other. The following topics are included:

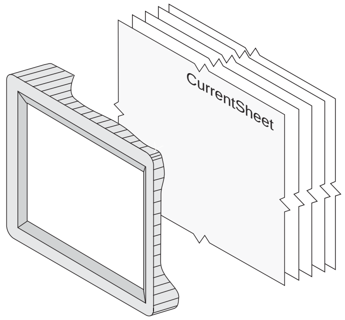
- Sheets
- Models
- Views, including Sheet into View and View Layouts
- Combining Sheets, Views and Models

Sheets

A sheet is an infinite planar area. Graphite allows you to have multiple opaque sheets within a drawing. A drawing can be a simple part or a complex assembly, and you can organize related geometry on as many sheets as the scope of the project calls for.

You can have as many sheets as you want but only one sheet can be visible at a time—the active sheet. You activate a sheet with the *Current* command in the Sheets dialog box in the Views menu. In Graphite one sheet is always current, the active sheet. It cannot be deleted.

If you have multiple sheets, imagine them arranged as a stack of sheets behind your computer screen. If you make a sheet current you bring it to the front. That is why you can see only one sheet at a time—the current sheet.



Multiple sheets are stacked behind the screen. The current sheet is always in front and the only visible one.

Using Sheets

Every document contains *Sheet 1*. You can use the *Sheets* command in the Views menu to create new sheets, delete, rename, or set the current work sheet. You may want to give the sheets distinctive names, rather than their default numbers, because it is much easier to know what's on a sheet if it's named, for instance, Detail View, rather than Sheet 2.

Sheets are Graphite's equivalent to pieces of paper, with the added function that Sheet 2 can display geometry associated with Sheet 1. When you are using detail views, layers won't serve your needs for printing or plotting because when you hide the layer containing the geometry the detail view is associated with, the geometry is also no longer visible in the detail view.

You can use sheets to display and print multiple pages of complex drawings. The following example explains the process:

- When you open a new document, Sheet 1 displays in the drawing area.
- You create a complex assembly and a detail view on Sheet 1.
- When it is time to plot the project, you find that the assembly fills all of the plotting area and there is no room for the detail view on the page.
- Select the detail view and choose *Cut* from the active View window menu (not from the Edit menu in the menu bar).
- Choose ***Views>Sheets*** and click New to create another sheet (Sheet 2).

- Click Current to make Sheet 2 the current sheet.

  A blank sheet displays.

- Choose ***Edit>Paste*** to paste the detail view on Sheet 2.

  The detail view displays on Sheet 2.

  You may have to do *a Zoom All* in order to see the detail view.

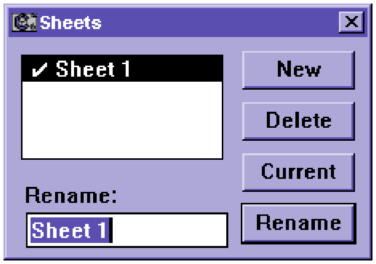
Finally, you can check to see that the detail view is still associated with the original assembly:

- Make Sheet 1 current and make a change to the area of the assembly that is in the detail view. (Remember: Crosshatching, Text, and Dimensions are not associated in a detail view.)

- Look at Sheet 2 and see that the change appears in the detail view on Sheet 2 and, of course, changes made in the detail view are also be reflected in the Sheet 1 view.

Sheets Command

This command in the Views menu allows you to create and name sheets and set the work sheet. The check mark (✓) in the list of sheets indicates the work (or current) sheet. The title bar for the document also indicates the active sheet (Document Name: Sheet Number). The name of the sheet only displays in the Title Bar if more than one sheet exists.



You may want to use several sheets to create related components and associated detail views. With the drafting board you use a new sheet of paper for a project; with Graphite you create a new sheet.

Tip:

Whenever you want to print a part related to one project separately, create a new sheet.

Creating a New Sheet

Click New to create another sheet. Name the sheet by typing the name in the Rename data field and clicking the Rename button.

Tip:

You can also type a name and then click New.

Note: Graphite automatically creates a new model when you create a new sheet. See a later section for more information on models.

Renaming a Sheet

1. Select the name of the sheet from the list box.

2. Type the new name.
3. Click Rename.

Deleting a Sheet

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

Graphite handles Models and Sheets separately. The geometry you create is the model—what you see on the sheet is only an image (view) of that model. So when you delete the sheet you delete only the *Sheet View* or *Detail View* of the model, but not the geometry (model) itself.

Note: The current sheet cannot be deleted.

Making a Sheet the Current Sheet

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

Tip:

Any combination of geometry such as lines, arcs, circles, dimensions, text, etc. you create with any of the creation tools in the tool palette or any tools in the Dimension palette is a Model.

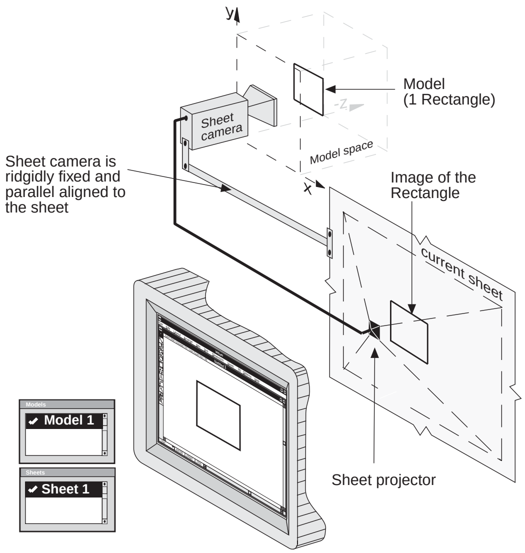
Models

Models are composed of the geometry you are creating. Even if it looks like the geometry is created directly on the sheet, it is not. What you see on the sheet is only an image of that model. The model itself is created in an infinite three-dimensional area we call Model space.

Projected Model Image

The image you see on the sheet is the projected Sheet View of the model picked up by the *Sheet Camera* which is aligned parallel to the sheet and looking from a top view, at the model.

When you use the *Zoom* commands in the Arrange menu or the **Zoom** tools from the tool palette you change the view scale of the model by zooming the Sheet camera like you would with a video camera.



Model 1 projected on the current Sheet

Multiple Models

There are two different ways to create models:

- with the *Sheets* command in the Views menu

  or

- with the *Model* command in the Views menu.

When you create a new sheet, Graphite automatically creates a new blank model where you can create geometry.

Tech Note:

The *Sheet Into View* command also creates a new model for the Sheet View. The existing model is placed in the detail view window. See a later section of this chapter for more information.

Creating a New Model with the Sheets Command

This task assumes that you have not created any additional Models and Sheets.

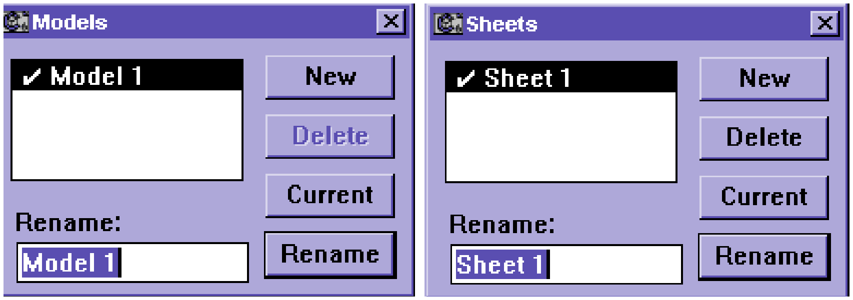
1. Select ***Views>Sheets***.

   The Sheets dialog box displays.

2. Select ***Views>Models***.

   The Models dialog box displays.

3. Since the Model dialog box is placed at the same location on the screen, move it to another location so you can see both dialog boxes.



   Now you can see that a new drawing in Graphite has one Sheet and one Model by default. The check mark in each dialog box shows that Sheet 1 and Model 1 are current.

4. Display the Sheets dialog box and click New to create a new sheet. You see also that a new model automatically appears in the Models dialog box (Model 2).

5. Make Sheet 2 current by highlighting it and clicking Current.

   If you have any geometry on Sheet 1, it disappears and a blank screen comes up. Also, the check mark moves from Sheet 1 to Sheet 2 and the check mark also moves automatically from Model 1 to Model 2.

   Model 2 automatically becomes current because Sheet 2 is related to Model 2. So if you make Sheet 2 current on the screen, then Model 2 has to be current as well.

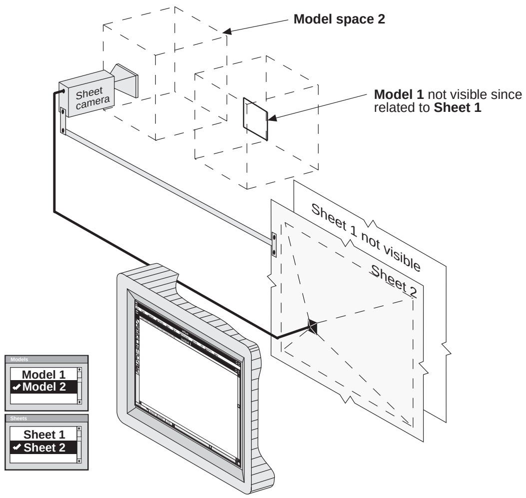
Sheet 2, now displayed on the monitor, is blank because nothing has yet been added to Model 2. As soon as you begin to draw, all geometry automatically adds to Model 2 since Model 2 is the current model space.

Relationship between Sheets and Models

When you create a new model with the *Model* command, a new model is created without creating a new sheet. So it is possible to have more models than there are sheets. You can only look at one model at a time in any view (sheet view or detail view) if the geometry is contained in different model spaces (i.e. Model 1 or Model 2). Think of models as a spatial area where geometry is located. You cannot view two spatial areas through one window at the same time.

You can create more models than there are view windows. So it is possible to have a model that is not displayed in any view. And you can have a single model displayed in many views at the same time.

When you make a Model current in a sheet view, it is like the Sheet camera is moved to the new model. When you make a Sheet current, the Sheet camera is moved to the related model whose image is projected on that sheet.



The **Sheet camera** moves automatically to **Model 2** when **Sheet 2** is made **current**

Models Command

This command in the Views menu allows you to specify models and switch between models to project them on the current sheet. You can display a single model on more than one sheet, but you can only see one model at a time on each sheet without using detail views.

Creating a New Model

Click New to create another model. You can name the model by typing the name in the Rename data field and clicking Rename.

Tip:

You can also type a name and then click New.

Renaming a Model

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

Deleting a Model

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

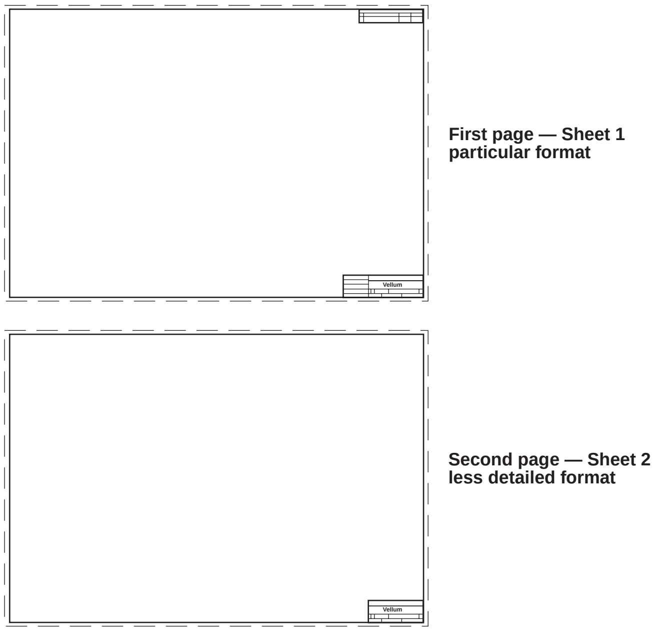
When you delete a model, you delete all of the geometry making up that model.

Note: You cannot delete the current model. A model which is still associated to a sheet cannot be deleted. As long it is related to a sheet the Delete button in the Models dialog box is not available.

Making a Model the Current Model

1. Activate the view you want (sheet view or detail view) by clicking on it.
2. Select the name from the list in the Models dialog box.
3. Click Current. A check mark shows the current model for the current view.

Multiple Sheets and Models



Generally, you create a model on a single sheet. When it is time to create drafting views of the model, however, you may need several sheets because your model is so large or so detailed that it requires more than one sheet of paper for the finished drawing.

For example, if you are designing a dental workstation, you could place the view on one page showing the chair, work tray, x-ray and drill arms.

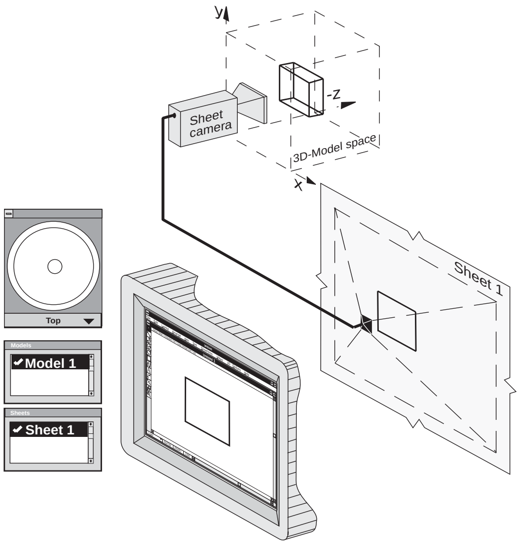
Then on subsequent pages, you can add the detail views of the x-ray arm and drill setup. In addition to this, your first sheet would use a particular format and the following sheets would use a different, less detailed drawing format.

Using Multiple Sheets

The key to using more than one sheet of paper is to use Graphite's model and sheet functions. In the example below, you will start with the front view of a model (Model 1) on a drawing format (Model 2) on Sheet 1. Then you open Sheet 2 and display all four views of Model 1 on it.

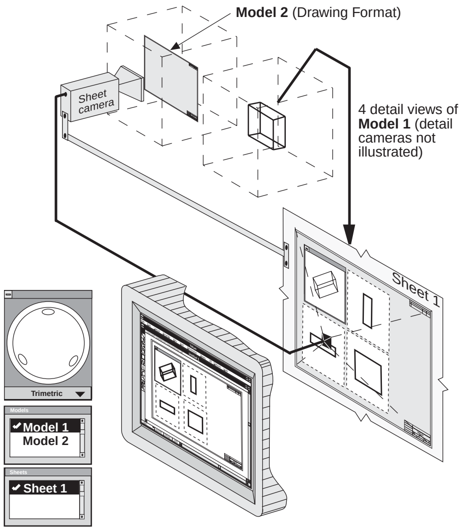
1. Create the part.

   Model 1 contains the part and its Top view is picked up by the sheet camera and displayed on Sheet 1.

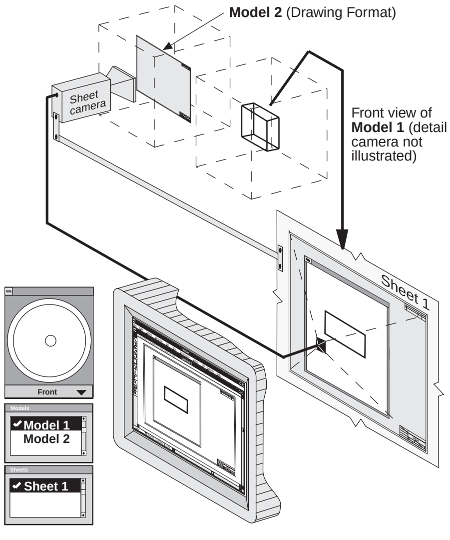


2. In the Views menu, use the *Sheet Into View* command to set up four views in the appropriate sized drawing format.

   The sheet camera aligns parallel to the current sheet and displays the Top view of Model 2 (the drawing format) on Sheet 1. Four detail view cameras display 4 views of Model 1 on Sheet 1 within the drawing format.



3. Use the *Cut* command in the view window menu to remove all the view windows, except the Front view.
4. Resize the remaining Front view window so that it fills the drawing format.



5. Choose ***Views>Sheets***.

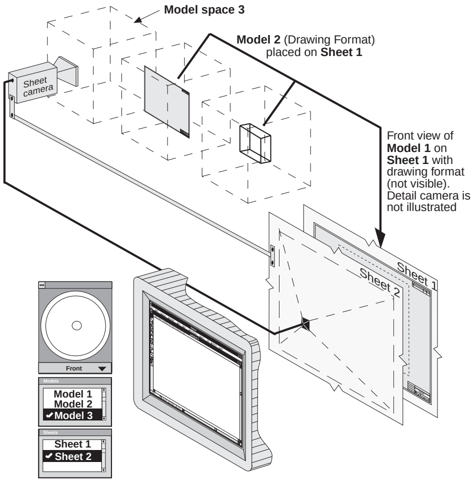
The Sheets dialog box displays.

6. Click New.

   Sheet 2 displays in the Rename box.

7. Click Current.

   Sheet 2 with the blank Model 3 displays in the drawing area.

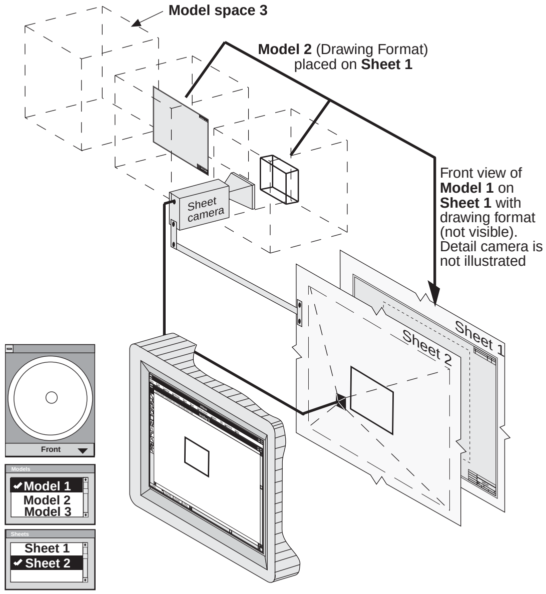


8. Choose ***Views>Models***.

   The Models dialog box displays. The blank Model 3 is the current model automatically created with Sheet 2, picked up by the sheet camera and displayed on Sheet 2.

9. Click Model 1 and click Current.

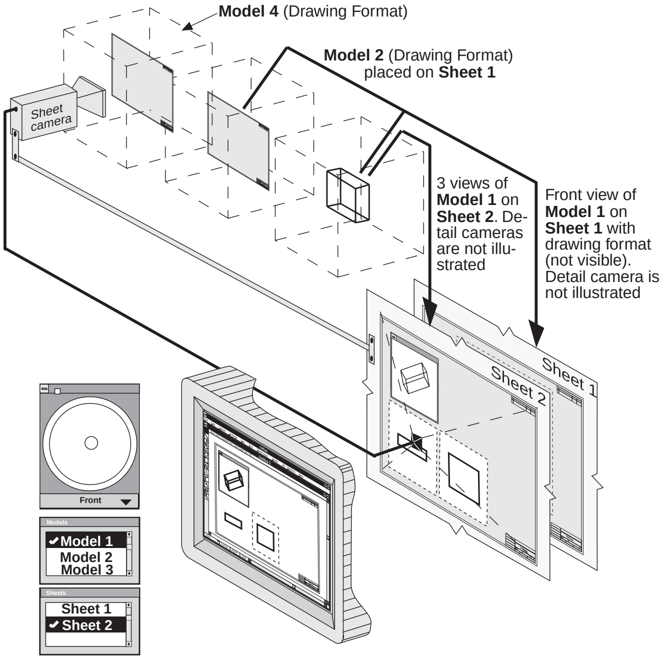
Model 1 is picked up by the sheet camera and displayed on the second sheet.



10. (Optional) Delete Model 3 since it is no longer needed. Model 3 was automatically created with Sheet 2 or

Another possibility would be to leave Model 3 active and paste the views of Model 1—which were removed with the *Cut* command at the beginning of this exercise— on Sheet 2.

11. Choose ***Views>Sheet Into View*** and select the appropriate format and views.

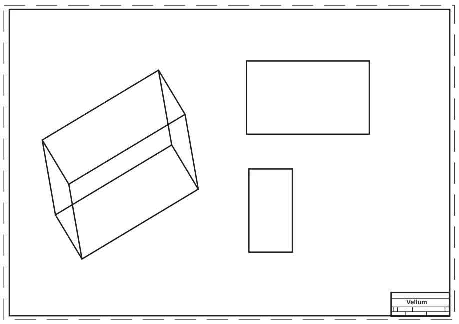


With the *Sheet Into View* command, you place views from the current Model 1 on the current Sheet 2. The sheet camera moves to Model 3 (automatically created with *Sheet Into View*) and displays the appropriate drawing format on Sheet 2. Sheet 1 with the front view of Model 1 lies behind Sheet 2 and is not visible.

Note: The sheet camera would be moved to Model 4 if Model 3 were not deleted in Step 10.

If you make Sheet 1 current, Sheet 1 moves forward and the sheet camera moves to Model 2 (drawing format) and displays the Front view of Model 1 within the drawing format, because Model 2 (drawing format) rests on Sheet 2.

12. Arrange the views as you like on the sheet.



Views

In the previous chapter, the concepts of Sheet Views and Detail Views were discussed.

Tech Note:

Regardless of the method you use to create a View Window, they all behave the same way. They all have a border which can be turned on and off in the Views menu with the *Show/Hide View Boundaries* command. The active view is displayed in a view window which has a title bar.

These two different views are easy to identify on the screen:

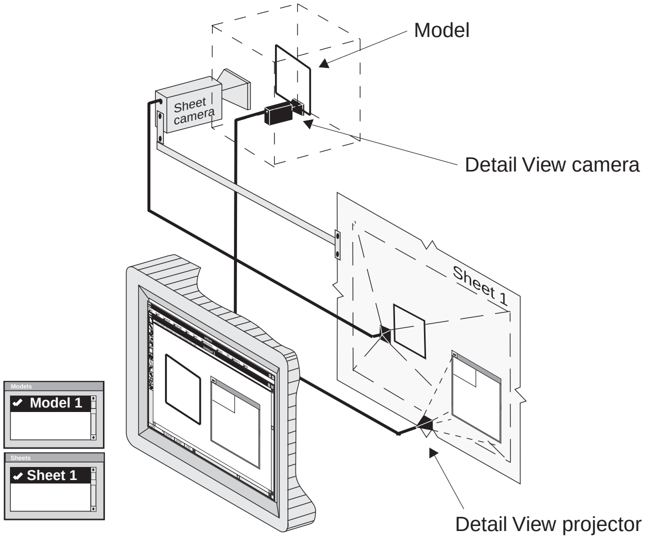
- Each item displayed inside a view window is a Detail View.
- The area outside (and underneath) all detail view windows is the Sheet View.

This section goes into greater detail on views and models, including Sheet into View.

More about Associative Detail Views

When you create a detail view with the **Detail View** tool on the tool palette or the *Sheet Into View* command in the Views menu, you see the same model through a second camera—the *Detail View Camera*.

The Detail View Camera behaves like the Sheet Camera except it displays its view of the model in a *View window* on the sheet.



Model with 1 additional **Detail View** camera projecting its view in its active **View window**.

Tip:

To move a view window to another Sheet, you cut the view window with the *Cut* command in the View Windows's control menu, make the other sheet active and place the view window with the *Paste* command in the Edit menu on the current Sheet.

All Views displaying the same model are associative. For example, when you change the geometry on the sheet, the geometry in the view window changes, and vice versa.

When you have several views displaying the same model they have to be associative, because no matter which view is active—the Sheet View or any Detail View—you are always editing the same model.

Activating Views

To make a view active you have to click on it. When you click on the sheet, the Sheet View becomes active. When you click in a view window, the Detail View becomes active.

Identifying Models

If you do not know which view displays which model, open the Models dialog box. When you click on the drawing area, the check mark moves to the model displayed on the sheet. If you click in a view window, the check mark moves to the model displayed in the detail view.

Tip:

This procedure is very useful if you have to identify models and sheets for renaming.

Scaling Views

Tip:

You can also change the scale of a detail view with the *Properties* command in the View window menu.

Zooming affects the Sheet View and the Detail View differently. Zooming in a Detail View with the SHIFT (Windows) or the CONTROL (Macintosh) key pressed, changes the view scale property of that view only. With the *Zoom* or *Stroke-Zoom* commands, you change only the screen display scale of the sheet and all Detail Views projected on that sheet but not the size of the geometry itself.

Text, Dimensions, Crosshatching, Fills and Detail Views

Where it is important that changes made to geometry in one view be reflected in the other views, the same is not true for text, dimensions, crosshatching and fills. The views would become very cluttered and rendered useless if all notations made in each view appeared in the others. For that reason, Graphite designed these elements so that they would not be associative, in line with standard drafting practice.

Sheet Into View Command

Tip:

Documents are more manageable if you delete all existing view windows before you use the *Sheet Into View* command.

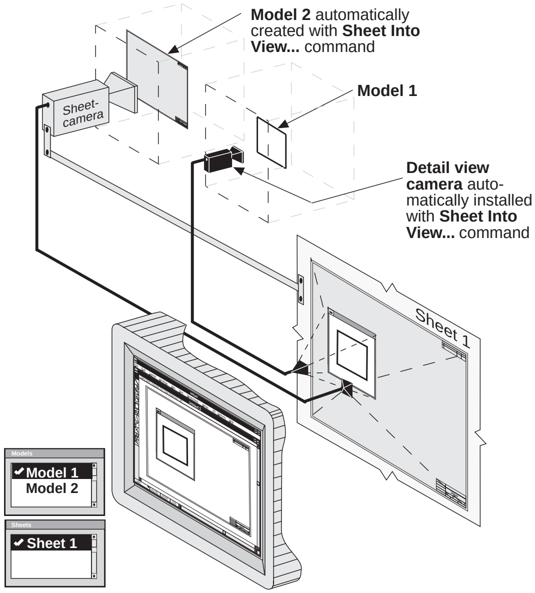
This command places all the geometry on a sheet including dimensions, text, crosshatching, and fills into one view window (the front).

The view window created with the *Sheet Into View* command behaves like any view window created with the **Detail View** tool. Although both view windows behave the same, the *Sheet Into View* command includes three automatic steps which are not performed when you create a view window with the **Detail View** tool.

The *Sheet Into View* command:

- creates an additional Model (blank or containing a standard drawing format if selected).
- moves the Sheet camera to that model (blank or with the drawing format) to display it on the current sheet

- puts all geometry including text, dimensions, crosshatching, and fills on the current sheet into a view window and displays it in the scale you have specified within the drawing format you have selected.



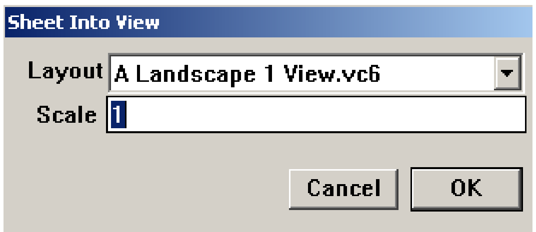
Layout with drawing format and one view loaded with **Sheet Into View...** command.

Tip:

Use the *Sheet Into View* command when you want to import a drawing format which is a different scale from your geometry.

Placing Geometry into a View using Sheet Into View

1. Draw the geometry.
2. Choose ***Views>Sheet Into View***.
3. Choose the layout you want from the Layout menu. You can choose from a view with no drawing format or a view in a standard drawing format.



4. Enter the scaling value in the Scale box. Factors greater than one enlarge the model in the view(s) and factors less than one reduce the model.
5. Click OK.
6. Choose ***Arrange>Zoom All***.

Tip:

You can add your own formats to this list as described Chapter 15, "Graphite Documents," or modify the layouts provided to meet your needs.

The geometry scales and the views appear as specified. If you specify a drawing format, it imports into the view and scales appropriately for the drawing size you have specified.

Adding Formats to the Sheet Into View Dialog Box List

You can create your own drawing formats and add them to the list in the Sheet into View dialog box.

1. Start a new document.
2. Select the **Detail View** tool and create a view window.
3. Display the Models dialog box and create a new model.
4. Click on the sheet and make Model 2 the current model in the Model dialog box.
5. Click in the view window and make Model 1 the current model in the Models dialog box.
6. Click on the sheet to activate Model 2.
7. Create the drawing format at a scale 1:1 or import one of Ashlar's formats and modify it.
8. Save the drawing with a proper name in the Layouts folder of Graphite.

   The Filename is listed on the Layout menu.

Tip:

You can also open one of the layouts in the Layouts folder of Graphite and change it. But be sure that you save that layout under a different name before you modify it.

See Chapter 15, "Graphite Documents," for more information on Formats and customization.

Erroneous Sheet into View

If you mistakenly invoked the *Sheet Into View* command and do not want your geometry in a view, use *Undo* to recover. If this is not done immediately, you must use the following method to manually back out.

Recovering from an Erroneous Sheet Into View

1. Delete all view windows on the current sheet by choosing Delete from the View window menu of each view.
2. Make Model 1 in the Model dialog box the current model for the Sheet View.
3. Delete Model 2.

View Orientation

You have many options for changing the view orientation within the view window.

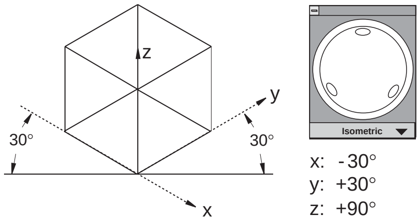
- Rotate the view with the Trackball.
- Choose ***3D>Views*** and select a view.
- Select a view from the pull-down menu on the Trackball.
- Set a view with the *Define View* command.
- Unfold a view with the *Unfold View* command.
- Set the view to the current work plane with the *View the Plane* command.

Define View Command

This command in the Views menu allows you to define an auxiliary view orientation or change the standard view orientations. The standard views cannot be changed unless the locked check box is clicked off. The default configurations are defined as follows:

Front The view of the x,z plane.

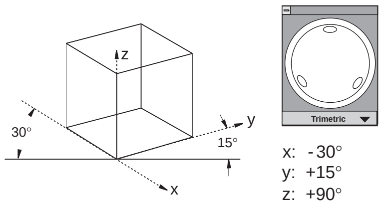
Isometric The view of the axes rotates as shown.



Right The view of the y,z plane.

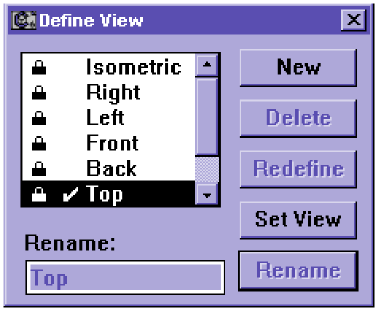
Top The view of the x,y plane.

Trimetric The view of the axes rotates as shown.



Specifying a New View

Follow these directions to specify a new view orientation.



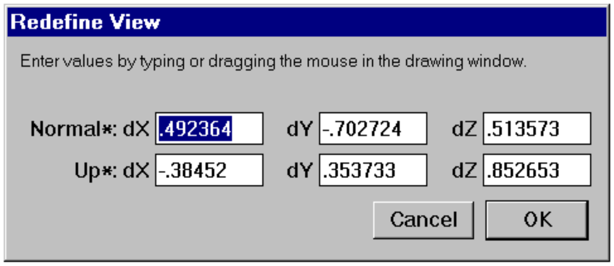
1. Choose ***Views>Define View***.

   The Define View dialog box displays.

2. Click New.

   The new work view is named View 1.

   The Redefine View dialog box displays the locations for the current view. If that view is the desired view, click OK. Otherwise, proceed with the specification. The *Normal* (line of sight) specification highlights in the dialog box.



3. Drag a line in the active view window to indicate the Normal (Line of Sight) vector.

   The *Up* data field becomes selected.

4. Drag a line in the active view window indicating the **Up** direction.
5. Click OK.

   The dialog box closes and the new view is defined.

6. Click Set View.

   The active view displays the new orientation.

7. Rename this view if you want to give it a more distinctive name.
8. Close the Define View dialog box if you have no other need to define work views.

Be aware that simply rotating the view does not alter the orientation of the work plane in 3D space.

Deleting a View Orientation

Select the view orientation to be deleted from the View list and click the Delete button.

Redefining a View Orientation

You have two methods to change the orientation of the axes of a view.

By Example

1. Manipulate the view orientation manually with the Trackball.
2. Choose ***Views>Define View***.
3. Select the view you want to redefine.
4. Click Redefine.
5. Click OK.

   The Redefine View dialog box closes.

6. Dismiss the Define View dialog box if you have no other need for it.

By Definition

1. Choose ***Views>Define View***.
2. Select the view you want to redefine.
3. Click Redefine.

   The Redefine View dialog box displays the settings as they appear on the screen with Normal (line of sight) selected.

4. In the active view window, drag the Normal (line of sight).

   The *Up* data field becomes selected.

5. Drag the direction for up.

   When you drag the normal vector with the mouse, know that the vector, from the beginning point, points directly at you, not away from you. So when this new view is made current, the normal vector points out of the screen, not into it.

6. Click OK.

   The Redefine View dialog box closes.

7. Dismiss the Define View dialog box if you have no other need for it.

Renaming a View Orientation

1. Click the view orientation to be renamed.

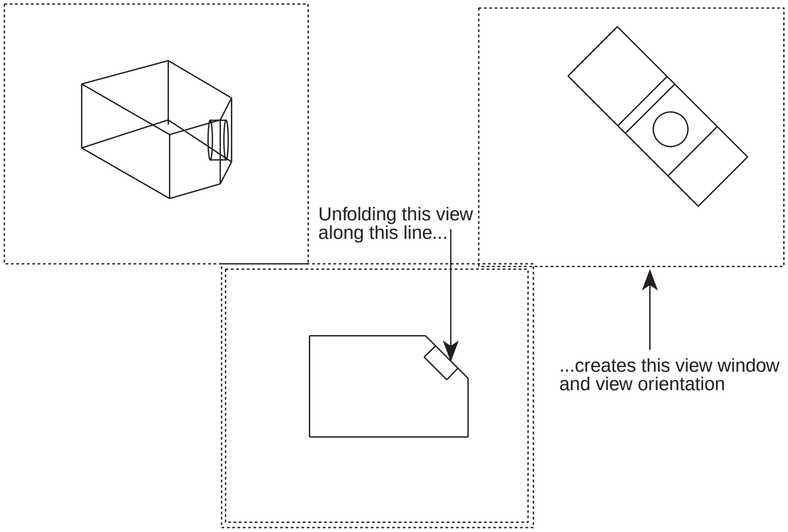
2. Type a new name.
3. Click Rename.

Renaming a Standard View Orientation

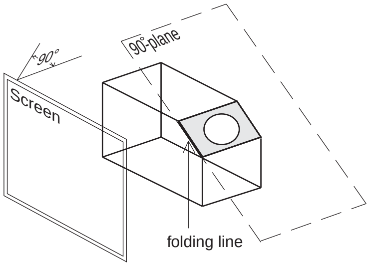
1. Click the standard view name in the list box.
2. Click the Locked box to toggle it off.
3. Type a new name.
4. Click Rename.

Unfold View Command

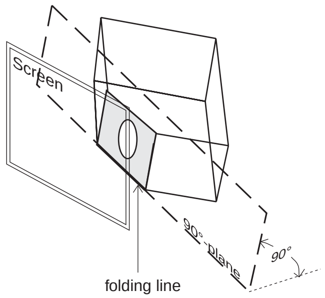
The command in the Views menu allows you to create a view orientation from the active view window by specifying a line about which to unfold the new view orientation.



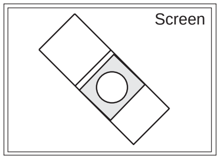
By specifying a line you define a 90°-plane along that line, which is coming towards you away from the screen. This plane is folded by 90° and creates the new view orientation from the active view.



before unfolding



after unfolding



Unfolded view

Note: You should use this command only in a view orientation aligned parallel to an object face. If you use it in a differently aligned view (object face is not parallel to the screen) this command will work correctly, but the result will be unpredictable in most cases.

Unfolding a View

1. Choose ***Views>Unfold View***.
2. Click the endpoints of the line to unfold on.

   You don't need an actual line, you can just indicate a vector.

   The active view window shows the view unfolded 90° from the line you specify.

Setting the Screen to the Work Plane

Occasionally, you might want to change your point of view to look directly at the work plane. You can choose ***Views>View the Plane*** to do this.

View the Plane Command

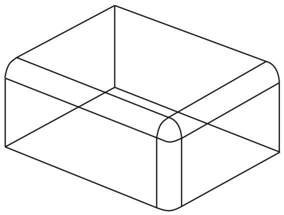
This command in the Views menu rotates the view orientation in the active view window to match the work plane. The view changes so the work plane becomes horizontal on the screen. The **y**-direction of the work plane becomes vertical, and the z-direction of the work plane becomes the line of sight, coming directly out of the screen.

To leave this view orientation, select a view from the Trackball pull-down menu or the Views submenu, or use the Trackball for free rotation.

Editing a View Without Editing the Model

The geometry within view windows is associative. When you make a change in one window, the change affects the model and therefore the geometry in all views. In the drafting phase of the design process, you may want to edit the geometry in one view window without making the same changes to the model. For example, you may want to clarify the view by removing a line or adding an arc visually, but you do not want to make the actual change in the model.

You can accomplish this by using the *Flatten View* command on the Views menu. When you flatten a view, the geometry is taken out of the view and placed back on the sheet, where it is no longer associated with the model. In this way, you can make changes to the geometry without changing the model or other views.



A typical situation for flattening a view is when a view should display fillets but you don't want the fillets to display in the model.

Trimming such intersecting fillets in 3D is a difficult problem that was addressed in a tutorial of the *Getting Started* section of this manual, but you will find Graphite makes this problem very easy to solve.

Flatten View Command

This command in the Views menu places a projection of the visible geometry in the active view window onto the sheet at full scale. When the view is flattened, all overlapping lines of equal length and all lines parallel to the line of sight are removed. The geometry is no longer associated with the model. If you make changes to the flattened geometry, those changes do not affect the model. If you make changes to the model, the flattened geometry is not changed.

1. Select a view window to make it the active view.
2. Choose ***Views>Flatten View***.

   A dialog box displays a warning message. Since flattening disassociates the current view from the model, you are given this opportunity to be certain you have selected the view you want to flatten and that you really want to flatten the selected view. All dimensions in the view are deleted because the 3D dimensions would not be correct in 2D geometry.

3. Click OK. The geometry is placed on the sheet at full scale.

Important:

- The geometry will also be flattened at the scale that you are viewing in. This scale is listed in the Control Menu of the Detail View.
- Text and crosshatching are treated like geometry by this command.
- Dimensions will no longer update if you make a change to the 3D model since the flattened geometry is disassociated, but they will change if you alter the flattened geometry. Also if you dimension your geometry after it is flattened, you will have to enter the desired values because Graphite will simply read the length of the projected flat lines. If you want to keep the dimensions, copy the view and flatten the copy, or group the geometry with the dimensions, then flatten the view.
- If you receive the following message, *The current view and the draft view have the same model*, you must change the model on the sheet. Click the sheet, outside all views, then choose ***Views>Models***. Click New and then click Current. Repeat the above steps to flatten the view.

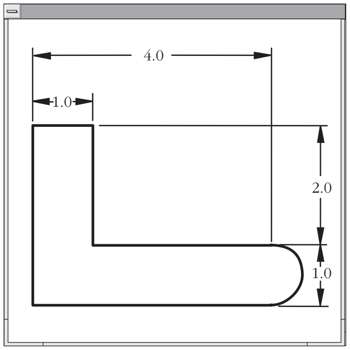
View Layouts

Included with Graphite are layouts set up with various views that you can choose using the *Sheet into View* command in the Views menu. You can also create your own view layouts. In all but the *Design.vc6*, each view has the same scale. In the *Design.vc6*, the Trimetric view is scaled as specified in the Sheet Into View dialog box and the other three views are scaled to 25% of the Trimetric view.

The default layouts available from the *Sheet Into View* command are as follows:

View Top

This layout shows the Top view of geometry, just as it is seen in 2D.



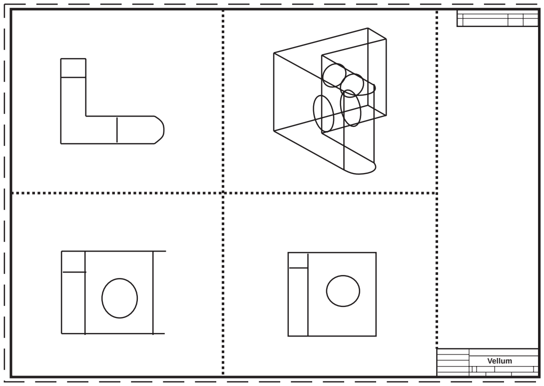
Views with Standard Drawing Formats

A Landscape 4 VIEW.vc6 (Windows & Macintosh); =
4 views for an A size sheet

B Landscape 4 VIEW.vc6 (Windows & Macintosh); =
4 views for a B size sheet

C Landscape 4 VIEW.vc6 (Windows & Macintosh); =
4 views for a C size sheet

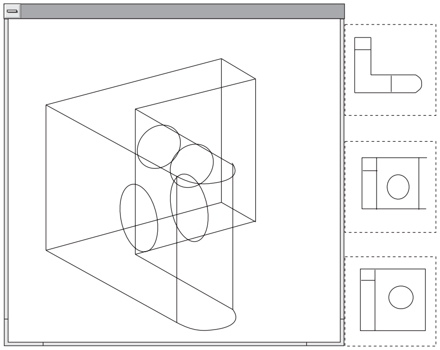
D Landscape 4 VIEW.vc6 (Windows & Macintosh); =
4 views for a D size sheet



Design.vc6 (Windows); Views Design 4 (Macintosh)

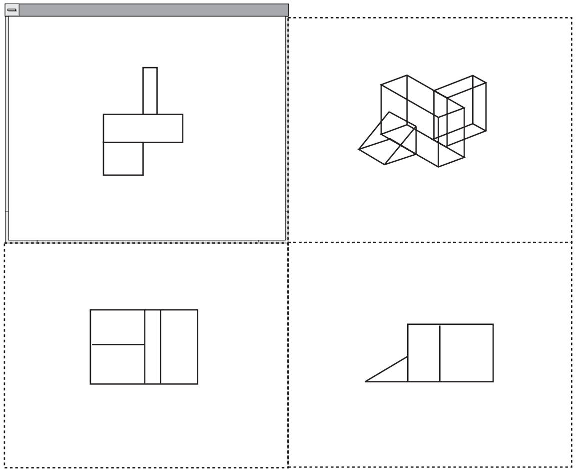
This layout creates a Trimetric view at full scale and Top, Front, and Right views at 1/4 scale.

The design layout is primarily for viewing the model as you design it. If you choose this layout, you may want to change the scale and rearrange the smaller views to prepare for plotting.



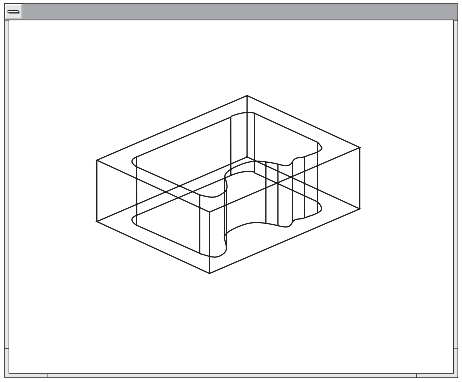
Draft.vc6 (Windows); Views Draft 4 (Macintosh)

This layout creates four full-scale views: Top, Front, Right, and Trimetric.



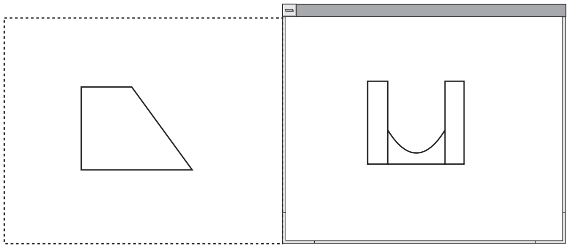
Trimtric.vc6 (Windows); View Trimetric (Macintosh)

This layout is a single Trimetric view at full scale.



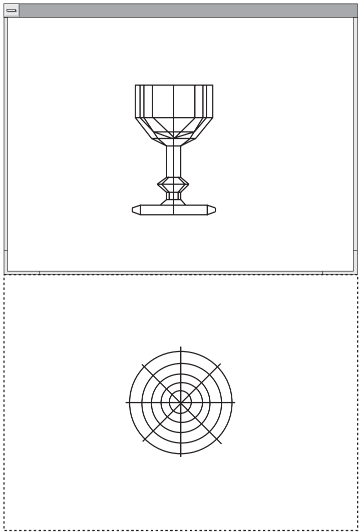
Frntrit.vc6 (Windows); Views Front and Right (Macintosh)

This layout displays 2 full-scale views, the Front and Right.



Frnttop.vc6 (Windows); Views Top and Front (Macintosh)

This layout displays 2 full-scale views, the Top and Front.



Creating Custom View Layouts

The specifications for the view layouts available from the Sheet into View dialog box are in individual Graphite drawing files stored in the Layouts folder in the Graphite folder. If you want to create your own layout or a customized drawing format, simply edit one of the existing drawings in the Layouts folder, or create a new drawing and save it in the Layouts folder. The filename will then appear in the pull-down menu in the Sheet Into View dialog box.

Method 1

1. Choose ***File>Open*** and open a Layout file in the Layout folder.
2. Save the file under a different name in the Layout folder with by choosing ***File>Save As*** before you do any changes.
3. Customize the layout to your needs.
4. Save the file.
5. Select ***Views>Sheet Into View***.
6. The new layout appears in the pull-down menu in the Sheet Into View dialog box.

Method 2

Another way to create your own layout is to design it in a new document.

1. Open a new document by choosing ***File>New***.

Tip:

You might want to look at one of the files in the Layouts folder to see how it is set up. Be sure not to make any unwanted changes to these layout files.

2. Create a view window with the Detail View function.
3. Make the current sheet active by clicking anywhere outside the view window on the sheet.
4. Choose ***Views>Model*** and create Model 2.
5. Make Model 2 the current model in the Models dialog box.
6. Duplicate the view window as often as you want.

   Duplicating or copying a view was discussed in Chapter 13.

7. Set each view window to the desired view orientation with the on-screen Trackball pull-down menu or the Views submenu in the Views menu.
8. Select *Properties* in the detail view window's control menu to scale each view window.
9. Save the new layout under a proper name in the Layout folder of the Graphite folder.

Drafting Methods with Views and View Layouts

Two different methods exist for creating 3D wireframe models:

- Start the model creation on the drawing sheet and add views later for editing and viewing the model from different angles
- Use a view layout from the beginning and create the model in different view windows simultaneously.

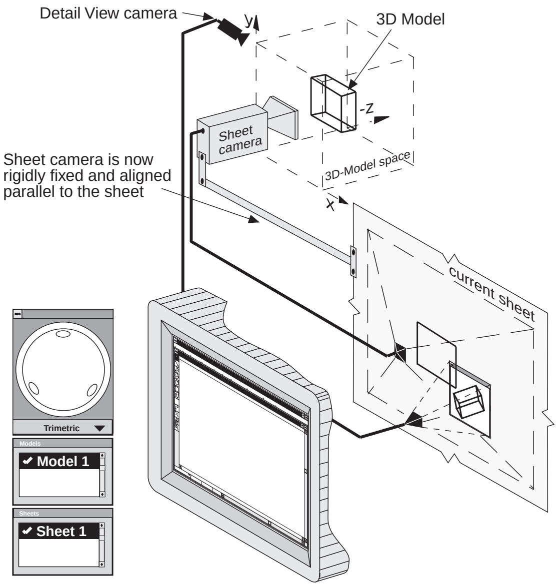
Method 1

Open a document and create the geometry on an empty sheet. With the on-screen Trackball and the view commands in the View submenu, you can rotate your model and edit it from different angles as long as you have not created any detail view windows.

If you want to view your geometry from different angles simultaneously you create detail view windows with the **Detail View** tool and place them on the sheet. The sheet view then shows an image of your model in the Top view on the sheet. In the view windows, you can display different view orientations of your geometry, like front, top, trimetric, etc.

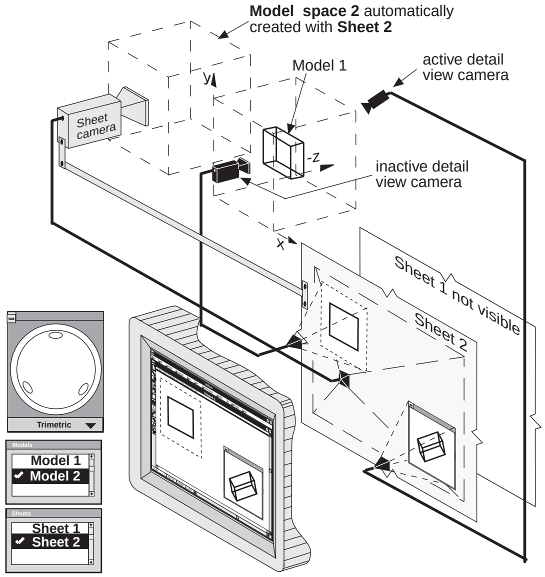
As soon as you create a view window, the sheet view of your geometry remains fixed in the Top view parallel aligned and rigidly fixed to the sheet. As a result the view angle can only be changed in the detail view windows.

The view orientation of a sheet with a detail view cannot be rotated with the Track-ball.



1 Model with 1 additional active **Trimetric** view window

The sheet view of your geometry may overlap the view windows on the sheet as shown in the upper graphic. You could create a new sheet with the Sheets dialog box in the View menu and *Cut* and *Paste* the view window from Sheet 1 onto Sheet 2 to get a clear view on the view windows.



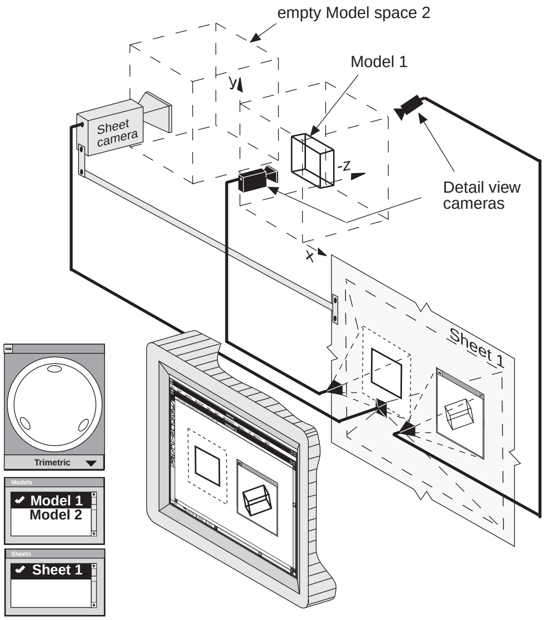
Right and **Trimetric** view window on **Sheet 2**.
Model space 2 is empty, **Sheet 1** is not visible.

Something similar, but more elegant, is done by the *Sheet Into View* command in the Views menu. *Sheet Into View* puts Model 1 into one or more views and puts Model 2 on the current sheet.

Method 2

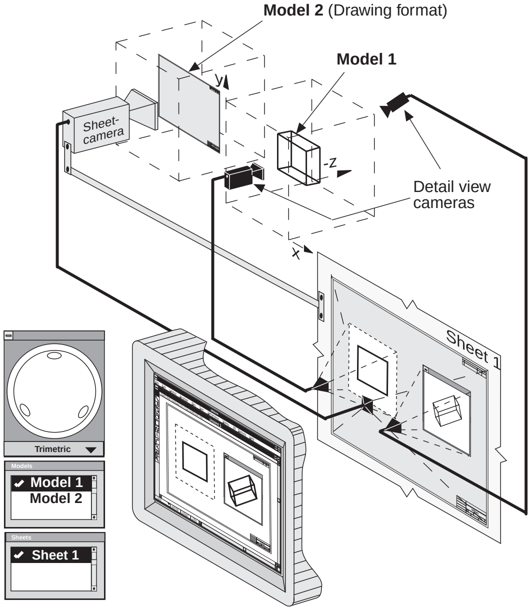
With this method you choose a view layout with the *Sheet Into View* command. The views provide a vantage point of the model. It places all geometry into one of the view layouts. You can even customize the view layouts.

Sheet into View puts Model 1 into one or more views as specified by your choice from the dialog box and creates Model 2 and projects it onto the current sheet.



Layout with views **Front** and **Trimetric** loaded with **Sheet Into View....**

- If you choose one of the layouts that contain a title block and a border format, that format becomes Model 2 and is projected by the Sheet camera in the Top view on the current sheet.



Layout with views **Front** and **Trimetric** including drawing forma
loaded with **Sheet Into View...**.

Combining Sheets, Views and Models

In general, you need a powerful CAD program for two tasks:

- The creation of your designs

  and

- the generation of 2D engineering drawings, or blueprints, in order to get the designs built.

The design phase is effortless and very productive with the help of Graphite's Drafting Assistant, Integrated Parametrics (Chapter 16), and the intuitive user interface itself.

For the second of these two tasks, you need very powerful and flexible drafting functions, since there are so many possible types of drawings that you might need to create. One of the specific features in Graphite that handles this so well is the combined power of Sheets, Views and Models.

For most of your daily work, it is not necessary to know anything about the relationship between Sheets, Views and Models. But for some tasks it is helpful to understand this relationship since it offers elegant solutions that were not possible without the combined power of features in Graphite.

Using the relationship between Sheets, Views and Models makes it easy to:

- place differently scaled views of a part, such as 1:50 in a standard drawing format with title block and borders that must be plotted at a scale of 1:1.
- create detail views if an object that is not associative to the original model.
- create customized drafting layouts.

  or

- recover from an erroneous *Sheet into View* command.

The first section of this chapter explained the use of Sheets, Views and Models. Each is a simple and straight forward operation. In the graphics you could see the environment, and how sheets, views, and models are set up in Graphite. What you still need to know is the exact definition of all engaged components and the rules that describe how they are interact.

Components

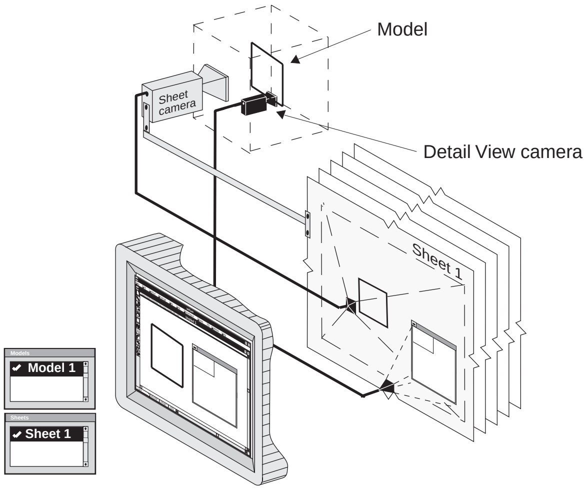
The environment in Graphite for handling all geometry you create can be described by five components—sheets, models, views, cameras, and projectors.

First, you have to know that all geometry you create is not created directly on the sheet you see on your computer screen but somewhere outside of the sheet in an infinitely large three-dimensional work space (Model space). All geometry is placed here as separate models. You only see images or views of these models on your sheet.

Sheets

A sheet is an infinite 2D planar area that displays an image of one or more models. The image of a model is picked up either by the Sheet camera and projected onto the sheet (the Sheet View) or by a Detail View camera and projected into a view window which is resting on the sheet (the Detail View).

- Sheets are arranged behind each other so you can see only one at a time.

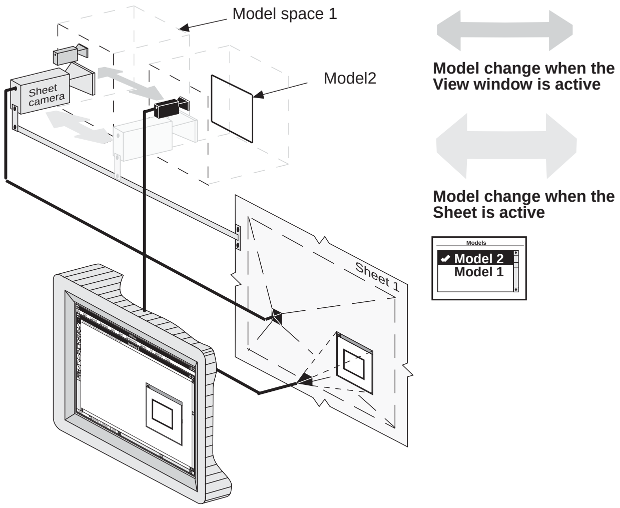


- You can create as many sheets as you want, but with each sheet you automatically create a blank model, similar to the drafting board when you start with a clean sheet.
- Independent of how many models exist, you can delete all sheets but one—the current sheet—since Graphite needs at least one sheet to display the models even if they are blank.
- When you delete a sheet, all Detail Views resting on that sheet are deleted.
- You can display as many Detail Views as you want on one sheet but only one sheet (Sheet View) at a time.
- When you change the current sheet, the related model is always activated. You can change this relation only by assigning another model to that sheet.

Models

A model is a collection of geometry, dimensions, text, fills and hatching. Models are placed in an infinite three-dimensional area. A model can be blank (then we call it Model space) just as it is when you launch Graphite and look at a blank sheet. Images of models are picked up by either the Sheet camera or by Detail View cameras and projected on the sheet. The view of the Sheet camera is called a Sheet View and the views of the detail cameras are called Detail Views.

- You can create and delete as many models as you want.
- When you change the current model, you move the related camera. The camera you move depends on what is active—the sheet or the detail view. If the sheet is active, the Sheet camera moves to the current model. If the detail view is active then the related Detail view camera moves.



Views

Views are the images picked up by cameras and projected on sheets. Graphite has two types of views—the Detail View and the Sheet View.

Detail Views

- Detail Views are picked up by Detail View cameras and always displayed in a view window which rests on the sheet. They have boundaries and display only a limited view of the model.

  You can create as many views as you want and move or copy and paste them to different sheets.

  When you delete a detail view, the Detail View Camera deletes. When you delete a sheet which contains a detail view, the detail view deletes also since the detail view rests on the sheet.
- You can place views from different models on one sheet, but each view can display only one model at a time.

- Once your create a detail view, the sheet camera remains stationary in the original x,y (world) orientation. You, then, will not be able to change the view of the sheet.

Sheet View

Tech Note:

The *Drawing Size* command changes only the plot scale but not the scale factor of the Sheet view. Even if the view is scaled to fit the paper format the scale factor remains 1:1 since the real size of the geometry is not affected.

- There is only one sheet view for each sheet. The sheet view is an infinite view picked up by the Sheet camera and displays everything on the sheet outside of all view windows.

  The sheet view cannot be deleted and needs at least one sheet to display its view.

  Since the sheet view, like the detail view, can display only one model at a time, you have to use detail views to show more than one model on a sheet.

- To activate a detail view you have to click in the view window; to activate the sheet view you have to click on the sheet outside of all detail views. If the Models dialog box is displayed, the related model is highlighted.
- The view orientation of a sheet becomes fixed when a detail view is created. The sheet camera remains stationary in the original x,y (world) orientation. You will not be able to change the view of the sheet.

Cameras and Projectors

Tech Note:

When you change the sheet, the Sheet camera is moved to the selected sheet and automatically displays it at the view scale which was current when the sheet was last changed.

There are two type of cameras—one Sheet camera and as many Detail View cameras as you create detail views.

- The Sheet camera is permanently installed as default and displays its view via the sheet projector on the sheet.

  To move the Sheet camera you have to perform a model change either by the *Models* command or by the *Sheets* command in which the Sheet camera moves automatically to the related model.

- Detail View cameras display their views via Detail View projectors in view windows which rest on the sheet.

  You can install as many Detail cameras as you want even if each one is looking at one model. You install Detail cameras by creating Detail views either by the **Detail View** tool from the tool palette or with the *Sheet Into View* command.

  You can move Detail View cameras to another model by performing a model change when the view window is active.

- Both cameras, the Sheet camera and a Detail View camera can look only at one model at a time.

Using Sheets, Views and Models

We listed many rules in the sections above and they all describe the same fact that Graphite administrates geometry (models), views and sheets separately. That has a lot of advantages. The most impressive example is if you delete a sheet displaying geometry. The geometry is not lost since you have not deleted the related model.

If you practice the following examples and implement them in your daily work, using Models, Sheets and Views will become second nature and you will never want to work without them.

Recovering from Deleting a Sheet Displaying Geometry

To restore the geometry displayed on a sheet you deleted unintentionally, proceed as follows:

1. Select ***Views>Sheets***.
2. Click New to create a new sheet.
3. Click Current to activate the new sheet and draw some geometry.
4. Make another sheet active by highlighting a different sheet in the Sheets dialog box and click Current.
5. Close the Sheets dialog box and select ***Views>Models***.
6. Search for the before-displayed geometry by activating each Model followed by a *Zoom All* command until the previous geometry displays on the new sheet.

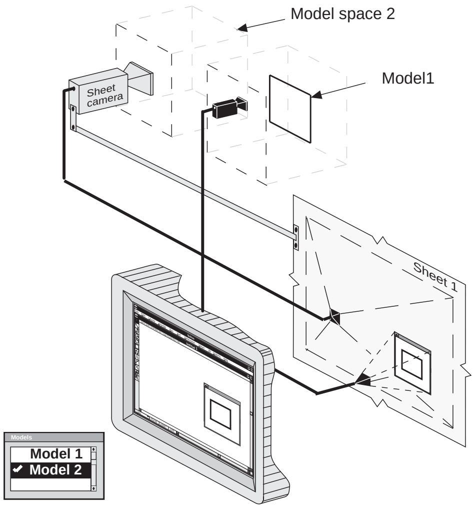
Displaying Three Components on One Sheet

Sheets, Views and Models are very helpful to display several components of a part created by different drafters on one sheet. The following simple example shows how to perform this task:

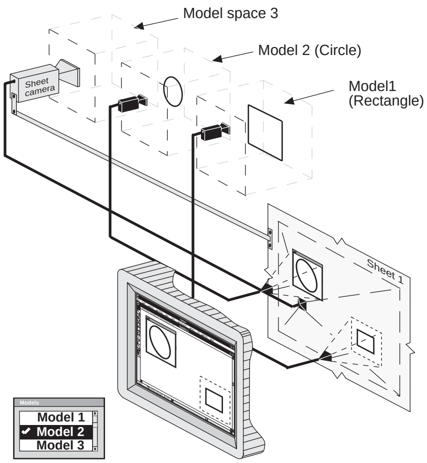
1. Open a new document.
2. Draw an ellipse.
3. Save the drawing as *ellipse.vc6*.
4. Open a new document and draw a rectangle.
5. Select ***Views>Sheet Into View***.

6. In that dialog box, choose the ViewTop layout, set the Scale to 1 and click OK.

   You now have one **Sheet** with one view window showing the Rectangle (Model 1). The Sheet camera aligns to Model 2 (created with the *Sheet Into View* command) displaying a blank model space on the sheet.



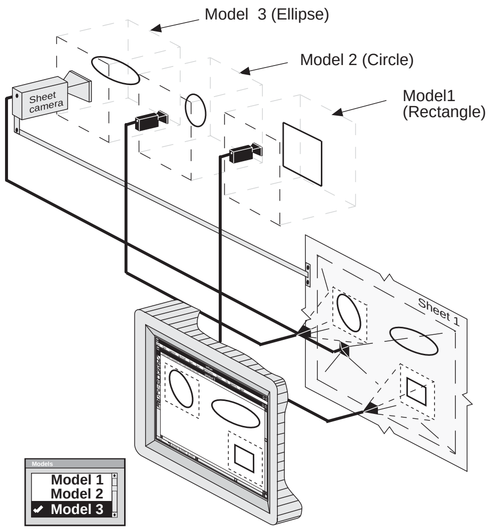
7. Make the sheet active by clicking on it and draw a circle.
8. Select *Sheet Into View* (the sheet still active) and choose Top view again with the Scale factor 1.



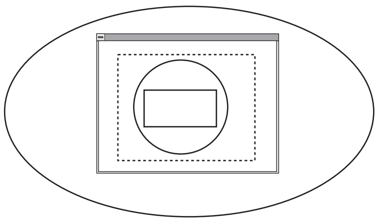
Now you have one Sheet with two view windows, one displaying the rectangle (Model 1) and one displaying the circle which became Model 2. The Sheet camera moves to the new Model 3 and an empty model space is displayed on the sheet.

9. Activate the sheet by clicking on it and select ***File>Import***.
10. Select the file *ellipse.vc6* and click OK.

The ellipse imports onto the current sheet and is added to Model 3 which was active when you performed the *Import* command.



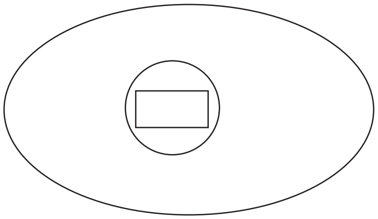
11. Arrange the two view windows around the ellipse and rearrange the rectangle and the circle with the *Zoom* commands in the Arrange menu like in the following graphic.



Tech Note:

In order to select overlapping view windows, *Auto Front* must be deactivated, as described in Chapter 13, "Viewing Geometry."

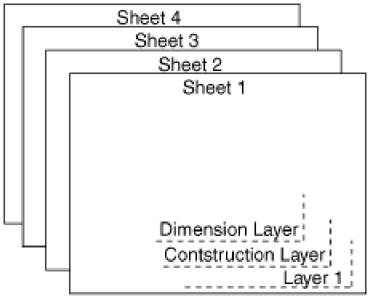
12. Activate the sheet and in the Views menu deactivate the *Show View Boundaries* command.



Graphite Documents

A document is a Graphite file. Whenever you open Graphite or choose New from the File menu, a new document appears in its window.

The sheets and layers features add a great deal of flexibility to your documents. They allow you to see various parts of a document individually or as part of the whole. Each document has one or more sheets, like the sheets of paper in a set of blueprints, and each sheet is made up of layers which can be hidden or displayed as needed.



Graphite Document

If you are new to computers, you may be too busy learning about Graphite and how the computer operates to think very much about how to organize your drawings. You should consider organization early, however, since eventually you will end up with a large number of drawings. Organization is particularly important if you are sharing files with other people.

The following topics are covered in this chapter.

- Using Documents, including opening, recent file list and saving
- Importing/Exporting
- Drawing at Full Scale
- Drawing Scale and Paper Size
- Drawing Formats
- Forms (Title blocks)
- Printing and Plotting

Using Documents

A drawing can be a simple part or a complex assembly. A new document opens as *Untitled* and remains untitled until you save it. Saving a drawing is explained later in this chapter.

The commands you use to manipulate documents are in the File menu.

New - CTRL+N (Windows); ⌘+N (Macintosh)

This command in the File menu creates a new Graphite document.

The new document has no name (the title bar shows Untitled 1), and is set with the default options, such as pen style or grid display.

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

Opening Files

You can open files using the *Open* command in the File menu in the program or by double-clicking on the file to launch the program and the file.

Open Command - CTRL+O (Windows); ⌘+O (Macintosh)

This command in the File menu opens an existing Graphite document.
The document appears in the drawing area maintaining the same settings as the last time you saved it.

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

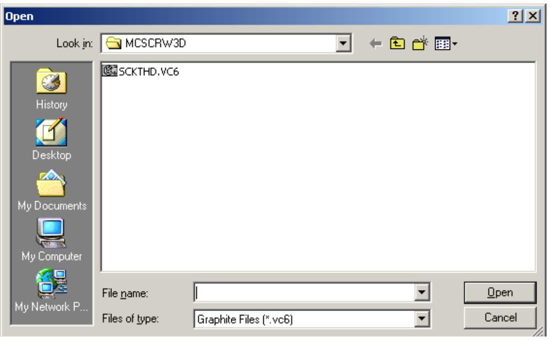
Tech Note:

Use the *Import* command if you want to use drawings of other format types, such as *DXF*, *IGES*, or *ASCII* text.

Opening a Document

1. Choose ***File>Open***.

   The dialog box appears.



   The current folder displays with the files and/or folders contained within.

2. Display the appropriate folder containing the document you want to open.

   For Windows, *.vc6* is the Graphite file extension.

3. Click the file name you want to open in the list box.
4. Click OK.

Opening a File when Graphite is not Running

You can open a file and launch Graphite at the same time by double-clicking on the file. The program launches automatically and the file opens. By using this technique, no untitled file opens.

Recent File List

Another way to access files that have been opened recently is through the *Recent File List* that appears in the File menu after the *Exit* command. This list contains the names and paths of the most recent files that have been opened by Graphite.

To open a file from the *Recent File List*, simply select the file from the File menu. If the file has been moved since it was last used and the path is no longer accurate, Graphite will ask you to locate the file by providing the standard Open dialog box.

Tech Note:

The number of files contained in this list can be set in the *Graphite.ini* file (Windows) or the recent .lst file in the Scripts folder (Macintosh). For more information, consult Chapter 5, "Basic Environment Settings."

Saving a Drawing

You can save a drawing by choosing ***File>Save*** or ***Save As***. The file is stored on your computer in the folder you specify.

Tip:

Use AutoSave so that Graphite automatically saves your work. Choose ***Layout>Preferences>AutoSave***.

Save Command - CTRL+S (Windows); ⌘+S (Macintosh)

This command in the File menu saves the current Graphite document to its original folder. If you want to save it to a different folder or with a different name, choose *Save As*. If you choose *Save* and the document has not been saved previously, the Save As tool dialog box appears automatically, allowing you to name the document and specify the folder in which to save it.

If you have named and saved the document before, a brief message appears when you choose *Save*, and the program pauses while it updates the information.

You should save frequently. Even though your drawing appears on the screen, it is not stored on the disk until you save it. Hours of work can be lost because of a power failure. It is also important to save before performing any intricate, multistep maneuver. In that way, if the result is not exactly what you had hoped, you can abandon the file by closing it without saving.

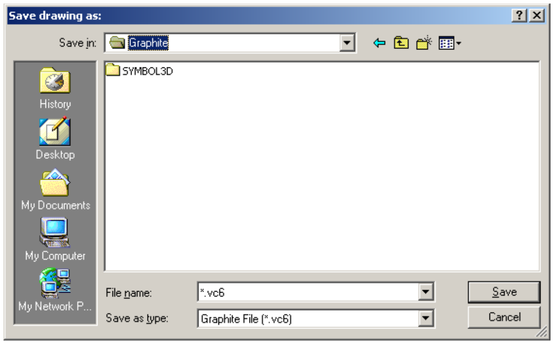
Save As Command

This command in the File menu saves the current document. A dialog box appears so you can name the current document, give it a different name, or save it to a different directory.

Saving in the Current Directory

1. Choose ***File>Save As***.

   The Save Drawing as dialog box appears.



2. If necessary, display a different folder.
3. Type the name you want to use in the File name data field.
4. Either press ENTER (Windows), RETURN (Macintosh) or click OK.

   For Windows, the filename extension *.vc6* automatically appends to the filename.

You can use the *Save As* command to make a backup of a document.

Memory Exhaustion Warning (Mac only)

If the size or complexity or your geometry uses all memory allocated to Graphite, the program will crash. The following dialog box will appear notifying you:

Before shutting down, Graphite will display a Save as dialog box allowing you to save the file first.

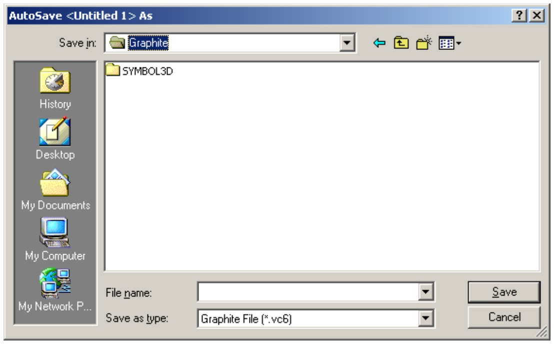
Making a Backup

You should make a backup of your work in case you make many changes and want to go back to the original version. You can choose ***File>Save As*** and save the file with another name. It's possible that your company makes periodic backups of your directories, and you can go back to a previous version that way.

AutoSave Command

This command is found under ***Layout>Preferences*** and directs Graphite to save a backup copy of your work periodically. If your computer hangs up for any reason, the work you did up to the last AutoSave will be recoverable.

In the AutoSave dialog box you can turn AutoSave on or off, and specify the time interval (in minutes) between AutoSaves. At the specified interval, Graphite creates an AutoSave file with the base of the original file name and a *.SAV* extension in the same folder as the original file. AutoSave prompts you for a file name before creating an AutoSave file for untitled documents (created by using the *New* command from the File menu). If you supply a file name, this file name will be associated with the document and the AutoSave file. If you Cancel, AutoSave will skip that document, but the prompt will appear again at the next AutoSave interval.



For Windows, files opened with the *READ ONLY* attribute are also autosaved.

Setting up AutoSave

1. Choose ***Layout>Preferences>AutoSave***. The AutoSave dialog appears.
2. Switch AutoSave on or off. The default setting is off.
3. Specify the time interval for AutoSaving in minutes in the AutoSave Interval data field. The interval can be from 1 to 60 minutes; the default setting is 15 minutes. An interval of 0 or less is equivalent to turning AutoSave off.
4. Click OK.

If *AutoSave* is ON your work automatically saves for the specified time interval. To save these settings permanently, choose ***Layout>Preferences>Save Preferences***. If you are working files including an untitled file open with no geometry, close that untitled file. Otherwise, with AutoSave on, the *Save As* command appears as the default setting.

Close Command- CTRL+W (Windows); ⌘+W (Macintosh)

This command in the File menu closes the current Graphite document (the one displayed in the top window). If other Graphite documents are open, they remain open when you close the current document. If you have made any changes since you last saved the current document, a dialog box allows you to save the changes. You can close the document with or without saving the changes.

For Windows, you can also close the document by double clicking the Control menu at the upper left corner of the title bar. For Macintosh, click the Close button once at the upper left corner.

Exit Command - CTRL+Q (Windows); ⌘+Q (Macintosh)

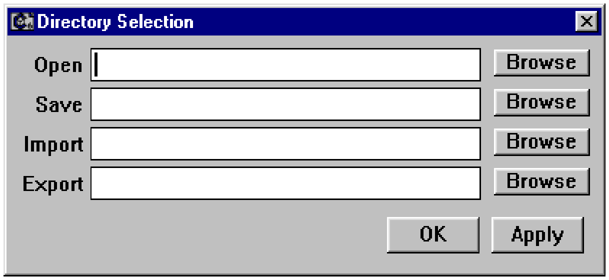
This command in the File menu closes Graphite. If you have made changes since you last saved, a dialog box allows you to save the changes. If you have more than one document open, an alert message allows you to save unsaved documents.

File Organization

Graphite provides you with the ability to automatically organize your files using the Directory function in *Preferences*. When you open, save, import and export files, Graphite goes to the directory or folder you specify.

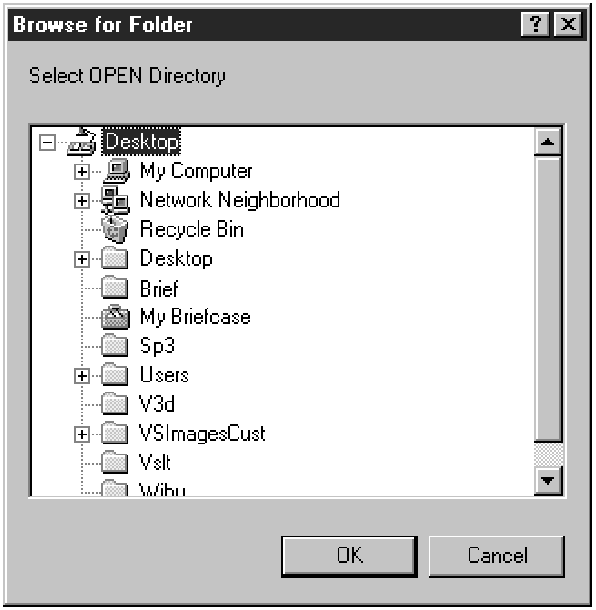
1. Choose ***Layout>Preferences>Directory***.

The Directory Selection dialog box appears with four fields for specifying the directory location.



2. Enter the directory path in each data field. You can either type the path or browse your computer for the folder and have the path entered automatically.

   To browse, click on the ellipsis (...) and the following dialog box appears:



   Locate the desired folder and click OK.

The path appears in the selected data field of the Directory Selection dialog box.

3. Repeat step #2 for all of the directories you want specified.
4. Click Apply to save the settings and close the dialog box.

Importing and Exporting

There are various reasons to use these features. You might want to bring a document in from another application to work on in Graphite. You might need to save a document in a format other than the Graphite format for use with another application. You also might need to import or export text to or from a Graphite document that's an ASCII text file. Graphite offers you a wide variety of options to accomplish this.

Import Command

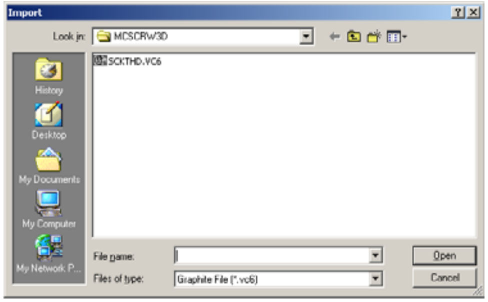
Tip:

When importing into Graphite, you may have to choose ***Arrange>Zoom All*** to see the imported geometry.

This command in the File menu imports a document and places it in the current model. The names of documents in a format Graphite can read are displayed in a dialog box that works like the Open dialog box.

1. Choose ***File>Import***.

   A dialog box appears similar to the Open dialog box.

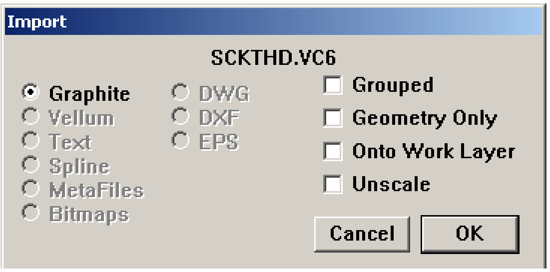


2. (Windows) Select the file type you want to import from the *Files of Type* list box. The list box shows all files of the selected type available in that folder.

   (Macintosh) Skip to the next step.

3. Double-click the File name you want to import.

   The Import dialog box appears.



4. Specify any import options you want.
5. Click OK.

The file appears in the drawing area. If you want to save the file in its original format after editing, you must choose the ***File>Export***.

The file formats Graphite can import are listed below:

Graphite	Ashlar-Vellum's native file format
Text	ASCII text file

DXF	AutoCAD's Data eXchange Format. R14 and ACAD2000 files can be imported.
DWG	AutoCAD's native file format. R14 and ACAD2000 files can be imported.
IGES	IGES version 4.0 Initial Graphics Exchange Specification. The translator also supports IGES entity 128 and 144 NURB surfaces and importing an unlimited number of entities.
EPS	Encapsulated PostScript format for printing to a postscript printer.
Spline	Text file, which Graphite converts to a spline.
MetaFiles	(Windows only) Windows Metafile Format
Bitmaps	(Windows only) Windows BitMaP Format
Pict	(Macintosh only) Macintosh native file format.

When you use the *Import* command, Graphite detects the format of the document and displays a dialog box showing you the format and allowing you to choose how to bring the geometry into Graphite.

Grouped	Brings the geometry into Graphite grouped so it is a single unit in the Graphite document.
Geometry Only	Brings in only the geometry, leaving any text, dimensions and crosshatching behind.
Onto Work Layer	The imported geometry is placed on the work layer rather than the layer on which it was originally created.
Unscaled	When checked, this option will use the drawing scale to resize your part. (i.e. a 12" line will become six inches with a 2:1 Drawing Scale.)

Tip:

You can ungroup the imported geometry by choosing ***Arrange> Ungroup***.

Importing DXF Files

When you import a DXF file, the geometry is constructed according to the units set in Preferences when you choose the *Import* command. Be sure to set the appropriate units before you import DXF geometry.

Importing ASCII Text from Another Document

1. Choose ***File>Import***.

   The Import dialog box appears.

2. Select the filename you want to import.
3. If *Text* is not selected, click the Text button.
4. Click OK.

   The text appears in a text box in the drawing.

Importing Splines

When you import a text file that contains the coordinates of a spline, Graphite creates the spline according to the imported coordinates.

1. Select ***File>Import***.

   Graphite displays the standard Open dialog box.

2. Select a text file that contains the coordinates for the spline.

   The Import dialog box displays.

3. Specify the import option *Spline*.
4. Click OK.

   Graphite begins creating the Spline.

Creating a Text File for Importing a Spline

1. Use a text editor, a word processor or a spreadsheet to create a text file.
2. Input X, Y and Z values for your spline coordinates. (In the case of Graphite enter X and Y values.)

   The text file should be tab or space separated. Each line ends with a return. Line feeds after each return should have no effect.

   The text file should conform to the following columnar format:

1	**1**	**0**
2	**2**	**0**

   You are allowed to specify decimal coordinates as well:

1.33	**1.1**	**0**

2.4 2.5 3.5678

Be sure that you press ENTER (Windows) or RETURN (Macintosh) after the last coordinate, if not Graphite will not import the coordinates specified in the last line.

3. Save the file as *Text only* and import into Graphite to create the spline.

How Importing Affects Some Objects

It is difficult to exchange data between different graphics programs because each program defines geometric objects differently. All entities created in one program may not translate accurately or at all depending on what the program supports. The following lists are specifics about entities for two translators.

DXF/DWG	**Graphite**
Blocks	Convert into groups.
Center Mark	Convert into lines.
Crosshatch	Convert to grouped lines.
Donuts	Convert to grouped Three Point arcs.
Double lines	Convert to lines.
Points	Convert into Points.
Polygons	Convert into grouped lines.
Polylines	Polylines and 3D polylines convert into lines.
3D Face	Converts into lines that define the polygonal boundary of the face.
View Ports	Not supported.

EPS	**Graphite**
Line	Line - all shapes and attributes are maintained.
Curve	Splines - curves convert into splines. All shapes and attributes are maintained.
Fill	Fill - all shapes and attributes are maintained.
Fill/line	Fill - all shapes and attributes are maintained.
Non-fill/Non-line	Not supported.

Text	Not supported.
Weight	Converts to the current weight.
Color (white/black)	Converts to the approximate color (7 colors as defined by Graphite 2.7).
Process color	Convert to the approximate color (7 colors as defined by Graphite 2.7).
Custom color	Converts to the approximate color (7 colors as defined by Graphite 2.7).
Pattern	Converts to black fill
Layer	Layer - layer name and elements are retained. Layers that contain no elements are not retained.

BMI	**Graphite**
File start	(FLST) Not supported
File end	(FLEN) Not supported
Model Text	(MTXT) Not supported
Plot Data Definition	(PLOT) Not supported
Attribute Table Def.	(ATRB) Not supported
Grid	(GRID) Grid
Beginning of Diagram	(BEGD) View, Model
End of Diagram	(ENDD) View, Model
Beginning of View	(BEGV) View, Model
End of View	(ENDV) View, Model
Ditto	(DITO) View, Model
End of Ditto	(DITE) View, Model
Sets Definition	(SETS) Group
Attrib. - Elements/Sets	(ELNM) Group additional information
Color Table Definition	(CLOR) Color; you can define the color table with the VelBmi.ini file.

Appendix	(APND) Drawing elements - additional information.
Txt Parm.-Symb. w/text	(SYMP) Symbol character (character height only, width is determined by height)
Super Text Option Parm.	(TXTP) Character elements; character height only, width is determined by height.
Point	(PONT) Not supported
Line	(LINE) Line
Circle, Arc	(CIRC) Circle, Arc
Ellipse	(ELPS) Ellipse
Polygonal Line	(MLTL) Line
Spline/Offset Spline	(LSPL) Spline
Linear Spline	(LSPL) Line
Beginning of Areafill	(BEGA) Fill, hatch
End of Areafill	(ENDA) Fill, hatch
Areafill Polygon	(PLOG) Fill, hatch
Areafill Elements	(STRK) Hatch; you can define the hatching pattern table with the VelBmi.ini file.
SuperText	(SUPT) Text

Vertical, horizontal, parallel and normal dimensions are identified as Dimensions. Others are identified in case of expansion. The next nine items fall within this category.

ANSI Dimension	(DIMA) Dimension or general elements
ISO Dimension	(DIMI) Dimension or general elements
End - Dimension expan.	(DIME) Dimension or general elements
Beginning - (ISO) Dim.	(GEBI) Dimension or general elements
End - (ISO) Dim.	(ENDI) Dimension or general elements
Arc - Dim. Line	(34, 35) (SACI) Dimension or general elements
Witness/Polyg. Dim Line	(34,35) (ISAR) Dimension or general elements

Line-Dimension Line	(34,35) (SLNI) Dimension or general elements
Text-Dimension Line	(34,35) (ISTX) Dimension or general elements
Balloon	(BALN) Text, Line; Grouping general elements
Arrow with Text	(AROW) Text, Line; Grouping general elements
Delta	(DELT) Text, Line; Grouping general elements
Section Arrow	(SECT) Line; Grouping general elements
Text Line	(TXTL) Line, Text; Grouping general elements
Circular Symbol	(DOT) Circle; Grouping general elements
Rectangular Symbol	(RECT) Line; Grouping general elements
Rived Type Symbol	(RIVT) Line; Grouping general elements
Triangular Symbol	(TRIA) Line; Grouping general elements
User Definition Symbol	(SYMB) General elements; Grouping general elements
End - User Def. Symbol	(SYME) General elements; Grouping general elements

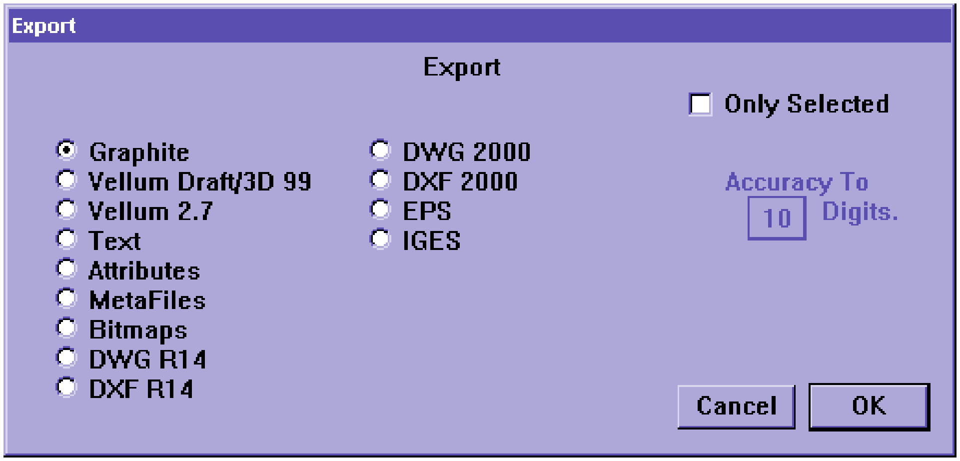
Export Command

This command in the File menu saves a document in the format you specify.

1. Choose ***File>Export***.

   The Export dialog box appears.

2. Click the desired file type. If you wish to export only selected objects on the screen, rather than the entire document, click *Only Selected*.



3. Click OK. The Export file dialog box appears.

   The Export dialog box is identical in both appearance and function to the Save As dialog box. For Windows, filename extension indicates the type of export file.

Types of Files

You can export the following file types:

Graphite	Ashlar-Vellum's native file format
DXF	AutoCAD's Data eXchange Format. R14 and ACAD2000 files can be exported.
DWG	AutoCAD's native file format. R14 and ACAD2000 files can be exported.
IGES	Initial Graphics Exchange Specification, version 4, the National Bureau of Standards format to standardize the exchange of graphics. The translator also includes the support of IGES entity 128 and 144 NURB surfaces and exporting an unlimited number of entities.
TEXT	ASCII characters. This option is available only if the **Text** tool is the active tool.

META	(Windows only) The format used by the Windows clipboard, is 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 *.WMF* automatically appends to the filename.
Bitmaps	(Windows only) Microsoft Windows Bitmap file format. The filename extension *.BMP* automatically appends to the filename.
Pict	(Macintosh only) Macintosh's Pict file format which uses object-oriented bitmaps or resolution independent graphics.
EPS	Encapsulated PostScript format for printing to a Postscript printer and for importing into compatible applications.

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

How Exporting Affects Some Objects

It is difficult to exchange data between different graphics programs because each program defines geometric objects differently. Even though the DXF and IGES formats standardize this exchange, some objects do not transfer exactly. For example, crosshatching is no longer associative although it does appear in exported files. The following table tells you what to expect when you export Graphite objects.

Graphite	DXF
Crosshatching	Supported.
Detail views	The view is lost, but the dimensions and text are retained.
Diameter dimension	The leader style may change.
GD&T symbol	The feature control frame converts to non-GD&T symbol representation.
Layers	Layers are maintained.

Lines with arrows	Lines with arrowheads at the start and/or end are converted into lines and solids.
Locked geometry	The geometry is retained, but it is no longer locked.
Line Weight	AutoCad 2000 only
Multiple sheets	Each sheet must be exported individually.
NURB splines	Convert to splines.
Radial dimensions	The leader style may change.
Smart Walls	The double lines convert to symbol representation.
Solid Fills	Convert into solids.
Symbols	Convert into blocks.
Tolerances	The tolerance is lost, but the dimension remains.
Graphite	**IGES**
Crosshatching	Converts to symbol representation.
Detail views	The view is lost but the dimensions and text are retained.
Diameter dimension	The leader style may change.
Fonts	Fonts are not maintained.
GD&T symbol	The feature control frame converts to a group of non-GD&T geometry.
Grouped geometry	The geometry is retained but it is no longer grouped.
Layers	The layers are lost but the geometry is retained.
Locked geometry	The geometry is retained but it is no longer locked.
Multiple sheets	Each sheet must be exported individually.
Radial dimensions	The leader style may change.
Smart Walls	The double lines convert to symbol representation.
Text	A multiple line note becomes many single lines.

Graphite	EPS
Line	Line - all shapes and attributes are maintained.
Circle	Curve - convert to a spline. All shapes and attributes are maintained.
Arc	Curve - converts to plural connected paths (a maximum of 8 paths and 45 degrees). All shapes and attributes are maintained.
Ellipse	Curve - convert to plural connected paths (a maximum of 8 paths). All shapes and attributes are maintained.
Spline	Curve - all shapes and attributes are maintained.
Fill	Fill - all shapes and attributes are maintained.
Hatch	Fill - patterns are not maintained.
Wall	Line.
Dimension	Not supported.
Text	Not supported.
Picts	Not supported.
Groups	Line, curves, fills - group attributes do not convert.
Layer	Layer - layer names, shapes, attributes and elements are retained. Layers that contain no elements are not retained.
Color	(7 colors as defined by v. 2.7) Color - converts to approximate color.
Weight	Line weight converts to 1 point.
Line patterns	Not supported - all lines are converted into solid lines.

All geometry is exported as individual elements, except in the following circumstances:

- Fill geometry
- Grouped geometry

Graphite	BMI
Sheets	Current view and model on the sheet only
Models	Current model only
Views	Current view only
Line	LINE
Circle, Arc	CIRC
Spline	SPLIN
Ellipse	ELPS
Text (textEd)	SUPT
Hatch	BEGA, ENDA, PLOG
Fill	BEGA, ENDA, PLOG
Smart Walls	LINE
Linear Dimensions	BEGI, ENDI, ISAC, ISAR, ISLN, ISTX
Radial Dimension	BEGI, ENDI, ISAC, ISAR, ISLN, ISTX
Diametric Dimension	BEGI, ENDI, ISAC, ISAR, ISLN, ISTX
Angular Dimension	BEGI, ENDI, ISAC, ISAR, ISLN, ISTX
Arc Length	BEGI, ENDI, ISAC, ISAR, ISLN, ISTX
Geometrical Tol.	General elements
Balloon	BALN, AROW
Group	SETS, ELNM, General elements

EPS Translator

This translator enables you to share data between Graphite and other eps applications. It was optimized for Adobe Illustrator 5.0J but will work with other applications that support the EPS file format. This translator allows you to import a postscript file into Graphite (text is not translated) and export drawing data.

Translator Notations

In postscript's expression, drawing data is described as a “path.” 2D drawing data is expressed by the values you give for the path from the startpoint to the endpoint. Straight lines and Bezier curves create such paths.

For example, a circle contains the following components:

- 4 connected Bezier curves (in Adobe Illustrator the path is structured by anchor points (startpoint, centerpoint and endpoint) and segments). If the circle is created using one tool, then the anchor points are connected.
- Path width: user-defined units or points (1pt=0.35 mm)
- Path Color: user-defined
- Fill inside Startpoint~Endpoint: user-defined
- Fill Color: user-defined

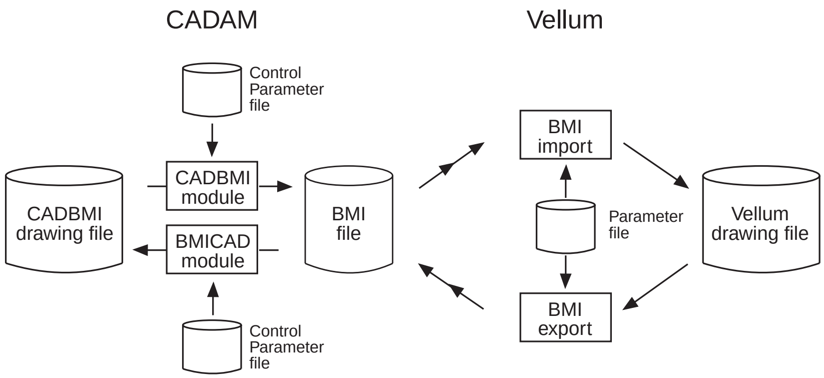
CADAM BMI Translator

*This translator is located in the Graphite “Unsupported” Folder.

The CADAM BMI Translator enables you to share data between Graphite and CADAM. The translator converts this data, using the BMI format, into a neutral file format of CADAM.

All recognizable entities will be converted with their attributes. For unrecognizable entities, only the shapes will be converted.

The translator works according to the graphic below.



For Windows, *VelBmi.ini* in the Graphite32 folder is the Control Parameter file. This parameter file is used in the translation of the geometric entities in your drawing.

Tips for the Best Possible Translation

In general, it is recommended that you show all your layers before you export from Graphite or any other CAD package so that you know what objects are exporting.

1. Graphite was designed to export only the current model. If you have Detail Views in your file, flatten them with the *Flatten View* command found in the Views menu. If you have dimensions already in the detail view and the view scale is set to 1, you can group the dimensions and geometry together before you flatten the view. This will bring the dimensions across to the current model for Export.
2. Graphite supports 256 layers. If you have a DXF file that contains more than 256 layers, the geometry that is on the additional layers will automatically be placed on the current work layer in Graphite. If you don't want this to happen, reduce the number of layers to a maximum of 256 in the application you will be creating the DXF file in.
3. Attributes in an AutoCAD file do not come into Graphite as text because Graphite does not support that entity type. If you want your attributes to come into Graphite via DXF, you need to convert those attributes to Text before you Export from AutoCAD.
4. If, when translating to AutoCAD the line patterns do not seem to come across or are distorted (too big or too small), change the AutoCAD variable *LTSCALE* to a smaller number to display the line patterns at an appropriate scale.
5. With some angular dimensions, the arrow may be displayed in AutoCAD in the wrong direction (pointing towards the text rather than towards the extension lines). If this happens, execute the *DIM* command in AutoCAD, select the affected dimensions, and use the *UPDATE* command to display the arrow correctly.

Convert Command

This command in the File menu starts the Convert batch translator which allows you to import and export multiple files in an unattended mode. It imports all files of the same type from one folder and saves these files in the selected format into a destination folder.

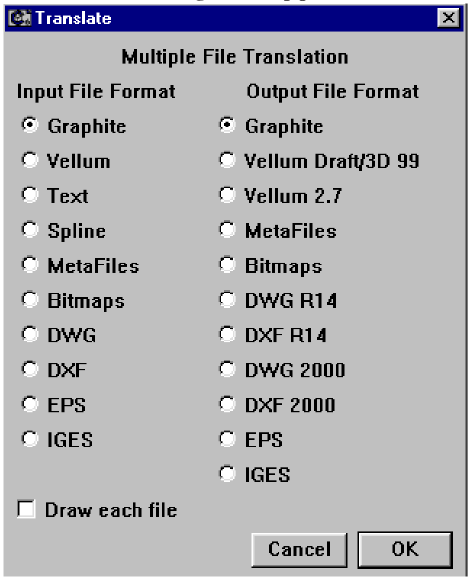
Converting Multiple Files

1. Create two folders.

One folder will be used to store the original files (the FilesIn folder), the other folder will store the converted files (the FilesOut folder).

2. Copy all files you want to convert into the FilesIn folder.
3. Select the ***File>Convert***.

   The Multiple File Translation dialog box appears.



4. Select the file type you are translating from the *Input File Format* column.
5. Select the file type you are translating from the *Output File Format* column.
6. Mark the *Draw each file* option, if desired.

   If you mark this option you will see each file during the conversion process. If this option is not selected, Graphite will open a blank drawing window, perform the conversion, close the blank window and repeat the process for each file in the folder.

   For large drawings or large quantities of drawings turning off *Draw each file* speeds up the operation.

7. Click OK.

   The Open file dialog box displays.

8. Open the folder (FilesIn) that contains the files to convert.
9. Select a single file and click Open.

The selected file must be of the same type as was selected for *Input File* format or Graphite will generate an error message.

10. Select a destination folder (FilesOut) for the translated files or create a new one.

    Specify a new extension to identify the converted files (it is reasonable to use the standard file type extension like *.dxf* for DXF files or *.igs* for IGES files. This is not necessary for the Macintosh but may help in file organization.

Graphite will convert every file from the Input folder that is of the same type.

If you select a Graphite file in the Input folder, and requested convert to DXF, Graphite will convert every Graphite file in that folder to DXF, and place those DXF files in the Output folder you selected. If the folder also contains IGES files, they will not be translated.

If you specify DXF as the *Output file* format, Graphite asks once if you want thick lines to be converted to Polylines. Whatever you specify will be valid for all DXF Files in that folder.

Drawing at Full Scale

Whether you are designing or drafting a highly detailed blueprint, you should create the geometry at its actual size. Graphite allows you to construct the part using full-scale specifications and then set the visual scale of the drawing. In this way, the part dimensions to its true-to-life measurements. Drawing at full scale has the following advantages:

- Scaling mistakes are eliminated
- Dimensions are automatic (you must dimension manually if you do not draw at full scale)
- Associative dimensions update when the object is edited (manual dimensions do not)
- The size relationship of imported parts is compatible.
- Calculations for 2D analysis are accurate (See Chapter 18 for more information.)

Once your project is drawn, you can dimension it, scale it visually, and size it to fit into a standard drawing format, if you wish. The actual size of the geometry remains constant unless you edit it.

Tech Note:

Zoom All magnifies or reduces all objects on your drawing to fill the screen—regardless of how big or small the objects are.

When you open a new Graphite document, the drawing area is a sheet that is infinitely large so that you can design anything at full size. For a simplistic example, here's how to draw and view a line 83 feet long:

1. Draw a line, specifying **83'** for the length.

   The line extends off the screen.

2. Choose ***Arrange>Zoom All***. The entire 83 foot line is visible on the screen.

Using the draw to scale/Zoom All method, you can create accurate full-scale drawings which are displayed at the magnification you choose.

Reminder: The actual size of an object is not affected by zoom magnification or reduction.

Scaling Line Patterns and Crosshatch Patterns

If your unscaled construction is very large or very small compared to the size of added text and dimensions, line patterns and crosshatch patterns will be inappropriately sized. When you set the scale for your drawing with the *Drawing Size* or *Sheet Into View* command, line patterns and crosshatch patterns are automatically scaled with the inverse scale factor you specified. The patterns appear at the appropriate size for the view scale of the geometry.

Scaling Text and Dimensions

If you are drawing objects that will be scaled visually, Text, Dimensions, Line patterns and Crosshatch patterns behave, when scaled, differently than the geometry you created. While geometry always scales with the specified scale factor, text and dimensions scale with the inverse scale factor you have set with the *Drawing Size* or *Sheet Into View* command.

While the inverse scale factor applies immediately to Line patterns and Crosshatch patterns as described in the section above, the inverse scale factor applies only to future text and dimensions added after scaling. Existing text and dimensions scale with the same scale factor as the object geometry. A 12 pt. text that was created at a 1:1 Drawing Scale will be 24 pt. on a page printed at a 2:1 scale. The size of the text will not change relative to the other geometry in the file.

If you want to keep the text size specified before scaling, choose the *Keep Text Size* and *Keep Dimension Text Size* options in the Drawing Size dialog box. This will change the size of the text relative to other geometry in the file. For example, 12 pt. text that was created at a 1:1 Drawing Scale will still be 12 pt. on a page printed at a 2:1 scale. The size of the text will be half as small as before changing the Drawing Scale, relative to other geometry.

Another method of controlling text size and dimensions while scaling is to calculate the needed text size manually as shown in the following example.

When calculating the text size for scaling manually the rule is: set the text size to the inverse of the scaling factor. (The text size affects both text and dimensions.) In that way, the text elements of the drawing are proportional with the size of the geometry.

For example, if an assembly is 83 feet wide, you may want to scale that assembly 1:100 (a scale factor of .01) to place it in a drawing format. This requires using text that is equally large so that you can read it visually when you scale the drawing. If you want the final text to be .156 inch in height, you must specify 100 x .156 for the text size in the drawing. With the manual calculation you have to make this specification before you begin dimensioning or annotating.

Tech Note:

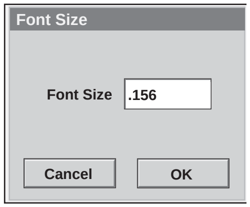
If you are scaling smaller, multiply the text size by the scale factor. If you are scaling larger, divide the text size by the scale factor.

Scaling Text and Dimensions Manually

1. Choose ***Text>Size>Other***.

   The dialog box displays the measurement of the text.

   The Font Size is set to .156 inch.

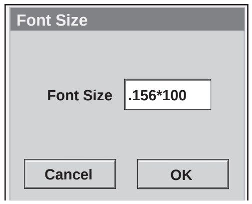


2. Click after the entry in the data field.

   The text cursor blinks in the data field.

3. Enter ***100** and click OK.

   Subsequent text and dimensions will be one hundred times the original size, or 15.6 inches. When you add dimensions, the text size will be appropriate for the part.



Tech Note:

The relationship between the object and the text should appear normal when you zoom. If it looks right, it is right.

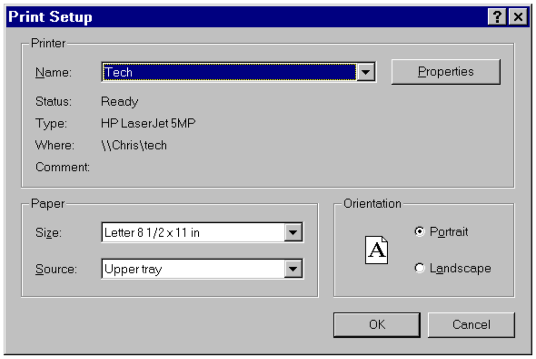
Drawing Scale and Paper Size

When you are ready to detail your design with line patterns and dimensions, or when you want to print or plot your design, you may need to scale the view of the drawing to fit the paper you will use.

For that you have to specify a printer or plotter, the paper format and orientation, and then the drawing size for the specified format.

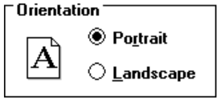
Print Setup - (Windows); Page Setup (Macintosh)

This command in the File menu allows you to set the page size, orientation and other options.

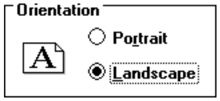


Setting up the Page

1. Select the plotter or printer you will use.
2. In the Paper Size menu, select the paper size.
3. Specify the orientation. *Portrait* is taller than wide.



   Landscape is wider than tall.



4. Click OK.

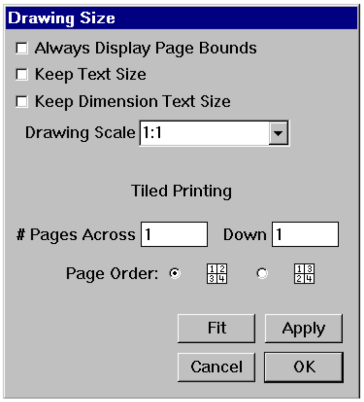
Drawing Size Command

This command in the Layout menu allows you to see the size of the maximum plotting/printing area relative to the drawing. It also allows you to specify the scaling of the drawing so it fits the paper size and orientation set by the *Print Setup* (Windows) or *Page Setup* (Macintosh) command.

Setting up the Drawing Size

If the geometry is larger than a standard piece of paper, or so small that it would be impossible to use it at full scale, you must set the Drawing Size. In many cases, you may want to use a standard drawing scale (such as 1/4 inch = 1 foot) for the drafting project.

1. Choose ***Layout>Drawing Size***. A dialog box appears.



2. Specify the scale. To specify a scale, use one of two methods (the ratio represents *Plotted size : Actual size*):

- Double-click the *Drawing Scale* data field and enter a scale (the ratio) you want to use. You can enter real values, such as **0.25" : 1'**, which Graphite converts to 1:48. Click Apply to see the effect of the scaling you specified.
- Press the arrow beside the *Drawing Scale* data field and drag to one of the standard ratios from the menu. Click Apply to see the specified scaling.

  The Cancel button on the Drawing Size dialog box doesn't abort changes if a Fit or Apply are chosen first.

  If you don't need a specific scale, you can click Fit to scale the drawing to fit the paper.

  With any of these methods, the geometry itself does not change scale; it only changes visually, not physically.

3. If you are printing multiple sheets and taping them together, follow the directions for "Tiled Printing" later in this chapter.
4. Click OK.

Tip:

When you scale your drawing, text and dimensions behave differently from the geometry you created as described under the "Scaling Text and Dimensions" section earlier in this chapter.

Setting the Visual Scale with the Sheet Into View Command

Once you have drawn and dimensioned a full-scale part, you can create a visually-scaled view of that part. The *Sheet Into View* command performs the details of this operation automatically.

1. Choose ***Arrange>Zoom All***.

   All objects are displayed on the screen.

2. Choose the ***Views>Sheet Into View***.
3. Enter the scaling value in the *Scale* data field of the Sheet Into View dialog box. Figuring out this value is similar to what you would do if you were drafting on paper, determining which scale to use so that the finished part fits on the paper size you are using.

Referral:

You will find an exact description of the *Sheet Into View* command in the Chapter 14, "Advanced Viewing Techniques."

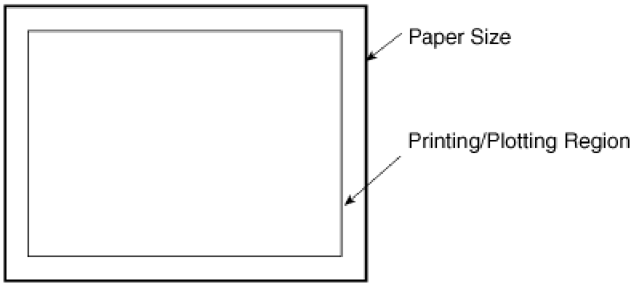
4. Click OK.

Your part is now in an independent view, scaled as specified.

Scaling does not change the actual dimensions of the part. You can verify this by selecting an object and then looking at the specifications of that object in the Edit Objects dialog box.

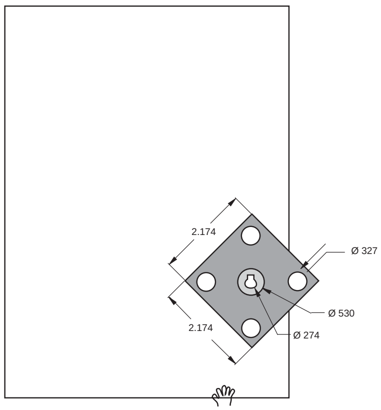
Printing/Plotting Region

When you choose *Drawing Size* and choose *Always Display Page Bounds* a gray rectangle appears in the drawing area. This rectangle represents the printing/plotting region of the page so you can see how your drawing fits the paper. The plotting region is smaller than the actual page size because most printers and plotters cannot plot to the edge of the paper, allowing room for the margins. If you do not want to see the plottable outline all the time, uncheck *Always Display Page Bounds*.



Graphite displays the Printing/Plotting region

The size of this region is based on the paper size and the printer or plotter drive currently selected.



When the boundary of the printing/plotting region appears, you can drag the boundary rectangle around the drawing area with the *Hand* pointer, as shown above. The cursor becomes the Hand pointer when you move the cursor outside of the dialog box.

To change the paper orientation while the Drawing Size dialog box is visible, choose ***File>Print Setup*** (Windows) or ***Page Setup*** (Macintosh) and enter the specifications you want. The new specifications appear in the drawing area.

Tech Note:

You can only pan when the Drawing Size dialog box is open. Once you close the box, you can no longer adjust the boundaries through panning.

Tiled Printing

If you are using a printer rather than a plotter, you may want to tape pages together to get a larger drawing than laser printers allow. Specify the layout for those pages by choosing ***Layout>Drawing Size***.

1. Choose ***Layout>Drawing Size***.
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).

Across then Down

1	2
3	4

Down then Across

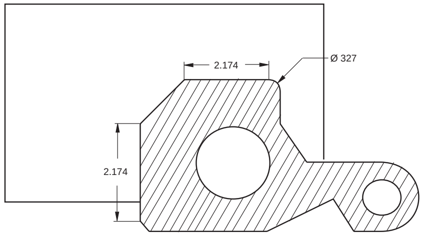
1	3
2	4

5. Click Fit.
6. Click OK.

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

Preview Layout

This command in the File menu shows the size and orientation of the printing area specified in *Print Setup* (Windows) or *Page Setup* (Macintosh) by displaying a black border. If you have not specified a printing/plotting size, the default size is used. To remove the page border display, select the command again or simply select a tool from the tool palette.



If the drawing does not fit the paper size, choose ***Layout>Drawing Size***.

Drawing Formats

When your part is complete and you want to place it in a formatted border, Graphite provides you with many options. You cannot only modify all drawing formats in the Layout folder of the Graphite folder but create your own drawing formats and design layouts as well.

Graphite distinguishes between *Drawing Formats* and *Design Layouts*. Drawing formats contain the *border line* of the drawing, the *cutting line* and a *Title block* for entries such as *name* and *scale* of the drawing. Design layouts contain a border and detail views.

- Drawing formats have to be created for each paper format such as A, B, C or D and for both paper orientations Portrait and Landscape.
- All drawing formats you find in the Layout folder of the Graphite folder are designed for Plotter devices, where the cutting lines correspond exactly with the dimensions of the selected paper format (for example 30 x 40 inch for the D Format) and the Border line is offset inside by a half inch according to the ANSI standard.
- When you use these plotter formats for your laser printer device you have to adjust them to the printing area of your printer, that could differ from printer to printer.
- Design layouts contain, in addition to the drawing format, one or more detail views that may display with the *Sheet Into View* command the content of a drawing at different view angles and scale factor.

Placing Drawing Formats

In Graphite you can place *Standard Drawing Formats* using two different commands:

- the ***Import*** command in the File menu

  or

- the *Sheet Into View* command in the Views menu.

Using the Import Command

If your part is placed in a detail view and it is visually scaled into a view, you can import a standard drawing format (either one of those supplied by Ashlar or one you created) in preparation for printing or plotting. Since you want the format to be full scale, it must be imported onto the sheet outside the view because the detail view is scaled and the sheet is not.

1. Choose ***Layout>Drawing Size*** to scale the drawing.

   The **Drawing Size** dialog box displays.

2. Mark the *Always Display Page bounds* option that allows you to compare the true size of the drawing with the displayed printing area of the drawing format.
3. Choose the ***File>Import.***

   The Import dialog box displays.

4. Mark the *unscaled* option.
5. Select, in the Layout folder of the Graphite folder, the desired drawing format such as:

   ViewB.vc6 (Windows) or *a Format B & View* (Macintosh) where **1View** stands for one detail view and **B** for the paper format.

   Important: You can ignore the number of detail views since they are only important when using the *Sheet Into View* command.

6. Click OK.

Using the Sheet into View Command

With the *Sheet Into View* command, Graphite imports not only a drawing format but also displays the drawing at the specified scale in one or several detail views, depending on the selected design layout.

1. Choose the *Sheet Into View* command in the Views menu.

The Sheet Into View dialog box displays.

2. Select in the *Layout* list box the desired drawing format such as:

   4ViewB.vc6 (Windows) or *a Format B & 4Views* (Macintosh)

   where **4View** stands for four detail views and **B** for the paper format. (This example is only available in Graphite 3D.)

3. Enter the desired Scale factor in the *Scale* data field.
4. Click OK.

   The drawing format is placed at a scale of 1:1 (at it's true size) onto the drawing area and the complete geometry of the drawing displays in one or several detail views at the specified scale factor. The number of detail views and the view angle (Top, Isometric etc.) depends on the selected design layout. (Only the Top view is available for Graphite Draft.)

Modifying Drawing Formats

All drawing formats shipping with Graphite in the Layout folder can be modified. When you want to create your own drawing formats it is recommended to modify existing formats and save them under a new name.

Creating Drawing Formats and Design Layouts for Laser Printers

1. Open in the Layout folder within the Graphite folder the drawing format:

   1ViewB.vc6 (Windows) or *a Format B & View* (Macintosh)

   where **1View** stands for the number of detail views and **B** for the paper format.

2. Save the drawing under a new name.
3. Choose ***Layout>Drawing Size***.

   The Drawing Size dialog box displays.

4. Mark the *Always Display Page Bounds* option.

   The Printing area of your active Laser Printer displays as a gray rectangle.

5. Click OK.

   The Drawing Size dialog box closes.

6. Select the grouped drawing format on the drawing area and ungroup the format by choosing ***Arrange>Ungroup***.

7. Adjust the border lines of the drawing format until they are placed exactly on top of the displayed gray rectangle for the printing area of your laser printer.
8. Delete all cutting lines since they are not required for print on a laser printer.
9. Hide the TitleBlocks layer.

   All Text Entries are hidden. See the next section for information about title blocks and text entries.

10. Select the complete drawing format by choosing ***Edit>Select All*** and group the format by choosing ***Arrange>Group***.

    Important: Text entries with an @ character in front must not be grouped since then the *Forms* command cannot identify these entries as title entries (see a later section, “Creating Forms”).

11. Activate the TitleBlocks layer.
12. Save your work.

Forms (Title Blocks)

By using the *Forms* command in the AutoText submenu of the Text menu, you can fill in text or values within a dialog box for the title blocks of all standard drawing formats that are saved in the Layout folder of the Graphite folder. Graphite allows you to create your own drawing formats that you can use with the *Forms* command. This is described in a later section of this chapter.

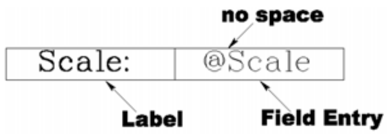
As long as no standard drawing format is placed in a drawing, the *Forms* command is not available. It can be only selected if you place a drawing format with a title block using the *Import* command in the File menu or the *Sheet into View* command in the Views menu.

Creating Forms (Title Blocks)

Before you can use the *Forms* command in the AutoText submenu to fill in your own drawing formats you have to prepare the title blocks of these formats.

To enable Graphite to identify the title blocks, the user must define *fields* in the drawing format. To define a field, create a regular text block, but type an @ character in front of the text. Any text following an @ character then becomes a field. For example, for an entry field *Scale* create a text block and type in “@Scale.” If you want a label in front of the field, create another text block without an @ character.

The following illustration shows two text blocks. One is the label (a standard text block), one is the field (a text block beginning with @).



Note: All field and label text blocks have to be placed on the TitleBlocks layer. (Create this layer using the *Layers* command as discussed in Chapter 13.)

Preparing Title Blocks with the Forms Command

1. Open one of the drawing formats in the Layout folder.
2. If no layer with the name TitleBlocks exists, create this layer by choosing ***Layout>Layers*** and make it the current layer.
3. Select the *Text* command from the function palette.
4. Create text blocks. Any text block beginning with an @ character will become a field, all others will become labels.
5. Assign all of the text blocks the desired font, style and size using the **Selection** tool from the tool palette.
6. Save the drawing format.

Using Forms

With a format already placed in the drawing, selecting the *Forms* command in the AutoText submenu brings up the Forms dialog box. This dialog box is similar to the Resolve dialog box in that it prompts the user for values for each previously defined field. Graphite replaces the field text blocks with whatever values are entered. Text attributes, such as font, assigned in the saved format are applied to the new values.

Filling in a Title Block

1. Place a Standard drawing format (use the *Import* or *Sheet into View* command). The form appears with all of the field labels as they were originally created.
2. Choose the ***Text>AutoText>Forms***.

   The Forms dialog box containing all entry fields of the title block displays.
3. Type in the desired parameters. You don't have to fill in all entry fields.
4. Click OK.

The Forms dialog box closes and all entry fields of the selected title block are filled in according to your specifications.

Editing Title Blocks

1. Select the fields containing values you want to update (or choose ***Edit>Select All***).
2. Select the ***Text>AutoText>Forms***.

   The Forms dialog box displays again.

3. Modify the entries of the title block.
4. Click OK.

   The Forms dialog box closes and the selected title block modified accordingly.

Printing or Plotting a Drawing

Graphite prints and plots on most printers and plotters supported by your computer. After you have followed the manufacturer's instructions for installing and setting up the printer or plotter, you can size your drawing for the paper you will be using.

Setting the Printer or Plotter and Paper Specifications

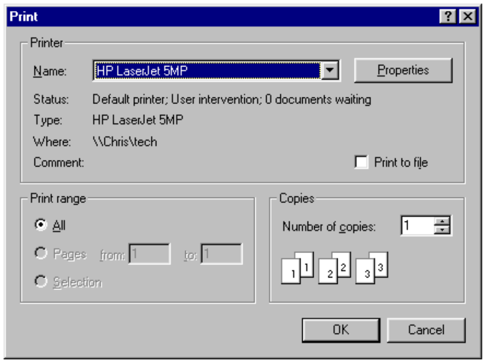
In the File menu, use the *Print Setup* (Windows) or *Page Setup* (Macintosh) to specify the printer or plotter, the paper size, orientation, number of copies and other options.

Once you have specified the page setup, you can set the visual scale of the geometry so that it fits the paper size.

Follow the instructions found earlier in this chapter to scale the geometry visually to fit the paper you specify in *Print Setup* (Windows) or *Page Setup* (Macintosh).

Print Command - CTRL+P (Windows); ⌘+P (Macintosh)

This command in the File menu prints or plots the current document as specified in *Print Setup* (Windows) or *Page Setup* (Macintosh).



The area printed or plotted is the portion that fits on the page size specified in *Print Setup* (Windows) or *Page Setup* (Macintosh) when the origin (0,0) is placed in the center of the page. Choose ***Layout>Drawing Size*** to scale your drawing to the appropriate size and reposition the print/plot region.

You can specify tiling (printing on several pages to be pasted together) through the *Drawing Size* command. If you have more than one sheet or have specified tiling, you may specify which page(s) to print/plot. Pages are numbered sequentially for tiling, then for additional sheets.

Plotting to a File

You can plot to a file rather than to a plotter or printer. In that way, you don't have to have a plotter attached to your computer. You can have someone else plot the drawing without having a copy of Graphite on the plotter's computer.

The type of plotter you choose when you are setting up the page determines the format of the plot file.

If you choose a PostScript printer, the file format will be Encapsulated PostScript; and the HPGL language is used when you select Hewlett Packard plotters. The computer that finally plots the file must have an application compatible with the file format of your printer or plotter.

Plotter Fonts

When you are using a plotter, you should specify the Plotter font for the text and dimensions on your drawing. You can also specify different text styles (such as italic or bold) in the Text menu and generate special characters and accents as described in Appendix B.

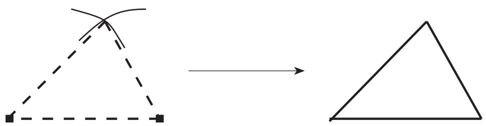
Parametrics

Graphite's integrated parametrics feature allows you to create geometry without regard to its actual measurements. When you resolve geometry parametrically, you specify values for the dimensions and Graphite redraws the geometry to your specifications. One example of parts well-suited to parametric definition are containers that vary in size according to the needs of the product line. Another might be hydraulic pistons that vary in size because of the duty loads. Still another might be valves that vary according to the diameter of the pipes to which they are attached.

This parametric feature is the basis for creating symbols that you would use in your drafting on a regular basis. See the next chapter for more detailed information.

The following topics cover the parametrics feature:

- Introduction to Parametrics
- Using Parametrics
- Parametric Drafting
- Parametric Problems
- Complex Parametric Drafting
- Parametrics and Grouped Objects



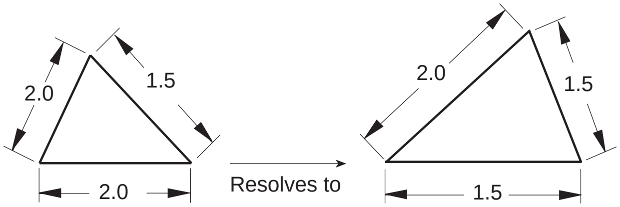
Introduction to Parametrics

In its simplest form, parametrics allows you to create a shape and then specify the exact measurements. Take, for example, the basic activity of constructing triangles.

Creating parametric geometry

1. Create the geometry.
2. Dimension the geometry using variables, constant values, and expressions. You may use as many as 253 parametric dimensions.
3. Resolve the parametric geometry.

Using this method, you can quickly draw a triangle to exact specifications.



Of course, parametrics can be much more complex when you use variables, as you will see later in this chapter. While creating geometry is straightforward, when a part becomes complex, parametric dimensioning requires some skill and comprehension of how Graphite treats geometry.

Using Parametrics

Tip:

The Graphite folder contains a folder of symbols that use parametrics. You might retrieve a few of the symbols by going to the File menu to the Symbols submenu and choosing the *Insert* command to become familiar with how parametrics works. See the next chapter for more information on symbols.

The parametric mechanism is *point-driven*. It locates the points and then connects them with the appropriate geometry, much like the childhood activity of *connect-the-dots*. It is not unlike drawing with a compass, triangle, and T-square.

To begin, determine where the endpoints are and then draw lines to connect the points. Or you might figure out where the center of an arc should be, where the arc begins and ends, and use a compass to connect the points.

Basic Rules

1. Define the geometry completely.

   Each object must relate to another object.

   Dimensions must define every aspect of the geometry. (The geometry may be over dimensioned according to standard drafting practice.)

   Do not include extraneous text as part of a dimension. (A single # is acceptable.

The parametric mechanism recognizes R # to mean a radius, measuring the real value of the geometry.)

2. Keep it simple. If the part is complex, construct small segments, resolving as you go, solving any parametric problem before proceeding.
3. Return to the original part between tests. Use the *Undo* command to revert to the original state of the parametric part between tests. In this way, you won't introduce an unwanted constraint.
4. Consider the following assumptions made by the parametric mechanism:
   - Horizontal and vertical lines maintain their orientation.
   - Connected lines remain connected.
   - Lines tangent to arcs remain tangent (if there is an endpoint at the tangency point).
   - Colinear lines remain colinear if they overlap or share endpoints.
   - Concentric arcs remain concentric.
   - A point of an object on a horizontal or vertical line will remain on that line.
5. Be aware of the relationships that the parametric mechanism cannot recognize:
   - Parallel lines may not remain parallel.
   - Perpendicular lines may not remain perpendicular.
   - Symmetrical geometry may not remain symmetrical.

Creating Geometry

When approaching the problem of resolving geometry, evaluate it as though you were going to draw it on paper. Remember the assumptions and requirements for geometric relationships as outlined in the previous Basic Rules section.

Parametric Dimensions

The essence of parametrics is in the dimensions. The dimensions may be the actual value (which you get by default), a constant which is not the actual value, a single variable, or an algebraic expression involving constants, variables, mathematical operators, functions, and conditional operators.

Constants are specific numeric values, such as specifying a radius as .25 (inch). Variable expressions may be as simple as a single letter such as *L* (for length), or a mathematical expression such as *2 * Dia* (where *Dia* may be the diameter). The arithmetic operators are *addition* (+), *subtraction* (-), *multiplication* (*), *division* (/), *mod* (the remainder after division—%), and *exponentiation* (**). See Appendix A for a list of functions and conditional operators.

However you choose to specify the dimensions, they must define all geometry and any relationships that exist between different parts of the geometry.

Creating Parametric Dimensions

1. Construct the geometry.
2. Display the dimension palette and choose the appropriate dimensioning tool.

   The Status Line displays a # symbol in the text data field to show that the dimensions are entered as actual measurements.
3. Click the geometry to dimension as usual.

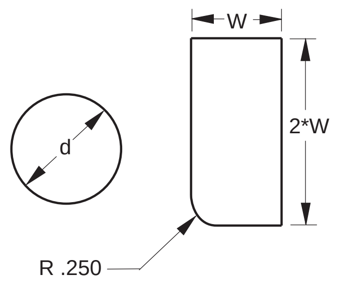
   The Text data field in the Status Line highlights.
4. Type whatever expression, variable, or constant you want to use for the dimension, and press the ENTER (Windows) or the RETURN (Macintosh) key. If you want the actual value to be used, do not change the # symbol in the text data field.

   Variables are case sensitive: *D* is not the same as *d*.

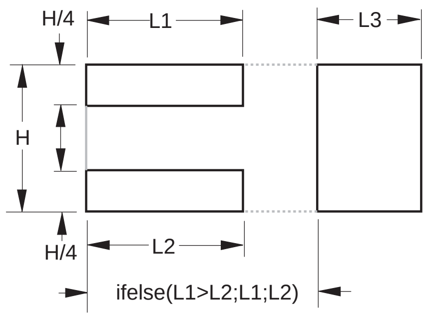
   The expression you type replaces the **#** symbol.

Examples of Parametric Dimensions and Conditional Expressions

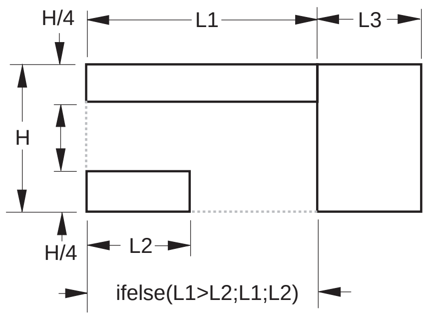
You can use the conditional operators as parametric dimensions. Create the dimension as usual and enter the conditional expression as text in the Status Line.



The example here shows a conditional *ifelse* expression.

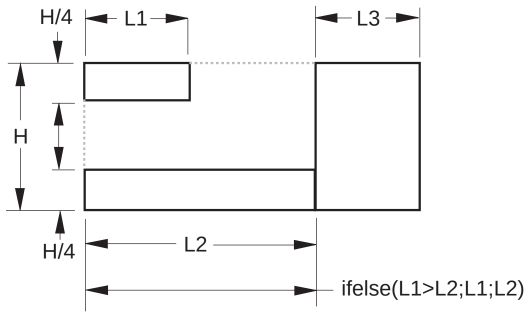


When the above *ifelse* statement resolves, the rectangle with the larger value (L1 or L2) touches the L3 rectangle.



The length represented by the *ifelse* expression equals L1 because L1 is greater than L2.

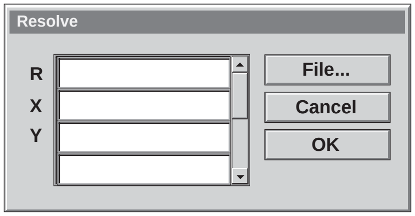
The length represented by the *ifelse* expression equals L2 because L2 is not less than L1.



Resolving the Parameters

Once you've constructed the geometry and added parametric dimensions, use the *Resolve* command to specify the values for the variables and redraw the geometry.

Resolve Command



This command in the Edit menu allows you to redraw geometry automatically to fit specified dimensions. This parametric feature enables you to draw a geometric shape without regard to measurements and then have Graphite redraw the same shape to the values you specify.

Tech Note:

Example: You can use a mathematical expression with a fastener symbol. If you want a numbered fastener (#2, #4, #6, etc.,) use the following formula for the thread size: **(N*13+60)/1000** where N is fastener number. For a #6 screw, the thread size equals **6*13+60/1000** or **.138**

Using the Resolve Command with Parametrics

1. Create the geometry.
2. Dimension all geometry by using variables, constant values and expressions. (You must dimension the essential, related geometry so Graphite can reconstruct the geometry. Graphite cannot identify parallel or colinear lines.)
   - Select the appropriate dimensioning tool.
   - Click the geometry to be dimensioned.
   - Enter an expression (such as **1.5, x, x+3*y**) in the text data field in the Status Line.
   - Press the ENTER (Windows) or the RETURN (Macintosh) key.
3. Select the geometry and dimensions to be resolved.
4. Choose ***Edit>Resolve***.
5. If necessary, enter the values you want to assign to the variables.

   You can enter mathematical expressions, fractions, and decimals in the data fields of the **Resolve** dialog box. You can use different units as long as you are specific, for example 2’6”.
6. Optional step: If you want to anchor a point on the geometry at a particular location, click the point. The point remains in the same location after you resolve the geometry. An example appears later in this section.
7. Click OK.

   If the parametric dimensions defined the geometry properly, the geometry redraws as you specified. The dimensions remain as variable expressions.

If Graphite cannot resolve the parametrics, a message box displays information about the problem. The “Parametric Problems” section, later in this chapter, describes typical problems you may encounter. If you have not given all of the dimensions required to draw the geometry, the geometry divides into unrelated groups. The missing information determines how the groups relate to one another.

An alert box shows how many groups exist. To see each group, click Next. Examine the groups shown to determine why the position of each group is not related to any other. A group consisting of a single point is particularly telling. (This investigation can require some clever thought because the solution may not be obvious.)

The *Resolve* command cannot resolve ellipses and splines, unless the ellipse or spline is contained in a group. See the section on Parametrics and Grouped Objects.

Specifying Parametric Variables in a Text File

If you frequently use parametrics which contain many variables, you can create a text file to specify the variables and avoid entering them in the Resolve dialog box each time you resolve the parametrics.

Tech Note:

If you use a spreadsheet like *Microsoft Excel* for creating the text file, you have to type in each cell the variable name and the value separated by a white space and export it as a text file.

```
Height 12
R 17
Diameter 5
```

```
Height
12
R
17
Diameter
5
```

```
Height 12 R 17 Diameter 5
```

Create a text file which alternates the variables and the values, using white space (SPACEBAR, TAB, or ENTER (Windows), RETURN (Macintosh)) between entries. Graphite assumes the first entry is a variable and the second is the value for that variable. Your text file might look like the examples shown here or any other arrangements).

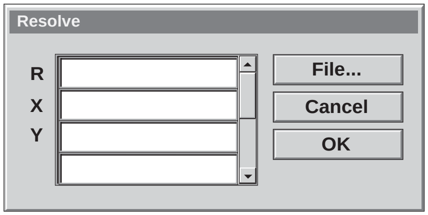
Tech Note:

If your text file contains more variables and values than you need to resolve the geometry, Graphite automatically imports only the needed variables and values.

Using a Text File for Parametric Variables

1. Create the text file using any word processor or spreadsheet and saving it as *Text only*.
2. In Graphite select the geometry and parametric dimensions.
3. Choose ***Edit>Resolve***.

   The Resolve dialog box displays.



4. Click File to specify the values for the variables automatically.
5. Select the text filename.

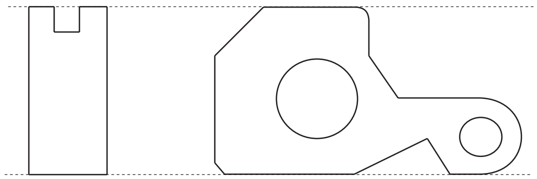
   The variables and values are read from the file.

6. Specify an anchor point, if you want one.
7. Click OK.

   The geometry resolves accordingly.

Anchoring the Geometry

You can anchor the geometry by clicking the anchor point when the Resolve dialog box appears. For example, in a tutorial exercise in the *Getting Started* section of the manual, you used parametrics to create the side view.



If you were using this side view as parametric geometry, you would want to anchor the upper-right corner so that it remains aligned with the front view after resolution.

Changing the Dimensions to Actual Measurements

Once you resolve parametric geometry, the dimension notations continue to show variables and constants. If you want the geometry to reflect the actual measurements, you have to edit the dimensions. Follow these steps to change a resolved parametric dimension to a real value:

1. Choose ***Edit>Selection Mask***.
2. Highlight the four dimension types in the box on the left.

   Only dimensions are selectable.
3. Choose ***Edit>Select All***.

   All dimensions are selected.
4. Choose ***Edit>Edit Objects***.
5. Change the entry in the text data field to a # symbol.
6. Click Apply.
7. Close the Edit Objects and Selection Mask dialog boxes.
8. Click the **Selection** tool to cancel the effect of the Selection Mask.

If you want different formats for different types of dimensions, select each type separately.

Parametric Drafting

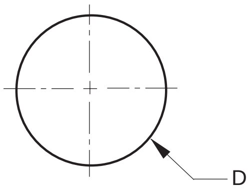
This section describes how to modify standard drafting practices to accommodate parametrics. It describes three examples with three topics to illustrate how to define parametric geometry completely.

Reminder:

- Define the geometry completely
- Dimension all geometry
- Relate all geometry
- Do not include extraneous text
- Return to the original part between tests

Dimension All Geometry

Center lines require consideration if they are to be resolved properly.



Problem

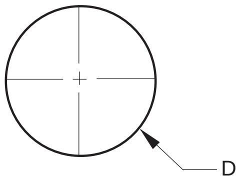
If you create the circle shown with two centerlines, parametrics can resolve the circle but not the centerlines because the *endpoints* of the lines are not defined.

Solutions

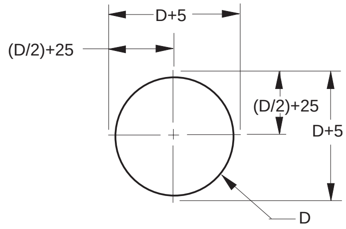
A. Using the Pen Style, Center

If you look at this problem as a connect-the-dots problem, you can see that the centerlines have no dots to define them.

1. You can construct the centerlines so their endpoints are *on* the circle, as shown here.



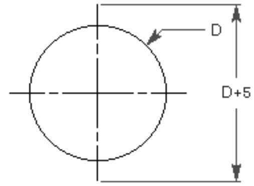
2. You can include dimensions for the centerlines. The dimensions should be variables based on the diameter of the circle so when you specify the diameter of the circle, the centerlines are drawn relative to the diameter.



B. Using the Circle Center Line Tool

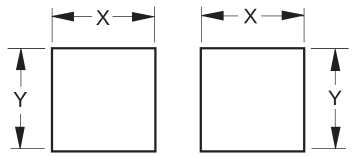
1. Use the **Circle Center Line** tool to place the center line on your circle.

2. Dimension the diameter of the circle and one of the center lines relative to the diameter. You can either dimension the center line from endpoint to endpoint or from the center of the circle to one endpoint of a center line.



Relate All Geometry

All parts of the geometry must be related.

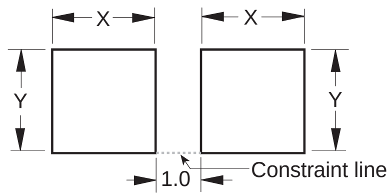


Problem

Both squares could be drawn by the parametric mechanism, but there is no way to determine their relative positions.

Solution

You can add a *constraint* line and dimension between the squares to connect the dots.



Additional Tips:

1. The added dimension is an example of using a constant value. Of course, you could have entered a variable expression such as $x+y$ for the constraint line.

2. Constraint lines are ordinary lines usually drawn with the **Single Line** tool. They can be any line style, but they should be different from the lines of the regular geometry. The Construction line style is a good choice.

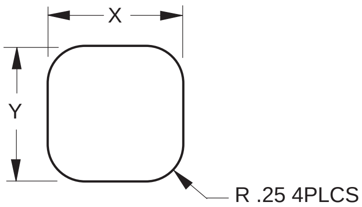
3. If you place the constraint line and dimension on a layer (named Constraints), you can hide that layer when you plot the drawing.

No Extraneous Text

You must dimension all geometry because the parametric mechanism does not understand text and cannot detect symmetry.

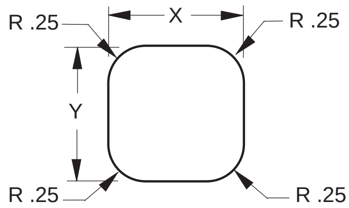
Problem

If you dimension the radius of a filleted corner of a rectangle as *R .25 4 PLCS* to indicate four fillets, the parametric mechanism doesn't understand 4 PLCS and doesn't know the corners are symmetrical.



Solution

Dimension each fillet.

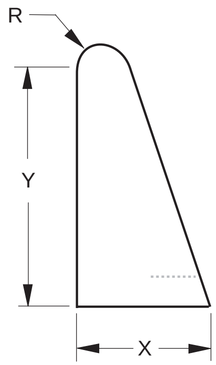


Return to the Original Part between Tests

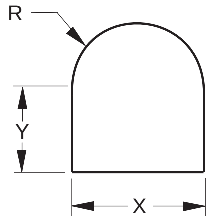
If you resolve parametric geometry more than one time, you can introduce unintentional constraints that will prevent you from using the parametric function to return to the original display of the part. Take the part below, for example.

Problem

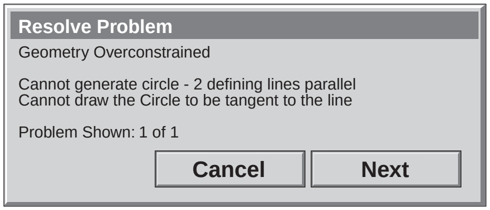
If you first resolve this geometry by specifying that *R* is half the measurement of *X* *(R=X/2)*, the resolved geometry would appear as shown in the lower illustration.



You would now be able to resolve this geometry only with the same *X* value of *R * 2* because the undimensioned line is vertical, and according to the basic rules, vertical lines remain vertical.



If you tried to use a value for *R* or *X* other than the *R = X/2* relationship, the parametric mechanism would display this problem message:



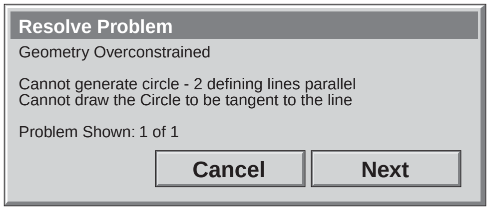
Solution

Use the *Undo* command to return to the original shape. (The *Undo* command is always available for any of the previous eight actions.) You could also solve this problem by changing the geometry.

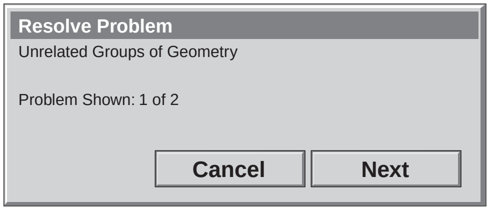
Parametric Problems

This section illustrates problems you may encounter when you resolve parametric geometry. It provides examples and shows you what caused the problem and how to solve it.

When you resolve parametric geometry, problems which prevent resolution appear in a message box, stating the type and number of instances of the problems.



When a message appears, the problematic geometry appears as thick lines and individual dots. The lines indicate the geometry that the parametric mechanism can draw. Dots indicate unknown points. When you click Next, the next problem is shown. Examination of the relationships between the problems can help you discern what must be done.



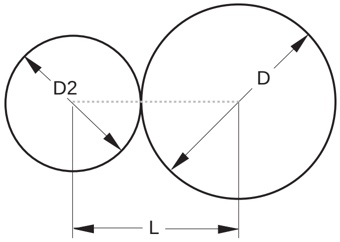
Important: Pay attention to single points! Remember, the parametric mechanism connects the dots. Frequently, one constraint or dimension can solve several problems at once.

Geometry Overconstrained

The basic rules say that you must define all geometry and relate every object to some other geometry within the parametric definition. When geometry is overconstrained, it is related in more than one way, so the geometry's specifications could be resolved to more than one answer.

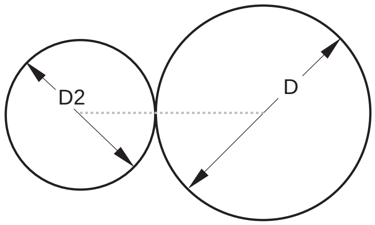
Problem

The diameters of the circles indicate the relationship between the circles adequately because the centerline remains horizontal and the circles are tangent to each other. The length variable causes the problem because the length between the centers of the circles may not correspond to the sizes of the connected circles. The centerline begins and ends at the centers of the circles.



Solution

Delete the Length dimension.

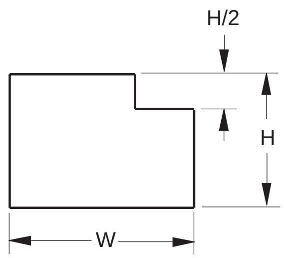


Unrelated Groups

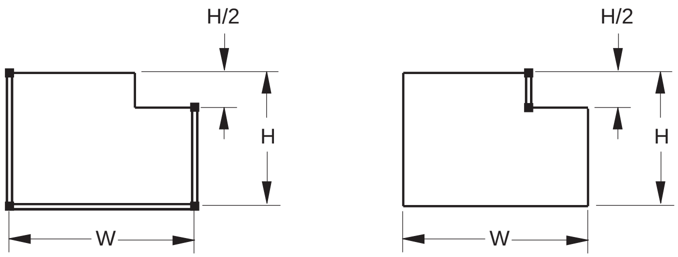
If you do not give enough dimensions, the parametric mechanism will find two or more groups of geometry, each of which is well-defined in itself, but not related to each other.

Problem

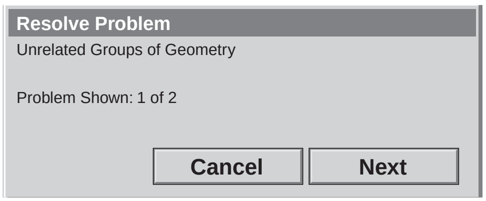
A simple example of this problem is illustrated on the left, where the short vertical line is not related to the rest of the geometry. When you try to resolve this geometry, you get an *Unrelated group problem* message.

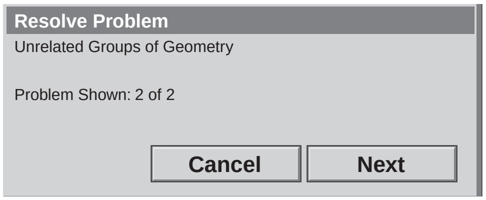


The first unrelated group appears in bold black lines. Click Next to highlight the second group.



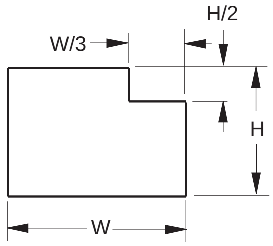
This problem is like the two squares shown earlier. Only one group could be drawn. The parametric mechanism does not know how to relate the two groups.





Solution

You must decide how you want to relate the groups. Usually, you have many options to accomplish this. For this example, adding the dimension *W/3* is one alternative.

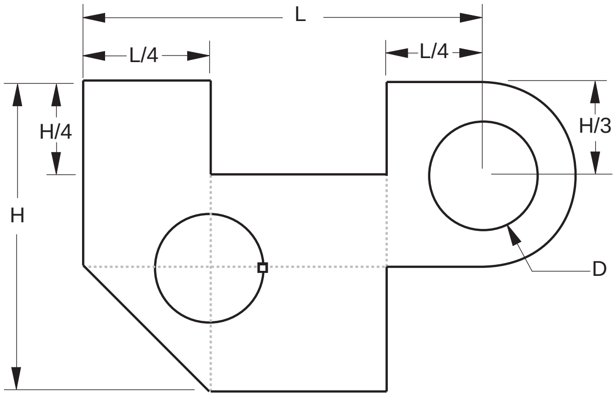


Unrelated Groups where One Group is a Single Point

A single point as an unrelated group is like a neon sign saying: *The problem is right here!*

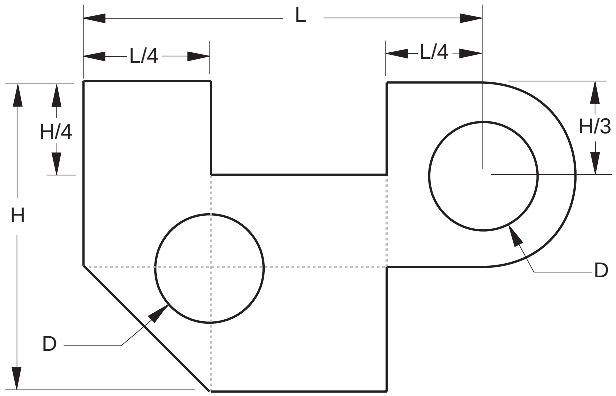
Problem

The *Resolve Problem* message may display a single point, such as the point on the left hole of this example. That point is the *endpoint* of the circle.



Solution

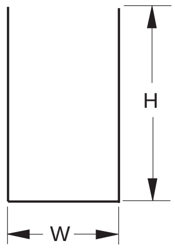
On closer examination, you can see that the circle is not dimensioned. Therefore, the solution is to dimension the circle.



Another Unrelated Group

Problem

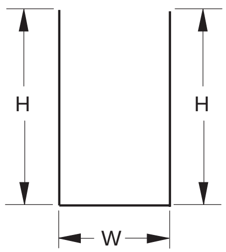
In this graphic, the height of the right side of the part is dimensioned. You assume that the height dimension applies to the right and left side. The parametric mechanism does not make that assumption.



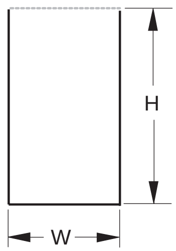
Solutions

There are three possible solutions to this problem:

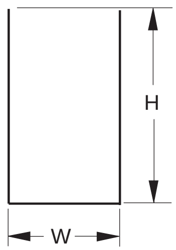
1. You can dimension the height on the left.



2. You can draw a constraint line to close the opening.



3. You can dimension the height from the top of the left line to the bottom of the right line. While this is not good drafting practice, it is very useful for parametric symbols.

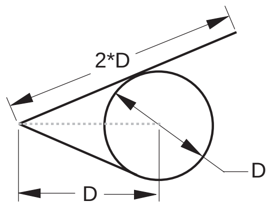


Assumptions Involving Tangency

The parametric mechanism can identify tangent points but not the tangency of geometry.

Problem

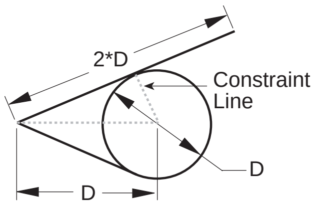
The *endpoints* of the lower line that is *tangent* to and ends at the circle can be determined. However, the upper line is tangent to and extends beyond the circle and cannot be defined without more information.



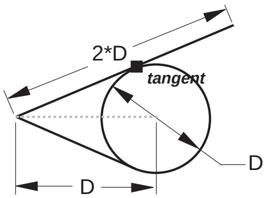
Solutions

Here are three possible solutions to the problem:

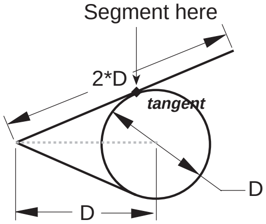
1. Draw a constraint line from the *center* of the circle to the point of tangency for the line that extends beyond the circle, as shown here.



2. Use the **Rotate** tool on the **Transformation** subpalette to move the *endpoint* of the circle to the *tangent* point of the line that extends beyond the circle.



3. Divide the line into two segments which join at the *tangent* point.

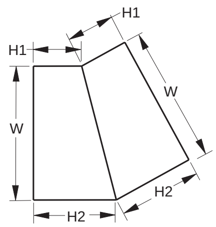


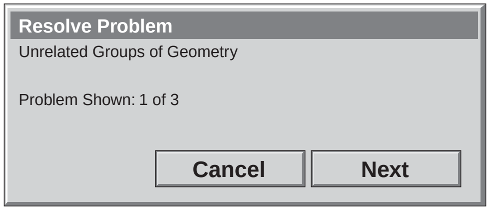
Relationships Involving Symmetry

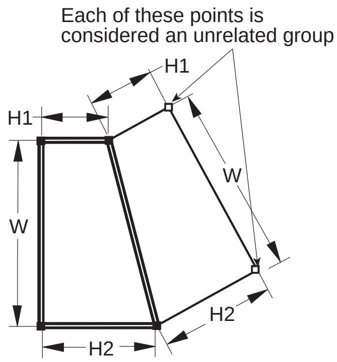
If you create geometry with a mirrored copy, the parametric mechanism cannot resolve and maintain the symmetry without more information.

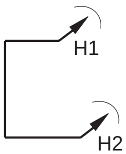
Problem

The problem message shows three unrelated *groups*: the trapezoid on the left and the two rightmost corners, as shown below.





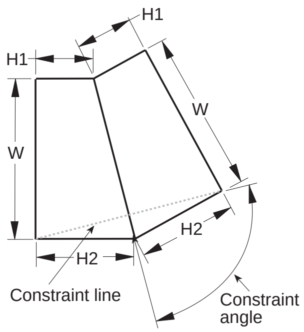




If parametrics swings an arc of the radius *H1* and *H2*, it doesn't know where the point is supposed to be on the arc. The point has a distance but no direction unless an angle dimension is added.

Solution

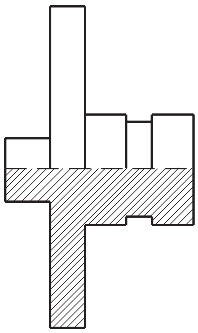
*S*ingle points that are considered a group provide a clue to the solution. In this example, the parametric mechanism does not know where those points are, relative to the original trapezoid. You can solve the problem by adding a constraint line and angle dimension between that line and the mirror line.



Complex Parametric Drafting

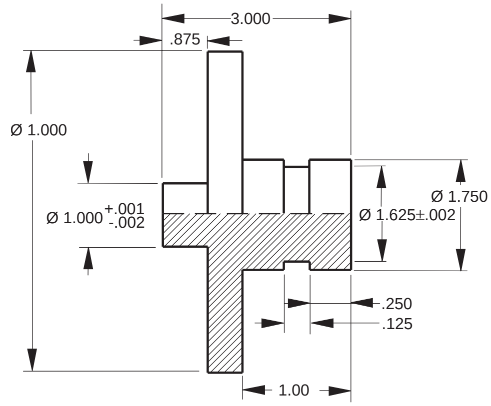
This section illustrates two complex examples of parametric drafting. These examples combine the information provided in this chapter so you can see how to use the parametric mechanism.

Example—Departures from Standard Drafting

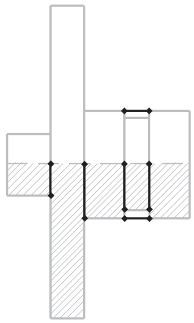


This example illustrates the parametric drafting practices that differ from standard drafting practices. The first illustration is the side view of an adapter without dimensions so you can see its shape clearly.

Here is the side view with typical dimensions added.

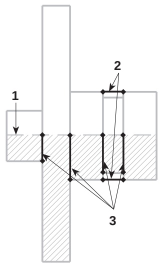


The illustration shows the side view with added constraint lines.

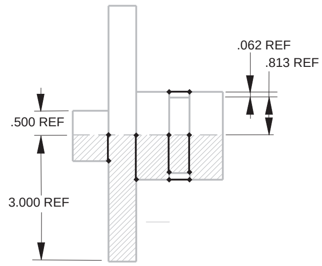


The next illustration shows the side view with numbered constraint lines.

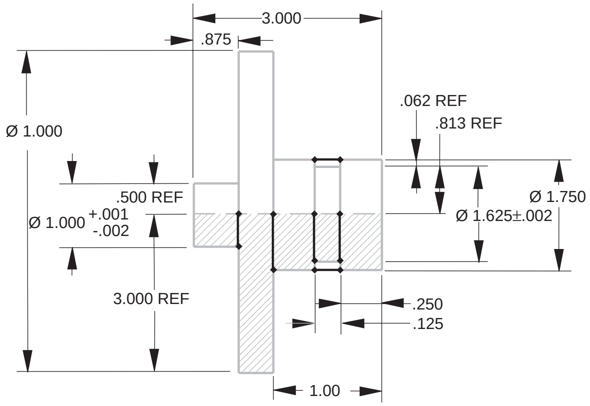
1. The centerline does not extend beyond the geometry.
2. Constraint lines define the edges of the groove. Parametrics does not recognize non-touching colinear lines.



3. Constraint lines connect the *end-points* of the upper and lower halves to relate the geometry. Parametrics does not recognize non-touching colinear lines.
4. Four dimensions are added to relate the geometry to the centerline. Parametrics doesn't recognize non-touching colinear lines.



The illustration below shows the completed parametric drawing.



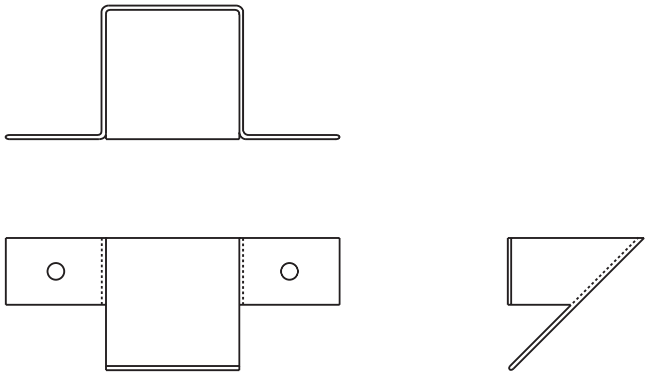
Additional Tips:

1. You should place the additional information on a separate layer so you can hide the layer when you plot the drawing.
2. If you use a yellow pen (which is hard to see against a white background) you won't confuse the additions with the actual geometry.

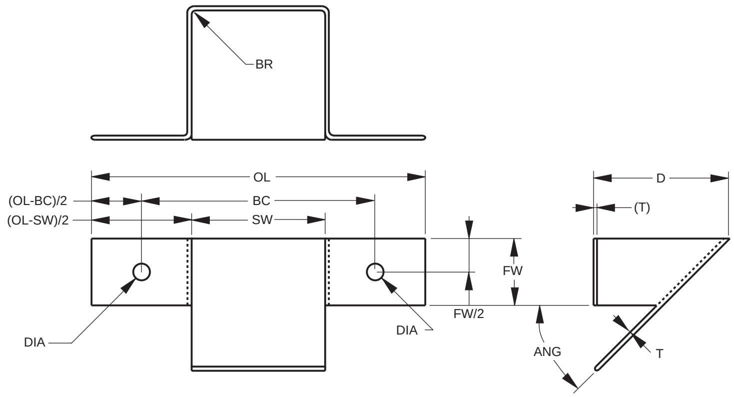
Example—Keep It Simple or Testing as You Go

When you have unconnected multiple views, the parametric mechanism can't determine the relationships between them.

The coin chute example below describes the parametric drafting of three views.

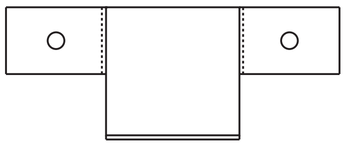


You can dimension these views according to standard drafting practice, as shown.

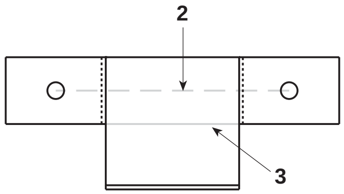


Section 1

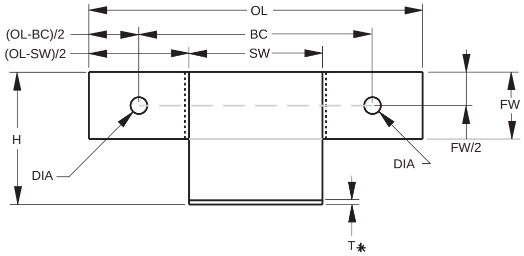
1. Construct and resolve one section at a time. Solve any problems with this bracket before moving on.



2. Add a centerline between the holes to maintain alignment.
3. Add a line connecting the lower edges of the flanges to keep the lines colinear.



The section below would resolve without problems, so you could go on to the next section.

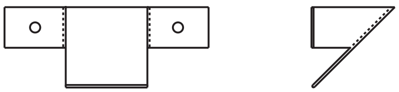


Once this view resolves properly, you can add the side view.

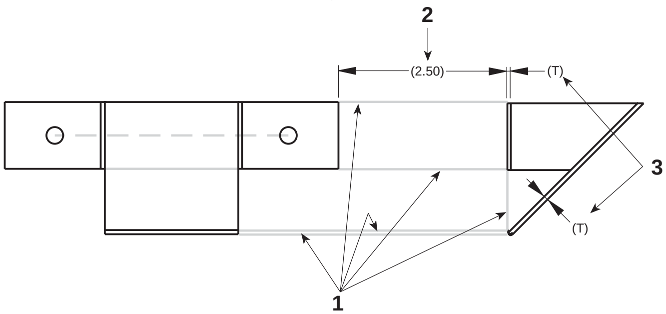
Since the side view determines height, you can delete the height (H) dimension and the lower thickness (T) dimension after you added the side view.

Section 2

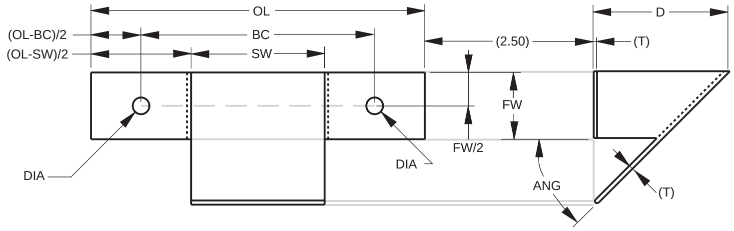
1. Add constraint lines to connect the geometry between sections.



2. Add a dimension to specify the distance between the sections. (See the two boxes problem earlier in this section.)

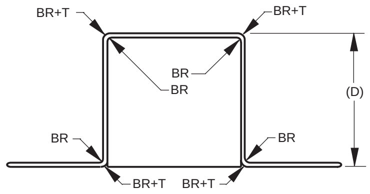


3. Add extra thickness (T) dimensions since all geometry must be defined.

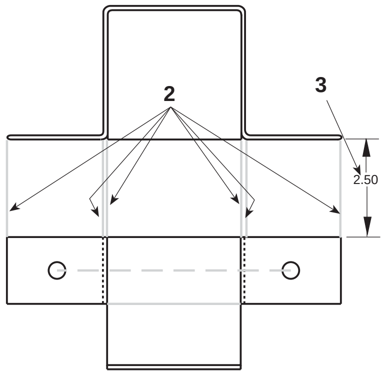


Section 3

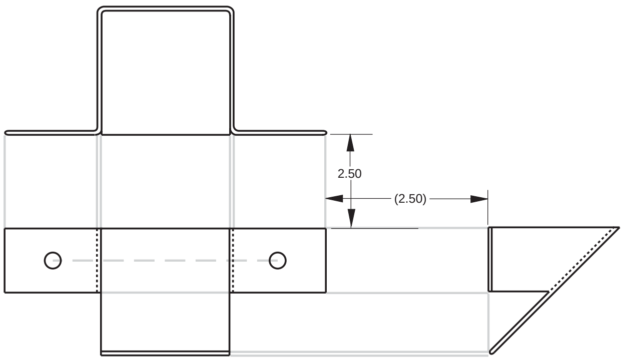
1. Dimension each fillet, as explained earlier.



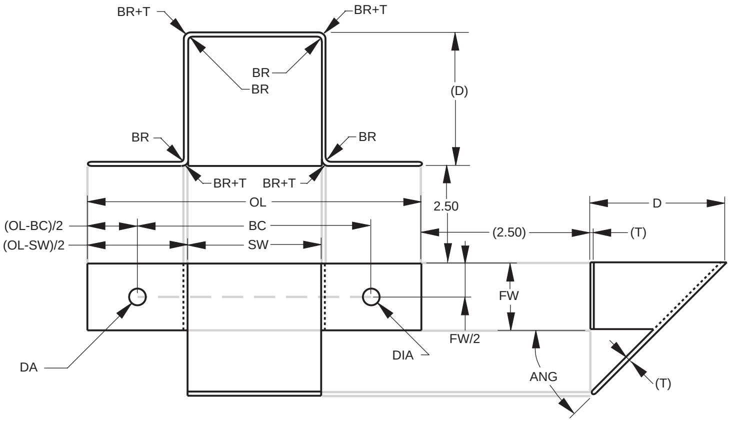
2. Relate this section to the rest of the geometry.
3. Add a dimension to specify the distance between the sections.



Three sections are related as shown below.



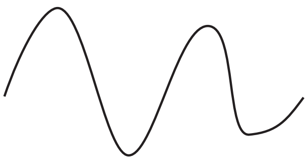
This is the completed parametric drawing.



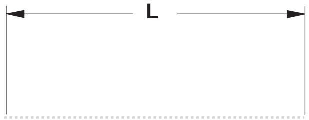
Parametrics and Grouped Objects

You can create many objects and then group them and treat them as a single parameterized object. There must be a framework on which the group sits. Dimension between two control points on the framework and then resolve the framework; the group changes accordingly, shrinking or expanding proportionately to the distance between the control points.

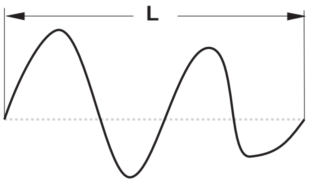
You can create a spline and group it.



Then create a framework line between the *endpoints* and dimension the line.



When you resolve for the length of the line (L), the spline changes accordingly.



Using Parametrics and Grouped Geometry

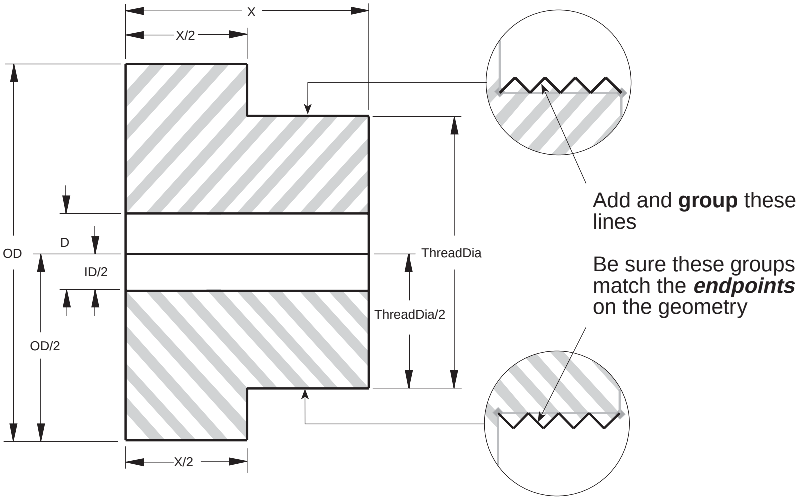
1. Create the geometry to be parameterized.
2. Select the geometry.
3. Choose ***Arrange>Group***.
4. Create a framework on which the grouped geometry sits.
5. Dimension between two control points. You can use only one variable per group.
6. Select the framework, the dimension and the group.
7. Choose ***Edit>Resolve***.
8. Enter the value for the distance between the control points.
9. Click OK.

If you include a dimension as part of the Group, the dimension will change when you resolve the Group. It will not be used as part of the parametric solver.

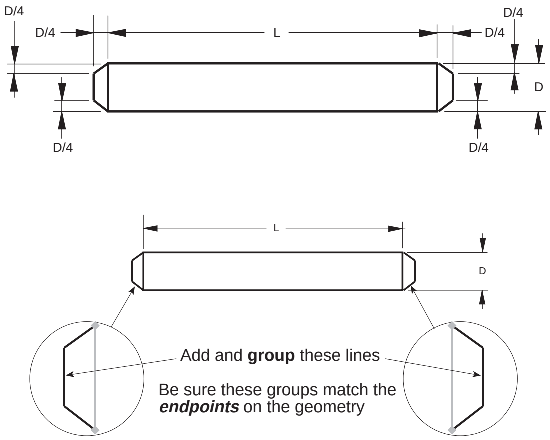
Rigid Links

You can group geometry into a rigid body and then attach the rigid body to parametric geometry in at least two points. When you resolve the parametric geometry, the rigid body undergoes the same geometric transformation.

The following example shows a real-life example of using parametrics with grouped objects. Begin with a basic shape that resolves properly, as shown in the part below. Then add the thread groups. When you resolve the part, the threads also resolve.



In the next example, you can see the usual method of parametric drafting followed by an example showing how using parametrics with groups can simplify making changes to the part.



Parametrics in 3D

The parametric feature works in 3D with some restrictions. You might consider its functionality as 2 1/2D since it functions properly in all planes parallel to the work plane.

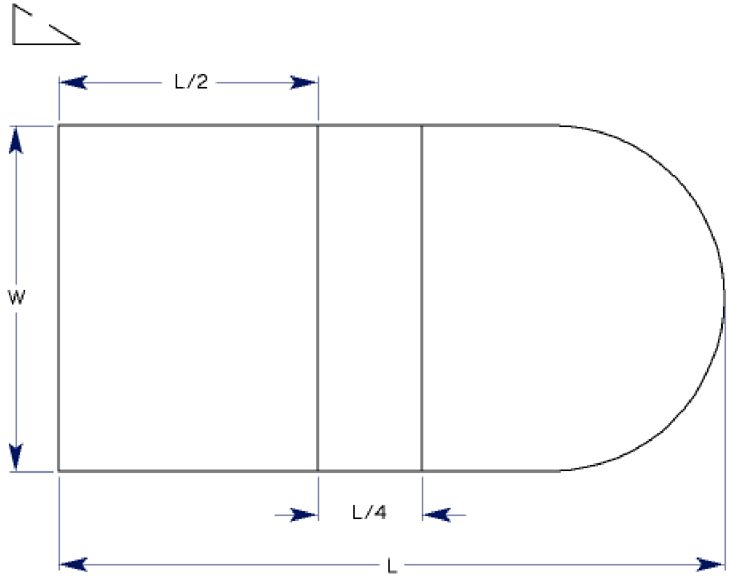
The parametric mechanism has two phases. In the first phase, parametrics ignores the z coordinates of the selected geometry and resolves the geometry as it does in 2D. Any dimension that is not completely in a plane parallel to the x,y plane is also ignored. In other words, the x and y coordinates are resolved and the z coordinates are not changed.

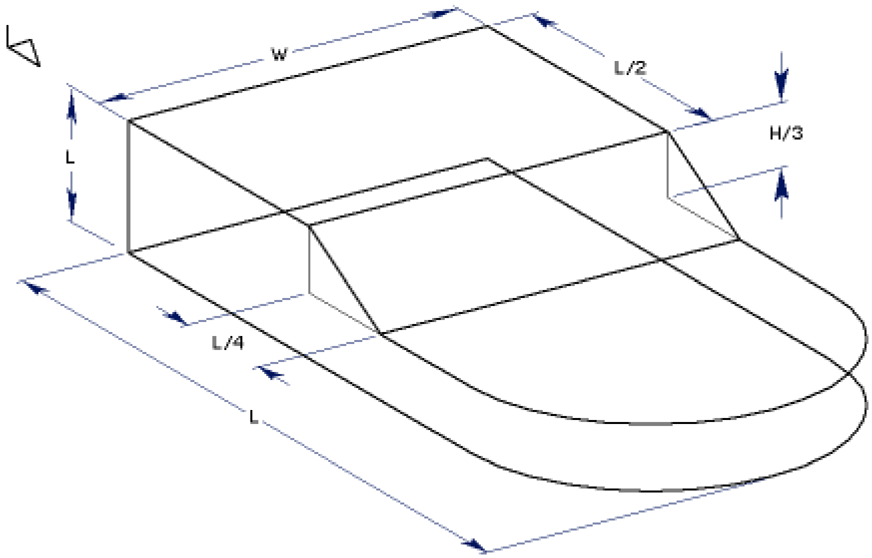
In the second phase, parametrics adjusts the z coordinates, as necessary. All linear dimensions parallel to the z axis are examined and every point in the selected geometry is considered to define a plane parallel to the x,y plane. The dimensions in the z direction define the required distances between these planes. Unlike the first phase, in the second phase parametrics is very tolerant of missing dimensions. If the dimensions do not completely define the distance between any two planes, then the distance is not changed.

In phase two, any line or arc that does not lie in a plane parallel to the x,y plane is modified appropriately because the defining points are adjusted by parametrics and the line or arc is changed accordingly. This includes all lines parallel to the z axis and any oblique lines at any angle to all three axes.

Parametrics deals with all geometry in the current work plane coordinates. Consequently, it is a good idea to set up a view that looks down the z axis of the work plane onto the x,y plane. In this way, you can see what the first phase of the operation "sees." This is particularly useful if you encounter problems resolving parametric geometry. Errors in parametrics are shown in bold lines drawn in the x,y plane and independent of the z values of the highlighted lines. The easiest way to see this is to look down on the x,y plane by choosing ***Views>View The Plane***.

In the drawing below, the first phase of parametrics will deal with only four dimensions W, L, L/2 and L/4. In the second phase, the H and H/3 dimensions are evaluated.





Symbols

In Graphite you can place any drawing as a symbol and open any symbol as a drawing. You can create symbol documents for generic, parameterized parts, and then bring them into your drawing, customizing the dimensions with each use. Symbols are particularly useful for creating a library of standard designs common to your work. For example, you can create libraries of windows, screws, nuts or bolts.

Graphite distinguishes between:

- *simple symbols*, which have neither parametric dimensions nor underlying value tables,
- *parametric symbols*, with editable dimensions

  and
- *symbol libraries*, that contain parametric symbols with underlying value tables.

All commands you use to manipulate symbols can be found by choosing ***File>Symbols...***.

Most of the symbols shipped with Graphite are parametric symbols. Parametric means that you can specify the symbol's final measurements just before placing it into a drawing. This feature allows you to use one symbol in countless variations.

The following topics are covered in this chapter:

- Using Symbols
- Placing Symbols
- Smart Symbols

- Symbol Libraries
- Creating Symbol Libraries

Using Symbols

The installation procedure copied the parameterized symbols in the Symbols folder of the Graphite folder. You may want to look at some of those symbols to get an idea of the type of symbols you can create with Graphite.

When you bring a symbol into a drawing, not only can you specify the location of a symbol to be added to your drawing, but you can also indicate its orientation. In order for this to work properly, a control point must be at the Origin (0,0) in the original symbol file.

As a special feature for use with parametrics, lines in the Construction pen style do not appear with the symbol when it comes into a document. You can use this feature to define your own smart windows and smart doors, as well as other symbols.

Creating a Symbol Document

1. Construct the geometry, using the Construction pen style for any lines you do not want to appear in the symbol when it comes into a drawing. Be sure to place one control point at (0,0) if you want to specify the location of the symbol when you bring it into a drawing.
2. Dimension all aspects of the part by entering a variable in the text data field in the status line of each dimension.
3. Choose ***Edit>Resolve*** to check the accuracy of the parametric dimensions.
4. Choose ***File>Save As***.
5. Name the document as you would with the *Save* command.

Placing Symbols

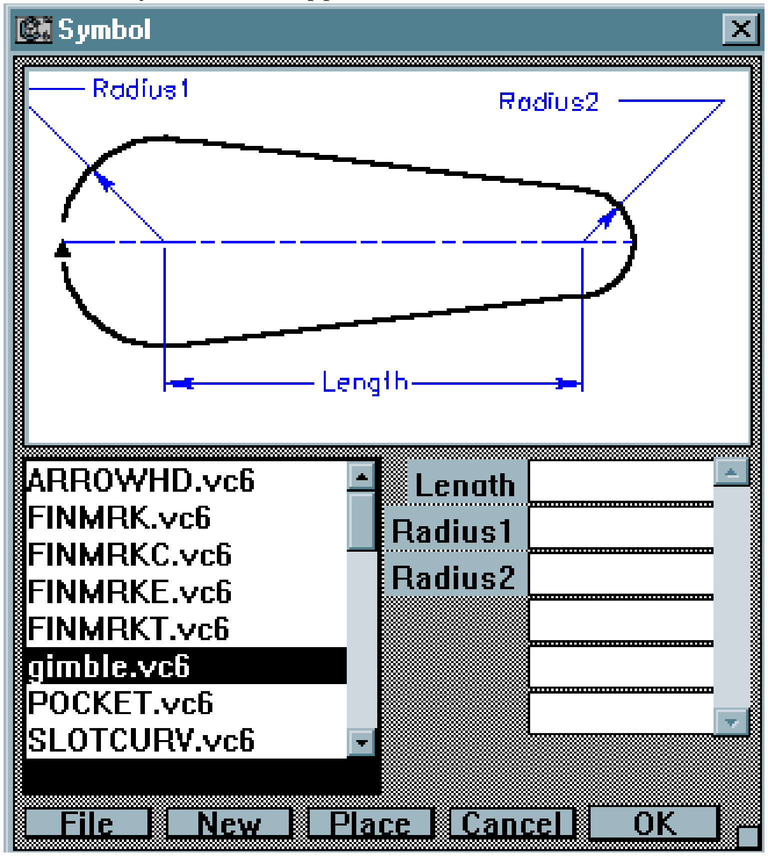
It is a very easy process to place symbols into your drawing. Using the ***File>Symbol...***, you place either symbols you created or those that came with the program.

Symbol Commands

When you select *Symbol* in the File menu, an open file dialog box appears. Using the dialog box you will navigate to and select a symbol directory. All the symbols in that directory are then loaded into the *Symbol Panel* and their file names display along the left side of the *Symbol Panel*.

Insert

Choose ***File>Symbol...*** and a standard open file dialog box displays. Once you select a symbol, the *Symbol Panel* appears.



The dialog box displays a preview of the symbol and lists the file names of all other symbols stored in that directory. The *Symbol Panel* is resizeable.

For parametric symbols, entry fields for all variables defining the symbol are displayed.

To place the symbol in your drawing click or drag a vector on the drawing area. If you click, the symbol inserts at the click location, in its original orientation. If you drag, the starting point of the drag specifies the *insertion point* for the symbol and the direction of the drag indicates the *orientation*.

Tech Note:

Click the small resize box in the lower right corner of the *Symbol Panel* to drag and resize it. Shift click to minimize the panel (for use on nonparametric symbols, such as nuts, bolts, valves, etc.).

Bringing a Symbol into the Current Document

1. Choose ***File>Symbol...***.

   The Open dialog box appears.

2. Select the symbol file you want to use and click **Open**.

   The Symbol dialog box appears displaying a preview of the currently selected symbol. All symbol files in that directory appear in the symbols list on the left side of the *Symbol Panel*.

3. Enter a value for each of the parametric dimensions.
4. Specify the location and orientation for the symbol.

   In the dialog box, a triangle appears on the geometry to indicate the origin or the point you are locating. If you click, the symbol inserts at the click location, in its original orientation.

   If you drag, the starting point of the drag specifies the insertion point for the symbol and the direction of the drag indicates the orientation.

   If you do not specify a location, you may have to scroll to see the symbol.

5. Click OK.

   The geometry resolves and appears in the current drawing at the location you clicked—sized as you specified.

   The symbol geometry is selected so you can move it to a new location.

6. Add regular dimensions if desired.

Tech Note:

You can use a text file to fill in parametric dimensions of a symbol. Create a text file with value pairings for each variable and its value. Then load the symbol and *click* on File in the *Symbol Panel*. The file automatically fills in the values you specified for each dimension.

If you want to see an enlargement of any part of the symbol within the viewing window, move the pointer to the area of interest and press the mouse button. The enlargement reduces when you release the mouse button.

Browsing a Directory of Symbols

1. Choose ***File>Symbol...***.

   The Open dialog box appears.

2. Select a symbol file from the directory you want to browse and click **Open**.

   The Symbol dialog box appears displaying a preview of the currently selected symbol. All symbol files in that directory appear in the symbols list on the left side of the *Symbol Panel*. (To change the selected symbol, ***click*** on a file name from the list along the left side of the *Symbol Panel*. The preview pane updates to display the currently selected symbol.)

3. Click New to load a different directory of symbols

Importing a Symbol

You can also use the *Import* to bring in the geometry and dimensions of a symbol. When you import a symbol, it comes into the current drawing at the size it was drawn; you can specify measurements by choosing ***Edit>Edit Objects*** or choose ***Edit>Select All*** and then choose ***Edit>Resolve*** to change the size.

Editing Parametric Symbols

Using the *Edit Objects* command you can edit parametric symbols placed in a drawing. After having selected the symbol, choose ***Edit>Edit Objects***, then you can edit the dimension of the symbol.

Smart Symbols

The Architect folder of the Symbols folder contains smart symbols for doors and windows. Smart window and door symbols are smart because they contain a smart wall segment. When you add one of these symbols to a smart wall, it breaks into the wall in the location you specify.

You should draw smart walls with the Construction pen style when creating your own smart symbol. (Geometry created with the Construction pen does not appear when it is brought into a drawing.) The invisible geometry is still functional in that existing smart walls in the drawing detect the invisible wall of the symbol.

You must draw a thick wall in line with or overlapping a thinner wall, so it hides the thinner wall. Therefore, to get a symbol to mask part of a wall, you should include a single wall that overlays the target wall. If you want to see an example, open one of the smart symbols in the Architect folder that was installed with Graphite.

Creating a Smart Symbol

1. Choose ***Pen>Style>Construction***.
2. Draw a smart wall thicker than the maximum thickness you expect to use and only as long as the final symbol will be.
3. Change pen styles to any style other than Construction.
4. Draw the symbol.
5. Choose ***File>Save As*** and save as you would any other symbol.

Using Symbols for Smart Windows and Doors

The Symbols/Architect folder that was installed with Graphite contains symbols for smart windows and smart doors. These symbols work like regular Graphite symbols in that they are parameterized and you can place and orient them on your drawing. In addition, they break into any smart wall they touch and the cut area of the smart wall fills in if you move or delete the smart symbol.

2D Analysis

The 2D Analysis provides sectional properties for the selected geometry—a feature that is useful for many calculations related to design and drafting. Architects can use the perimeter and area values for calculating material requirements; engineers can use the moments of inertia for stress analysis; manufacturers can use the centroid for balancing parts for turning. The following topics are covered in this section:

- 2D Analysis
- Calculations

2D Analysis Command

This command in the Layout menu displays the statistics on the selection—the length of the perimeter, the enclosed area, the center of gravity (centroid), and moments of inertia. The selection must be a closed figure.

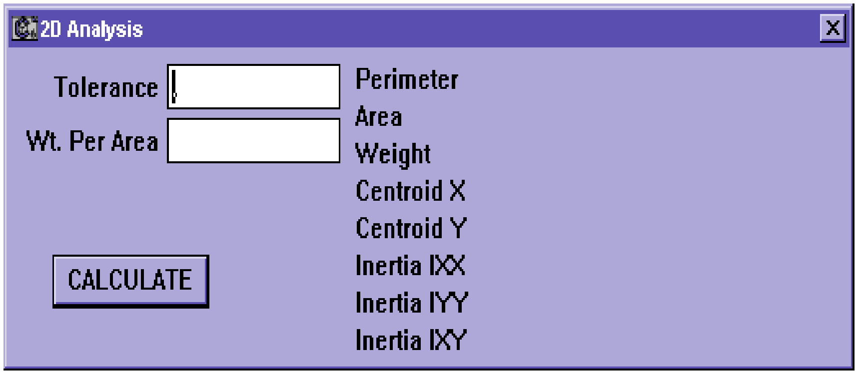
Tech Note:

The closed figure is an 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.

Performing 2D Analysis

1. Select the geometry that defines a closed boundary. You may want to use the **Tracer** tool to select the perimeter.
2. Choose ***Layout>2D Analysis***.

The analysis is performed and the values appear in the dialog box.



Tech Note:

Once you crosshatch the area and you see that your selection is correct, you can choose *Undo* to remove the crosshatching and still maintain the selection.

Tech Note:

This tolerance value has no effect on the precision of the resultant values. The precision of the result is set by choosing ***Layout>Preferences>Units.***

3. If necessary, enter changes for the tolerance and weight per area values.
4. Click CALCULATE.

The analysis recalculates and the new values appear in the dialog box.

The following items are included in the dialog box:

Tolerance

A setting to determine the accuracy of calculations for curved objects. The smaller the tolerance is, the higher the accuracy will be. The default value is 0.01. You can change the value by entering a new number.

The 2D analysis mechanism analyzes arcs, circles, ellipses and splines as straight line segments that deviate from the true curve by no more than the stated tolerance value. The smaller the number, the greater the accuracy; however, the greater the accuracy, the longer the calculation time.

Weight Per Area

A multiplier used to determine the actual weight of a part of constant thickness defined by a selected boundary. The weight of the part is the product of the Area multiplied by the Weight Per Area.

The default entry is 1. You can change the value by entering a new number. You can find the proper number for this entry in a handbook published by most material vendors, particularly for sheet steel and aluminum. Enter the weight for the thickness of the proposed material and click CALCULATE.

For example, if you had a square that you wanted analyzed as if it were a cube three inches thick, you would enter **3 * the material's density** in the weight per area box.

Perimeter

The length of all segments which define the selected boundary. This is the only value that appears if the boundary is not closed.

For circles, arcs and other curves, the accuracy depends on the tolerance setting.

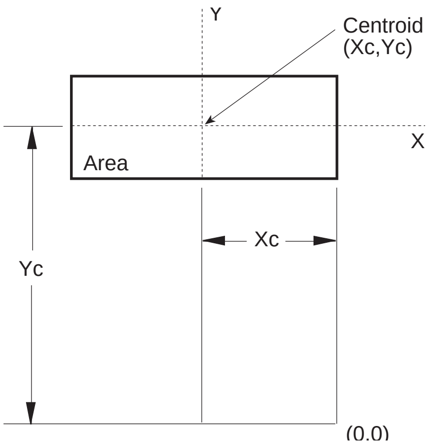
Area

The surface area enclosed by the selected boundary.

For circles, arcs and other curves, the accuracy depends on the tolerance setting.

Weight

The product of the *Area* multiplied by the *Weight Per Area*.



Centroid X

The X coordinate of the center of mass defined by the selected boundary.

Centroid Y The Y coordinate of the center of mass defined by the selected boundary.

Inertia IXX The moment of inertia about the central X-X axis which is parallel to the X-axis.

$IXX = \int (y\text{-}Yc)^2 dA$

Inertia IYY The moment of inertia about the central Y-Y axis which is parallel to the Y-axis.

$IYY = \int (x\text{-}Xc)^2 dA$

Inertia IXY The product of inertia about the centroid.

$IXY = \int (x\text{-}Xc)(y\text{-}Yc) dA$

Calculations

Graphite calculates the values for 2D Analysis before the dialog box appears. If you make a change in the *Tolerance* or the *Weight Per Area*, you must click CALCULATE to recalculate the statistics.

The 2D Analysis mechanism evaluates the boundary in the same way the crosshatching mechanism does; for example, Graphite considers a circle inside a bounded area to be a hole. Crosshatching does not fill the hole and 2D Analysis does not include the area of the circle in the area calculation.

Displaying the Centroid

1. Select the geometry defining the part.
2. Choose ***Layout>2D Analysis***.

   The 2D Analysis dialog box appears.
3. Choose ***Layout>Construction***.

   The Construction dialog box appears.
4. In the *Angle* field in the Construction dialog box, enter **0;90**. (Be certain to separate the numbers with a semicolon.)
5. Enter the value for Centroid X in the *X* field of the Construction dialog box.
6. Enter the value for Centroid Y in the *Y* field of the Construction dialog box.
7. Press ENTER (Windows) or RETURN (Macintosh).

Construction lines display, intersecting at the centroid of the selected part.

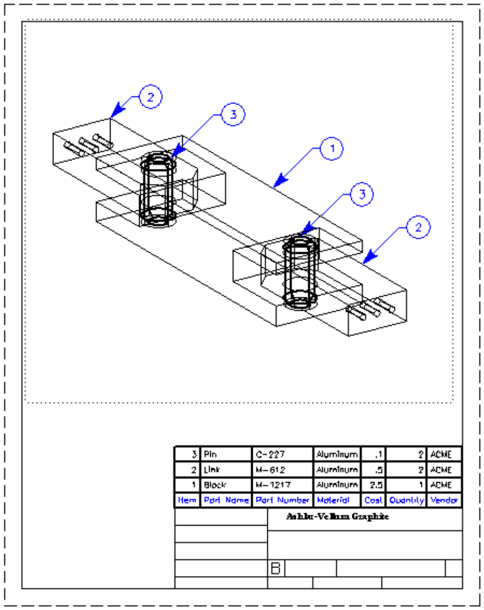
Bill of Materials

Bill of Materials is an important extension to CAD programs for tracking and listing parts all the way through the product design and manufacturing process. In the product manufacturing industry every part produced has some type of distinguishing mark—typically the part number—either stamped, etched, embossed, or silk screened onto the part. The manufacturer also needs to document the design and track the characteristics of each part like material, cost of manufacturing, and outside vendor code during the product development process.

The following topics are covered:

- Introduction
- Attributes
- Item Numbers
- Bill of Materials Table

Introduction



Graphite's Bill of Materials Extraction utility gives you the ability to attach characteristics (attributes) to the parts being designed. From these attributes you can extract a Bill of Materials table. This Bill of Materials table can be printed separately or with the drawing.

In addition, you can export the Bill of Materials to other applications.

The Bill of Materials Extraction utility is completely integrated in Graphite and combines powerful functionality with an easy to use interface.

Generating a Bill of Materials is a two step process:

- defining and assigning attributes to object geometry in drawings

  and

- extracting user-defined and predefined attributes in the form of lists or ASCII-files for export purposes.

Tip:

You can decide to use the BOM package on new projects only and assign attributes to the parts on an as-needed basis for new designs.

When using the Bill of Materials Extraction utility for the first time, you should start by interviewing all the groups within your organization or company to determine which characteristics need to be tracked. The next step is to assign these characteristics to an existing catalog of parts, typically Graphite symbol files. These parts can then be used to create an assembly or installation drawing. These attributes can then be extracted into a Bill of Materials table or exported for use in other applications.

The Bill of Material distinguishes between two kinds of attributes:

User-defined Attributes You define and assign attributes to object geometry after you have created the geometry.

Predefined Attributes These are numerical attributes like the perimeter or area of a circle which are automatically defined by creating the geometry.

Non-numerical attributes like line color and line style are not recognized by the Bill of Materials.

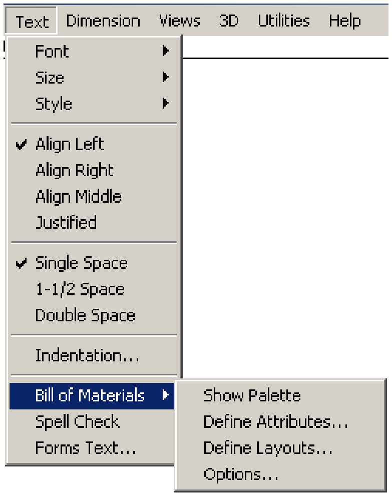
Attributes and Objects

Each object you create in Graphite automatically has different kinds of attributes: *numerical attributes* like the perimeter and the area of a circle, and *non-numerical attributes* like the line color and the line style.

With the Bill of Materials Extraction utility you can now assign User-defined Attributes to objects, like a Part Name or Part Number.

Defining Attributes

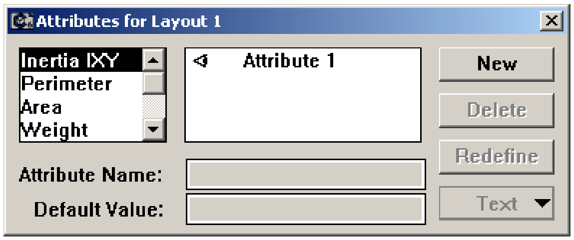
Before you can assign attributes to objects you have to define them first. For defining attributes you choose ***Text>Bill of Materials>Define Attributes***.



Attributes Command

This command allows you to define, delete, redefine and activate attributes. Active attributes (attributes which can be assigned) are indicated by an “eye” mark in front of the attribute name. Locked attributes (their values cannot be changed during assignment) are indicated by a lock icon in front of the check mark.

The *Define Attributes* command displays the following dialog box:



Attributes are defined in the Define Attributes dialog box by three parameters:

Attribute Name
In this entry field you specify a general name for the attribute like name, material, serial number, etc.

Default Value
This entry assigns a specific value to the attribute. So you could assign the value of steel to the attribute of material. The value of an attribute can be changed in the Status Line before assigning it to object geometry or in the Edit Objects dialog box after it has been assigned.

Text
In this pop-up menu you select a format for the attribute value. You can choose between three attribute formats: *Text*, *Number* and *Qty*.

Text is the default format and is the choice for all alpha-numeric values (Text entries) like Name, Material, etc.

Number is the choice for all values which are expressed by numbers which can be used for further calculations. A part number like *S3456* or *252-49* would be formatted as Text, since this value would never be used for any kind of calculation. If you use hyphens or dashes in your Part Number, you must use the *Text* format, not *Number*.

An attribute would be formatted as *Qty*. if the Bill of Material should calculate the number of identical objects in a drawing. Normally this format only makes sense for an attribute which is named *Qty*. and has the default value of one (1).

The Define Attributes dialog box contains the following buttons:

New
Clicking New creates a new attribute like Attribute 1, 2, 3, etc. You can rename the attribute by typing the name in the *Attribute Name* entry field and clicking Redefine.

Delete
Clicking Delete deletes the selected attribute from the attribute list.

Redefine — Clicking Redefine does three things: First it renames a selected attribute with the name you have specified in the *Attribute Name* entry field; second, it assigns a value to the attribute which may be entered in the *Default Value* entry field; and third, it assigns the format for the attribute you selected from the *Format* pop-up list.

Defining Attributes

1. Click New to create a new attribute.

   In the *Attribute Name* entry field the name Attribute 1 displays.

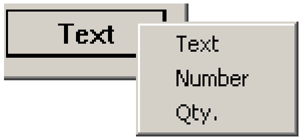
2. Rename the attribute by typing a name in the *Attribute Name* entry field, like COST.

3. Specify a value for the attribute in the *Default Value* entry field.

   The value you enter here can be changed in the Status Line during assignment, like 2.50. If you use the $ sign, the format must be *Text*, not *Number*.

4. Select a format for the new attribute. In the examples given above, the format is *Number*.

   Click the *Format* pop-up menu and drag the mouse pointer (with the left mouse button pressed) to the desired format. As soon as you release the mouse button the selected format displays in the pop-up menu.

5. Click Redefine.

   The new attribute name appears in the attributes list, and the specified value and format is assigned to the attribute.

6. Close the Define Attributes dialog box by clicking the Close button (Windows) or the Close box (Macintosh).

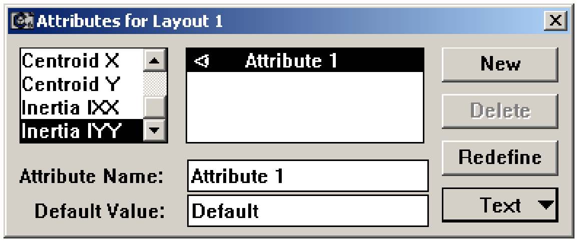
Show or Hide Attributes

Only visible attributes (indicated by a eye mark beside the attribute name) can be seen during assignment. The user cannot edit hidden attributes.

1. Select an attribute to show or hide from the list box. The eye mark besides the attribute name appears for shown objects or disappears for hidden objects.

Locking and Unlocking Attributes

To lock or unlock an attribute simply click in front of the attribute's name. The lock icon, in front of the name, appears or disappears.



The values of locked attributes cannot be changed during assignment.

Changing the Characteristics of an Attribute

Choose the ***Text>Bill of Materials>Define Attributes*** to open the Define Attributes dialog box.

1. Select the attribute whose name, value or format you want to change.
2. Type a new name or value, or choose a new attribute format.
3. Click Redefine.

   The new attribute name appears in the attributes list and the new value and format are assigned to the attribute.
4. Close the Define Attributes dialog box by clicking the Close button (Windows) or the Close box (Macintosh).

Deleting Defined Attributes

Choose ***Text>Bill of Materials>Define Attributes*** to open the Define Attributes dialog box.

1. Select the name of the attribute from the list box.
2. Click Delete. The selected attribute deletes.

Saving Attributes

With the Redefine button you rename an attribute or assign a new value or format to the attribute.

If you want to save all attribute definitions permanently, choose ***Layout>Preferences>Save Preferences***. Close Graphite and relaunch it for the new preferences to take affect.

Predefined Attributes (Graphite Attributes)

Predefined Attributes are attributes that Graphite defines. These attributes are calculated and displayed using 2D Analysis (see Chapter 18). Choose the ones you want from the list by double clicking.

- Perimeter
- Area
- Weight
- Centroid X
- Centroid Y
- Inertia IXX
- Inertia IYY
- Inertia IXY

For predefined attributes enter the attribute's name, since their values and formats are predefined. These attributes are displayed in the Status Line but cannot be changed, since Graphite calculates the values automatically.

Assigning Attributes

You assign attributes using the **BOM** tools. You activate **BOM** tools like any other tool in the Graphite tool palette.

An object's attributes are inaccessible if the object is grouped, so that the group itself can have attributes. The object's attributes are made accessible by ungrouping the group. Ungrouping a group with attributes removes the group's attributes.

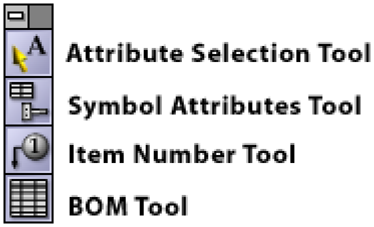
BOM Tools

Graphite’s **BOM** tools are on a separate palette which you can drag around the drawing area. The palette displays by choosing ***Text>Bill of Materials>Show Palette***.

Show Palette/Hide Palette

When you choose ***Text>Bill of Materials>Show Palette*** or ***Hide Palette***, the **BOM** tool palette display or hides.

You can also remove the tool palette by double-clicking (Windows) or single clicking (Macintosh) the button in the title bar of the palette.



Attribute Tools

With the attribute tools you assign attributes to object geometry or mark an object as a symbol. When you select one of these tools all attributes of the current BOM layout display in the Status Line. Hidden attributes do not display in the Status Line.

The attribute values shown in the Status Line can be entered or modified if they are not locked. If they are locked, the attribute's value is greyed out.

Attribute Selection Tool

With this tool you assign attributes of the current layout to a selected object.

Assigning Attributes

1. Choose ***Text>Bill of Materials>Define Layouts***.

   The Define Layouts dialog box displays.

   For more information about the Define Layouts dialog box, see the "Bill of Materials Layouts" section later in this chapter.

2. Select a BOM Layout.
3. Close the Define Layouts dialog box by clicking the Close button (Windows) or the Close box (Macintosh).
4. Select ***Text>Bill of Materials>Define Attributes***.

   The Define Attribute dialog box displays.

5. Hide all attributes which you don't want to assign to object geometry in the cur-

rent drawing by deselecting the “eye” next to that attribute.

6. Close the Define Attributes dialog box by clicking the Close button (Windows) or Close box (Macintosh).
7. In the **BOM** tool palette, select the **Attribute Selection** tool. The Message Line reads, *User Attribute Tool: Select Geometry…[Return = Append Attribute, Ctrl+Return = Remove Attribute {Windows} or Option+Return = Remove Attribute {Macintosh}].*
8. Select the object to which you want to assign attributes.
9. Enter the desired values in the entry field in the Status Line.

   Attribute values which are locked cannot be edited in the Status Line.
10. Press the ENTER (Windows) or RETURN (Macintosh) key.

    Important: The attributes will be assigned only when you hit ENTER (Windows) or RETURN (Macintosh).

Tip:

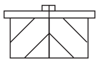
If you want to assign attributes to an object made up of several individual objects—like a rectangle, (made up of four single lines)—group the object before assigning attributes. Otherwise, the attributes assign to each individual object selected.

Symbol Attributes Tool

This tool marks a symbol with assigned attributes.

Graphite doesn't recognize any difference between drawings and symbols. They are only opened by different commands:

- Drawings are opened by choosing ***File>Open***.
- Symbols are inserted by choosing ***File>Symbol>Insert***.



Attribute	Value
Name	Wedge
Material	Steel
Quantity	1

Symbol attribute tables cannot be edited. They must be deleted and reinserted.

Marking an Object as a Symbol

With the **Symbol Attributes** tool you mark an object with a small table containing all assigned attributes. This table will be interpreted when the object is placed as a symbol.

When you insert this object with the *Symbol* command all attributes contained in this table will be listed in the Symbol dialog box. Values of unlocked attributes can be edited before the symbol is placed—without the attributes table—in the drawing area.

Symbol Attributes do not attach to the objects in the symbol file like object attributes. Instead, Symbol Attributes attach to the symbol when the symbol is inserted into a drawing.

If you open a marked symbol by choosing ***File>Open*** the attributes table appears in the drawing area with the symbol.

1. In the **BOM** tool palette, choose the **Symbol Attributes** tool. The Message Line reads, *BOM Symbol Attribute Tool: Select Insertion Point.*

2. Enter the values for the attributes in the Status Line.
3. Click to place the Symbol Attributes table in an empty spot in the drawing area.

   A table with all assigned attributes appears with the values entered in the Status Line.
4. Save the object with the Symbol Attributes table.

   The attributes in this table attach to the symbol when placed using the **Symbol** command, and the assigned values can be changed in the Symbol dialog box. There should be only one symbol table per file.

If you place a symbol holding object attributes but no Symbol Attributes table in a drawing, the object attributes do not display in the Symbol dialog and do not get attached to the symbol when placed in the new drawing. To access the object attributes, the symbol must first be ungrouped.

An object can have both object attributes and symbol attributes, and their values do not have to be the same.

Editing Attributes

Assigned attributes display in the Status Line, where the default values can be edited.

Showing Attributes

1. In the **BOM** tool palette, select the **Attribute Selection** tool.

2. Move the mouse pointer over the drawing area. As soon as the pointer comes near an object with assigned attributes, these attributes and their respective values automatically display in the Status Line.

   Important: Attributes must be visible (“eye” mark) and the proper layout must be active, as indicated in the Define Attributes dialog box and in the Define Layouts dialog box in order to display assigned attributes in the Status Line.

Changing the Values of Attributes

Tip:

Values of attributes can also be edited in the Edit Objects dialog box. To do so you select the object, choose ***Edit>Edit Objects*** change the value in the Edit Objects dialog box and click Apply.

1. In the **BOM** tool palette, select the **Attribute Selection** tool.

2. Move the mouse pointer near an object with assigned attributes.

   The attributes and their respective values display in the Status Line.
3. Select the object with the **Attribute Selection** tool.
4. Change the respective values in the Status Line. To remove an attribute value, delete the value in the entry field.
5. Press the ENTER (Windows) or RETURN (Macintosh) key.

   The changed values are assigned to the selected attribute.

Removing Attributes

1. In the **BOM** tool palette select the **Attribute Selection** tool.

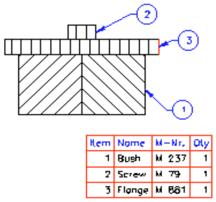
   

2. Move the mouse pointer near an object with assigned attributes.

   The attributes and their respective values display in the Status Line.
3. Select the object with the **Attribute Selection** tool.
4. Press the CTRL+ENTER keys (Windows) or the OPTION+RETURN key (Macintosh) simultaneously. All visible and unlocked attributes are removed.

Item Numbers

The **BOM** tool palette allows you to assign item numbers to objects which will be automatically included in the Bill of Materials table.



Assigning Item Numbers

Item numbers are assigned with the **Item Number** tool in the **BOM** tool palette.

Do not use Graphite's **Balloon** tool from Graphite's **Dimension** palette to assign Item Numbers for objects that you want listed in the Bill of Materials. Item Numbers created with the **Balloon** tool are not recognized by Graphite's Bill of Materials Extraction utility.

Item Number Tool

With this tool in the **BOM** tool palette you assign Item Numbers to objects. These Item Numbers will automatically be included in the Bill of Materials.

Using the Item Number Tool

1. In the **BOM** tool palette, select the **Item Number** tool. The Message Line reads, *Item Number Tool: Pick BOM geometry to point to... [Ctrl = Renumber].*
2. Three entry fields display in the Status Line; Item, Width and Size.
3. Enter the starting item number for the selected object into the Item field.

   Item Numbers can also contain alpha characters—A, B, C, etc.
4. Click the **Item Number** tool near the object you want to label.
5. Click with the **Item Number** tool in the drawing area to indicate the position of the Item Number balloon.

   The Item Numbers are placed at the location you clicked.

If you place additional position balloons the Item Number automatically increases. The default Item Number always increments from the last Item Number used.

If you assign an Item Number to an object, already assigned to another object, the item numbers adjust as follows:

- If the new Item Number is lower than the highest already assigned Item Number, all higher Item Numbers increase.
- If the new Item Number is higher than the highest assigned Item Number, the existing Item Numbers remain unchanged.

You cannot assign more than one Item Number to an object; if you do, the original Item Number balloon deletes. To get the original Item Number back, do an *Undo*.

Adjusting Item Number Balloon and Text Size

The following procedure describes how to adjust the Item Number balloon and Text size before you assign any Item Numbers.

1. In the **BOM** tool palette, select the **Item Number** tool.

   The entry fields, Item, Width and Size, display in the Status Line.
2. Press the TAB key twice and enter the desired width for the Item Number balloon. The units for the indicated width are based on the unit specified in the Units dialog box from the ***Layout>Preferences>Units***.
3. Enter the desired size.
4. Select the desired text size.

   The text size of the Item Number balloon is now set.
5. Press the ENTER (Windows) or RETURN (Macintosh) key.

   The size of the Item Number balloon is now set.

Editing Item Numbers

Tip:

An Item Number's font is specified in the Font submenu of the Dimension menu.

Item Number, Text Size and the Size of the Item Number balloon can be changed.

Editing the Size of an Existing Item Number Balloon

1. Select the desired Item Number balloon.
2. Choose ***Item Number Tool***.

Tip:

The shape of the Item Number balloon can also be changed in Edit Objects. If you renumber existing Item Numbers, all Item Number balloons revert to the default circle balloon.

3. In the Status Line, enter the new diameter in the width field.
4. Press the “Enter” or “Return” key.

   The diameter of the selected Item Number balloon updates accordingly.

Editing the Text Size

1. Select the respective Item Number balloon.
2. Choose ***Edit>Edit Objects***.
3. In the Status Line, enter the new Text Size in the size field.
4. Press the “Enter” or “Return” key.

   The text size of the selected Item Number balloon update accordingly.

Item Number text size can be changed dynamically by selecting the Item Number balloon and changing the size in the Dimension menu.

Deleting Item Numbers

1. Select the desired Item Number balloon.
2. Select ***Edit>Delete*** or press the BACKSPACE (Windows) or DELETE (Macintosh) key.

   The selected Item Number will be deleted.

Renumber Existing Item Numbers

1. In the **BOM** tool palette, select the **Item Number** tool.

   The entry fields, Item, Width and Size, display in the Status Line.
2. Enter in the Item field the new starting number for the Item Number, press the “Enter” or “Return” key.
3. Press the CTRL (Windows) or OPTION (Macintosh) key and keep it pressed.
4. Click the Item Number balloons in the desired order.

   The Item Number balloon will split. The upper half of the symbol shows the old Item Number and the lower half of the symbol the new Item Number.
5. Release the CTRL (Windows) or OPTION (Macintosh) key.

   All Item Numbers renumber.

Item Numbers and Object Copies

The **Item Number** tool automatically recognizes if an object is a copy of an object to which an Item Number is already assigned as long as the original object has an attribute of the type *Quantity*. The **Item Number** tool automatically assigns the identical Item Number of the original object to the object copy.

Bill of Materials

After you have assigned attributes to objects you create a Bill of Materials. The Bill of Materials will contain all Item Numbers and their associated attributes.

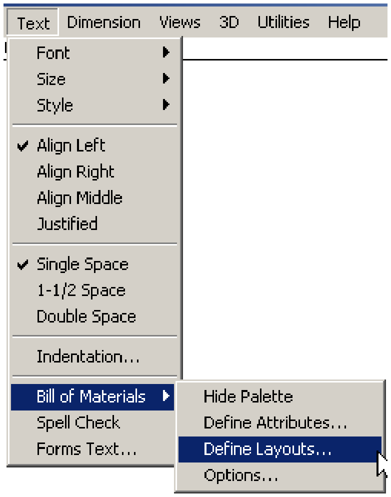
3	Pin	C-227	Aluminum	1	2	ACME
2	Link	M-612	Aluminum	5	2	ACME
1	Block	M-1217	Aluminum	2.5	1	ACME
Item	Part Name	Part Number	Material	Cost	Quantity	Vendor

The Bill of Materials will be placed on the current sheet. You can move it to another location, copy it to another sheet or export it into an ASCII file to use it in other programs like spread sheets or text editors.

In order to create a Bill of Materials you must define a layout first.

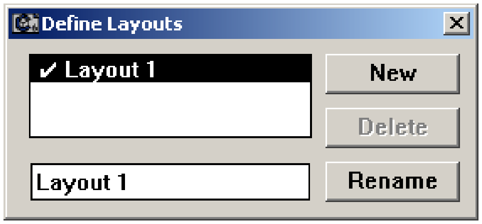
Bill of Materials Layouts

In conjunction with the Define Attributes dialog box, the *Layouts* command found by choosing ***Text>Bill of Materials>Define Layouts*** allows you to define different layouts for the Bill of Materials.

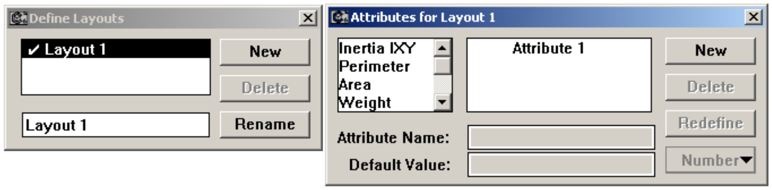


Layouts Command

The *Layouts* command, in the BOM submenu of the Modules menu, displays the following dialog box:



Layouts differ only by the attributes to which they relate. A layout contains all the visible attributes listed in the Define Attributes dialog box. (Visible attributes are indicated by a check mark.) Therefore, the Define Attributes dialog box should be displayed when defining a layout.



Defining a New Layout

1. Choose ***Text>Bill of Materials>Define Layouts***.

   The Define Layouts dialog box appears.

2. Click New.
3. Rename the layout in the Layout Name entry field.
4. Click Rename.
5. Double click to activate/set.
6. Select the ***Text>Bill of Materials>Define Attributes***.

   The Define Attributes dialog box displays.

7. Define all the attributes you want to use in different layouts.
8. Make all the attributes visible that you want to save under one layout name.

All active attributes in the Define Attribute dialog box will automatically be saved under the current layout name for the current Graphite session. If you want to save layouts permanently, choose ***Layout>Preferences>Save Preferences***.

Editing Layouts

1. Choose ***Text>Bill of Materials>Define Layouts***.

   The Define Layout dialog box displays.

2. Double click to activate/select (the “check” mark shows which is active.)
3. Click Current.
4. Select the ***Text>Bill of Materials>Define Attributes***.

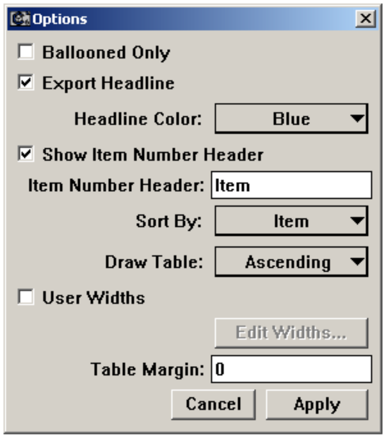
   The Define Attribute dialog box displays.

5. Show or Hide the attributes as desired (visible attributes are indicated by an eye mark).

All changes made in the Define Attribute dialog box will automatically be saved under the current layout name. If you want to save the changes permanently, choose ***Layout> Preferences>Save Preferences***.

Layout Options

For all BOM layouts you define you can set some preferences globally.

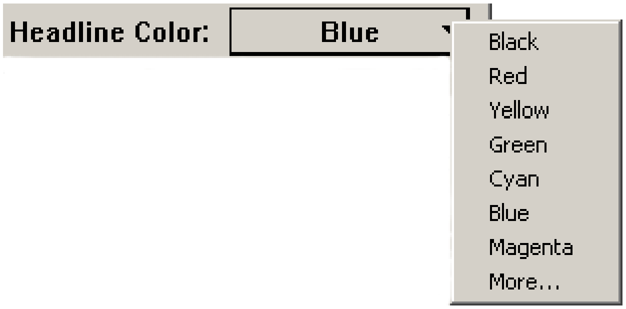


When you select the ***Text>Bill of Materials>Options*** this dialog box displays:

Ballooned Only	When checked, this option places only BOM items in the table that have an associated BOM Balloon in the file.
Export Headline	Draws the header line in the table when checked.
Show Item Number Header	Marking this option creates an Item column as the first column.
Item Number Header	In this entry field you name the Item column.

Table Margin In this field you can enter a value for the margin between the text and the vertical column guides.

Headline Color In this pop-up menu you choose a color for the title text.



To choose a color you click into the pop-up menu and drag to the desired color. As soon you release the mouse button the selected color displays in the pop-up menu.

Sort by In this pop-up menu you select the attribute by which the Bill of Materials should be sorted. Click into the pop-up menu with the left mouse button pressed and drag to the desired attribute. As soon as you release the mouse button, the selected attribute displays in the pop-up menu.

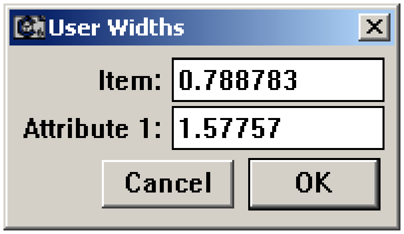
User Widths When you mark this option the Edit button activates. Clicking Edit displays a dialog box in which you can define the column width of the Bill of Materials table.

Draw Table This pop-up menu lets you select whether the Bill of Materials is drawn from the *Top Down* or from the *Bottom Up*.

User Widths

In the User Widths dialog box you specify the width of the BOM columns individually.

When you check the option *User Wodths* in the Options dialog box the Edit button activates. Clicking Edit displays the following dialog box:



In this dialog box all attributes of the actual layout appear. For each attribute the respective column width displays. The displayed values for the column widths are calculated by the word length of each attribute and the margin you indicate. You can change the width for each column individually. All values you specify are based on the units set in the Units dialog box (***Layout>Preferences>Units***).

Attribute Order

The order by which the attributes displays in the Bill of Materials is determined by the order the attributes were defined in the Define Attribute dialog box and cannot be changed in this box.

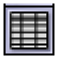
Tip:

Macintosh users: you can drag the attribute names in the Attribute List in the Define Attributes dialog box to change the order.

Creating A Bill of Materials

A Bill of Materials will contain all the Item Numbers and their associated attributes. The generated Bill of Materials is an object in Graphite like any other object geometry. That means it can be moved, copied, deleted and edited. You create a Bill of Materials with the **Bill of Materials** tool in the **BOM** tool palette.

BOM Tool



This tool, in the **BOM** tool palette, generates a Bill of Materials of the current model and places it on the current sheet. Its insertion point is based on the Draw Table setting in the Options dialog box.

Creating a Bill of Materials for the Entire Drawing

1. Choose ***Text>Bill of Materials>Show Palette***.

The **BOM** palette displays.

2. Select the **BOM** tool. The Message Line reads, *BOM Tool: Pick Insertion Point [Ctrl = Selection Only {Windows}, OPTION = Selection Only {Macintosh}]*.

3. Click in the drawing area to indicate a location where you want to display the Bill of Materials.

   The Bill of Materials of the current layout appears at the selected location.

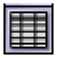
   The Bill of Materials uses the font currently selected in the Text menu.

   It can be moved, copied or deleted like any other object in Graphite.

Creating a Bill of Materials of Selected Objects

1. Select all objects you want to display in a Bill of Materials.
2. Choose ***Text>Bill of Materials>Show Palette***.

   The **BOM** palette displays.
3. Select the **BOM** tool.



4. With the CTRL (Windows) or OPTION (Macintosh) key pressed, click in the drawing area to indicate where you want to place the Bill of Materials.

   A Bill of Materials containing the selected objects will be created and placed at the location you indicated with the last mouse click.

Editing Bill of Materials

1. Select the Bill of Materials with the **Selection** tool.

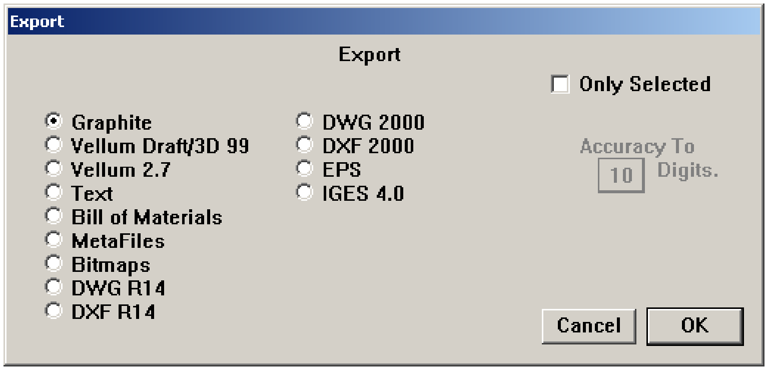
2. Choose ***Arrange>Ungroup***.

The Bill of Material ungroups into text and line objects that can be edited.

Exporting Bill of Materials

Attributes assigned to objects in a drawing can be exported into an ASCII file to be used in other programs like text editors or spread sheets.

To export a Bill of Materials, select the ***File>Export***. The displayed Export dialog box contains the export option *Attributes*.



Exporting Bill of Materials

1. Select ***File>Export***.

   The Export dialog box displays.

2. Mark the *Attributes* option.
3. Select the option, *only selected*, if you want to export attributes only from selected objects and not from the entire drawing.
4. Click OK and provide a file name and location.

The Item Numbers and attributes of the entire drawing or of the selected objects only export into an ASCII file.

GD&T

GD&T stands for Geometric Dimensioning and Tolerancing, a U.S. government standard. This GD&T capability is compliant with ANSI Y14.5 (1985), DIN, ISO and JIS standards.

Since this annotation may be new to you, this section includes an overview of GD&T standards and a brief explanation of the components of a GD&T label, using the true position of holes as an example. The topics covered include:

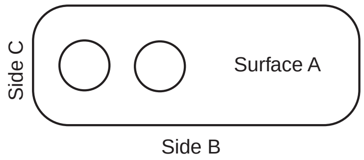
- Background
- Alignment Information
- Bonus Tolerance
- Basic Dimensions
- GD&T Feature Control Frame
- Geometric Characteristics
- The GD&T Label

Background

Before World War II, a single vendor did most military manufacturing. However, during the war, it became a matter of national security to diversify weapons manufacturing so a single plant was not the only source of a vital part. When more than one manufacturer interpreted specifications and blueprints, difficulties arose.

When several companies manufacture the same part, each must drill holes within the same tolerance for the parts to be interchangeable. Even though manufacturers thought they were following the tolerance specifications, the parts were not interchangeable. The problem arose from the order in which measurements were made.

Alignment Information

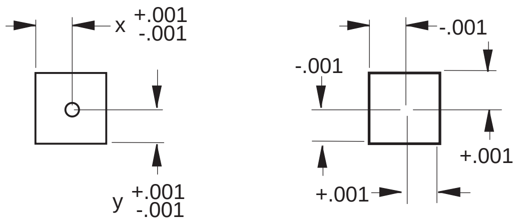


A part was usually laid on a flat surface (A) and aligned first with one straight edge (B), then with another (C).

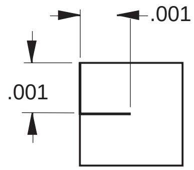
If one manufacturer aligned with edge B, then edge C, but another aligned with edge C, then edge B, the resulting measurements would probably be different. The GD&T standards tell a manufacturer the order of alignment.

Bonus Tolerance

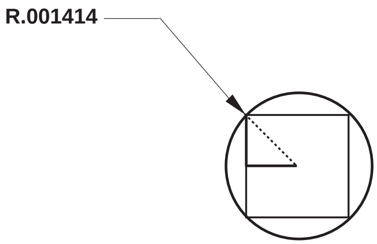
In addition to the alignment information, a manufacturer gets a bonus from GD&T tolerances. In conventional drafting, the tolerance for hole drilling results in a square target area. For example, examine the plate on the left below. The positional tolerance for the hole is ±.001. The target area (magnified and shown on the right) is from .001 left of the perfect center to .001 right of the perfect center, and .001 above the perfect center to .001 below the perfect center.



GD&T provides a circular target area for the hole. First measure from the center in one direction by the distance of the tolerance; then measure the same distance perpendicular to that point.

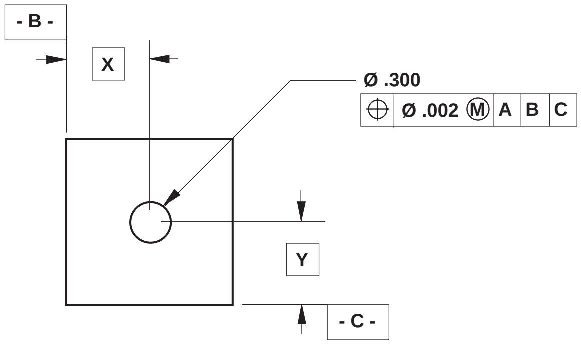


You can then use the distance from the endpoint of the second line to the perfect center as the radius of a circle to create a circular target—the bonus tolerance.



In that way, the target area for the hole's center is the area circumscribing the square of standard dimensions.

The standard information (basic dimensions) about the target area appears in a box with a GD&T feature control frame.



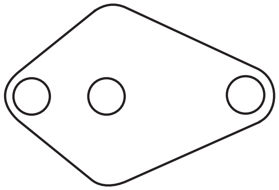
Basic Dimension

Since the tolerance for a dimension is in the feature control frame, GD&T dimensions are also basic dimensions and appear in a box. The value defines the theoretically perfect location and implies that the tolerance is in the feature control frame.

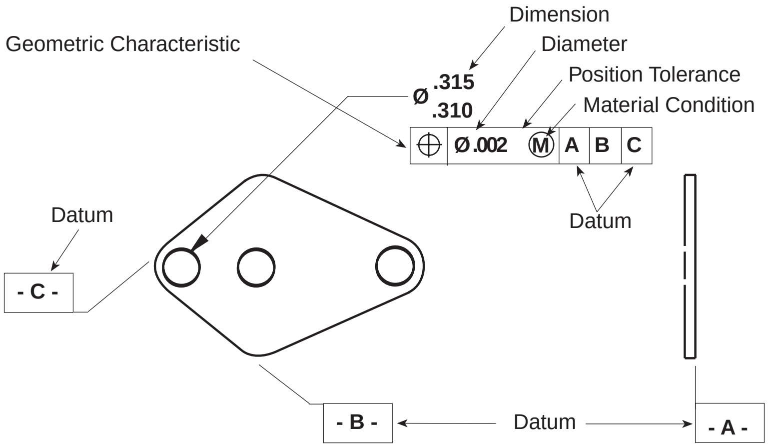
To specify the basic dimension format, in the Dimension menu, go to the Linear and Angular submenus, and choose [xxx] (basic).

GD&T Feature Control Frame

This section discusses the GD&T labeling of the left hole in the plate below.



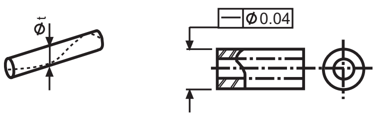
Typically, the GD&T feature control frame appears as shown below:



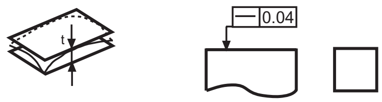
Geometric Characteristics

The first section of the feature control frame contains the geometric characteristic symbol. The feature control frame on the previous page describes True Position—how to locate the circular target area for the hole. The possible characteristics are as follows:

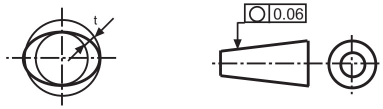
— **Straightness** The axis of an item of revolution or element of a surface is a straight line.



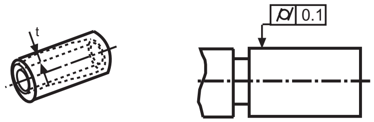
⏥ Flatness All elements of a surface in one plane.



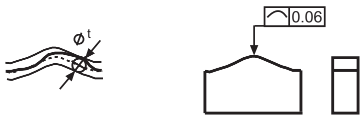
○ Roundness All points of a surface of revolution equidistant from an axis that is intersected by a plane perpendicular to that axis.



⌭ Cylindricity All points of a surface of revolution equidistant from a common axis.

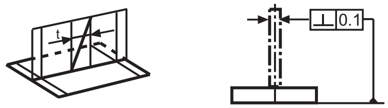


⌒ Profile of Line A 3D element projected to a plane.

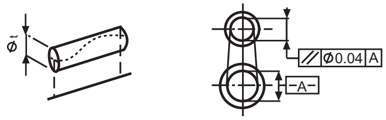


∠ Angularity An axis or planar surface at a specified angle from another axis or datum plane.

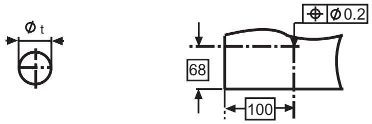
⊥ Perpendicularity An axis or planar surface at a right angle to a datum axis or datum plane.



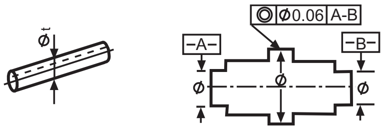
// **Parallelism** — An axis or planar surface equidistant along its length to a datum axis or datum plane.



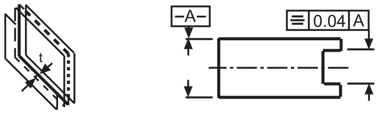
⌖ **True Position** — A zone of tolerance for a center axis or center plane.



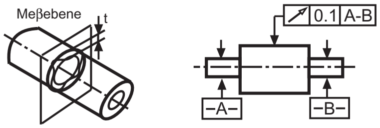
◎ **Concentricity** — The axes of all elements in a cross-section share a common datum axis.



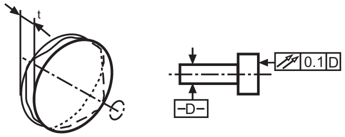
≡ **Symmetry** — The location of a feature relative to a center plane.



↗ **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.



Tip:

If the diameter symbol is appropriate for the geometry, you should display it. It is usually not used with flatness, roundness, cylindricity, profile of line or surface, angularity, circular or total runout.

Diameter

The second section of the feature control frame specifies whether the tolerance applies to a circular zone. If it does, you should specify the diameter symbol (**Ø**) from the pop-up menu.

Position Tolerance

The third section of the feature control frame defines the tolerance allowed under the stated conditions.

Material Condition

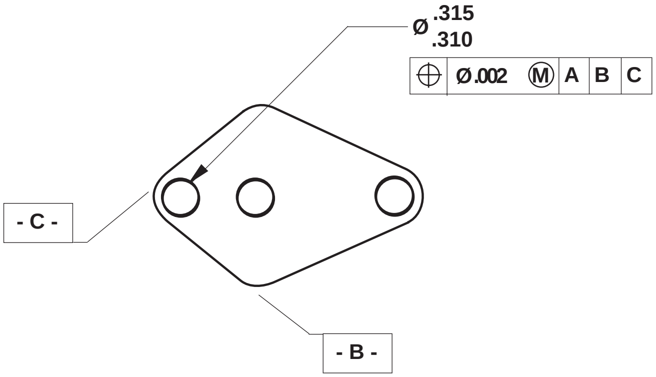
The fourth section of the feature control frame provides the material condition for limiting the tolerance. You can use any of three material conditions: Maximum, Least and Regardless of Feature Size.

Maximum Material Condition

Maximum Material Condition (MMC) indicates that a feature contains the maximum amount of material within the specified tolerance limit. For drilling holes, MMC

Tip:
Maximum Material Condition is the most commonly used symbol for parts.

means that the drill bit is accurate—it drills the smallest hole. For shafts, MMC results in the largest dimension.



In the example above, this material condition symbol means that the tolerance in the feature control frame (.002) applies to the location of the smallest hole (.310 in this case). As the hole size nears its upper limit (.315), the location of the hole increases from .002 to .007 (the tolerance for the location, .002, plus the tolerance for size, .005).

Least Material Condition

Tip:
Least Material Condition is most frequently used to control wall thicknesses of parts, as well as for tool fixtures and inspection gauging.

Least Material Condition (LMC) indicates that a feature contains the minimum amount of material within its specified tolerance limit. For holes, the swept area is the largest and the drill bit drills the largest hole. For shafts, the tolerance is the smallest allowable dimension.

In the example above, this material condition symbol means that the tolerance (.002) applies to the location of the largest hole (.315). As the hole size nears its lower limit (.310), the tolerance for the location of the hole increases from .002 to .007 (the tolerance for the location, .002, plus the tolerance for size, .005).

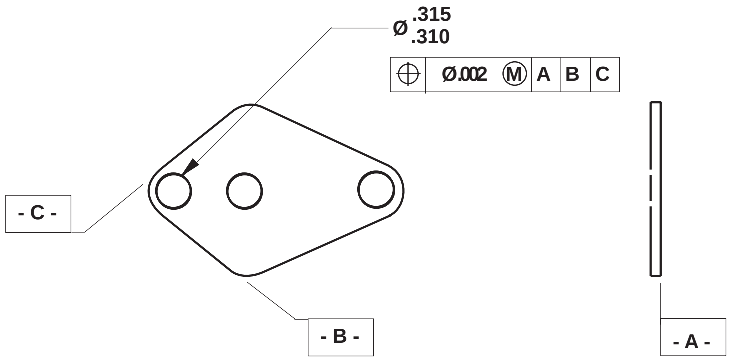
Regardless of Feature Size

The Regardless of Feature Size (RFS) material condition means the tolerance applies regardless of the size of the feature (within the specified tolerance limit). For holes, the location tolerance is the same regardless of whether the hole is the smallest or largest allowable size. For shafts, the roundness tolerance is the same regardless of the shaft diameter.

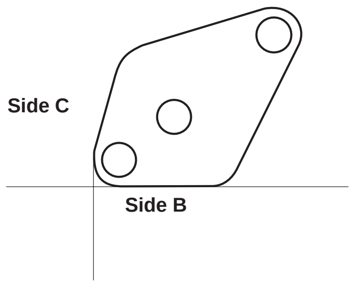
In the example above, this material condition symbol in the feature control frame (.002) applies to the location of the hole whether the hole is at its smallest (.310) or largest (.315). This material condition would control an axis in space, so you probably wouldn't use Regardless of Feature for this hole.

Datum

The last three sections of the frame show the alignment order to position the part. This is easier to see in a part which does not have perpendicular sides. A typical engineering drawing might appear as shown below:



For proper drilling alignment and measurement, this part would lie on a flat surface (Surface A), with Side B pushed against the first straight edge, and then Side C pushed against the other straight edge.



GD&T tells a manufacturer which surface to align first. If you align the part with Side C before Side B, the holes do not line up in the same way as they would if you align with Side B before Side C.

The rule for points of contact per surface is as follows:

Surface	*Point of Contact (minimum)*

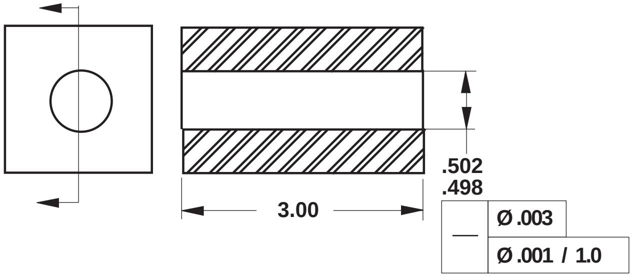
1st	3
2nd	2
3rs	1

Datum Material Condition

When showing a datum in a feature control frame, you also have the option of indicating a material condition. Such a modifier should only be specified for a datum that is an axis or centerplane of a feature, such as a hole or boss. Material Condition modifiers do not apply to planes such as the straight side or flat bottom of a part.

Composite GD&T Symbols

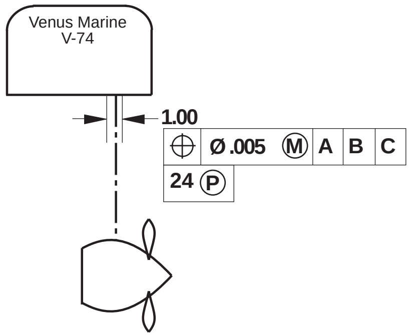
A composite symbol uses more than one line of the feature control frame to specify more than one tolerance for the same feature.



The first line of the feature control frame means that the hole must be straight throughout within a cylindrical tolerance of .003 diameter for the total of its length, 3.00 inches. The second line of the frame means that the hole must be straight for any 1.00 inch portion within a cylindrical tolerance of .001 diameter. Graphite creates the composite frame automatically if the same symbol is in *Line 1* and *Line 2* of the dialog box.

Projected Tolerance Zone

This entry box of the feature control frame specifies an area above the actual part where the tolerance of a feature should still apply if the feature were extended or projected into the area. For example, if an outboard motor has a hole in the housing, the hole has to be straight not only through the housing, but also down into the propeller assembly, so that the drive shaft aligns properly.



In the motor example, the tolerance for the hole should be checked **24** inches from the actual hole.

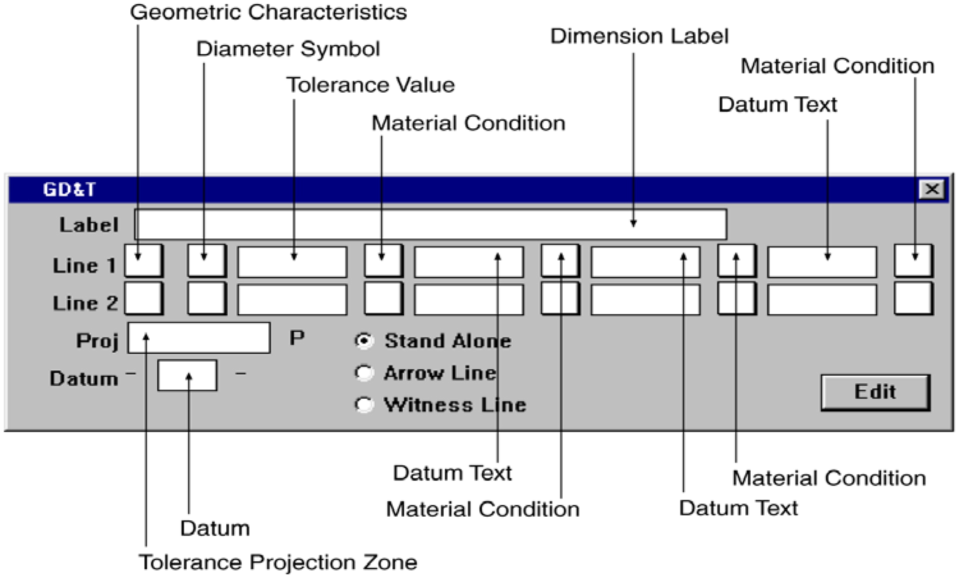
The GD&T Label

GD&T Command

This command in the Dimension menu creates a label—a feature control frame for showing dimensions, alignment, and tolerances. The dialog box looks like a complex GD&T symbol into which you may enter the information you need.

The GD&T label uses Plotter fonts so the font size corresponds to ANSI standards, since the sizes of other fonts are not always consistent with ANSI standards. If you want the size of your dimensions to be the same as the font in a GD&T label, you must select the dimensions and change the font size or choose the plotter font.

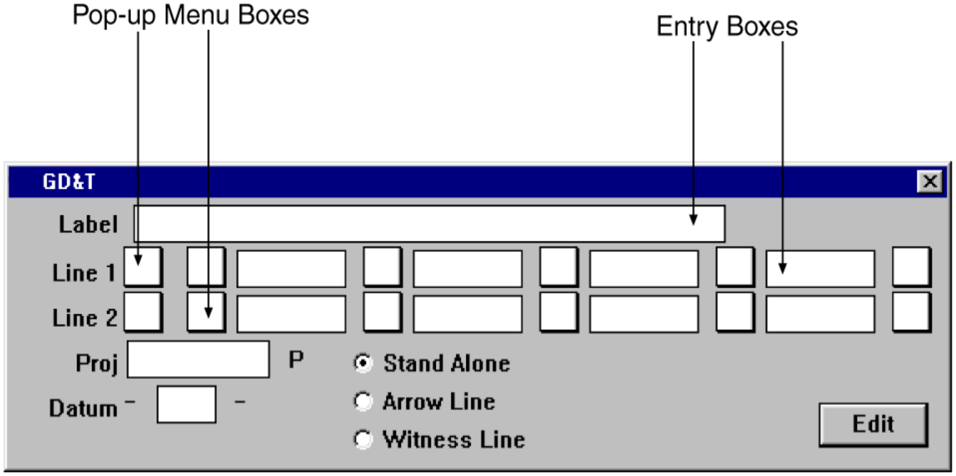
The GD&T dialog box resembles a GD&T label. For 3D geometry, GD&T labels are created parallel to the work plane.



The square boxes are pop-up menus from which you can select symbols. You can use a single entry, a line, or a combination of lines as needed to specify the GD&T dimensions for your part.

Creating a GD&T Label

1. Choose ***Dimension>GD&T***. The GD&T dialog box appears.



2. Click a button (Stand Alone, Arrow Line, or Witness Line) to indicate the way you want the GD&T symbol connected to the geometry you're labeling.
3. Enter the appropriate data to create the GD&T label. You can make two types of entries in this dialog box:

   a. Select symbols from pop-up menus that appear when you press the square buttons.

   b. Enter text in the rectangular fields by clicking to place a text cursor, and then typing the entry.
4. Indicate the location of the label. The Message Line displays the instructions you need.

   • If you want a Stand Alone label, click the location for the upper-left corner of the GD&T label.

   • If you want 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 upper-left corner of the label.

Editing a GD&T Label

Once you have created a **GD&T** label, you may wish to make changes.

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

   The GD&T dialog box appears, displaying the current GD&T information.
3. Make the changes you want in the GD&T dialog box.
4. Click Edit.

   The GD&T label changes.

Using an Editing Shortcut with GD&T

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 the first field to display the Geometric Characteristic menu.
2. Drag to the X symbol.

   All entries in the line disappear.

Appendix A: Operators and Units

This appendix describes mathematical and conditional operators.

Mathematical Operators

Entry boxes that accept numbers also accept mathematical, trigonometric and exponential operators in the form:

funcName (arg1; arg2; . . . ; argN)

Operator	Example	Operator	Example
Addition	3+.450	Arctangent	atan(1;1)
Subtraction	3-.500	Log (Base 10)	log(7.25)
Multiplication	3*.725	Natural Log (Base e)	ln(8.5)
Division	3/5.25	Remove Fractional Part	truncate(6.125)
Square Root	sqrt(8.75)	Absolute Value	abs(-47+16)
Parenthetical	3/(5*2/4)	Smallest Larger Integer	ceiling(5.25)
Scientific Notation	4e-3	Largest Smaller Integer	floor(12.75)
Exponentiation	exp(2;7)	Negative Value	neg(11.12)

Sine of Angle	sin(15)	Round	round(11.12)
Cosine of Angle	cos(30)	Random # in range 0 to 2_{31}	rand()

Math operators are case-sensitive; so be sure to type them in lower case when you use them. You can use these operators to edit objects, make entries in the Status Line and specify values in dialog boxes.

Conditional Expressions

Conditional expressions are particularly useful with parametrics when an expression depends on a condition. For example, a flange might depend on the diameter of a pipe but should never be smaller than some minimum size. You can use two function forms of conditional expressions, *ifelse* and *RNG* (range).

ifelse (condition; expression1; expression2)

<table>
<tr><th>Condition</th><th>The result of a boolean expression:</th></tr>
<tr><td>==</td><td>equals</td></tr>
<tr><td>!=</td><td>not equals</td></tr>
<tr><td>></td><td>greater than</td></tr>
<tr><td>>=</td><td>greater than or equals</td></tr>
<tr><td><</td><td>less than</td></tr>
<tr><td><=</td><td>less than or equals</td></tr>
<tr><td>&&</td><td>logical and</td></tr>
<tr><td>||</td><td>logical or</td></tr>
<tr><td>!</td><td>logical not</td></tr>
<tr><td>expression1</td><td>The value of the ifelse function if the condition is true.</td></tr>
<tr><td>expression2</td><td>The value of the ifelse function if the condition is false.</td></tr>
</table>

The following examples show boolean expressions and an ifelse statement:

<table>
<tr><td>Length > 5</td><td>The length is greater than 5</td></tr>
<tr><td>(L <3) || (W >=2)</td><td>Either the length is less than 3 or the width is greater than or equal to 2.</td></tr>
</table>

3*ifelse (A>B; A; B)	Evaluate three times the maximum of the variables A and B, returning 3*A if A is larger than B and 3*B if B is larger than A.

The range function is a more general form of conditional expression.

RNG (T; V1; N1; V2; N2; V3; N3 ... Vi; Ni; Vi + 1)

In this function, the first argument (T) is tested against every N value in the other arguments and returns the value (V) that lies between the two N values that bracket T. Each of the test N values must be greater than the preceding N.

For example, the expression

RNG (A; -1; 10; 0; 20; 1)	returns	–1 if A is less than 10
		0 if 10 <= A <201
		1 if A >= 20

Units of Measure

You can use different units of measure such as inches ("), feet ('), inches and feet (x'y"), millimeters (mm), centimeters (cm) and meters (m). You can also mix the units in the mathematical expression as long as you label the units properly—for example, **10" + 25.4 cm**.

Nanoseconds

Graphite is well-suited for computer hardware design, such as diagrams of timing devices. For example, you can draw in nanoseconds (a billionth of a second). Designs which include nanoseconds typically consist of wave forms made of horizontal and vertical lines which mimic the functions of a timing device.

The concept for using nanoseconds is to relate time with distance.

1. Specify millimeters as the unit of measure by choosing ***Layout>Preferences>Units***.
2. Choose ***Layout>Preferences>Grid***.
3. Set the grid spacing to 34 and the number of subdivisions to 2.

   A megahertz is approximately 34 nanoseconds and this becomes the base cycle, with two tick marks for drawing half cycles.
4. Turn on the grid.

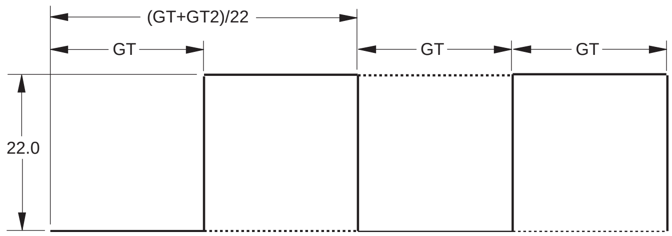
5. Use the **Connected Lines** tool to draw the wave form of the desired specifications.
6. Use the **Expand/Shrink** tool to stretch the wave form to half and full cycles.

Using Parametrics for Wave Forms

Tip:

You can place the *constraint lines* on another layer and hide them if you want.

You can use parametrics to speed up this process, as shown below. The dotted lines are constraint lines.



Decimal Indicators

International numeric keypads have a comma in place of the period because Europeans use commas in decimal expressions.

Graphite can handle both American and European decimal entries in the Status Line (24,5 or 24.5); however, the display is controlled by the Numbers Control Panel.

Keyboards and Command Keys

Since keyboards vary from country to country, the Graphite command keys may not respond as described if you are using a non-American keyboard.

Appendix B: Special Characters

You can use special characters and accents which are available with your computer. Usually these characters are described in an appendix of the user manual. Many symbols and characters are from the keyboard.

Windows:

If necessary, unlock the keypad with the NUM LOCK key, and then hold down the ALT key and enter the numeric code for the character you want.

The character appears in the current font (including Graphite's Plotter fonts). The following list includes common symbols which are available in all fonts:

Accent	Key Combination
°	ALT 0176
ø	ALT 0216
±	ALT 0177

Tech Note:

Numeric values must be entered with an enhanced 101 keyboard using the separate number pad.

Macintosh:

To see the characters associated with a particular font, choose *Key Caps* from the Apple menu and select the font from the Font menu. When you press the ⌘, OPTION, SHIFT, or CONTROL key or a combination of these keys, the keyboard graphic changes to show which characters are available.

In addition to those characters you see in KEY CAPS, you can use the OPTION key to generate international accents in the current font, including Graphite's Plotter fonts.

Accent	Key Combination
é	OPTION+e then letter to be accented
è	OPTION+` then letter to be accented
ñ	OPTION+n then letter to be accented
ü	OPTION+u then letter to be accented

The following list includes common symbols which are available in all fonts.

Accent	Key Combination
°	OPTION+ SHIFT 8
ø	OPTION+ O (letter or number)
±	OPTION+SHIFT =

Appendix C: DWG Notes

The DWG/DXF Translator utilizes the same *.CFG* files as the previous DXF translator. For more information about the *.CFG* files, entity translation to and from AutoCAD and Tips for translation, read the *DXF_r12.doc* file located in the folder where Graphite is installed.

This document is sorted by *TYPE*. So if you have a problem with dimensions, look for the Dimensions section in this list and see if your problem is listed.

Important: Before you attempt to export, always save your file in the most current version of Graphite. If you are exporting a file created in an earlier version of the program, open the file in the most current version, save it, close it, then reopen in the most current version.

Dimensions

Exporting Files

1. It is recommended that you use a Plotter font for your dimensions for the best possible translation into AutoCAD.
2. Dimstyles with the following tolerance settings translate in the manner listed below. It is recommended that you use one of the DIMSTYLES marked good for the best quality translation.

   (The test was done using a Plotter Font mapped to the standard TXT font in AutoCAD.)

DIMSTYLE	RESULTS
xxx	good
xxx +/- tol	good
xxx =upper/-lower	tolerance will be missing
yyy/xxx	dimension text will be missing
yyy-xxx	good
xxx [basic]	good
xxx	good

3. If you want text symbols (i.e. degree or diameter symbols) to export with your dimensions, create the dimensions in Graphite using a Plotter font.

Importing Files

1. Dimensions in AutoCAD that contain multiple colors (extensions lines, arrow-heads and text are different colors in AutoCAD) come into Graphite as black. If your dimensions are different colors, change them in AutoCAD to one of the colors that Graphite supports before importing the file.
2. It is recommended that you import DXF/DWG drawings into Graphite with the option *Dimensions Imported as Graphite smart dimensions* selected. If you do not, some dimensions may be lost.

Geometry and Other Objects

Exporting Files

1. Line types can come in at the wrong display scale in AutoCAD12. Change the *LTSCALE* variable to something more suitable. .1 works well. The lines will then display more appropriately.
2. Edited line patterns do not come across to AutoCAD via DXF or DWG.

Importing Files

1. Line patterns may or may not be visible when coming in from AutoCAD depending on the size of the drawing or geometry. If the patterns are too small, choose ***Pen>Pattern>Edit Pattern*** to change the pattern spacing to a better visual representation on the screen.

AutoCAD is a registered trademark of Autodesk Inc..

Appendix D: Program Settings & Files

For your convenience, this appendix includes the default settings present when you first install the program. It also includes a list of all the folders and files that come with the program.

Default Settings

The default settings are contained in the *Prefs.VC6* (Windows) or *Graphite Prefs* (Macintosh). file Change these setting by opening this file, making the changes and then saving the file. See Chapter 5, Basic Environment Settings for more information.

Once your new preferences are saved, the previous default settings are no longer available. The following are the default settings that came with the program according to the pull down menus:

Edit menu

Selection Mask	All Selected
Selectable Points	On

Layout menu

Preferences Submenu	
Units	
Precision	0.001

Inches		
* All others off		
Selection	Color	Red
Invert	White	
DA	On	
Autosave	Off	15 min.
Layers	Construction, Dimension, Layer1, Constraint	

Arrange menu

Divide	
# of pieces	2
Show Points	On

Pen menu

Style	Outline
Color	Black
Weight	0.010
Pattern	Solid
Arrowheads	Standard
Arrow at Start	Off
Arrow at End	Off

Text menu

Font	Plotter (Windows); Geneva (Macintosh)
Size	12 pt
Style	Normal
Align Left	On
Single Space	On

Dimension menu

Linear	.xxx
	xxx
Linear Tol	.xxx
Angular	xx.xx° (Windows); Whole Degrees (Macintosh)
	xxx
Angular Tol	xx.xx° (Windows); xx.x° (Macintosh)
Text	Horizontal
Font	Plotter

Size	9 pt
Style	Normal
Color	Blue
Weight	.002
Layer	dimension
Arrowhead	Standard
Arrow Size	
Dot Size	.125
Length	.25
Height	.125
Side	.258
Angle:	28° 4' 21"
Witness Lines	Line at Start, On
	Line at End, On

Views menu

Sheets	Sheet 1
Models	Model 1
Views	Top
View the Plane	Off
Draw View Boundaries	On
Show Trackball	Off
Windows	Untitled

3D menu

Planes	Top
Show Triad	Off

Utilities menu

Macros	Empty
Navigator	Unchecked

Program Files and Folders

The following are the folders and files that come with Graphite.

Windows:

Graphite Folder

Files:

Tech Note:

For Windows, you have the option of installing Graphite to any folder. Within that folder are the files and folders listed here.

Adinit.dat, graphite.chm, graphite.dll, graphite.exe, Graphite.feb, graphite.ini, graphite.txt PREFS.VC6, Solobug.exe, SoloBug.hlp, SsceBase.dll

Folders:

Ansi Folder: 12 files

Fonts Folder: 1 files

Xcmd Folder - 11 files

Scripts Folder: 2 files

Spell Folder: 9 files (.clx and .tlx file types)

Layouts Folder: 40 files (Graphite)
4viewb.VC6, 4viewc.VC6, 4viewd.VC6, 4viewsa.VC6, design.VC6, draft.VC6, frntrit.VC6, frnttop.VC6, trimtric.VC6, Viewtop.VC6, 1viewa.VC6, 1viewb.VC6, cformat.VC6, dformat.VC6.

Samples Folder: 2 folders
Samples2D Folder: 14 files

Symbols3d Folder: 2 folders (Fastener and Features)
Fastener Folder: 2 folders (Fastenr and Fastenrm)
Fastenr Folder: 4 folders (Mcscrw3d, Nuts3d0, Pins3d0, Washer3d)
Mcscrw3d Folder: 7 files
Nuts3d0 Folder: 1 file
Pins3d0 Folder: 2 files
Washer3d Folder: 3 files
Fastenrm Folder: 2 folders (Mchscrwm, Nutsm0)
Mchscrwm Folder: 7 files
Nutsm0 Folder: 1 file
Features Folder: 2 folders (Assorted, Holes)
Assorted Folder: 6 files
Holes Folder: 5 files

Symbols Folder: 6 folders (Architct, Assorted, Fastenr, Fastenrm, Holes, Template)
Architct Folder: 2 folders (Plandoor, Planwin)
Plandoor Folder: 7 files
Planwin Folder: 7 files
Assorted Folder: 11 files

Fastenr Folder: 4 folders (Mchscrw, Nuts, Pins, Washers)
Mchscrw Folder: 14 files
Nuts Folder: 2 files

Pins Folder: 3 files
Washers: 6 files
Fastenrm Folder: 2 folders (Mchscrwm, Nutsm)
Mchscrwm Folder: 14 files
Nutsm Folder:2 files
Holes Folder: 14 files
Template Folder: 2 folders (Fluidpwr, P&id)
Fluidpwr Folder - 8 files
P&id Folder: 3 files

Macintosh:

Graphite Folder

Files:

Graphite, Graphite Prefs, Graphite.feb, Graphite.ini, DXF IN 12, DXF OUT 12, VELDXF.CFG, DXFVEL.CFG, DXLATE, IDEAS OUT, IGES2IN, IGES2OUT, Graphite/EPS.

Folders:

Fonts Folder: 4 files
DIN Folder: 3 files
Ansi Folder: 12 files

Layouts Folder: 18 files (Graphite)
View Top, View Trimetric, Views Design 4, Views Draft 4, Views Front & Right, Views Top & Front, aFormat A & 4Views, AFormat B & 4 Views, aFormat C & 4Views, aFormat D & 4Views, aFormat A & View, aFormat B & View, aFormat C & View and aFormat D & View
Scripts Folder: 5 files
ambient.vli, parallel.vli, point.vli, spot.vli and recent.1st

SYMBOLS Folder: 6 folders
ASSORTED Folder: 11 files
ARCHITCT Folder: 2 folders (PLANDOOR, PLANWIN)
PLANDOOR Folder: 7 files
PLANWIN Folder: 7 files
TEMPLATE Folder: 2 folders (FLUIDPWR, P&ID)
FLUIDPWR Folder: 8 files
P&ID Folder: 3 files
FASTENRM Folder: 2 folders (MCHSCRWM, NUTSM)
MCHSCRWM Folder: 14 files

NUTSM Folder: 2 files
FASTENR Folder: 4 folders (MCHSCW, NUTS, PINS, WASHERS)
MCHSCW Folder: 14 files
NUTS Folder: 2 files
PINS Folder: 3 files
WASHERS Folder: 6 files
HOLES Folder: 14 files.

SYMBOL3D Folder: 2 folders (FASTENER, FEATURES)
FASTENER Folder: 2 folders (FASTENR, FASTENRM)
FASTENR Folder: 4 folders (MSSCRW3D, WASHER3D, PINS3DO, NUTS3DO)
MCSCRW3D Folder: 7 files
WASHER3D Folder: 3 files
PINS3DO Folder: 2 files
NUTS3DO Folder: 1 file
FASTENRM folder: 2 folders (MCHSCRWM, NUTSMO)
MCHSCRWM Folder: 7 files
NUTSMO Folder: 1 files
FEATURES Folder: 2 folders (ASSORTED, HOLES)
ASSORTED Folder: 6 files
HOLES Folder: 5 files

Samples Folder: 1 folder
SAMPLES2D Folder: 18 files

Glossary

Accelerators	Keyboard Equivalents that invoke commands rather than using the mouse to choose from menus.
Annotation	Text on drawings, including notes, crosshatching and dimensions.
Apple HSL Display	This is a standard color wheel for the Macintosh with the option to specify Hue, Saturation and Lightness.
Apple RGB Display	This is a standard color display for the Macintosh with the option to specify Red, Green and Blue percentages.
Alignment	1. The justification of text with either or both margins even or text centered within the designated text space. Alignment is set in the Text menu. 2. The spatial relationship of different sets of geometry to each other along a common line.
Alignment Angle	The angle of the Drafting Assistant's automatic construction lines. The specification is set by choosing ***Layout>Preferences>Snap*** .
Arc Tool	Draws a part of a circle.

ASCII	An acronym for American Standard Code for Information Interchange. This is the format for text characters to be imported into Graphite. Most word processors allow you to save text in ASCII format.
Arrow Tool	Used for selecting objects to be operated on with subsequent commands. Also used to move selected geometry.
Associativity	A link between an object and its dimensions or between views. If the object is changed, the dimensions automatically change to match. In associative views, such as a detail view, when geometry is changed in one view, all views of the same geometry are changed accordingly.
Attributes	The data fields associated with a particular object that completely define that object. Data such as Layer Name, Pen Style, Color, X-Y-Z Coordinates and even data for a B.O.M. such as Cost, Item-Name, Quantity, Description, etc.
Automatic Dimensions	This tool automatically places dimensions on lines, circles, ellipses and arcs with one mouse click on the geometry.
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 geometry that defines the limits for operations such as trimming, relimiting, crosshatching, 2D analysis, or fill.
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 objects.
Chain	This entity is created when you flatten a surfaced object using the *Flatten View* command. Once an object is flattened these chain entities can be edited.
Characteristics	See Attributes.
Circle Tool	Draws circles specified by the radius, the diameter, or tangency to other entities.
Circle Center Line	This tool places a center line on circles.
Circumference	The distance around a circle along its edge: =2_r
Circumscribed	Enclosing a circle. In circumscribed polygons, the *midpoint* of each side of the polygon touches an imaginary circle (i.e. the polygon exactly surrounds the 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 Lines	Lines, displayed as dotted or gray lines, that you use for exact alignment. The Drafting Assistant creates dynamic, temporary construction lines, or 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.
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 for 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 line 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 circle, arc, ellipse, 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.
Custom Colors	(Windows only) This button in the color display allows you to define 16 additional colors to show in the partial color display.
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 your own 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.
Delta	A change, usually in position.
Detail View Tool	The tool that creates a separate view of the geometry you indicate, at the scale you specify.
Dialog Box	A specification box that appears in response to certain commands. A dialog box allows you to provide information that qualifies the execution of those commands.

Diameter	The distance across an arc or circle, passing through the center.
Dimension	A graphic object that displays the distance between two points. A measurement of an object. Also a pen style.
Divide	To segment a line or curve into equal parts.
Double-click	To press and release the mouse button twice, quickly.
Drafting Assistant	A unique Graphite feature which displays feedback notations and construction lines to aid with snap, alignment and constraint operations. The patented 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	An acronym for Data Exchange Format-a format of AutoCAD files.
Drawing	A drafted, 2-dimensional drawing of a model. Usually a drawing including an orthogonal view.
DWG	AutoCAD's native file 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.

EPS — Encapsulated PostScript format for printing to a PostScript printer and for importing into compatible applications.

File — An individual document.

Fillet Tool — The tool that allows you to create an arc of a specified radius tangent to entities.

Flatten — To convert a view of a 3D model into a 2D plane. Flattening disassociates a view from a model, allowing independent editing.

Font — The assortment of type used in text.

Forms — This command enables you to create standardized title blocks.

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 would not be visible in a solid part. 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.

IGES — An acronym for Initial Graphics Exchange Specification. The U.S. Department of Commerce, National Bureau of Standards issues IGES, which is intended to be the industry standard among CAD/

	CAM systems for data exchange in a neutral file format. Ashlar Graphite supports IGES Version 4.0.
Import	To load a non-Graphite 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.
Isometric	A top, front and right view in which the X-axis is at -30° from the screen horizontal, the Y-axis is 30° from the screen horizontal and the Z-axis is vertical.
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, parallel lines, or smart walls.
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.
Math Operator	The mathematical, trigonometric, or exponential expression used to indicate values. All value entry boxes in Graphite accept the math operators listed in Appendix A.
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.
Model	The geometry that is incorporated into a view. The model is the geometry that a view sees.
Non-planar	Surfaces or points that do not lie in a two dimensional plane.
NURB	Non-Uniform Rational B-splines—the type of splines Graphite 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 kinks won't develop as the spline is altered. Second, localized control of a complex curve is provided, so 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.
Object Rotation	Use this command to rotate geometry around any axis.
Origin	The 0,0,0 location on the drawing area. When a new document is opened, 0,0,0 is located in the middle of the screen. The coordinate symbol displays at the origin when the grid is turned on. You can change the origin at any time.
Orthogonal	A 2D view of a 3D model which does not show perspective, usually the top, front and right view of the model.
Palette	A group of tools. The general tool palette is always displayed to the left of the drawing area. You can open the dimension tool palette from the **Dimension** menu and drag it around the screen.
Parametrics	Integrated parametrics is available through the *Resolve* and *Symbol* commands. Once you create

geometry, you can add dimensions as variables or real values. When you choose *Resolve*, 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 current document.

Perpendicular At a 90° angle. The Drafting Assistant displays a notation when the current construction is at a 90° angle to an object.

PICT The Macintosh graphics file format which uses object-oriented bitmaps or resolution-independent graphics.

Pick To select a location or object by clicking it.

Planar Surfaces or points that lie in one two dimensional plane.

Plot To draw a part on paper using a plotter.

Plotter A computerized drawing device for hardcopy output.

Point A location for constructing geometry.

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 and rotate it around a reference point.

Polygon Tool The tool that draws regular polygons, that is, objects 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.
Reload	This command is used to load commands and functions into Graphite.
Remove Dots	This command removes lines of zero length.
Resolve	The command that draws geometry according to the specific measurements you indicate. See Parametrics.
Right-hand rule	A memory aid for the relative directions of the positive axes. With your right palm upturned, the thumb (X) points right, the index finger (Y) points straight ahead and the middle finger (Z) points up. If you move your hand to indicate the X and Y axes, you can easily see the direction of the Z axis.
Right-hand rule	A memory aid in which the thumb of the right hand points in the direction of the positive axis of rotation. The fingers curve in the direction of rotation.
Segment Tool	The tool used to divide a line or curve at specific boundaries.
Selection Tool	The tool that selects objects. Usually, subsequent actions affect only the selection. Also used to move selected geometry.

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.
Smart Symbol	A symbol created with the **Smart Wall** tool which is automatically incorporated into a smart wall.
Smart Wall	A tool to create special double walls automatically, trimming overlapping intersections.
Snap	The *Preferences* command that allows you to set the specifications for the Drafting Assistant.
Snap Command	This allows you to activate or deactivate different snap modes of 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.
Stretch Tool	Used to stretch your geometry along one axis and at a specified angle.
Symbol Libraries	Groups of symbols of the same type. These were either included with Graphite or created by you.
Symbol	The command that retrieves a predefined drawing. This command may use Graphite's integrated 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.
Textblocks	This command allows you to create a standard text block for multiple use.

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.
Text Lines Tool	Used to place text within a geometrically created horizontal line.
Toggle	To switch between two conditions, for example, *Hide Grid/Show Grid*.
Transformation	The tools to move, rotate, expand or shrink, or mirror an object or group of entities.
Trim	To shorten or remove a portion of a line.
Trimetric	A top, front and right view in which the Z-axis is vertical, the X-axis is at -30° from the screen horizontal and the Y-axis is at 15° from the screen horizontal.
Undo	The command that reverses the last editing or creation action. You can Undo (and Redo) the last eight actions.
Unfold	To open a model 90° along an axis.
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.
Wireframe	A 3-dimensional representation showing boundary lines, edges and intersections, but not surfaces.
World plane	The work plane used at the beginning of the construction of a model. Also know as the world coordinate system.

Work plane	The x,y plane used for 2D objects which has an origin of 0,0,0 for all data input. Sometimes referred to as the user or work coordinate system.
Zoom	The tool or command that allows you to magnify or reduce an image.

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