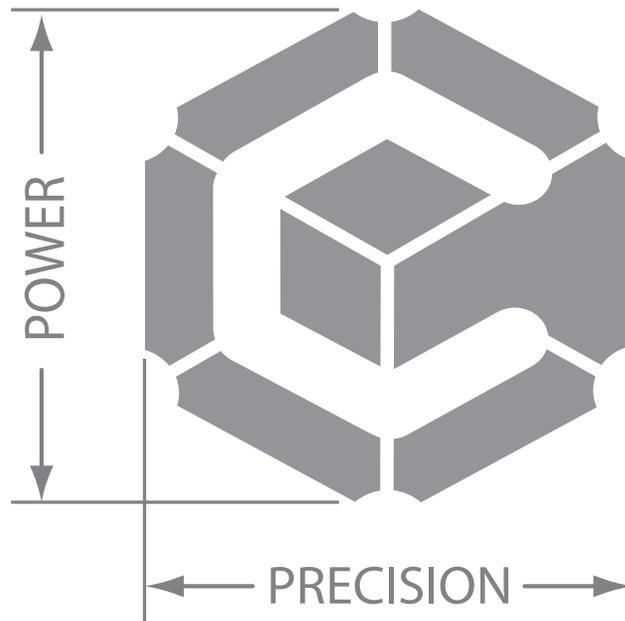




ASHLAR•VELLUM®

Software that works the way you think



GRAPHITE

Getting Started - Graphite v 7

GRAPHITE™

March 2003

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Installing & Launching Your Program

Each license has its own unique Serial Number and Registration Code. Installation instructions are included in the Readme text file.

Serial Number and Registration Code

This Designer Elements program requires a *Serial Number* and an *Registration Code*. If your registration code was not included with this package, call Ashlar's Help Desk at 1-800-966-2349 to receive the code. Be prepared to provide your Serial Number when you call. When you launch **this program** for the first time, enter your *Serial Number* and *Registration Code* in the data fields of the dialog box. If you do not wish to register immediately, or experience difficulties receiving your registration code you can press the Later button. Graphite will run without a registration code for a limited time.

Technical Support

All customers have 30 days of free technical support starting from the date you receive your Registration Code.

Starting Your Program

1. Start your computer.
2. Locate the program icon and double-click it to launch the program.

Ending Your Program

1. Save and close all open files.
2. Choose **File>Exit** (Windows) or **File>Quit** (Macintosh).

Tutorial Introduction

The exercises in the next chapters will familiarize you with the tools, features and commands of Graphite. You will discover how quickly you can learn and use Graphite for your design needs. The exercises demonstrate Graphite's most powerful features so you can see how the patented Drafting Assistant simplifies Computer-Aided Design and Drafting (CADD). You will soon integrate what you already know about design and drafting with the power of a computer, making your work faster, more accurate and more creative!

If you are new to computers, CADD may feel awkward and uncomfortable at first. You may mutter to yourself, "I wouldn't even have to think to put this on paper!" Drawing on a computer requires a different thought process but Graphite lets you leverage your drafting experience. Once you see what you can do and how easy it is to make changes such as adapting a standard part to new specs, we think you'll agree that learning this new process is worth the effort. Revisions can be done quickly and easily without spending hours redoing whole sets of drawings.

So, be prepared to look at things differently. Go through these exercises, and then begin to adapt Graphite's power to your personal work style.

Chapter Layout

The exercises are organized in chapters according to the tools and features used. The chapters are ordered with the most basic exercises in the beginning and proceed to more complex exercises in the later chapters.

Margin notes provide alternative ways of doing a procedure or may refer to another section or chapter for related information.

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 displayed as ***Pen>Color>Define***.

Margin Notes

Graphite includes margin notes that provide you with information that may help you use the program. There are three types of margin notes: Tip, Tech Note and Referral. These notes are 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 provides additional technical information that may help when using a tool.

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 (**Dimension Palette** tool names (**Single Line** tool); Keyboard-entered text; Definition terms (as shown in these style conventions)

Italic

Terms used for the first time in a chapter; (*control points*); Drafting Assistant notations (*midpoint*);

	tool and dialog box options (<i>Angle</i> data field); book references (<i>User Guide</i>); Message Line directions (<i>Single Line: Pick the beginning point.</i>); margin note headings (<i>Tip</i>); menu commands (<i>Zoom Previous</i>); filenames (<i>prefs.ini</i>); stand alone extensions (<i>.dwg</i>); directory names; drawing names
<i>Bold and Italic</i>	Command series (<i>Layout>Preferences>Snap</i>)
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)

Step Conventions

The numbered tasks in the exercises describe the activity you are to perform, and the bulleted steps beneath the numbers tell you how to accomplish the task. If you already know how to accomplish the task, you should do it without following the bulleted directions, then proceed to the next numbered task.

Exploring

Some users like to go off on their own to explore while going through the exercises of a tutorial. This is an excellent way to learn more about Graphite. If you are adventuresome, open a new document for your explorations and then switch back to the tutorial document when you want to continue with the exercises.

Occasionally, the tutorial may verify a position or entry that seems obvious to you. If the condition is vital to the next step and you might have inadvertently deviated from the tutorial path, verification (for example, “the x,y location is 0,0”) has been added to ensure that you get the correct result from the exercise.

There’s More than One Way

Graphite often provides more than one way to perform a task. This tutorial describes only one method at a time and may show you a different way to do the same task later. When you start to develop a preference, feel free to substitute your own method for whatever is suggested here, provided you're certain that your method produces the same outcome as the tutorial.

Graphics

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

Basic Terminology

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 may look like the current tool.



Arrow Pointer



Selection Arrow



Center-Point Circle

To move the pointer, move the mouse on the mouse pad.

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

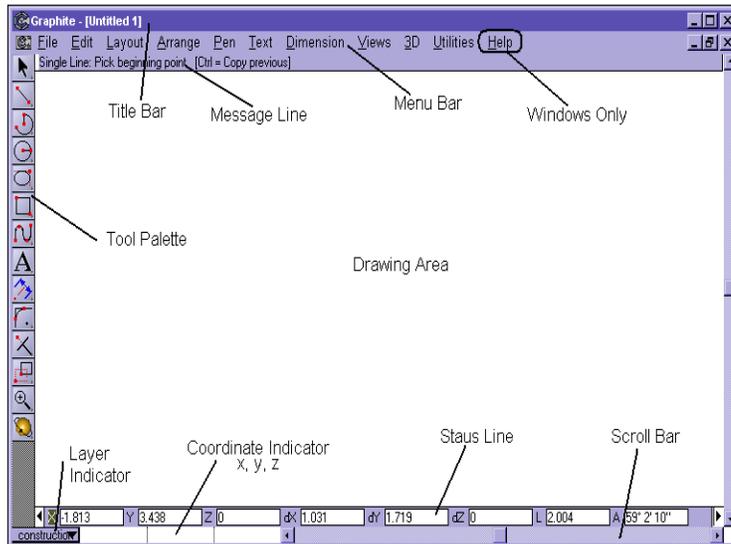
Click the mouse button twice, quickly in succession.

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

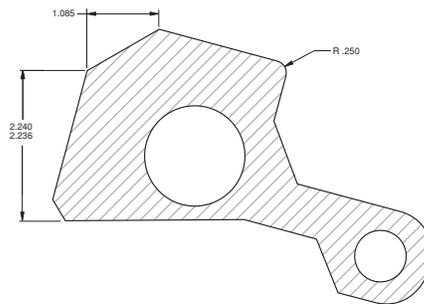
Allow you to move around a drawing so you can see different sections of it through the Graphite window. The scroll buttons allow you to move one line at a time.

Work Layer Indicator

Displays the name of the current layer and provides a menu for changing the work layer.

Graphite Basics

In these exercises you will construct this part using computerized fillets, chamfers, crosshatching and dimensions. You will perform the following tasks:



- Open Graphite
- Choose from menus
- Create geometry
- Change geometry
- Use the patented Drafting Assistant
- Create construction lines
- Save a document
- Stroke to create construction lines
- Stroke to zoom

- Create chamfers and fillets
- Construct circles
- Trim
- Change the characteristics of lines
- Dimension
- Crosshatch
- Stretch
- Rotate
- Print

Exercise 1: Starting Graphite

In this exercise you will start Graphite.

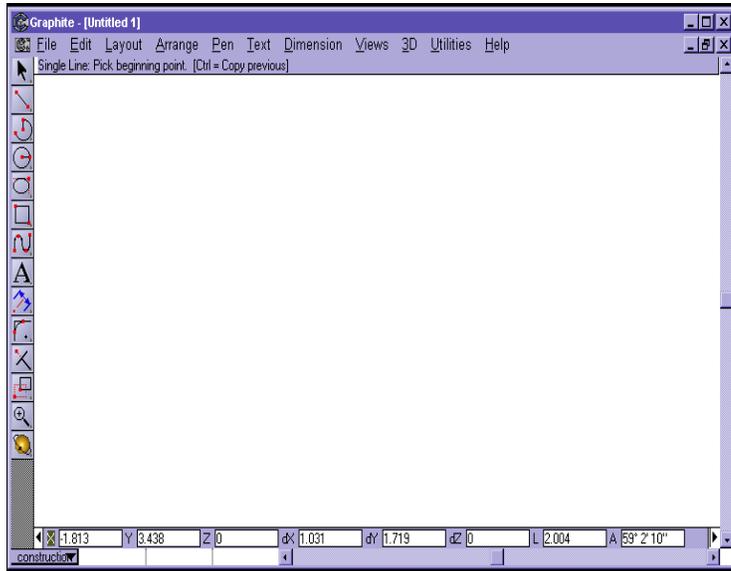
1. Start Graphite.

If you have already started Graphite, please select ***File>Exit*** (Windows) or ***Quit*** (Macintosh).

- Double-click the *Graphite* icon in the Graphite folder.



Graphite opens, displaying an untitled document with an empty drawing area.

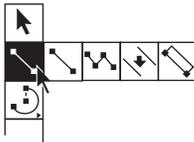


2. Familiarize yourself with the menus and tool palette.

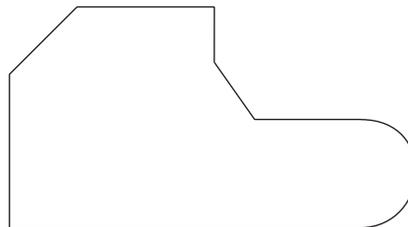
Exercise 2: Drawing a Part

In this exercise you will construct the basic shape of the part.

1. Select the **Connected Lines** tool.
 - Move the pointer to the **Single Line** tool icon on the tool palette and press on the **Single Line** tool.



The subpalette appears.



Tech Note:

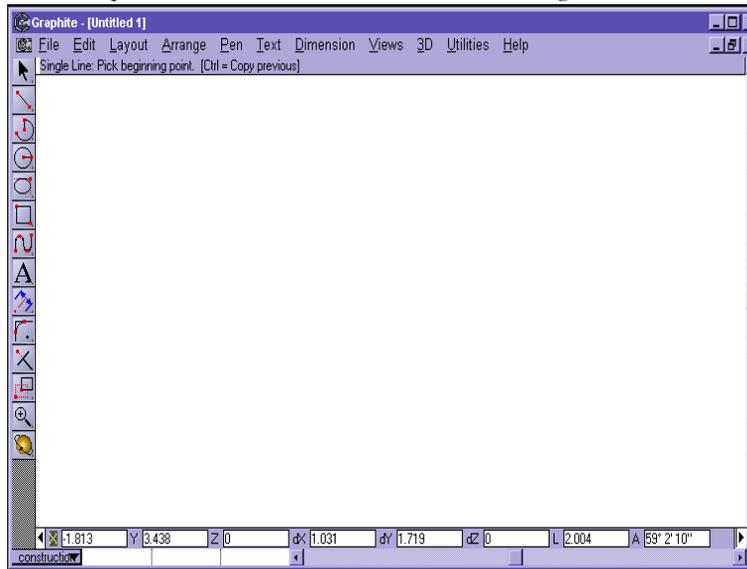
Notice that the icon on the left of the subpalette is part of the tool palette display and the other icons on the subpalette represent the choices you can make. The icon on the palette changes to reflect your last tool choice, but the icons in the subpalettes are always in the same order; regardless of which tool is on the palette.

- Drag across the subpalette until the **Connected Lines** tool highlights.



The Message Line tells you that the **Connected Lines** tool is selected and reads, *Connected Lines: Pick beginning point*. It remains the active tool until you choose another tool.

2. Draw the first vertical line 2.25 inches long.
 - Position the pointer in the lower left of the drawing area.



Tech Note:

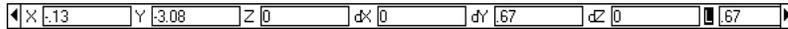
The Drafting Assistant's construction lines aid in precise placement. The predefined construction lines are vertical, horizontal and at a 45° angle to existing geometry points. When the pointer is near such a location the construction line appears and the word *on* appears next to the pointer.

- Click to set the first point.
- Move the pointer up until the Drafting Assistant's vertical construction line appears.



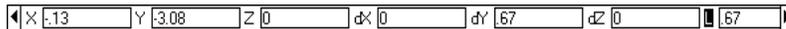
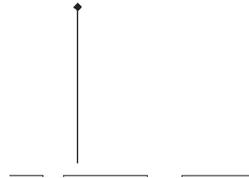
- Move the pointer a couple of inches from your last point (you don't need to be exact), and when *on* appears on the construction line, click to set the new point on the construction line.

At the bottom of the Graphite window, the Status Line displays these data fields. Notice that the Length (L) box is highlighted.



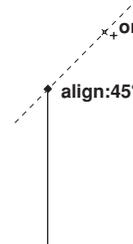
- The numbers on your screen may not match the numbers in this graphic.
- Use either the keypad or the numbers at the top of the main keyboard to type **2.25** and press ENTER (Windows) or RETURN (Macintosh).

The length (L) goes into the Status Line data field and the line is redrawn to exactly 2.25 inches, beginning at the first position you indicated.



3. Construct the second line 1.25 inches long at a 45° angle from the end of the first line. (Angles are measured from horizontal, not from the previous line.)

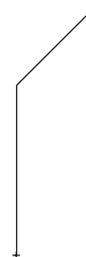
- Move the cursor up and to the right at a 45° angle using the Drafting Assistant construction line as shown above. Click to set a new point an inch or so from your last point.



- Type **1.25** and press ENTER (Windows) or RETURN (Macintosh).

The line segment is constructed at a 45° angle.

4. Construct a horizontal line 2 inches long.
 - Move the pointer to the right to display *on* for the horizontal construction line.
 - Click a few inches from your last point.



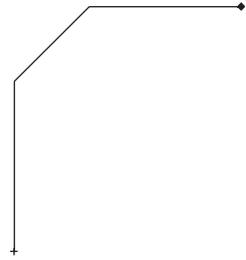
Tech Note:

The Drafting Assistant doesn't require you to be exact, only close enough to display the feedback. Once you click, the Drafting Assistant locks onto the exact location.

Whatever you type will automatically go into the Length Status Line field.

- Type **2** and press ENTER (Windows) or RETURN (Macintosh).

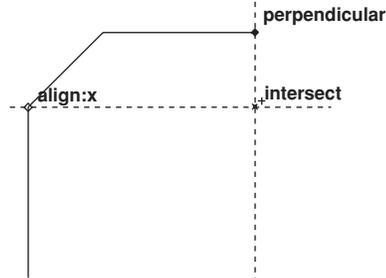
A 2-inch horizontal line is drawn.



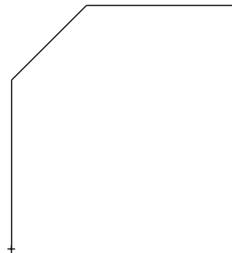
5. Construct a vertical line that ends on the horizontal construction line through the upper *endpoint* of your first line.
 - Move the pointer straight down until *intersect* appears, then click to set a point at this intersection.

Tip:

Whenever you move the pointer around the screen, the Drafting Assistant examines the existing geometry and displays relevant information about your current position.



A vertical line is drawn to the Drafting Assistant's intersection marker.



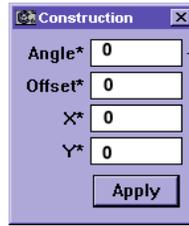
Tech Note:

The current tool remains in effect while you choose commands from the menu.

6. Use a command from a menu to create a construction line beginning at the endpoint of the last line and at an angle of -55° and then use it to draw a 1-inch line.
 - Choose **Layout>Construction**.

The dialog box appears.

When the pointer is in the dialog box, the angle data field highlights. You can enter an *Angle* for the construction line or an *Offset* from the point specified by the coordinates. In this step you are entering an angle for the construction line.

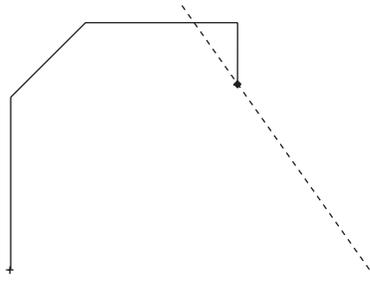


Change this number to -55

Tech Note:

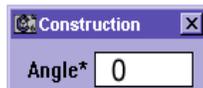
The X,Y coordinates displayed represent the location of the last position you selected, which is the location you need for this step. You can change the coordinates whenever you want.

- Enter **-55** (be sure to type the minus sign).
- Click Apply.



A permanent construction line appears as a dotted line on the screen.

- Click the Close box of the Construction dialog box in the upper-right corner (Windows) or upper left corner (Macintosh).

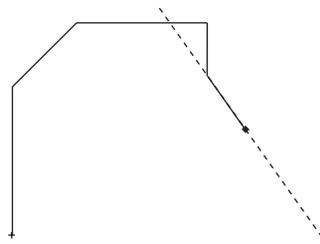


Close box (Windows)

The dialog box closes.

- Click *on* the construction line, about an inch from your last point.
- Type **1** and press ENTER (Windows) or RETURN (Macintosh).

The line is constructed at a -55° angle.



7. Construct a 1.5-inch horizontal line.

Tech Note:

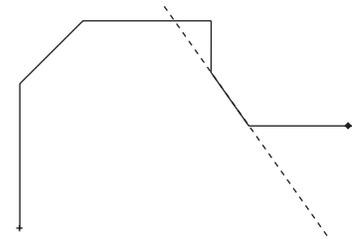
The Status Line data fields accept mathematical operators so that you can add, subtract, multiply, divide and group values within parentheses. See Appendix A in the *User Guide* section for a complete list of mathematical operators.

Tech Note:

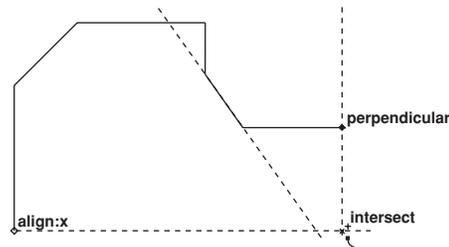
The Message Line tells you how you can modify the behavior of Graphite's drawing tools. For example, if you hold down the CTRL (Windows) or OPTION (Macintosh) key while using the **Connected Lines** tool, you draw an arc instead of a straight line. The pointer changes into the shape of an arc to signify this optional drawing mode.

- Move the pointer to the right to display the horizontal construction line through the *endpoint* of the last line.
- Click.
- Type $(2+1)/2$ in the Length Status Line data field and press ENTER (Windows) or RETURN (Macintosh).

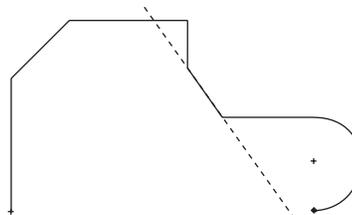
The 1.5 inch line appears.



8. Construct an arc from the right *endpoint* of the last line to a point horizontally aligned with the first point of this exercise.
- If part of the construction is now off-screen, choose **Arrange>Zoom All** to display the entire part on the screen.
 - Hold down the CTRL (Windows) or the OPTION (Macintosh) key, then move the pointer straight down until intersect appears. The pointer changes to an arc.



- Click the mouse button.
- Release the CTRL (Windows) or OPTION (Macintosh) key.



The arc appears.

9. Close the figure.

- Move the pointer horizontally to the left to display the *endpoint* notation for the first line you created.
- Double-click to set this point and signal that the connected-line figure is complete.

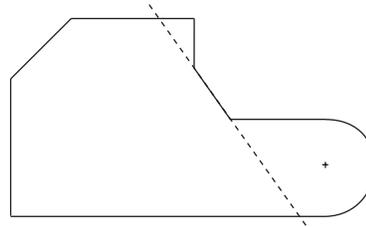
The outline of the part is now complete.

10. Save the drawing.

- Choose **File>Save**.

A dialog box appears so that you can name the drawing.

- Type **part1** and press ENTER (Windows) or RETURN (Macintosh).



Tech Note:

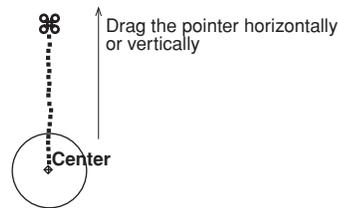
It is important to save often. Power failures and mistakes can destroy hours of work if you don't save frequently. It is important to save before you try any new multistep operation so that you can revert to the original if you don't like the results.

Accomplishments

- Selecting a tool from the tool palette
- Constructing a part with the **Connected Lines** tool
- Using the Drafting Assistant
- Choosing from a menu
- Creating a construction line
- Saving a document.

Exercise 3: Stroke Commands

Before proceeding with the drawing, examine the use of Graphite's stroke commands for creating construction lines, zooming, and displaying points. For stroke commands, hold down the CTRL and SHIFT keys (Windows) or the ⌘ key (Macintosh) and drag the ⌘ pointer across the screen or click as needed.



Drag

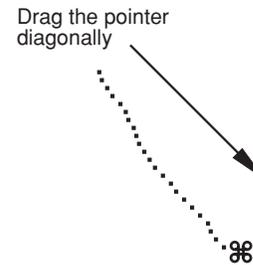
Vertically

Horizontally

Construction Lines

A vertical construction line through the first point of the stroke.

A horizontal construction line through the first point of the stroke.



Drag Diagonally

Upper left to lower right

Lower right to upper left

Upper right to lower left

Lower left to upper right

Zoom

Zoom-in enlargement centered over the stroked area.

Reverses zoom-in stroke to the previous magnification.

Zoom-out reduction, the current screen reduces to the size of the area defined by the stroke.

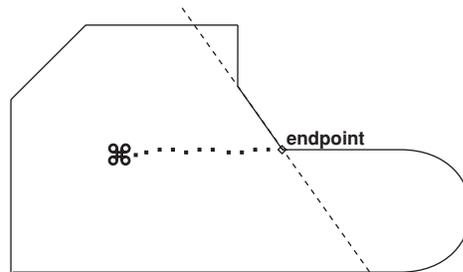
Reverses zoom-out stroke to previous magnification.

Click*On object***Point Display**

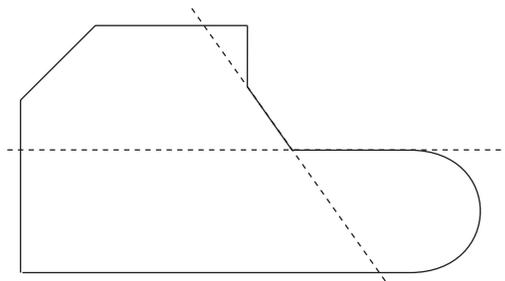
The display of the object's points is turned on or off.

You can use stroke commands while you are using any tool. The possible effects you can obtain with a stroke command depend on the direction you move the pointer. Even though you can create construction lines and zoom in other ways, the stroke commands come in handy because you can use them while you are in the process of using tools from the palette.

1. Create a horizontal construction line.
 - Hold down the CTRL+SHIFT keys (Windows) or the ⌘ key (Macintosh).
 - Position the pointer at the lower end of the -55° line so that *endpoint* or *intersect* appears.
 - Drag left horizontally. The pointer trails a dotted line as you drag. Don't worry if the line isn't straight.



A horizontal construction line appears. Now, you have two construction lines which remain until you delete them.



2. Add a vertical construction line.
 - Hold down the CTRL+SHIFT keys (Windows) or the ⌘ key (Macintosh).

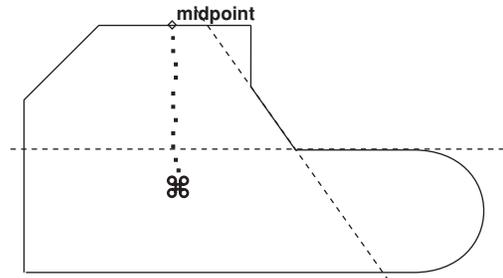
Tech Note:

These construction lines are placed on the construction layer and can be removed by choosing **Layout>Delete Constructions**. Since the work layer is transparent, you can see everything on the Construction layer.

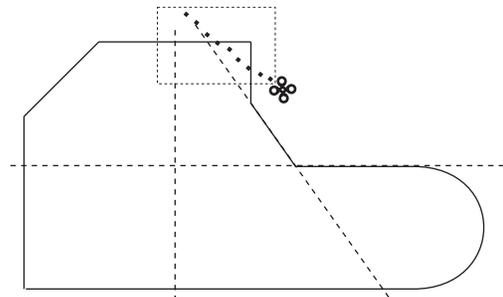
Tech Note:

Since all tools work at any magnification, you can repeat this action for further magnification. But for now, proceed to the next exercise.

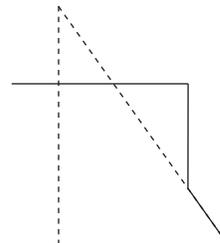
- Position the pointer at the *midpoint* of the uppermost horizontal line.



- Drag down vertically. A vertical construction line appears. You'll use these two construction lines later in the tutorial.
3. Use the stroke command to zoom in on a corner of the drawing so that you can fillet it in the next exercise.
- Hold down the CTRL+SHIFT keys (Windows) or the ⌘ key (Macintosh).
 - Drag as shown, from above and to the left of the upper-right corner of the drawing across the corner.



The corner magnifies.

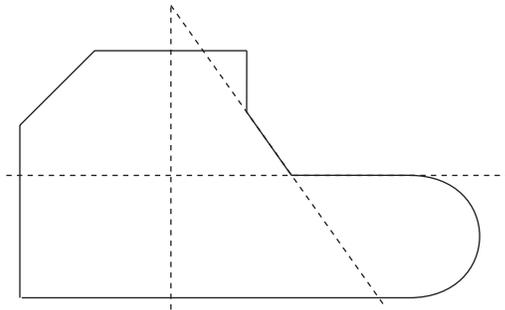


Accomplishments

- Creating a construction line by using a stroke command
- Zooming in with a stroke command

Exercise 4: Fillet and Chamfer

In this exercise you will fillet one corner of the part and add a chamfer to another.

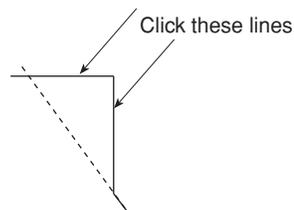


1. Fillet the corner you zoomed in on in the last exercise.

- Click the **2-Entity Fillet** tool, in the tenth subpalette of the main tool palette. The Message Line reads, *2-Entity Fillet: Pick first entity [Shift = Corner, Ctrl = No trim (Windows) or Option = No trim (Macintosh)].*



- Click the horizontal and vertical lines that intersect to form the corner.



The fillet is drawn with a radius of .25 inches, and the corner is automatically

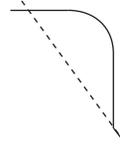
Tip:

If the geometry is large, you can hold down the SHIFT key and click inside the corner for single-click filleting. You will use this technique when you create the chamfer in Step 3.

Tech Note:

Fillets are automatically constructed with a .25 radius, but you can change the radius in the Status Line, just as you changed the line length earlier. If you don't want to trim a fillet you can hold down the CTRL (Windows) or the OPTION (Macintosh) key while you click the lines that you do not want trimmed away.

trimmed away.



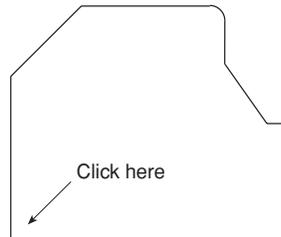
2. Return the drawing to its original size.
 - Hold down the CTRL+SHIFT keys or the ⌘ key (Macintosh).
 - Drag the pointer from the lower right to the upper left of the drawing area.
 - Release the CTRL and SHIFT keys or the ⌘ key (Macintosh).
3. Add a chamfer to the lower-left corner of the part.
 - Press the **Fillet** tool to display the subpalette.
 - Select the **2-Entity Chamfer** tool. The Message Line reads, *2-Entity Chamfer: Pick first entity to chamfer [Shift = Corner, Ctrl = No trim (Windows) or Option = No trim (Macintosh)].*

Tip:

It doesn't matter where you start this stroke or how far you drag if you only want to restore the original magnification.



- Hold down the SHIFT key.
- Click inside the lower-left corner of the part.



The chamfer is drawn .25 inch from the original corner.

4. Save your work.
 - Choose **File>Save**.

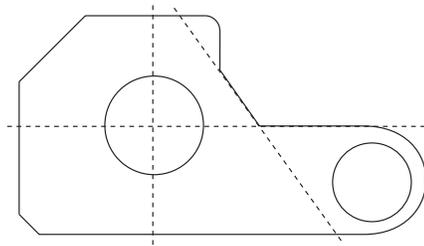
Since the drawing is already named, the part is saved without displaying the dialog box. The length of time it takes to save depends on the complexity of the part you are drawing.

Accomplishments

- Constructing a fillet
- Zooming out by using a stroke command
- Constructing a simple chamfer

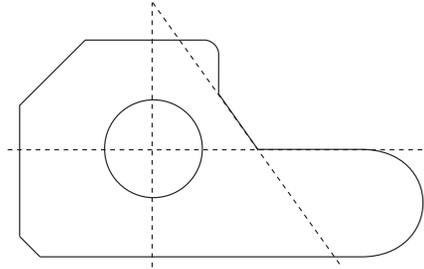
Exercise 5: Adding the Holes

In this exercise you will add two holes to the part, one of which will be offset from a specified location.



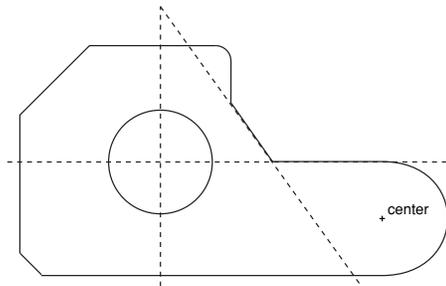
1. Construct a hole, 1.5 inches in diameter, centered at the intersection of the vertical and horizontal construction lines you created in *Exercise 2*.
 - Click the **Center-Point Circle** tool. The Message Line reads, *Center-Point Circle: Pick center [Ctrl = Copy previous (Windows) or Option = Copy previous (Macintosh)]*.
 - Click the *intersect* point of the horizontal and vertical construction lines to indicate the center of the circle.
 - Move the pointer an inch or so in any direction and click.
A circle is drawn with the center point at your specified location.
 - Enter **1.5** in the D (Diameter) data field of the Status Line.

- Press ENTER (Windows) or RETURN (Macintosh).



The circle is redrawn with a diameter of 1.5 inches.

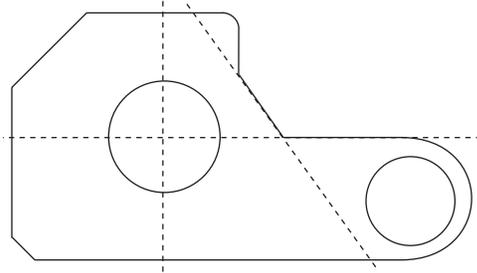
2. Construct a 1-inch hole that is offset .06 inches in the negative x-direction from the center of the arc of the part.
 - Move the pointer over the arc until the Drafting Assistant displays the + indicating the *center* of the arc.
 - Move the pointer to the + to display the *center* feedback.



The **Center-Point Circle** tool is still the current tool.

- Click to position the center of the circle.
- Enter **1** in the D (Diameter) data field but do not press ENTER (Windows) or RETURN (Macintosh).
- Click in the X data field of the Status Line and place the text cursor at the end of the existing entry.

- Type **-.06** after the value in the X data field (don't forget the minus, you are creating an equation from the existing X value) and press ENTER (Windows) or RETURN (Macintosh).



The circle is drawn with a diameter of 1 inch and its center is offset by $-.06$ inches.

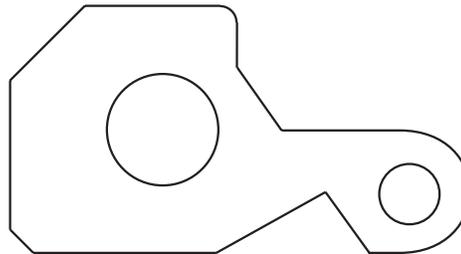
3. Save your work.

Accomplishments

- Constructing circles
- Using point offsets

Exercise 6: Making Changes

In this exercise you will modify the part you've been constructing. First, you'll add a cut-out section to the bottom edge, and then you'll change the diameter of one of the holes.



1. Add the cutout, beginning at the midpoint of the bottom edge and extending to the -55° construction line.
 - Select the **Connected Lines** tool.

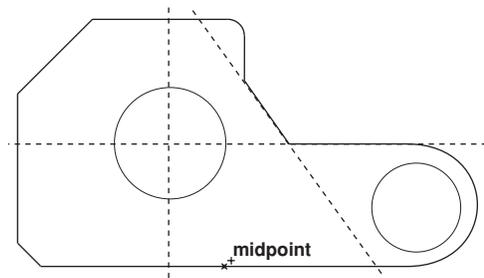
Tech Note:

The cursor must be at the end of the existing entry. Otherwise the circle will not be located properly.

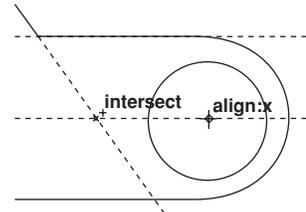
Tip:

You see here that you don't actually have to draw the circle, just click the center and enter a value for the diameter.

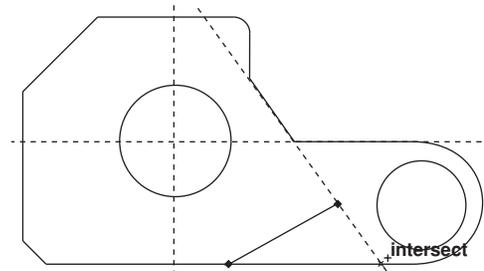
- Click at the *midpoint* of the bottom line of the part.
- Move the pointer to the arc to display the *center* point of the arc.



- Move the pointer along the -55° line until the *intersect* point of the *center* point and the -55° line appears.
- Click.

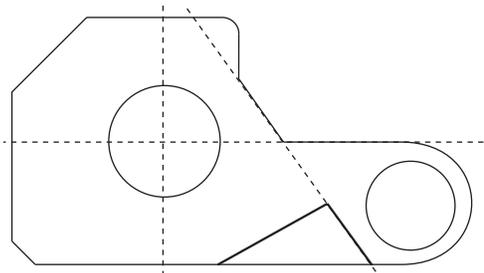


- Move the pointer to the *intersect* point of the -55° construction line and the bottom line of the drawing.



- Double-click.

The lines are drawn and remain selected. They must be selected to perform the next step.



2. Remove the unnecessary portion of the horizontal line.

Tech Note:

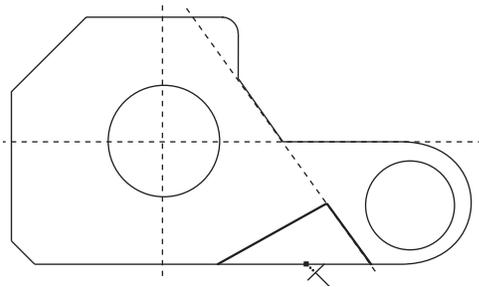
An object is automatically selected when its construction is finished. Both of these new lines are selected because you are using the **Connected Lines** tool, and both lines are the result of a single process. Once they are deselected, they are considered individual objects.

- Select the **Simple Trim** tool.



The Message Line reads, *Simple Trim: Pick section to trim [Shift = Select boundary, Ctrl = Relimit (Windows) or Option = Relimit (Macintosh)].*

- Position the Trim pointer dot over the line segment you want to discard.



- Click.

The line segment trims.

3. Delete the construction lines since you no longer need them.

- Choose **Layout>Delete Constructions**.

The visible horizontal, vertical and -55° construction lines created in this tutorial are removed. The Drafting Assistant's dynamic, on-the-fly construction lines continue to display when you move the pointer near geometry.

4. Change the diameter of the hole on the right to .75 inch.

- Click the **Selection** tool. The Message Line reads, *Select: Select [Shift = Extend, Ctrl = Copy (Windows) or Option = Copy (Macintosh)].*



- Click the smaller circle.
- Choose **Edit>Edit Objects**.
- Click the word diameter in the Edit Objects dialog box.

Tech Note:

In order to use the **Simple Trim** tool, you must first select the lines that define the boundaries of the geometry you want to trim away, not the line you want to trim. In this case the lines are already selected.

Tech Note:

If you are making only one change, you can press ENTER (Windows) or RETURN (Macintosh); otherwise, you should not press ENTER (Windows) or RETURN (Macintosh). First, make all the changes you want, then click the Apply button.

Tip:

The arrow beside the *Weight* command in the menu indicates a submenu.

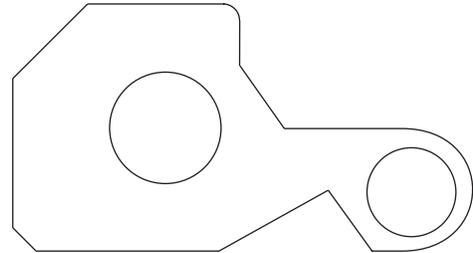
Tech Note:

The selected lines don't change to green until you deselect them.

- Type **.75** and press ENTER (Windows) or RETURN (Macintosh).

The original circle is redrawn with a .75-inch diameter.

- Click the Close box to close the Edit Objects dialog box in the upper right corner (Windows) or in the upper-left corner (Macintosh) of the dialog box.

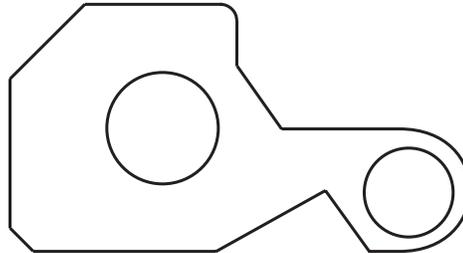


5. Change the Weight and Color of the lines.

- Double-click the **Selection** tool.

Everything in the drawing is selected.

- Choose **Pen>Weight**.
- Choose *0.016* for the pen weight.



All lines are changed to a medium pen weight.

- Choose **Pen>Color>Green**.
- Click in the drawing area where there is no geometry, deselecting the lines so that you can see the new color.

The lines in the drawing are displayed in green.

6. Save your work.

Accomplishments

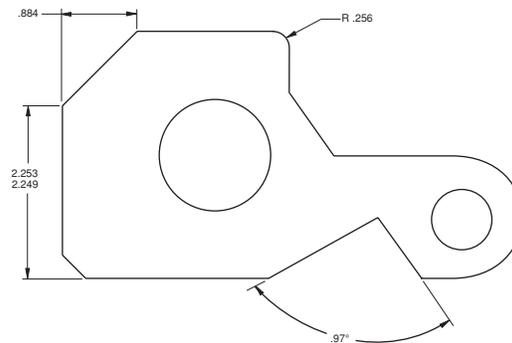
- Trimming
- Deleting construction geometry

- Editing geometry with the Edit Objects command
- Selecting all objects
- Changing the pen weight
- Changing the pen color

Exercise 7: Dimensions

In this exercise you will add some dimensions to the drawing.

1. Dimension the horizontal length of the 45° line.



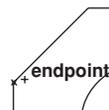
- Choose **Dimension>Show Palette**.

The **Dimension** palette appears.

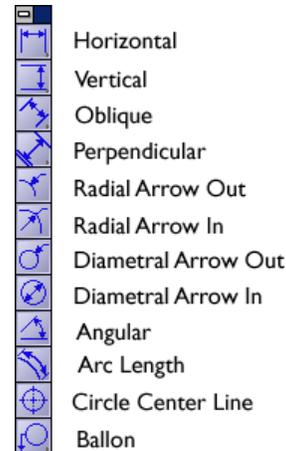
- Select the **Horizontal** Dimension tool. The Message Line reads, *Horizontal: Pick first dimension point.*



- Select the left end of the line to be dimensioned by clicking the lower endpoint of the 45° line.



The *hot spot* on the pointer moves to the right.



Tech Note:

Dimensions automatically go on the Dimension layer rather than on the work layer. This is a unique feature of dimensions and the Dimension layer. If you have removed the Dimension layer, you should create a new one to complete this exercise.

If you hide the Dimension layer, then the dimensions will be placed on the current work layer.

Tip:

Since the layers are transparent, you see the geometry and dimensions on both layers at the same time.

Tech Note:

Take a moment to examine the pointer that appears. The dot on one leg of the pointer marks the *hot spot* for the action you perform.



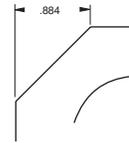
The location of the dot indicates which side of the object to select. The dot changes positions as you use the dimensioning tool. This type of pointer is a smart pointer because it gives you important information in a multi-step process. Many of Graphite's tools use the smart pointer.

Tech Note:

The order of the selection determines the placement of the dimension text. If you select in the order shown on the *smart pointer*, the text appears above horizontally dimensioned geometry or to the right of vertically dimensioned geometry. If you select in the opposite order, the text appears below or to the left of the selected geometry.

- Click the upper *endpoint* of the 45° line.

The dimension appears, but the text is in a location you may want to change.

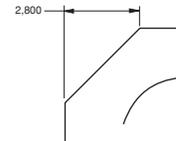


- Move the pointer to the dimension text to display the 4-way Move symbol.



- Drag the text to the left horizontally until it is about **.5** inch from the left leg of the dimension.

The dimension text is repositioned.



- Dimension the vertical line on the left.

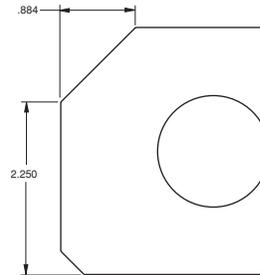
- Select the **Vertical** Dimension tool. The Message Line reads, *Vertical: Pick first dimension point.*



- Click the lowest point of the chamfered corner.



- Click the upper *endpoint* of the vertical line.



The dimension appears. You can add tolerances at this point.

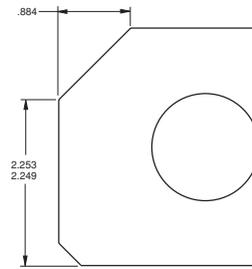
- Choose **Dimension>Linear** and select *yyy/xxx (limits)*.

Text Upper Lower

A new group of Status Line data fields appears at the bottom of the screen.

- Click the Upper Status Line data field.
- Type **.003** and press ENTER (Windows) or RETURN (Macintosh).

The dimension now reflects a .003 upper tolerance and -.001 lower tolerance.

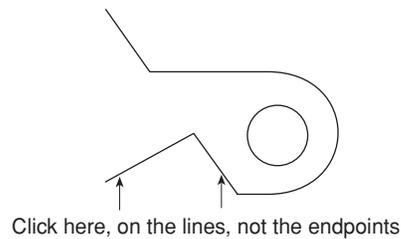


3. Dimension the angle of the cutout.

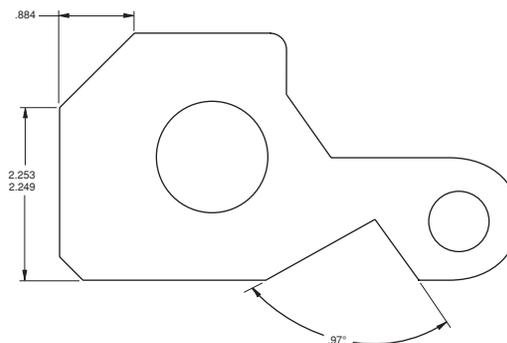
- Select the **Angular** Dimension tool. The Message Line reads, *Angular: Select Line.*



- Click the arms of the cutout, near the lower ends of the lines.



The angle is measured from the endpoint nearest the location you click.

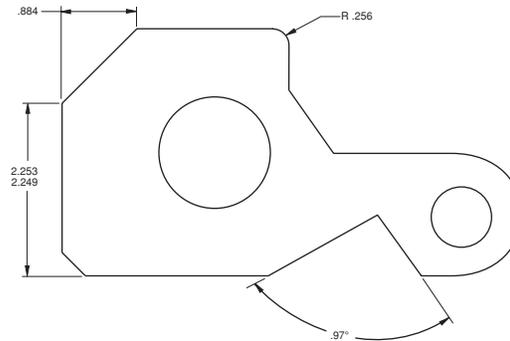


4. Add a radial dimension to the fillet.

- Select the **Radial Arrow Out** Dimension tool. The Message Line reads, *Radial Arrow Out: Select arc/circle.*



- Choose **Dimension>Linear** and select *xxx* from the submenu to return to dimensions without tolerance.
- Click outside (but near) the filleted corner. Make sure the *on* notation appears on the arc.



Tip:

Radial dimensions are created with a single click of the mouse. The text appears on the side of the arc that you click

The radial dimension appears on the side of the arc where you clicked.

- Choose **Dimension>Hide Palette.**

The **Dimension** tool palette disappears.

5. Remove the angular dimension.
 - Click the **Selection** tool.
 - Click the text portion of the angular dimension.
 - Press the DELETE key. The angular dimension deletes.
6. Save this part with a different name (*part1a*), so that you can use it later for an advanced exercise.
 - Choose **File>Save As.**

The Save File dialog box appears with *part1* listed in the Filename box. Windows also displays the current filename, *part1.vc6*, in the data field.

- Click after the 1 of part1 in the data field.
- Type **a** and press ENTER (Windows) or RETURN (Macintosh). The part saves with the new name, *part1a*. Windows also has a *.vc6* extension.

Accomplishments

- Adding dimensions
- Moving dimension text
- Adding tolerances
- Saving a version of a part with a different name

Exercise 8: Crosshatching

With the part fully drawn and dimensioned, you can add crosshatching.

1. Select the boundaries that define the area to be crosshatched.

- Double-click the **Selection** tool.

All geometry is selected.

2. Crosshatch the part, indicating Steel.

Choose **Pen>Crosshatch**.

- Click on the radio button for ISO crosshatching.
- Select *Steel* from the list of patterns. The display box shows the crosshatching exactly as it will appear in the part.
- Click Apply.

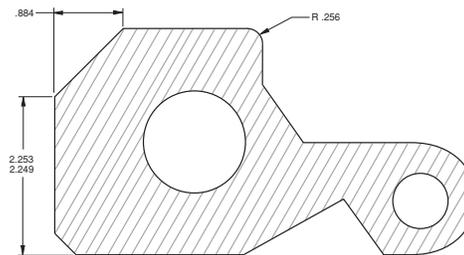
The part is crosshatched.

- Close the dialog box.
- Click anywhere in the drawing area to deselect the part.

3. Save the part once again as *part1*.

- Choose **File>Save As**. The file-name listed is *part1a*. (Windows also has the *.vc6* extension)
- Click after the *a* in the data field.
- Press the BACKSPACE (Windows) or DELETE (Macintosh) key once to remove the *a* and press ENTER (Windows) or RETURN (Macintosh).

Since you can't save two documents with the same name in the same directory, you are asked if you want to overwrite the existing file named *part1.vc6*.



Tech Note:

Graphite automatically excludes dimensions from the crosshatching process.

Tip:

You can choose **Pen>Hatch** to apply the current cross-hatch pattern to the selected area.

- Click OK. The original version of *part1* is replaced with this crosshatched version.

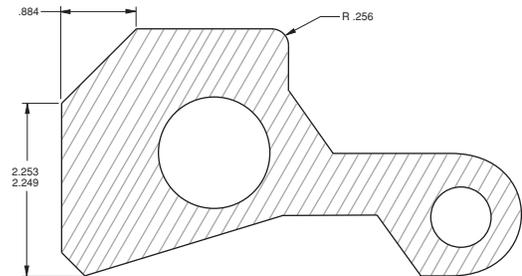
Accomplishments

- Crosshatching a part
- Saving and replacing an existing version
- Renaming a file

Exercise 9: Stretching

In this exercise you will change the basic outline of the part and see how the crosshatching automatically redraws to accommodate the change.

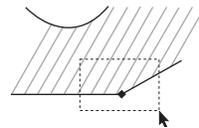
1. Drag a selection fence around the point where the left point of the cutout joins the horizontal line.



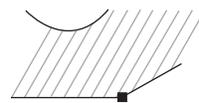
- Click the **Selection** tool.
- If any part of the drawing is selected, click anywhere in the drawing area to deselect the part.
- Position the pointer above and to the left of the point.
- Drag to a location below and to the right of the point.

Tech Note:

To drag, press and hold down the mouse button, drag the mouse to the desired location and release the button.



The point is selected, as shown below. If you don't see the square selection point, choose **Edit>Selectable Points** and select again.

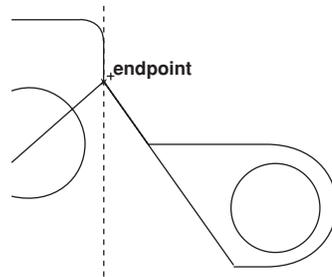


2. Stretch the part so that the left side of the cutout is horizontal.

- Move the pointer to the selected point until the pointer displays the 4-way Move symbol.



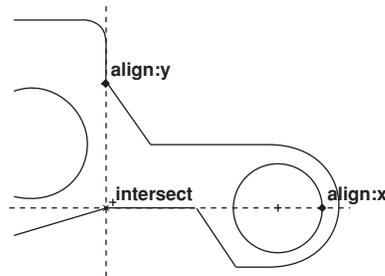
- Drag the point upward to the top of the -55° line, so that the vertical construction line appears. **Do not release the mouse button.**



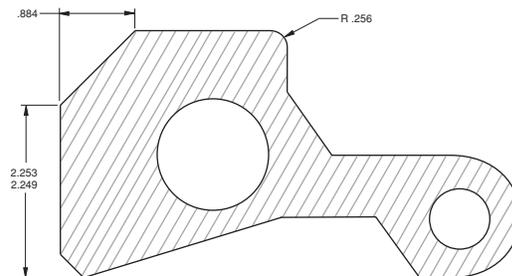
Tech Note:

Moving the pointer over this *endpoint* activates it so that the Drafting Assistant's construction line passes through it.

- With the mouse button still pressed, drag downward to the *intersect* point of the Drafting Assistant's construction lines as shown below.



The part is redrawn and the crosshatching is updated, once you release the mouse button.



3. Save your work.

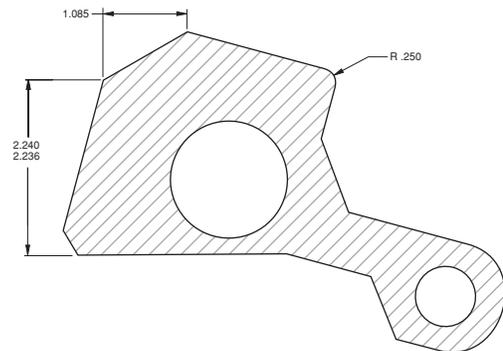
Accomplishments

- Selecting a point
- Using a selection fence
- Activating a point
- Stretching a part

Exercise 10: Rotating

Now that the part is complete, you can rotate it so that the left line of the newly created cutout is horizontal.

1. Specify the rotation.
 - If the part is not selected, double-click the **Selection** tool.

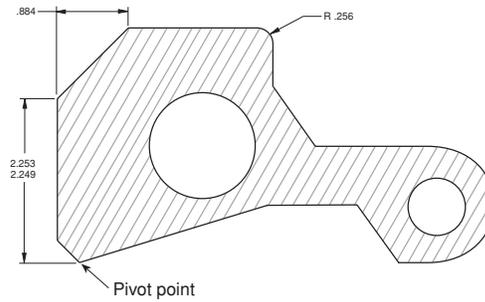


- In the **Transformation** subpalette, select the **Rotate** tool.

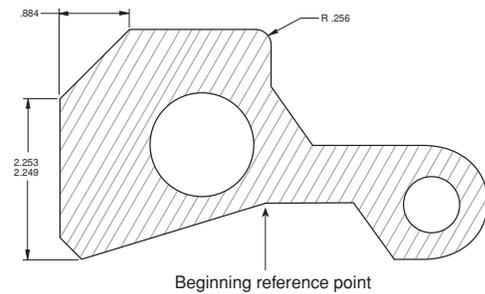


The Message Line reads, *Rotate: Pick center of rotation [Shift = Select, Ctrl = Copy (Windows) or Option = Copy (Macintosh)].*

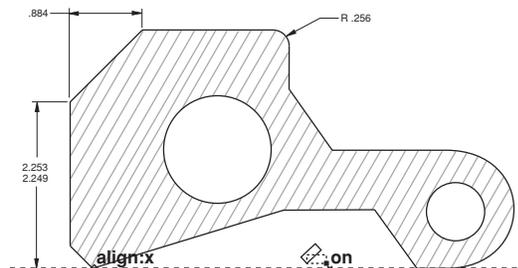
- Specify the *pivot point* (the center of rotation) by clicking the lower *endpoint* of the left side of the cutout.



- Specify the *beginning reference point* (the point that is to move) by clicking at the control point you moved in the last exercise.



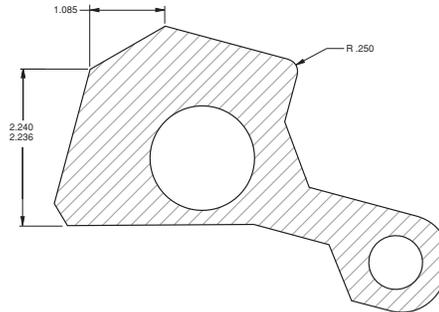
- Specify the *ending reference point* by clicking on the horizontal construction line across the bottom of the part.



Tech Note:

Notice that the dimensions changed to reflect the new orientation; this is an example of associativity. Graphite maintains an extensive database describing all geometry so that dimensions and locations can be updated quickly.

The part rotates.



Tech Note:

Zooming does not change the actual measurements of the part.

2. Reduce the visual display of the part.

- Select the **Zoom Out** tool to display the entire part on the screen.



The Message Line reads, *Zoom Out: Pick area to zoom [Ctrl = Zoom In (Windows) or Option = Zoom In (Macintosh)]*. The Scale data field in the Status Line displays the current scale.

- Click a location near the larger hole.

The magnification of the part decreases and the location you clicked is in the center of the screen.

3. Save your work.

Accomplishments

- Rotating a part
- Observing associative dimensions
- Zooming out to a specific location

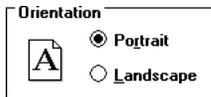
Exercise 11: Printing the Drawing

For your drawing to be useful, you need to transfer it to paper. If your drawing extends past the boundaries of the paper you are using, you can scale it before printing.

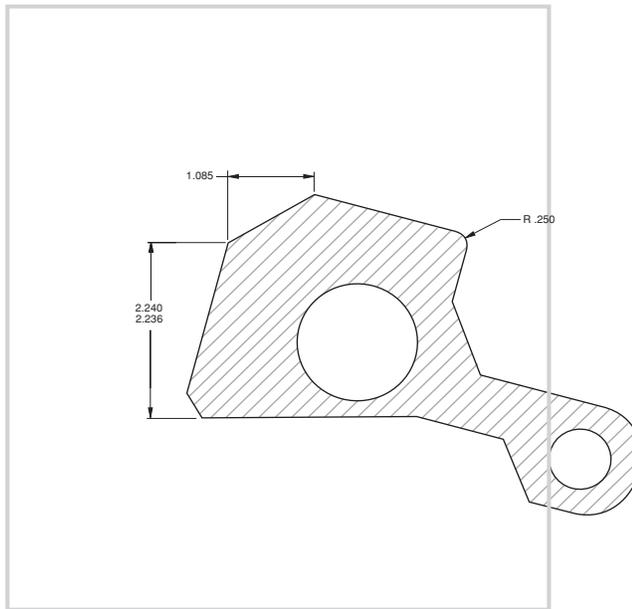
1. Specify the page orientation and paper size.

- Choose **File>Print Setup** (Windows) or **Page Setup** (Macintosh).

- If necessary, specify *Portrait* orientation.



- If you have a plotter, specify the appropriate paper size.
 - Click OK.
2. Specify the exact area to be printed.
- Select **Layout>Drawing Size**.
 - Click *Always Display Page Bounds* to display the gray box representing the maximum plotable area of the page.



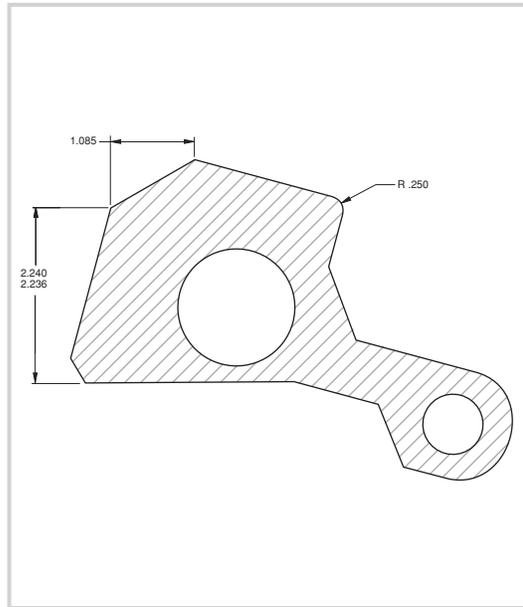
Tech Note:

The page outline appears in the drawing area at all times if the *Always Display Page Bounds* option is on. When this option is off, the page outline appears only when the Drawing Size dialog box is open or you can select **File>Preview Layout**.

Tech Note:

Your drawing may not look like the right graphic because you may have specified a different printer, plotter, or paper size.

- Click Fit.



The drawing border is scaled and redrawn so that the part fits on the paper size with the orientation you specified in *Print Setup* (Windows) or *Page Setup* (Macintosh).

- Click OK to close the dialog box.
3. Print the drawing.
 - Choose **File>Print**.

The drawing is sent to the printer or plotter.

4. Close the document.
 - Choose **File>Close**.

You are asked if you want to save.

- Click OK.

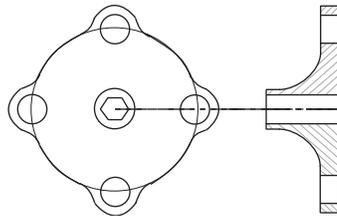
The document closes. (When you open another Graphite document, a new Graphite window appears.)

Accomplishments

- Displaying the paper size in the drawing area
- Scaling the drawing
- Printing the drawing

Additional Features

In this section you use some of the more advanced Graphite features. While many features are demonstrated, you should look through the *User Guide* chapters for other features which might be useful to you. In some cases, the exercises do not create a useful part but only show you how to use a feature, such as how to trim two lines to make a corner or how to construct a line tangent to a circle. In other cases you create real parts, such as the front and side view of a flange.



The features covered in this section include:

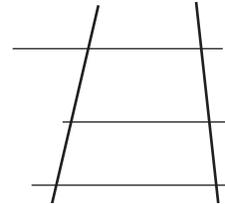
- Corner trim
- Trim
- Relimit
- Text
- Tangent lines
- Perpendicular lines
- Origin (0,0)
- Polar duplicate
- Polygon
- Parallel lines
- Mirror transformation
- Bolt circle
- Parametrics
- Splines

- Circle fillets
- Smart Walls

Exercise 1: Trimming and Relimiting

In this exercise you will investigate some advanced construction techniques. Graphite allows you to create corners and trim or relimit lines. When you trim a line it shortens to its intersection with the selected boundary. Relimiting allows you to extend or shorten a line to the limiting boundary.

1. Open a new Graphite document.
 - Choose **File>New**.
2. Explore the **Simple Trim** tool.
 - Create an approximation of the lines here.
 - Hold down the SHIFT key and select the two lines that appear bold in the graphic.
 - Select the **Simple Trim** tool. The Message Line reads, *Simple Trim: Pick section to trim [Shift = Select boundary, Ctrl = Relimit (Windows) or Option = Relimit (Macintosh)].*



Tip:

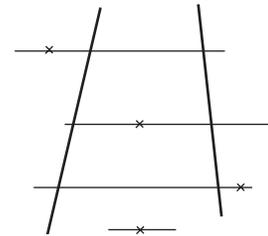
The exact figure is not important. You are only learning about the functions of the **Trim** and **Relimit** tools.



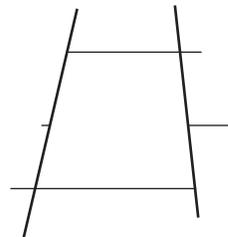
Tech Note:

If you trim an entire entity, a prompt will come up asking if you want to delete the entire item. Graphite is making sure you want to remove that piece of geometry before it performs the trim.

- Click the locations indicated here.



The lines are trimmed.



Tech Note:

For **Trim**—select what you want to throw away.

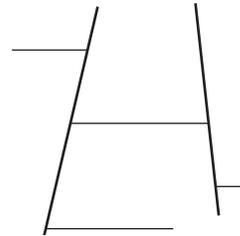
For **Relimit**—select what you want to keep.

3. Explore the **Relimit** tool.

- Select **Edit>Undo** four times so that the lines are restored.
- Select the **Relimit** tool. The Message Line reads, *Relimit: Pick section to retain [Shift = boundary, Ctrl = Trim (Windows) or Option = Trim (Macintosh)].*

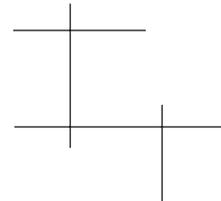


- Click the same locations with the **Relimit** tool.
- The lines are extended to the boundaries.



4. Explore the **Corner Trim** tool.

- Create an approximation of the lines below.
- Double-click the **Selection** tool to select all lines.



Tech Note:

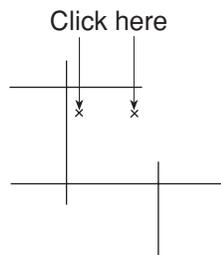
If you make mistakes, you can use *Undo* and *Redo* to go back to a previous version.

- Select the **Corner Trim** tool from the **Trim** subpalette.



The Message Line reads, *Corner Trim: Pick first entity to trim [Shift = Corner, Ctrl = No Trim (Windows) or Option = No Trim (Macintosh)].*

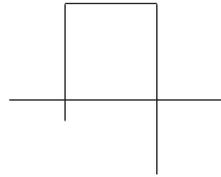
- Hold down the SHIFT key and click inside the top corners as shown in the following graphic.



Tip:

You can either click each line individually or use SHIFT-Click, and click inside the corners.

The lines are trimmed to create corners.



- Select the **Simple Trim** tool.



- Click the parts of the lines to be trimmed away (those that extend past the corners) to create the figure below.



5. Delete all geometry.
 - Choose **Edit>Select All**.
 - Press the BACKSPACE (Windows) or DELETE (Macintosh) key.

Accomplishments

- Using the **Corner Trim** tool
- Observing the difference between trim and relimit
- Deleting geometry

Exercise 2: Text

In this exercise you will explore using text.

1. Create a text block.
 - Choose the **Text** tool.



- Drag a box in the center of the screen—the size isn't important.



- Click the Width Status Line data field.

Tip:

Once you leave the **Text** tool, you can change the size of the text-entry box by dragging a selection fence around the control points on the right or left side of the box and dragging the points.

- Type **3** and press ENTER (Windows) or RETURN (Macintosh).

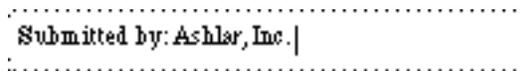


2. Change the text characteristics to *10-point Roman* (Windows) or *Courier* (Macintosh).

- Choose **Text>Font>Roman** (Windows) or **Courier** (Macintosh).
- Choose **Text>Size>10**.

3. Type this text: **Submitted by: Ashlar Inc.**

- Press ENTER (Windows) or RETURN (Macintosh).

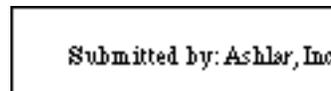


4. Draw a box around the text.

- Select the **Rectangle** tool.

The text entry box disappears.

- Drag a box around the text so that the text is centered within the box.



5. Group the box and the text so that they can be treated as a single unit.

- Double-click the **Selection** tool.
- Choose **Arrange>Group**.
- Drag the box around to see that the text and the box act as a single entity.

6. Type a list of notes.

- Create another text box, 2.5 inches wide.
- Change the text characteristics to accommodate a plotter:

Font: **Plotter**
 Size: **.156**
 Style: **Normal**

Tech Note:

Control points are the points defined when you first created the text block and represent the endpoints of the text box.

Instead of 10-point Roman or Courier you can use any font listed in **Text>Font**.

Tech Note:

The text box appears only when the text is selected. The box you are drawing now will always be visible.

Tech Note:

When you want several entities to act as one, group them. In that way you won't accidentally select one, when you meant to select the components of the group.

- Type the following without pressing ENTER (Windows) or RETURN (Macintosh).

1. The materials list is included on a separate sheet.

1. The materials
list is included on
a separate sheet.

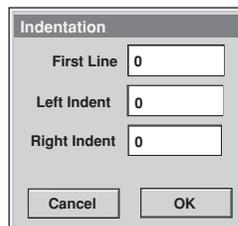
- Press ENTER (Windows) or RETURN (Macintosh) to begin a new line.
- Type the following:

2. Tolerances are specified as noted.

1. The materials
list is included on
a separate sheet.
2. Tolerances are
specified as noted.

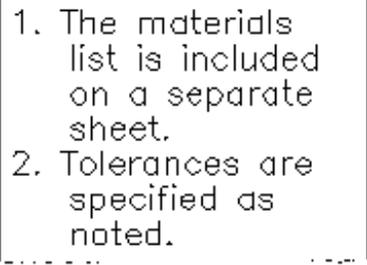
Use the BACKSPACE (Windows) or the DELETE (Macintosh) key to make corrections as you type. Also, notice word wrap—the text automatically wraps to the next line when a word extends past the right margin.

7. Change the indentation so that the text aligns under itself rather than under the number.
 - Use the **Selection** tool to select the note text.
 - Choose **Text>Indentation**. The Indentation dialog box appears.



- Click the *Left Indent* data field to highlight it.
- Type **.35**.

- Click OK.



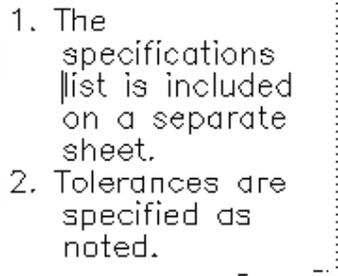
1. The materials
list is included
on a separate
sheet.
2. Tolerances are
specified as
noted.

The text shows a hanging indent.

8. Make changes to the text.
 - With the text still selected, click the **Text** tool.
 - Double-click the word, *materials*.

The word, materials and the space after it are highlighted.

- Type **specifications** and press the SPACEBAR once.

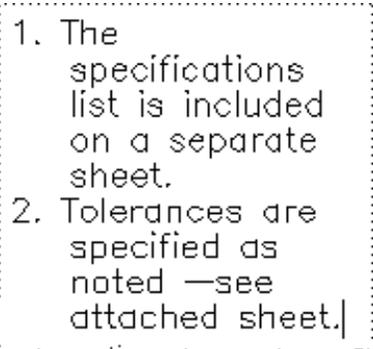


1. The
specifications
list is included
on a separate
sheet.
2. Tolerances are
specified as
noted.

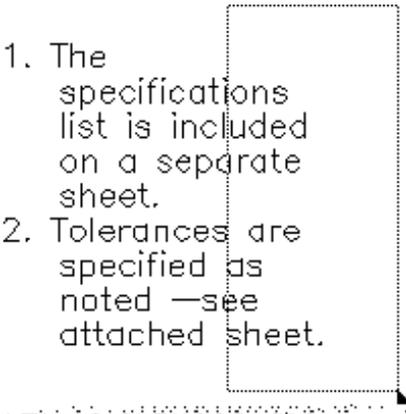
- Click after the *period* that ends the second sentence.
- Press the BACKSPACE (Windows) or the DELETE (Macintosh) key.

The period deletes.
- Press the SPACEBAR once and type the following:

—see attached sheet.

- 
1. The specifications list is included on a separate sheet.
 2. Tolerances are specified as noted —see attached sheet.

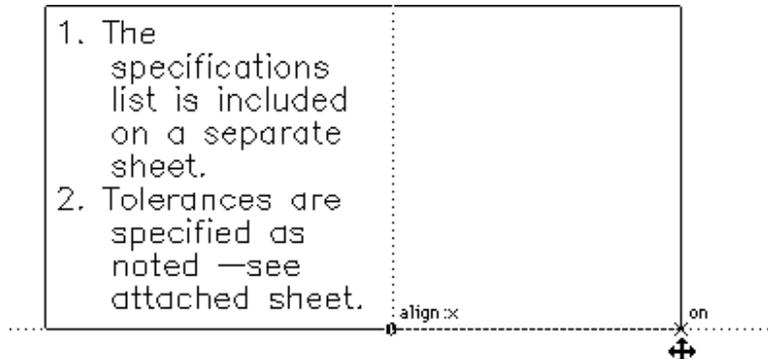
9. Change the size of the text box.
 - Click the **Selection** tool.
 - Drag a selection fence around the right side of the text box.

- 
1. The specifications list is included on a separate sheet.
 2. Tolerances are specified as noted —see attached sheet.

The upper and lower corner points of the box are selected.

1. The specifications list is included on a separate sheet.
2. Tolerances are specified as noted —see attached sheet.

- Drag the control points about one inch to the right.



The area resizes and the text is redraws.

1. The specifications list is included on a separate sheet.
2. Tolerances are specified as noted —see attached sheet.

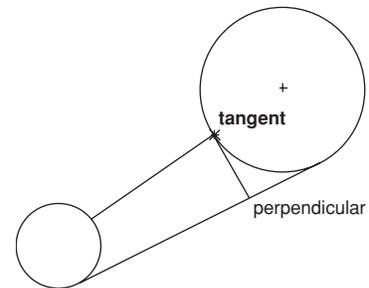
10. Explore the text processing functions until you feel comfortable with the tools, then delete all the text you created in this exercise.
 - Choose **Edit>Select All**.
 - Press the BACKSPACE (Windows) or the DELETE (Macintosh) key to delete all selections.

Accomplishments

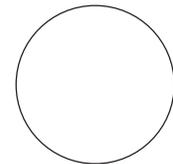
- Entering text
- Changing text characteristics
- Grouping entities
- Specifying a hanging indent
- Making changes to text
- Changing the size of the text area

Exercise 3: Tangent and Perpendicular Lines

In this exercise you will explore drawing tangent and perpendicular lines.



1. Draw two circles like those shown here using a circle tool.



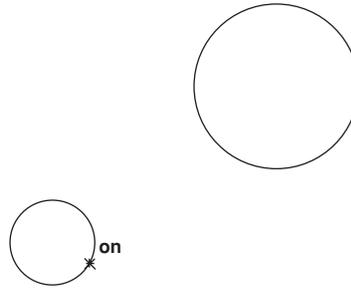
2. Construct a line tangent to the lower edges of both circles.
 - Click the **Single Line** tool. The Message Line reads, *Single Line: Pick beginning point [Ctrl = Copy previous (Windows) or Option = Copy Previous (Macintosh)]*.



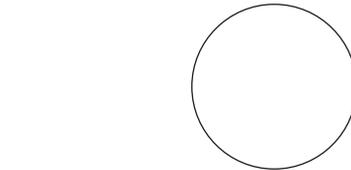
Tech Note:

The Drafting Assistant must display *on* and not other circle notations such as *quadrant*.

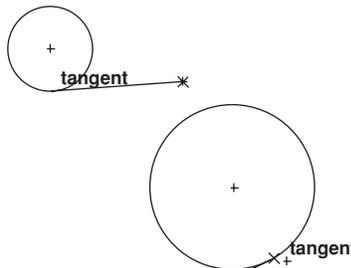
- Move the pointer to the circle on the left until the *on* notation appears. It cannot display a *quadrant* notation.



- Drag the pointer away from the circle at approximately a 45° angle until the *tangent* notation appears as shown in the following graphic. Do not release the mouse button.



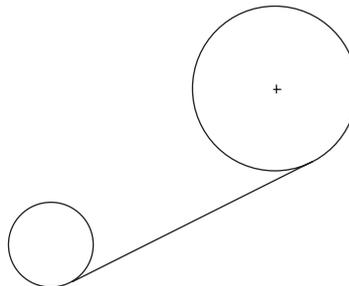
- Drag the pointer to the lower edge of the large circle until the *tangent* notation appears.
- Release the mouse button.



The tangent line is drawn.



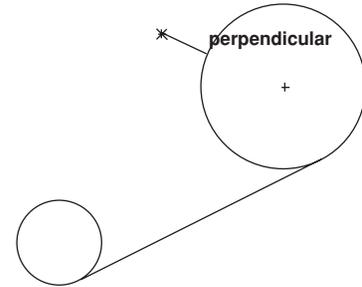
3. Construct a line perpendicular to both circles.
 - Press the mouse button when *on* appears on the large circle.



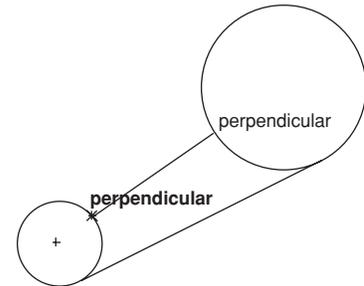
Tip:

Notice that the line you drag stays tangent to the circle wherever you drag it.

- Drag directly away from the circle at approximately a 90° angle until the *perpendicular* notation appears.

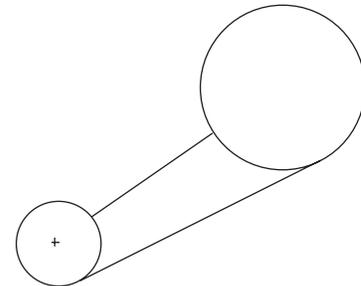


- Drag to the left circle until the *perpendicular* notation appears.



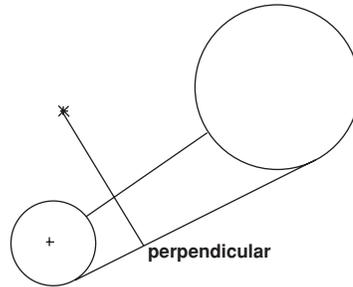
- Release the mouse button.

The line is drawn perpendicular to both circles.

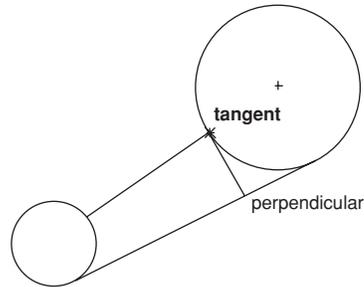


4. Construct a line perpendicular to the lower straight line and tangent to the larger circle.
 - Move the pointer to the lower straight line and press the mouse button when *on* notation appears.

- Drag at a **90°** angle from the line to display the *perpendicular* notation. Do not release the mouse button.



- Drag the new line along the lower straight line to the large circle until *tangent* appears.



The line is drawn tangent to the large circle and perpendicular to the lower straight line.

5. Save the file if you wish.

Accomplishments

- Constructing tangent lines
- Constructing perpendicular lines

Tip:

As you have seen, tangent and perpendicular lines can be pulled off circles and other lines, as well as ellipses, splines and arcs.

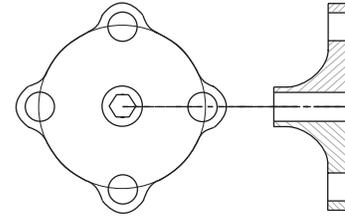
Tip:

You will notice that as you drag along the lower line, the active line remains perpendicular.

Exercise 4: Rotational Copy

In the next four exercises you will construct a flange with a side view.

For this exercise, draw a flange 3 inches in diameter containing four 1-inch lugs each with 0.5-inch holes. Center a 0.5-inch octagonal hole in the .75-inch hub of the flange.



1. Open a new document.
2. Choose a different pen style.
 - Choose **Pen>Style>Visible**.

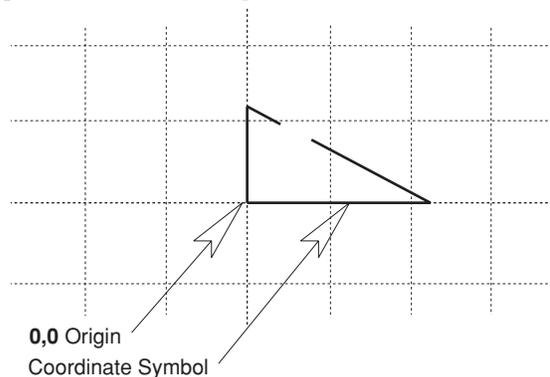
The pen style is now solid, black lines .02 inches wide.

3. Draw a 3-inch circle centered at 0,0.
 - Choose **Layout>Show Grid**, so that you can see the origin.

The grid appears on the drawing area.

Tech Note:

This step is not necessary, it merely shows you the grid and origin point. When the grid is on, lines you draw snap to the divisions on the grid.



- Choose the **Center-Point Circle** tool.
- Type **3** in the Status Line and press the TAB key.
- Type **0** and press TAB.
- Type **0** and press ENTER (Windows) or RETURN (Macintosh).

The 3-inch circle is drawn, centered at 0,0.

- Choose **Layout>Hide Grid**.

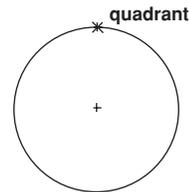
The grid and coordinate symbol disappear.

4. Draw a 0.5-inch lug hole centered at the top of the 3-inch circle.

The **Center-Point Circle** tool is still the current tool.

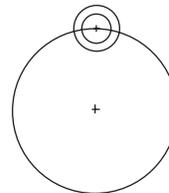
- Click on the 12 o'clock *quadrant* notation.
- Type **0.5** in the Status Line and press ENTER (Windows) or RETURN (Macintosh).

A 0.5-inch circle is drawn, centered on the top of the 3-inch circle.



5. Draw a 1-inch circle with the same center as the .5-inch circle.
- Type **1.0** and press ENTER (Windows) or RETURN (Macintosh).

Another circle is drawn, centered at the same place and with a diameter of 1 inch.



6. Create fillets with the default .25-inch radius where the lug joins the flange.

- Click the **2-Entity Fillet** tool.

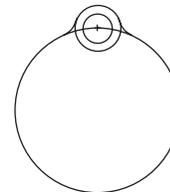


- Hold down the SHIFT key and click  between the two curves, as shown by the x in the graphic below.



- Repeat the process on the other side of the lug.

The fillets are complete.



Tech Note:

As you have seen, tangent and perpendicular lines can be pulled off circles and other lines as well as ellipses, splines and arcs.

7. Trim the lug.
 - Click the **Selection** tool.

The last fillet you created is still selected.

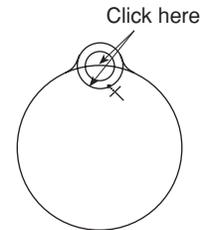
- Hold down the SHIFT key and click the other (unselected) fillet.

Both fillets are selected to be used as the boundaries for trimming.

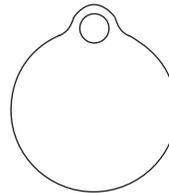
- Click the **Simple Trim** tool.



- Click the bottom of the 1-inch circle and the top of the 3-inch circle.



The circles are trimmed between the fillets.



Tech Note:

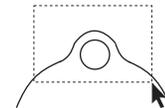
Only the objects inside the fence are selected.

8. Construct a total of four lugs for the flange.

- Click the **Selection** tool.
- Drag a fence around all entities that make up the lug.

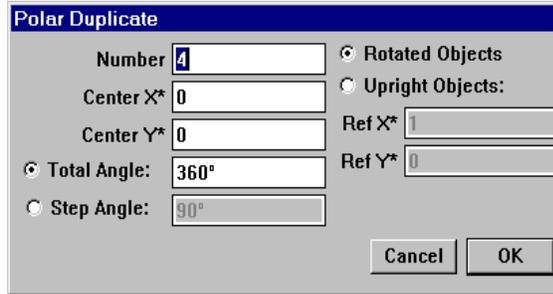
The lug is selected.

- Choose **Edit>Polar Duplicate**.

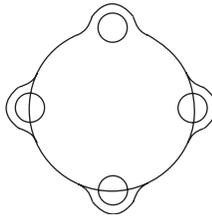


The dialog box should be set to rotate **4** objects, with

the Center X, Y coordinates set at **0,0**.

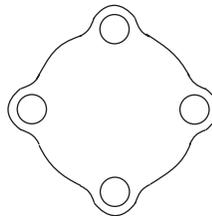


- Click OK.



The lug is copied, but the 3-inch circle must be trimmed.

- Trim the circle inside the copied lugs.
 - Use the **Selection** tool and the SHIFT key to select the fillets for the three copied lugs.
 - Choose the **Simple Trim** tool and trim the circle between the fillets.



The lugs are complete.

- Construct a .75-inch circle in the center of the flange.
 - Select the **Center-Point Circle** tool.

Tech Note:

The asterisks in the dialog box indicate that you can enter coordinates into the Center X and Center Y data fields or you can specify a location with the mouse. To use the mouse, first click the Center X data field. Then click the location for the center on the drawing area.

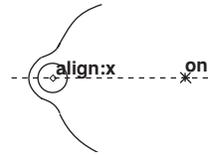
Tech Note:

For this exercise, you want the default location of 0,0 so you don't need any entries in this data field.

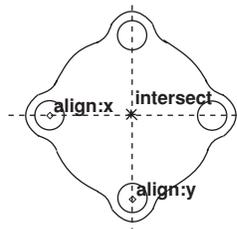
Tech Note:

If you click something you don't want to select, continue holding down the SHIFT key and click again to deselect it.

- Move the pointer to display the horizontal construction line through the left bolt hole.



- Display the vertical construction line for the lower bolt hole and click at the intersection of these two construction lines.



- Type **.75** and press ENTER (Windows) or RETURN (Macintosh).
11. Add the .5-inch (outside measurement) octagonal hole within the .75-inch hub.
- Select the **Inscribed Polygon** tool. The Message Line reads, *Inscribed Polygon: Pick center of polygon [Ctrl = Copy previous (Windows) or Option = Copy previous (Macintosh)].*

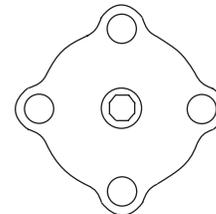


Confirm that the Status Line shows the X,Y coordinates as **0,0**. If this is not the case, enter **0** into each of these data fields.

- Type **.5** in the Diameter data field (do not press ENTER or RETURN).
- Press the TAB key to select the Sides data field.
- Type **8** and press ENTER (Windows) or RETURN (Macintosh).

The octagonal hole is drawn and the flange is complete.

12. Save the part, naming it *flange*.
- Choose **File>Save**.
 - Type *flange* and press ENTER (Windows) or RETURN (Macintosh).



Accomplishments

- Displaying the grid
- Placing geometry at the origin
- Filletting circles
- Making multiple selections
- Trimming unnecessary geometry
- Rotating and copying the lug
- Constructing an inscribed octagon

Exercise 5: Constructing a Side View

Create a side view of the flange that is 1.5 inches thick at the hub and .25 inch thick at the lugs.

1. Zoom out from the part.

- Click the **Zoom Out** tool.

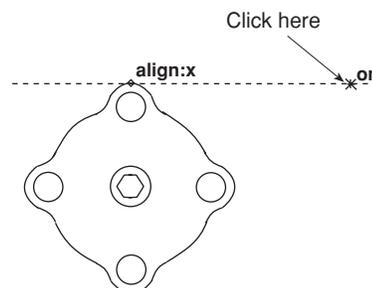
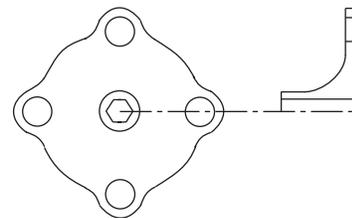


- Enter **.80** in the Status Line.

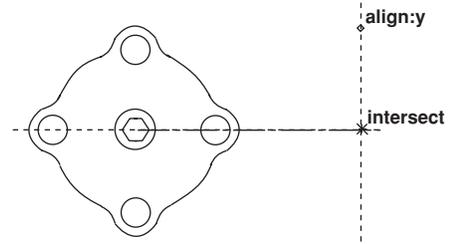
2. Construct a vertical line for the side view that is the same length as the distance from the top of the flange to the center.

- Click the **Single Line** tool.

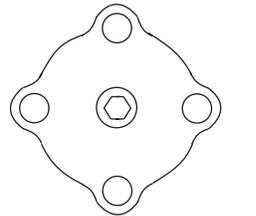
- Move the pointer to the top of the flange to display the construction line and move to the right the approximate distance shown below and click *on* the construction line.



- Move the pointer to the center of the right bolt hole to “wake it up;” then move to the right to display the horizontal construction line. Click at the *intersect* point of the horizontal and vertical construction lines.



The first vertical line for the side view is drawn.



3. Construct the remaining vertical lines of the side view.

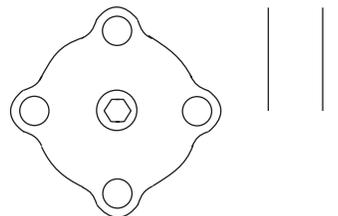
- Click the **Parallel Lines** tool. The Message Line reads, *Parallel Lines: Drag new line off existing line.*



- Drag a line to the right of the vertical line in the side view.
- Type **1.25** and press ENTER (Windows) or RETURN (Macintosh).

The parallel line is drawn 1.25 inches to the right of the original line.

- Click the original vertical line.
- Type **1.5** and press ENTER (Windows) or RETURN (Macintosh).



The vertical lines are complete.

4. Construct the horizontal lines for the side view.
 - Select the **Single Line** tool.

- Draw a horizontal line across the top of the right arm of the side view.



- Continue drawing the horizontal lines aligned with these locations and in the order shown below:

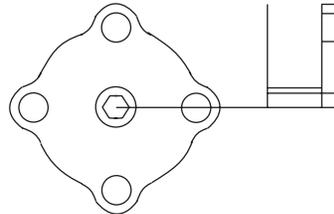
Top of bolt hole

Bottom of bolt hole

Top of hub

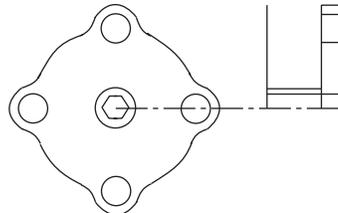
Top of octagon

Center of flange



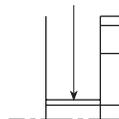
The lines are drawn and the *centerline* is selected. If it is not, use the **Selection** tool to select the line from the center of the flange to the side view.

- Choose **Pen>Style>Center**.



- Choose the **Selection** tool.
 - Click outside the flange to deselect everything.
 - Choose **Pen>Style>Visible**.
5. Trim the excess lines.
- Save the drawing in case things don't go as expected.
 - Zoom in on the side view only.
 - Select the trim boundary.

Select this line



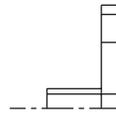
Tip:

You can use the CTRL+SHIFT keys (Windows) or the ⌘ key (Macintosh) or Stroke zoom, dragging from the upper left to the lower right across the side view. You can also select the **Zoom In** tool and drag a fence around the side view.

Tech Note:

If zooming takes you places you didn't want to go, choose **Arrange>Zoom All** and all geometry appears and fills the window.

- Trim the vertical line on the left that extends above the selected horizontal line.



The vertical line trims.

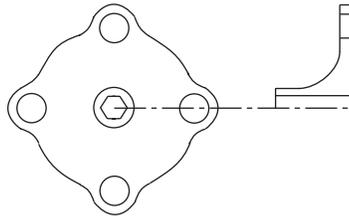
- Trim the vertical line that descends from the inside corner to the centerline.



The trimming is complete.

- When the drawing looks right, save it again.
6. Create the hub fillet with a radius of .75 inch.
 - Select the **2-Entity Fillet** tool.
 - Enter **.75** in the Radius data field.
 - Hold the SHIFT key down and click inside of the corner of the side view.

The fillet is redrawn to a .75-inch radius.



- Zoom out to see the entire flange.

Accomplishments

- Constructing a side view
- Creating parallel lines
- Creating a centerline
- Trimming corners

Tech Note:

If you have had trouble, and the drawing is not right, close this document without saving it. To close without saving, click the Close box icon, then click No when asked if you want to save. Next, choose **File>Open** and open the flange document again, and repeat Exercise 5.

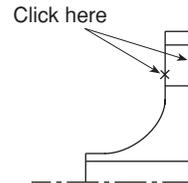
Exercise 6: Advanced Crosshatching

In this exercise you will crosshatch the side view. Since the side view isn't a single closed figure, you will have to segment some of the lines for the crosshatching to work properly.

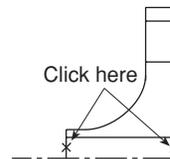
1. Zoom in on the side view.
2. Select all geometry for the side view by dragging a selection fence around the side view.
3. Use the **Segment** tool to break the vertical lines at the intersection of the horizontal lines. This is necessary to define a closed boundary to crosshatch.
 - In the **Trim** palette, choose the **Segment** tool. The Message Line reads, *Segment: Pick entity [Shift = Select boundary, Ctrl = Current pen (Windows) or Option = Current pen (Macintosh)].*



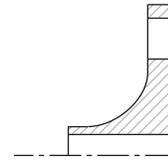
- Click the vertical lines to be segmented.



- Click the lines representing the center hole.



4. Crosshatch the solid sections of the side view.
 - Click the **Selection** tool.
 - Click anywhere in the drawing area to deselect the side view.



Tip:

You can also use the **Tracer** tool to select these unsegmented boundaries. You might practice both methods of selecting lines.

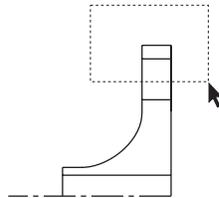
Tech Note:

To understand this process, consider how you created the side view. The right vertical line is a single line which is met by the horizontal lines forming the bolt hole and center hole. Graphite cannot determine the boundary because the horizontal lines touch the vertical lines within the closed figure. In this exercise, you'll segment the boundary lines so they can be selected to form two closed shapes.

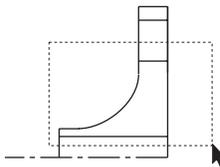
Tech Note:

Although no visual change has occurred, the vertical lines have been segmented—broken into shorter, connected lines.

- Drag a selection fence around the top part of the side view.

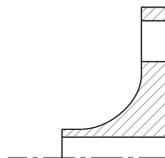


- Hold down the SHIFT key and drag a selection fence around the lower solid section of the side view.



Both sections are selected.

- Choose **Pen>Crosshatch**.
- If the ISO patterns are not already displayed, click the ISO radio button.
The Iron pattern is already selected and displayed in the pattern box.
- Click the *Spacing* data field.
- Enter **.1** and click Apply.
- Close the dialog box.



The side view is crosshatched.

5. Save the part.

Accomplishments

- Segmenting lines
- Crosshatching complex figures

Exercise 7: Mirror Images and Bolt Circles

In this exercise you will create the bottom half of the side view and add a bolt circle to the front view of the flange.

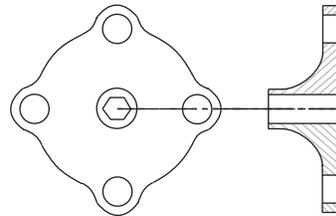
1. Create the bottom half of the side view.
 - Drag a selection fence around the entire side view.
 - Select the **Mirror** tool in the twelfth sub palette of the main tool palette. The Message Line reads, *Mirror: Pick beginning of reference line [Shift = Select, Ctrl = Copy (Windows) or Option = Copy (Macintosh)].*



You are asked to specify a reference line.

- Hold down the CTRL (Windows) or the OPTION (Macintosh) key and click on the centerline at two places in the side view.

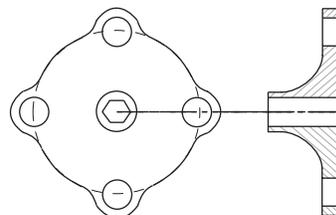
The side view is complete.



Tech Note:

CTRL (Windows) or OPTION (Macintosh) plus click causes the transformation tool to make a copy and transform the geometry.

2. Add a bolt circle to the front view.
 - Select the **Selection** tool and click anywhere to deselect the side view.
 - Choose **Pen>Style>Center**.
 - Select the **Center-Point Circle** tool.
 - Drag from the center of the flange to the center of one of the four bolt holes.



- Use the **Selection** tool and click in the drawing area to deselect the circle.
 - Choose **Pen>Style>Visible**.
3. Save the part and close the document.

• Close the current Graphite window.

You are asked if you want to save the document.

- Click Yes.

The document saves and its window closes. You may see other windows if other Graphite documents are open.

Accomplishments

- Creating a mirror image
- Creating a bolt circle

Exercise 8: Parametrics

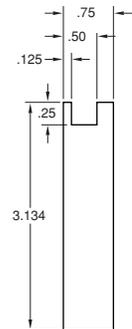
In this exercise you will construct a side view of the part you constructed in the Basic Graphite section. You will use one of Graphite's most powerful features—Parametrics. With parametrics, you can construct geometry in the shape you want and then Graphite automatically redraws the geometry with specific measurements.

1. Begin by opening the part that you saved earlier.

- Choose **File>Open**.
- Double-click the filename *part1a*.

If you saved the file in a different folder or gave it a different name, select the filename accordingly.

2. Zoom the drawing so that you can see the part and have room for the new construction.
- Choose **Arrange>Zoom All**.
 - In the tool palette, select the **Zoom Out** tool.
 - Click anywhere on the left side of the part.



3. Hide the dimensions you created to make it easier to construct the side view. (Note the placement of the dimensions so that you do not build the side view on top of them.)

- Choose **Layout>Layers**.

- Select the Dimension layer.

The Show button toggles to Hide.

- Click Hide.

The dimensions are no longer visible on the drawing.

- Close the dialog box.

4. Construct a side view that represents the final shape you want; don't be concerned with proportions or measurements.

- Choose the **Pen>Style>Visible**.

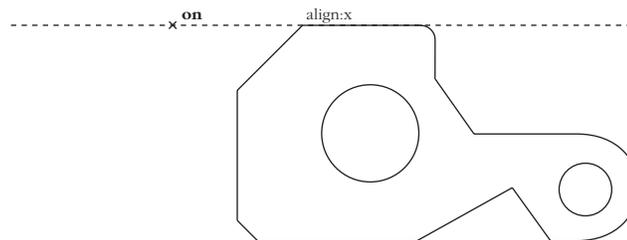
- Select the **Connected Lines** tool.

- Drag the pointer over the top *endpoint* of the 45° line to activate the horizontal construction line from the top of the part, but do not click.

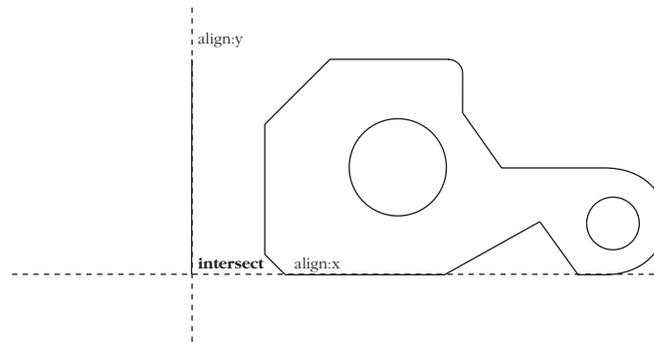
The point on the part activates so that the Drafting Assistant displays a dynamic construction line through it to assist in constructing the side view.

- Move the pointer to the left a short way along the construction and click the first point. Be sure an *on* notation appears.

- Drag a line to the bottom endpoint of

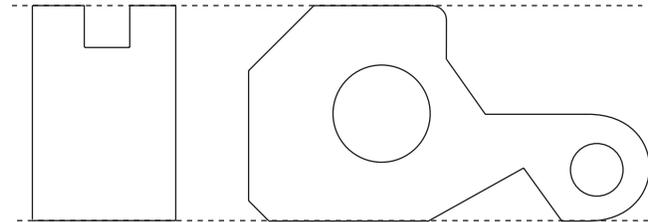


the chamfer to “wake up” a horizontal construction and drag to the left along the construction line until an intersect notation appears.



- Create the shape on the left below, approximating the horizontal measurements.

5. Save the part.
6. Dimension the vertical components of the side view, using baseline dimensions.



- Choose **Layout>Layers**.

- Select Dimension and click Show.

The Dimension layer appears.

- If necessary, choose **Dimension>Show Palette**.

- Select the **Vertical Base Line** Dimension tool. The Message Line reads, *Vertical Base Line: Pick first dimension point.*



- Select the top-left corner then the bottom-left corner of the side view.

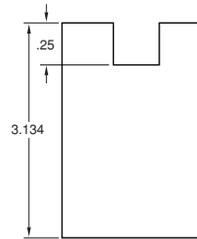
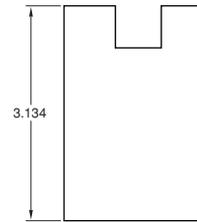
Tip:

In this case, since everything is measured from the baseline, you select the baseline first, regardless of the proper order for displaying the dimension. This means that the dimension will not be displayed in the proper location, but you can drag the dimension once it appears.

Tech Note:

In this example, only the vertical dimension displays its real value—it measures the actual height of the part. You will specify the values of the other dimensions.

- Drag the dimension to the left so that another dimension can be placed between it and the side view, as shown .
- Dimension the depth of the notch, by selecting the bottom of the left side of the notch. (Remember the baseline is already selected.)
- Drag the dimension text to the left between the view and the last vertical dimension.
- Enter **.25** in the Status Line and press ENTER (Windows) or RETURN (Macintosh).



Tip:

This entry goes into the Text data field on the Status Line, specifying the value you want rather than the current value.

7. Dimension the remaining components of the side view.

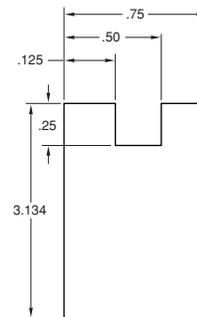
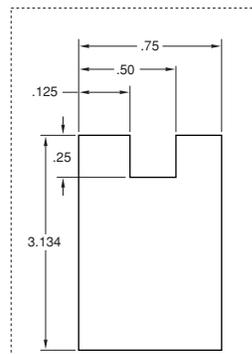
- Select the **Horizontal Base Line** Dimension tool. The Message Line reads, *Horizontal Base Line: Pick first dimension point.*

Enter the values as shown. If you begin at the left corner, the dimensions will be properly placed.

8. Save the part.

9. Resolve the parametrics for the side view.

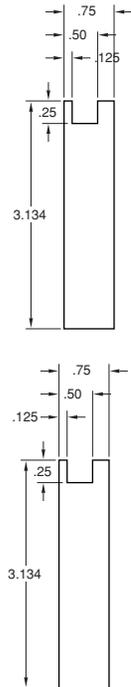
- Drag a selection fence around the side view, including the dimensions.



Tech Note:

If you make a mistake, press the BACKSPACE (Windows) or DELETE (Macintosh) key if the dimension is still selected. If it is not selected, select it with the **Selection** tool and then press the BACKSPACE (Windows) or DELETE (Macintosh) key.

- Choose **Edit>Resolve**. The Resolve dialog box appears.
 - Click the upper-right corner of the side view so that corner stays aligned with the front view.
 - Click OK. The part redraws using the new measurements.
10. Move the .125 dimension text to a better location.
- Deselect everything by clicking in an empty space with the **Selection** tool.
 - Select the .125 dimension text.
 - Drag the text to the left so that it is outside the extension lines, as shown below.
11. Save the part.

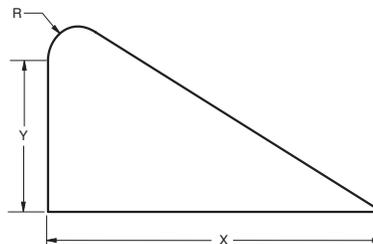


Accomplishments

- Using layers
- Entering specific dimensions
- Resolving parametrics

Exercise 9: Variable Parametrics

In this exercise you will use parametrics with variables. When you use variables, you can draw a part and then assign different values for the dimensions. In this way you can recreate the part repeatedly with different specifications.

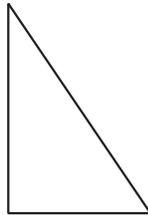


1. Open a new document.

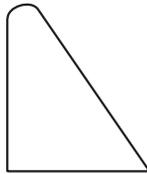
- Choose **File>New**.

A new Graphite document opens.

2. Draw the triangle here.



3. Fillet the upper corner of the triangle.

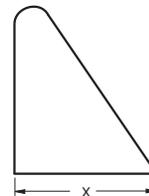


4. Dimension the triangle using variables.

- If the **Dimension** tool palette is not visible, choose **Dimension>Show Palette**.
- Select the **Horizontal** Dimension tool. The Message Line reads, *Horizontal: Pick first dimension point.*



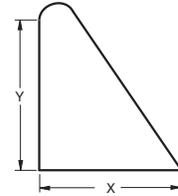
- Dimension the horizontal line, clicking the right side first.
- Type **X** in the Status Line and press ENTER (Windows) or RETURN (Macintosh).
- Select the **Vertical** Dimension tool. The Message Line reads, *Vertical: Pick first dimension point.*



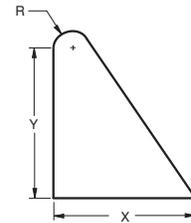
Tip:

If the palette obscures your construction, move it by dragging the title bar.

- Dimension the left vertical line, clicking the lower endpoint first.
- Type **Y** in the Status Line and press ENTER (Windows) or RETURN (Macintosh).



- Using the **Radial Arrow Out** Dimension tool, dimension the radius, using **R** as the variable.

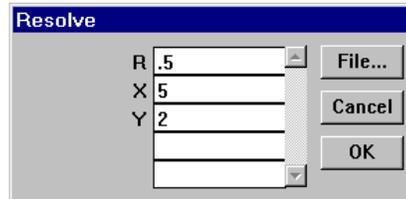


Tech Note:

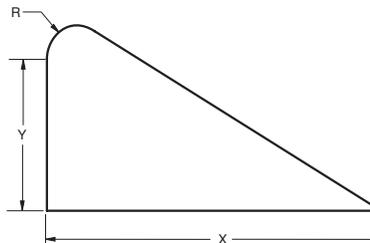
The next step won't work if there is extraneous geometry in the document and you select everything. If that is the case, instead of double-clicking the **Selection** tool, drag a selection fence around the triangle and its dimensions.

5. Resolve the geometry to specific dimensions.

- Double-click the **Selection** tool to select everything.
- Choose **Edit>Resolve**.



- Type **.5** for R and press the TAB key.
- Type **5** for X and press the TAB key.
- Type **2** for Y and click OK.



The geometry is redrawn to the dimensions you entered.

6. Save the document in the Symbols folder of Graphite.

- Choose **File>Save As**.
- Display the Symbols folder.
- Type **Test** in the Filename data field.
- Click OK.

The document saves as *Test* in the Symbols folder.

7. Use a symbol.

- Choose *New* from the File menu.

A new Graphite document displays.

- Choose **File>Symbol...**
- The Open dialog box appears.
- 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*.
- Enter a value for each of the parametric dimensions.
- 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.
- Click Place.
- 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.
- Add regular dimensions if desired.

Tip:

With the **File>Symbol...** any Graphite document can be brought into a current document as a symbol. Files do not have to be saved in the Symbols folder.

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.

Tech Note:

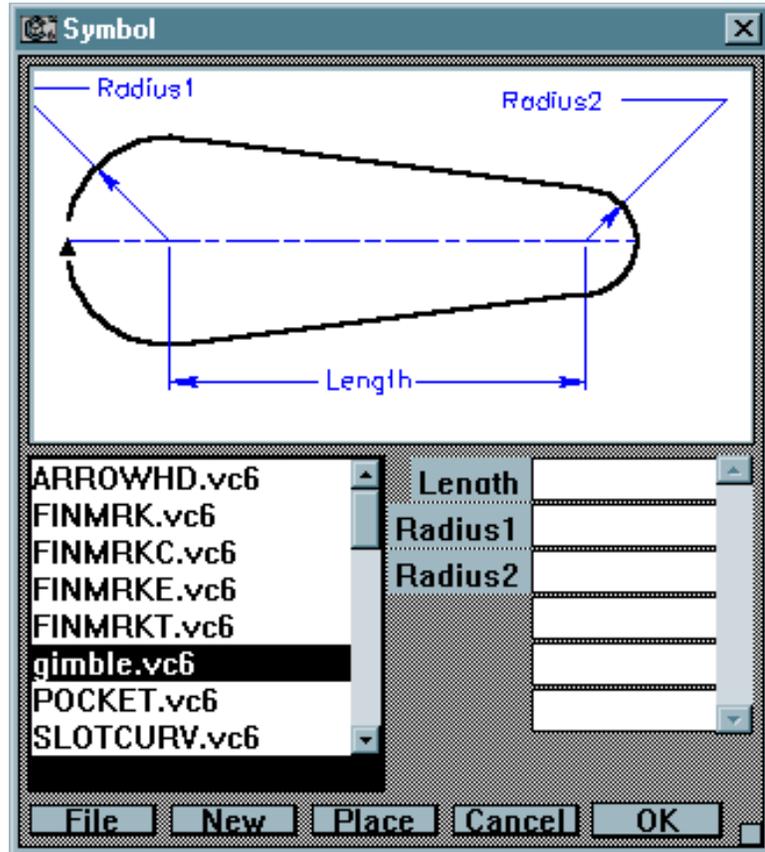
You can see an enlargement of any portion of the symbol by moving the pointer to the area you want enlarged and pressing the mouse button.

Tech Note:

If you enter values that cannot be used for the resolution, an error message appears. If this happens, choose Symbol again and enter different values.

The Symbol box appears with the shape of the symbol you selected.

8. Save or discard the documents, as you wish



Accomplishments

- Dimensioning with variables
- Resolving parametrics
- Creating symbol files
- Using symbols

Exercise 10: NURB Splines

In this exercise you will create and edit splines and observe the difference between splines constructed through specific points and those constructed from vectors.



1. Open a new document.
2. Select the **Through-Points Spline** tool. The Message Line reads, *Through-Points Spline: Pick control point [Double-click last point]*.



3. Click points such as those shown below, double-clicking the last point.



A NURB spline is drawn through the control points you specify.



4. Select the **Vector Spline** tool. The Message Line reads, *Vector Spline: Pick control point [Double-click last point]*.

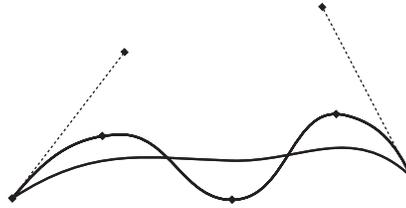


5. Click the same points (the *vertex* notation appears at the points you clicked for the first spline). A different curve is constructed using the vectors for calculation.



6. Edit the control point in the middle of the original spline.
 - Select the original spline.
 - Choose **Layout>Show Points**.

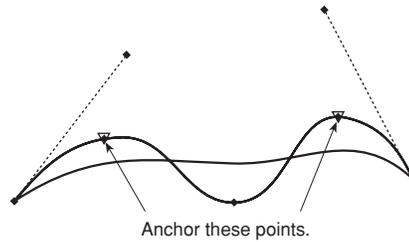
The control points and the endpoint slope controls are displayed.



- Select the **Lock Spline Control Point** tool. The Message Line reads, *Lock Spline Control Point: Pick control point.*



- Click the control points on both sides of the middle control point, as shown here.

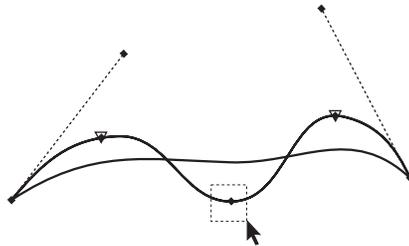


The points are locked.

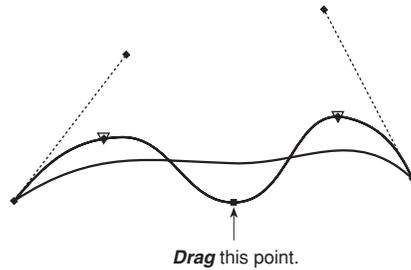
- Deselect the spline.
- Select the middle control point.

Tech Note:

If you can't select this point, choose **Edit>Selectable Points** and reselect the control point.



- Drag the middle control point to new locations and observe how the spline is redrawn.

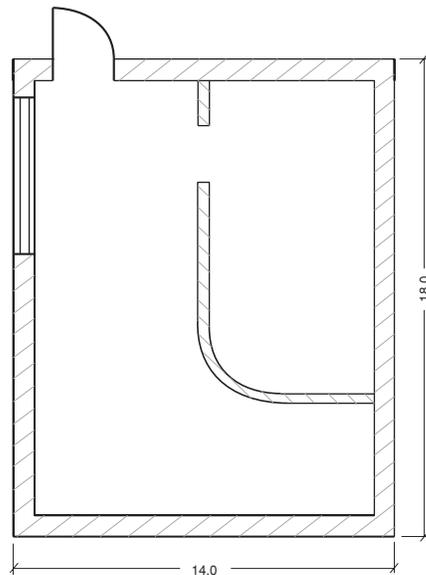


Accomplishments

- Creating a through-points spline
- Creating a vector spline
- Editing splines

Exercise 11: Smart Walls

In this exercise you will use the **Smart Wall** tool to create a simple architectural drawing, a floor plan with outside measurements of 14 feet by 18 feet.



Tech Note:

You can use different units of measure in the Status Line entries. If the selected units is not feet, be sure that you specify the unit such as 2' for two feet or 2 cm for two centimeters.

1. Open a new Graphite document.
2. Set the units to feet.
 - Choose **Layout>Preferences>Units**.
 - Click *Feet* and click OK.
3. Create two new layers: one for the interior walls and one for the exterior walls.
 - Select **Layout>Layers**.
The Layers dialog box appears.
 - Click New to create another layer.
 - Rename the layer by typing **Interior walls** in the Rename data field and click Rename.
 - Click New to create another layer.
 - In the Rename data field, rename the layer by typing **Exterior walls** and click Rename.
 - Select *Exterior walls* and click Set Layer, to make *Exterior walls* the current layer.
4. Draw the exterior walls.
 - In the **Line** subpalette, select the **Smart Wall** tool. The Message Line reads, *Double Line: Pick start point [Shift = Flip]*.

 - Enter **4"** in the T Status Line data field to indicate walls that are **4** inches thick.
 - Drag a horizontal line.
 - Enter **14** in the L Status Line data field and press ENTER (Windows) or RETURN (Macintosh).
The line extends off the screen.
 - Choose **Arrange>Zoom All**.


The entire line is visible.

Tech Note:

Click twice on the down arrow to scroll the horizontal wall upward on the screen.

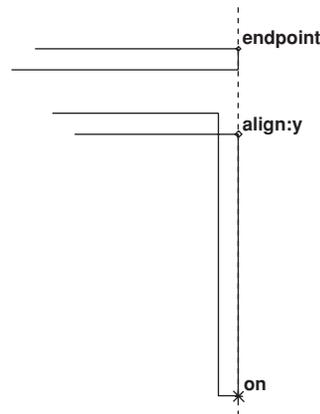
- Press the SHIFT key and drag down vertically from the *endpoint* of the line at the upper-right corner. (The first graphic below shows the point to select and the second shows what happens when you drag.)

It is critical that you hold down the SHIFT key and drag to construct the wall so that the wall thickness is not added to the length of the first wall.

- Enter **18** and press ENTER (Windows) or RETURN (Macintosh).

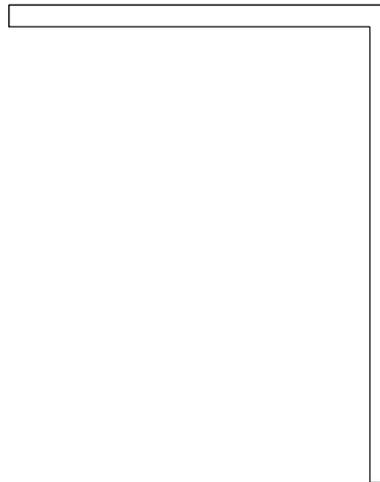
A corner is made between the first and second lines, and the second line extends off the screen.

- Choose **Arrange>Zoom All**.



Tech Note:

Selecting **Layout>Show Points** helps you to align the defining points of a wall precisely to the defining points of the previous wall segment.



Tech Note:

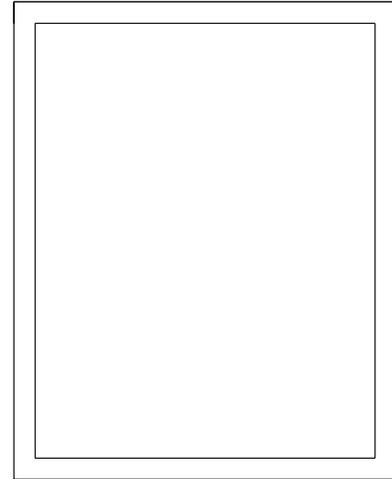
In the Edit Objects dialog box you can change the defining axes of selected walls.

- Complete the rectangle, using the SHIFT key while dragging the third wall.
5. Change the current work layer with the Work Layer Indicator box at the lower left of the drawing area.
- Click anywhere in the drawing area, to deselect all walls.
 - Press the mouse button on the box where the *Exterior walls* layer is showing and the menu displays.
 - Drag to *Interior walls*.

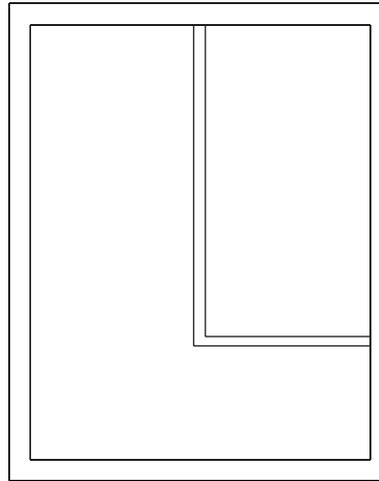
Interior walls is now the current work layer.

6. Create a drawing like the one below with two interior walls (3 inches thick) which don't merge with the exterior walls.
- Move the pointer along the inner edge of the upper horizontal wall until the notation *midpoint* displays.
 - Drag a vertical wall segment down about 10 inches long.
 - Type **12** in the L (length) Status Line data field.
 - Tab to the Thickness Status Line data field and type **3"** for the wall thickness and press ENTER (Windows) or RETURN (Macintosh).

The horizontal wall is not merged with the vertical wall segment, since they are not on the same layer.



- Drag a horizontal wall to connect the vertical wall segment with the right vertical exterior wall.
7. Construct a 2 Foot hole through the vertical interior wall, 2 feet distant from the upper horizontal wall.
- Move the mouse pointer to the inner left corner of the upper horizontal exterior wall.
 - When the *endpoint* notation displays, hold down the CTRL+SHIFT keys (Windows) or the ⌘ key (Macintosh) and drag horizontally toward the center, releasing the mouse button anywhere.



Tech Note:
Only walls on the same layer merge.

A stroke construction line displays through the inner edge of the upper exterior wall.

- Select the **Parallel Lines** tool.



- Drag the construction line about 2' towards the middle of the room.
- Enter 2 for the delta distance in the Status Line and press ENTER (Windows) or RETURN (Macintosh).

The parallel line is drawn 2 feet below the upper exterior wall.

- Drag the new construction line about 2' towards the middle of the room.
- Enter 2 for the delta distance in the Status Line and press ENTER (Windows) or RETURN (Macintosh).

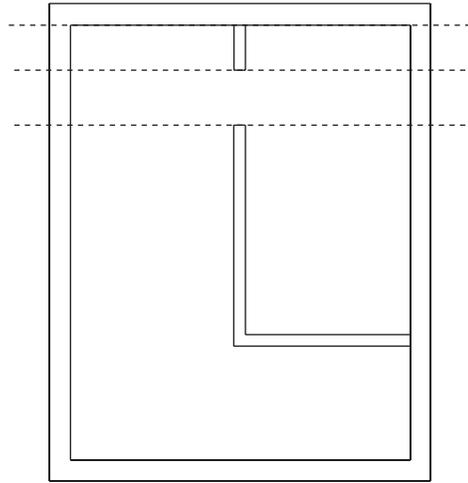
The second parallel line is drawn 2' below the second construction line.

- Select the lower two parallel construction lines with the **Selection** tool.
- Select the **Simple Trim** tool from the tool palette.



- Click the **Simple Trim** tool on the vertical interior wall between the two construction lines.

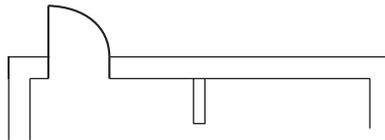
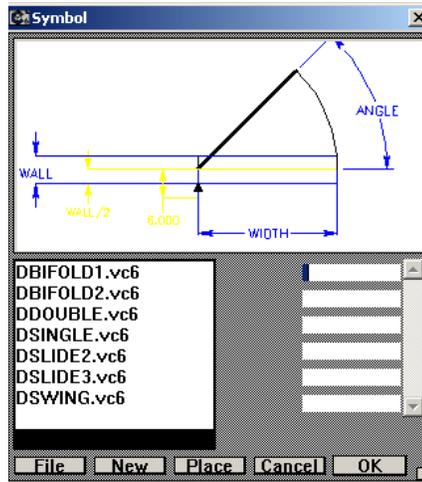
The interior wall trims and the wall segment between the two construction lines deletes.



8. Add a 3-foot door beginning at the left 10% point of the upper horizontal exterior wall and extending towards the center.
 - Set the layer to *Exterior walls* by moving the pointer to the Work Layer Indicator box where *Interior walls* is shown and dragging to the layer.
 - Choose **Layout>Preferences>Snap**.
 - Enter **10** for the *% Point*.
 - The Drafting Assistant will snap to *10%* points as well as *midpoints*.
 - Choose **File>Symbol...**
 - Go to the Symbols folder>Architect folder>Plandoor folder.
 - Choose the *dsingle.vc6* symbol (Windows) or the *Single* symbol (Macintosh) and click Open.

The Symbol dialog box appears so that you can specify the width of the door.

- Enter **90** for the *Angle* of the door opening and press the TAB key.
- Enter **4"** for the *Wall* thickness and press the TAB key.
- Enter **3** for the *Width* of the door.
- Drag the mouse horizontally, starting at the *% point* on the left side of the upper inside wall, towards the center, and release the mouse button anywhere. The drag indicates the orientation of the door.
- Click Place.



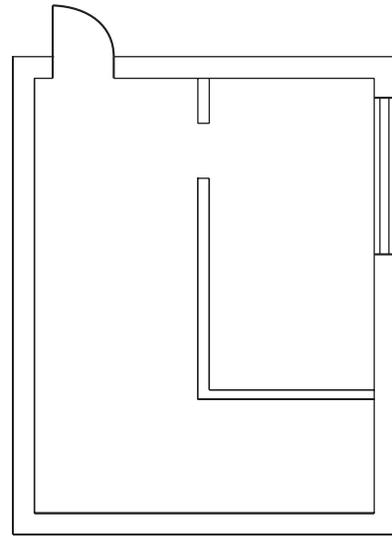
The wall geometry is redrawn to accommodate the door.

9. Add a 5 foot window beginning at the 10% point of the right wall and extending toward the center.
 - Choose **File>Symbol>Insert** and go to **Architect folder>planwin** (Windows) or **Plan Window** (Macintosh) folder. Open *wcasmnt2.vlm* (Windows) or *Casement 2* (Macintosh).
 - The Symbol dialog box appears so that you can specify the width of the window.
 - Enter **4"** (the Wall thickness) and press TAB.
 - Enter **5** (the Width of the window).
 - Drag the mouse vertically, starting at the *% point* on the inside wall of the right wall, towards the center, and release the mouse button anywhere.

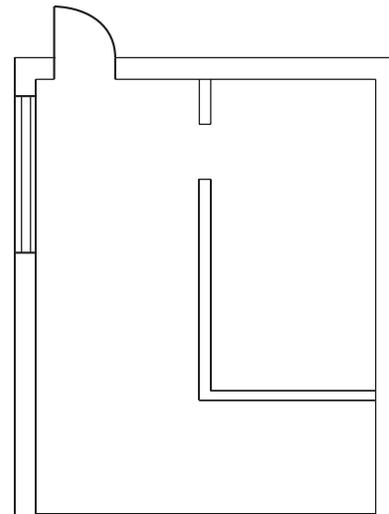
- Click OK.

The wall geometry is redrawn to accommodate the window.

10. Move the window and observe how the walls are automatically reconstructed.
 - Select the window and place the pointer on the top left corner of the window.



- Using the Drafting Assistant's dynamic construction line, drag it to the opposite wall.



11. Set the paper size and drawing scale.
 - Choose **File>Print Setup** (Windows) or **Page Setup** (Macintosh).
 - Select paper size **A** (or **8.5 x 11** for laser printers). Click OK.
 - Choose **Layout>Drawing Size**.

Tech Note:

The available paper sizes depend on the printer or plotter you have specified.

The Drawing Size dialog box displays.

- In the Drawing Scale data field, type **1":2'** (this entry is the same as 1:24)
- Click OK.

12. Segment the exterior walls.

- Select the two symbols (door and window).
- Choose the **Segment** tool from the tool palette.
- With the **Segment** tool, click the wall segments near the door and the window symbols.

The walls are segmented.

13. Crosshatch the exterior walls.

- Select all visible wall segments of the exterior walls by clicking with the **Selection** tool.
- Choose **Pen>Hatch**.

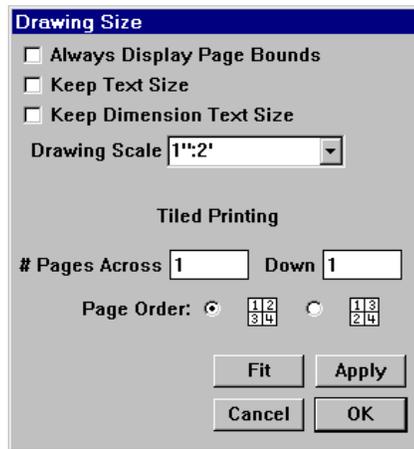
The exterior walls are crosshatched.

14. Ungroup the interior walls.

- Select the lower vertical and the horizontal interior wall.
- Choose **Arrange>Ungroup**.

Both wall segments are ungrouped and lose their smart features.

15. Fillet the corner of the two interior wall segments.

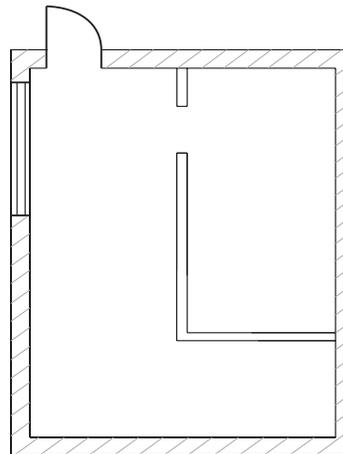


Tech Note:

The Scale data field accepts combinations of units of measure.

Tech Note:

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.



Tech Note:

You can fillet or chamfer smart walls only if you first ungroup them. Since smart walls will lose their smart features by ungrouping them, you should place all symbols before you fillet any smart wall.

- Click the **2-Entity Fillet** tool.



- Type **1** in the Status Line for the radius of the fillet.
- Click on the two interior lines of the walls to fillet the two ungrouped walls.
- Type **1.2** in the Status Line for the radius of the fillet for the outer two lines.
- Click on the two exterior lines of the walls to fillet the two ungrouped walls.

The interior walls are filleted.

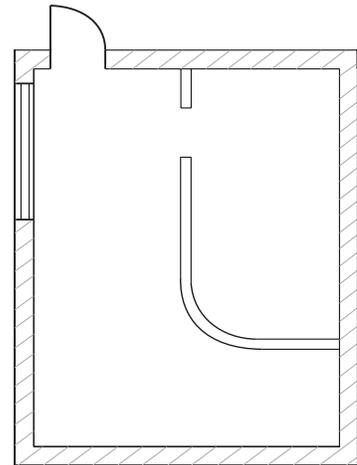
16. Crosshatch the interior walls.

- Select the upper vertical interior wall segment and all lines building the filleted interior walls.
- Choose **Pen>Crosshatch**.

The Crosshatch dialog box appears.

- Enter **-45°** for the hatch angle.
- Click OK.

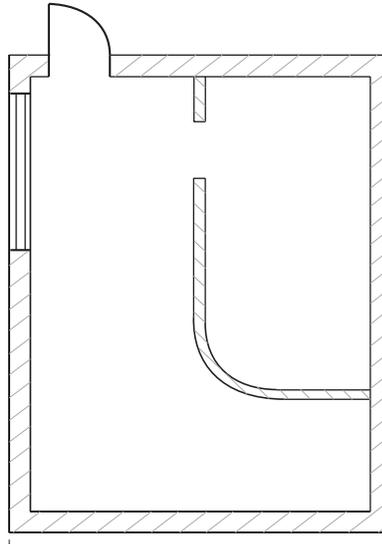
The interior walls are crosshatched.



17. Dimension the sides of the building.
18. Save or discard the file, as you wish.

Accomplishments

- Using the **Smart Wall** tool
- Using smart window and door symbols
- Filleting walls
- Crosshatching walls
- Scaling a drawing



Drafting Techniques

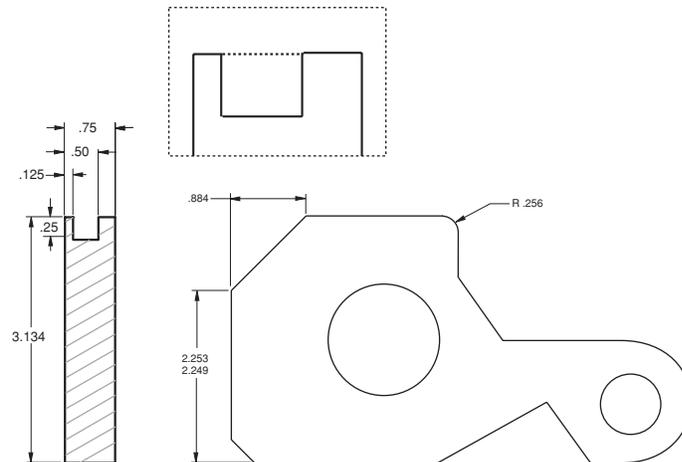
In this section, you will investigate Graphite's advanced drafting procedures, including detail views, drawing to scale and GD&T. In these exercises you will:

- Create a detail view
- Observe associativity
- Scale the drawing
- Import a drawing format
- Create a GD&T label

Exercise 1: Creating a Detail View

In this exercise, you will create a *Detail View*. A detail view is an enlarged or reduced view of part of the drawing. Detail views are dynamically linked to the part

from which they are derived, and modifications made to the part are also made to the detail view. This functionality is called associativity.



1. Create a detail view of the notch from Exercise 8 in the last chapter of the tutorial, using a 2-to-1 ratio.
 - Open the file named *part1a*.

- Click the **Detail View** tool on the **View Control** subpalette. The Message Line reads, *Detail View: Enter view scale then pick first corner of viewing frame.*

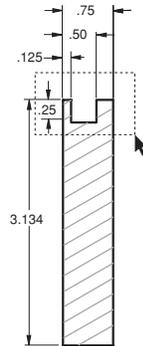


The Scale data field highlights in the Status Line.

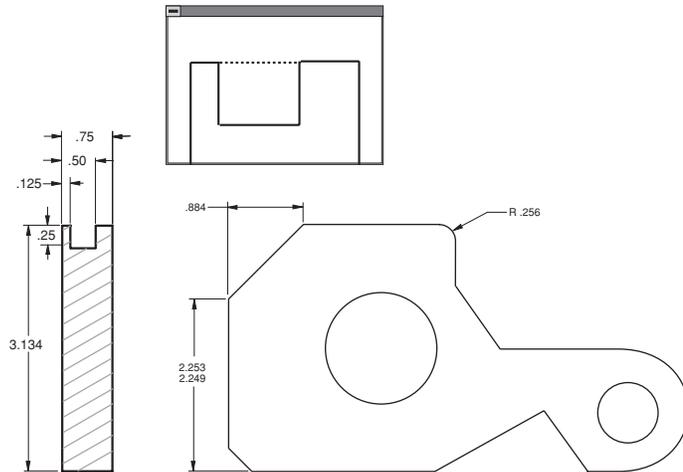
Scale

- Type **2**.

- Drag a frame around the top of the side view, including the notch.



- Drag the frame to a new location.



Tech Note:

Note that the detail view is not dimensioned. Cross-hatching, dimensions and text are view dependent—they appear only in the view in which they were created according to standard drafting practice.

The enlarged geometry appears in the detail view frame.

2. Fillet the right corner of the notch in the detail view and observe that the geometry in the frame is, indeed, a view of the same part.

- Click the **2-Entity Fillet** tool.

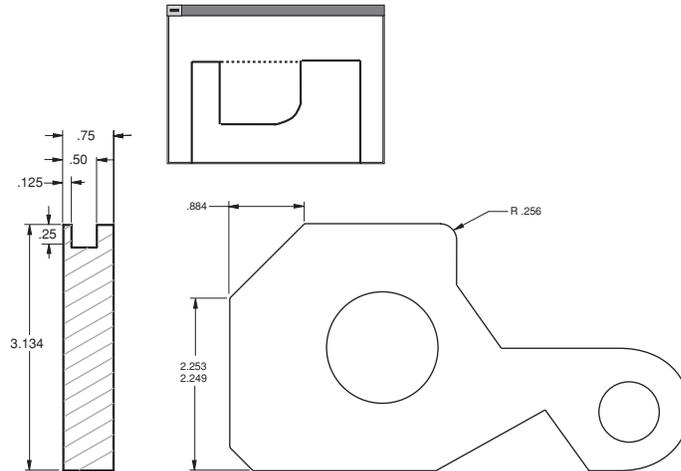


- Enter **.125**.

- Select the two lines that form the lower right corner of the notch in the detail view.

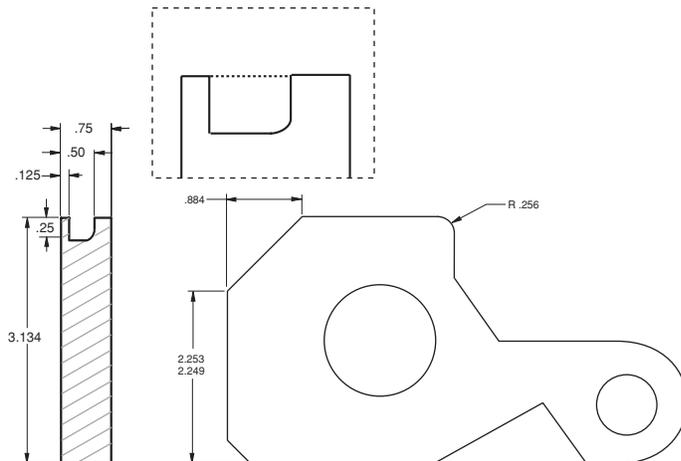
Referral:

If you want to know more about working with views, see Chapter 13, "Viewing Geometry," in the *User Guide*.



Both the part and the detail view are redrawn with the fillet.

3. Crosshatch the side view.
 - Deselect the view by clicking outside the view with the **Selection** tool.
 - Crosshatch the side view with the Steel crosshatch pattern.



The side view is crosshatched but the detail view is not.

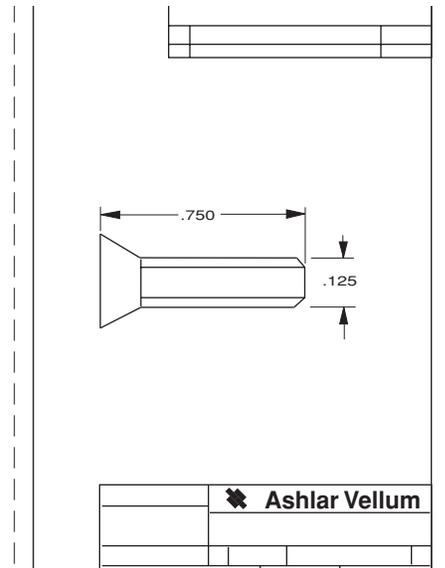
4. Save or discard the file, as you wish.

Accomplishments

- Creating a detail view
- Observing associativity

Exercise 2: Drawing to Scale

In this exercise you will use the Symbol feature to display a full-scale part. Then you will dimension and scale the part for placement on a standard drawing format. This exercise uses a simple part to demonstrate the procedure you should use to construct more complex full-scale parts. Drawing at full scale has several advantages, the most important of which is that you can start your work without having to specify paper formats or other parameters which would only distract from the design task.

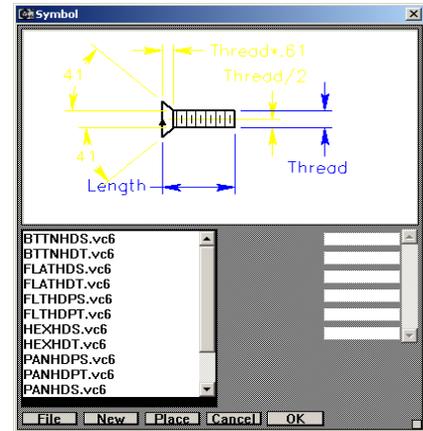


1. Draw a side view of a Phillips-head screw which is .75 inch long and has a thread diameter of .125 inch.
 - Choose **File>Symbol>Insert**.
 - The standard files dialog box appears.
 - Change to the Symbols folder.
 - Change to the *fastenr* (Windows) or *Fasteners* (Macintosh) folder.

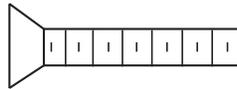
- Change to the *mchscrw* (Windows) or *Machine screws* (Macintosh) folder.
- Open the *flthdps.vlm* (Windows) or *Flat head-Phillips-side* (Macintosh) symbol document.

The Symbol dialog panel appears so that you can browse the various types of screws until you chose one and specify the measurements of the screw.

- Type **0.75** for the *Length* and press the TAB key.
- Type **0.125** for the *Thread* and click OK.



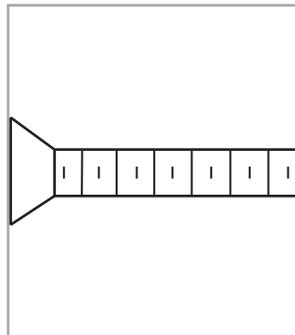
The screw appears.



2. Set the specifications for the paper size you will be using.
 - Choose **File>Print Setup** (Windows) or **Page Setup** (Macintosh).
 - Select the paper size, *size A* for a plotter or *US Letter* size for a laser printer.
3. Set the visual scale of the geometry.
 - Choose **Layout>Drawing Size**.
 - Click Fit.

Tech Note:

The scale is approximately 10:1 but the geometry is visually too large for the page, so you should reduce the scale to 6:1.



- Enter a scale of **6:1** and click OK.

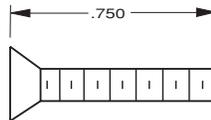
The size of the geometry is more appropriate for the size of the page boundary.

- Since the geometry looks like the right size for the page, click OK.

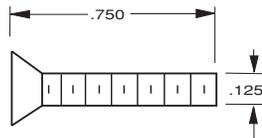
4. Dimension the screw.

- Choose **Arrange>Zoom All** to display the screw at a larger size. This does not effect the actual measurement.
- Choose the **Horizontal** Dimension tool.
- Click the opposite ends of the screw.

The dimension displays



- Dimension the diameter of the thread.



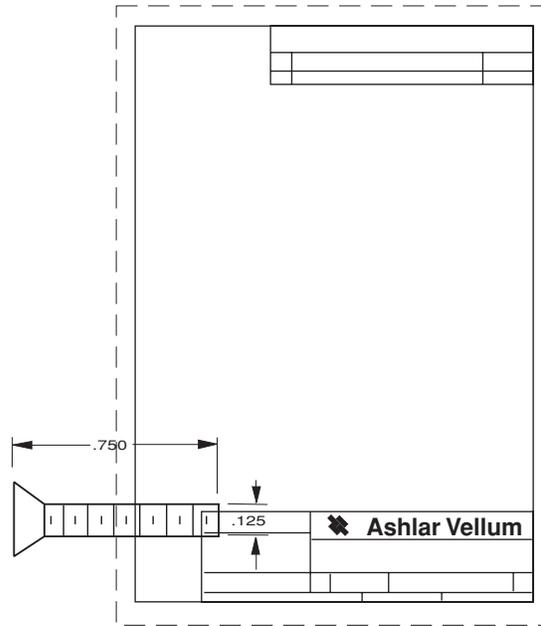
5. Place the scaled drawing into a standard A-size drawing format.

- Choose **Arrange>Zoom Out**.
- Choose **File>Import**.
- Open the Samples2D folder in the Samples folder.
- Click the *aFormatA* file in Samples2D folder.

Tip:

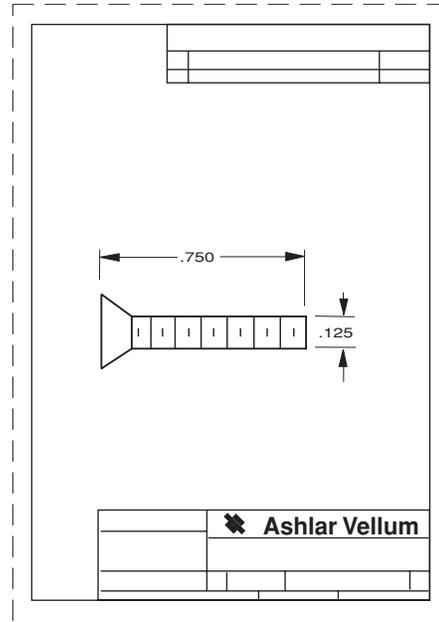
Zooming a part does not change its measurements, it magnifies or reduces it to display the specified portion on the screen. In this way you can draw a part full scale, whether it is 100 feet long or 5 millimeters wide.

- Click *Unscale* and OK to import the unscaled drawing format.



The format is imported to the sheet and is selected.

- Drag the format until the screw is properly positioned on it.
6. Plot the drawing.
 - If your plotter or printer is set up and if you want a copy of this drawing, choose **File>Print**.
 - The drawing is plotted.
 7. Save or discard the file, as you wish.

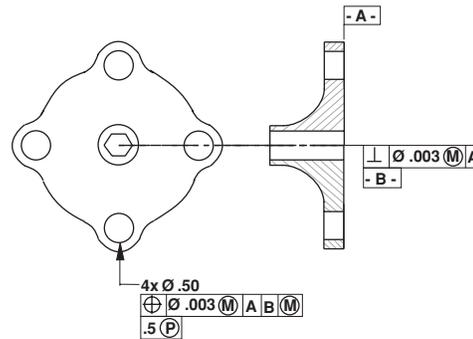


Accomplishments

- Using a symbol
- Scaling a drawing
- Importing a drawing format

Exercise 3: GD&T

In this exercise, you will add a GD&T label and feature control frames to the flange construction from Chapter 4, “Additional Features.”



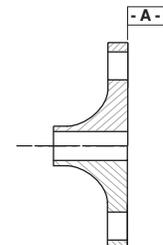
1. Open the flange document showing the flange and the side view.
2. Create a datum label A for the upper-right corner of the side view.
 - Choose **Dimension>GD&T**.
 - In the *Datum* field, type **A**.
 - Select the *Witness Line* option.

Tip:

Move the GD&T dialog box if necessary.



- Click the upper-right corner of the side view to indicate the geometry to attach the witness line to.
- Click a location on the Drafting Assistant construction line above the corner to place the GD&T label.

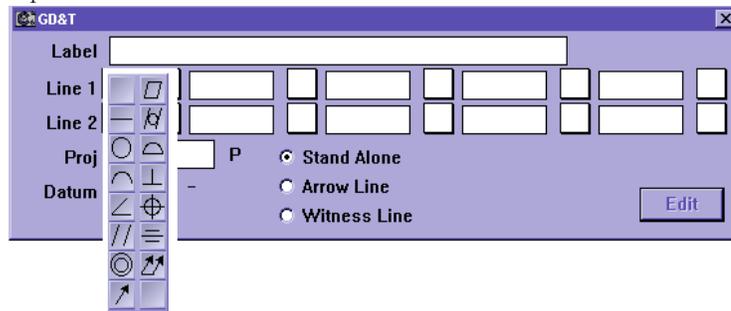


Tech Note:

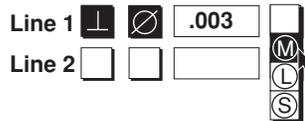
All GD&T text uses one of the Plotter fonts so that the sizes meet ANSI standards.

3. Create a feature control frame for perpendicularity for the hub of the side view.

- On *Line 1*, in the first field, select the *Perpendicularity* (\perp) symbol from the pop-up menu.

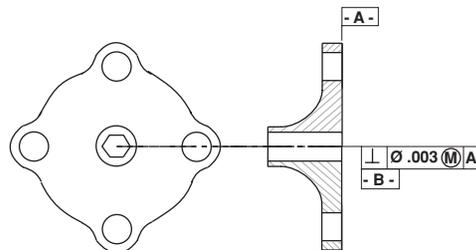


- On *Line 1*, in the second field, select the *Diameter* (\varnothing) symbol from the pop-up menu.
- On *Line 1*, in the third field, enter **.003**.



- On *Line 1*, in the fourth field, select the *Maximum Material Condition* (\textcircled{M}) symbol from the menu.
- On *Line 1*, in the fifth field, enter **A**.

- Change the entry to **B** in the lower datum line.
- Specify *Witness Line* if it is not already specified.
- Click the *endpoint* of the bottom of the hub in the side view to indicate the position for the witness line.
- Click on the Drafting Assistant's horizontal construction line to indicate the position of the feature control frame.



- Create a feature control frame for true position for the lower lug of the flange.

- In the *Label* line of the GD&T dialog box, enter **4X Ø .50** to show that the diameter applies to four places.

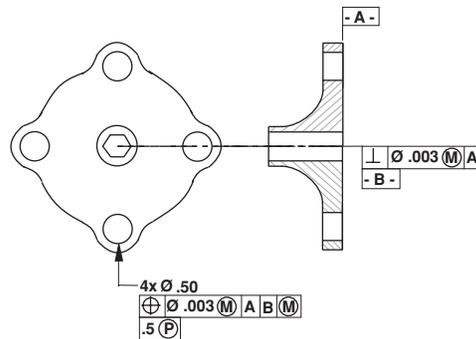
To get the Ø symbol for Windows press the ALT key and type **0216** on the number keypad of the extended 101 keyboard. To get the Ø symbol for the Macintosh, press SHIFT+OPTION+O.

- On *Line 1*, in the first field, select the *X* at the bottom-right of the pop-up menu to clear the entries in that line.
- Select the *True Position* symbol from the menu.



- On *Line 1*, in the second field, select the *Diameter* (Ø) symbol from the menu.
- On *Line 1*, in the third field, enter **.003**.
- On *Line 1*, in the fourth field, select the *Maximum Material Condition* (Ⓜ) symbol from the menu.
- On *Line 1*, in the fifth field, enter **A**.
- Skip the sixth field.
- In the seventh field, enter **B**.
- In the eighth field, select the *Maximum Material Condition* (Ⓜ) symbol from the menu.
- Enter **.5** in the *PROJ* (Projection) field.
- Delete the entry in the lower *Datum* line.
- Specify *Arrow Line* if it is not already specified.

- Drag from the hole of the bottom lug on the flange straight down to indicate the position of the arrow and the label.



5. Edit the second feature control frame.
 - Select the second frame you created.
 - Choose **Dimension>GD&T**.
 - Change the .003 entry to **.005**.
 - Click Edit.

The text changes in the frame.

6. Save or discard the file, as you wish.

Accomplishments

- Creating a GD&T label
- Editing a feature control frame

Constructing a Wedge Die

Up to now, all the exercises have been two dimensional. This exercise introduces you to the third dimension. You will discover how easy it is to use. All of the tools used in 2D space are used in 3D.

In this chapter, you will construct a Wedge Die part and then create a 2D drawing with four views (Top, Front, Right and Trimetric).

Since working in Graphite is a smooth transition from 2D to 3D, you can start the drawing in 2D.

Creating the Wireframe Geometry

1. Begin with a new document and draw a rectangle. The rectangle you will end up with is about four times as tall as it is wide. It should be almost as tall as the screen.
 - Select the **Connected Lines** tool.

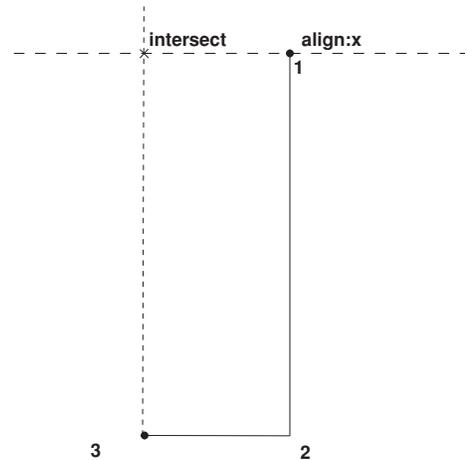


Constructing a Wedge Die

- Draw the first two lines by clicking at points 1, 2 and 3.

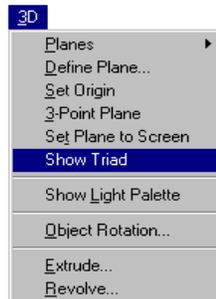
Move up and show the *intersect*, but don't click this point yet.

2. Display the Triad and change to a Trimetric view.
- Choose **3D>Show Triad**.



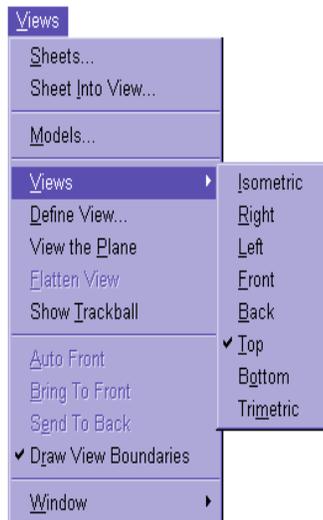
Tip:

As you have seen before, the Drafting Assistant gives you the alignments you need to complete the shape.



This command displays an orientation marker in the corner of the screen.

- Choose **Views>Views>Trimetric**.

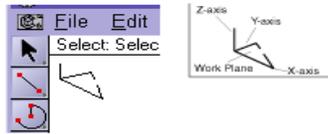


Tech Note:

When you start Graphite the Top view is automatically selected.

When you began drawing, you were in the Top view. Now, the triad has

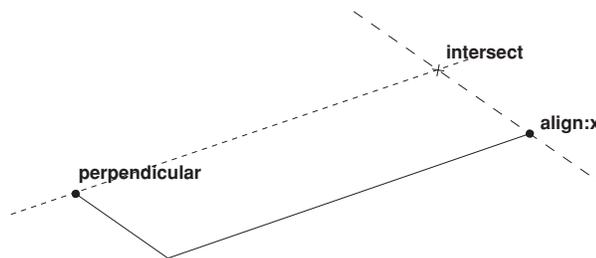
changed to show a new orientation.



The longest of the three lines is always the x-axis, the next longest is always the y-axis and the shortest line is always the z-axis.

The broken line forms a triangle that indicates where the work plane is. In this case, it is flat on the bottom, between the x- and y-axis. The rectangle has tilted down to match the triad.

Now you can line up the same *intersect* point as before, but now you are looking at it from a 3D orientation.



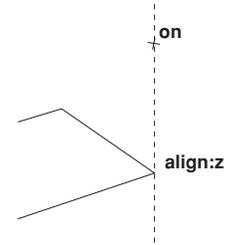
- Click the final two points, to close out the rectangle but **don't** double-click at the last point. You want to continue with the **Connected Lines** tool, and you want the rectangle to remain highlighted so the Drafting Assistant can align with it as you continue.
- Now click on the Scroll Up arrow a few times, to move the rectangle near the bottom of the screen.



3. Use the Z-Drafting Assistant to draw in the Trimetric view.

Constructing a Wedge Die

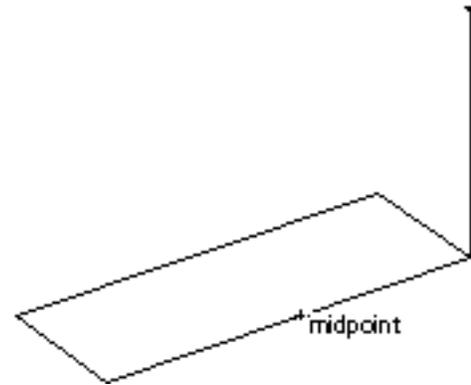
- Position the cursor above the first point of the rectangle, without clicking, to show the Z-Drafting Assistant.



Tip:

Here you see the first of many unique features of Graphite. This is the Z-Drafting Assistant. In other words, the Drafting Assistant is working in the third dimension. This provides you with automatic alignments off the x-axis, the y-axis and the z-axis.

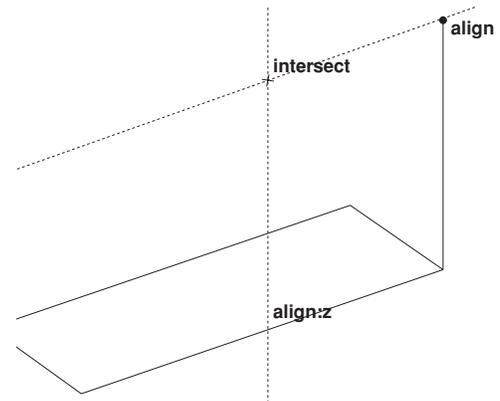
- Move up along the z-construction line and click the end point so the line is about 2/3 as tall as the rectangle is long. Then, without clicking, go down and touch the *midpoint* of the line, as shown in the following graphic.



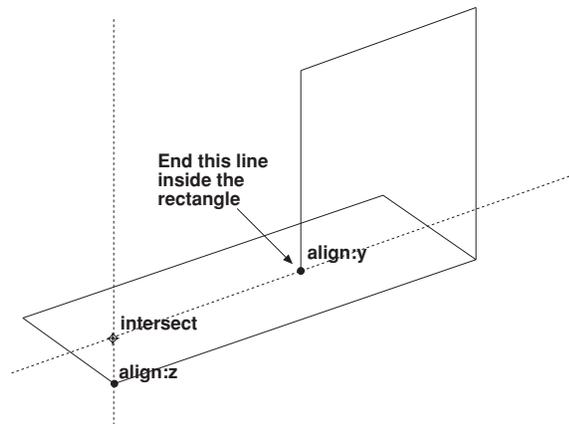
Tip:

After the Drafting Assistant displays the *intersect* you can begin moving around in 3D space and create this block by just sketching, and letting the Drafting Assistant find the alignments.

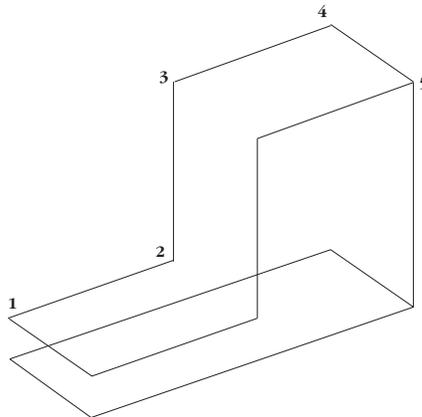
- Come up the z-construction line, and locate the intersect shown below.



- Click the *intersect* shown above. Then, move down along the z-axis, and end this line somewhere inside the rectangle as shown. Then move to the left along the y-axis and locate the *intersect* shown here. Click this point.



- Continue with the **Connected Lines** tool, and pick the five points shown here. Make sure you use the Drafting Assistant construction lines to get the proper alignments. When you get to point 5, double-click this point to end the line.



Tip:

As you pick each of these points, notice how the Drafting Assistant senses where you are and what other geometry you are near. It automatically provides you with all the alignment construction lines you need to draw this part.

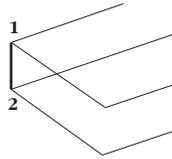
What you are seeing here is one of the unique features of Graphite; the ability to move around freely in 3D space and just pick points, almost as if you were sketching. But having those points end up precisely aligned in all three axes is something that really sets Graphite apart.

4. Complete the shape of the wedge by moving copies of the selected lines.

- Select the **Single Line** tool.



- Draw a single line from point 1 to point 2.



- Select the **Selection** tool.

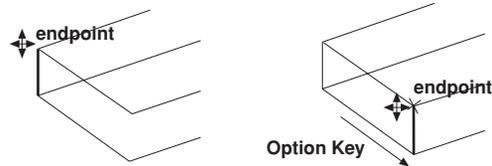


- With the line still selected, place the pointer at point 1. Hold down the CTRL (Windows) or OPTION (Macintosh) key and drag a copy of that line to the endpoint of the top line, as shown on the next page.

Notice that the Drafting Assistant helps you by automatically snapping to the endpoints at the start and the end of your move.

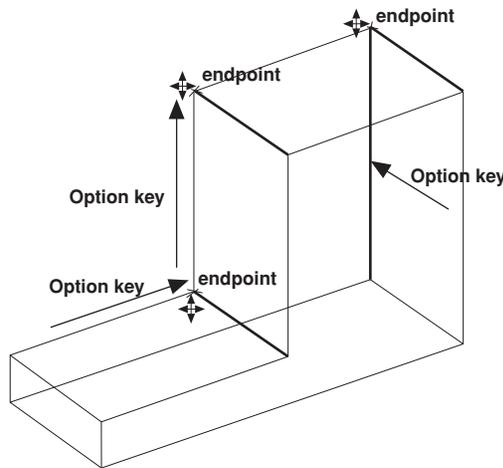
Tech Note:

Notice that when the pointer gets near the selected line, it changes from the **Selection** tool to the 4-Way Move symbol. This allows you to move the selected item(s), as well as make copies while you move it.



- With the **Selection** tool, select the line shown below.

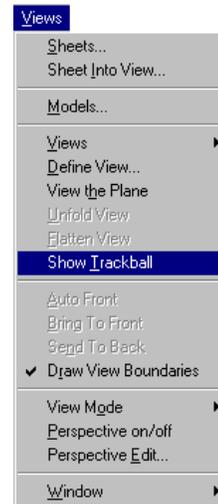
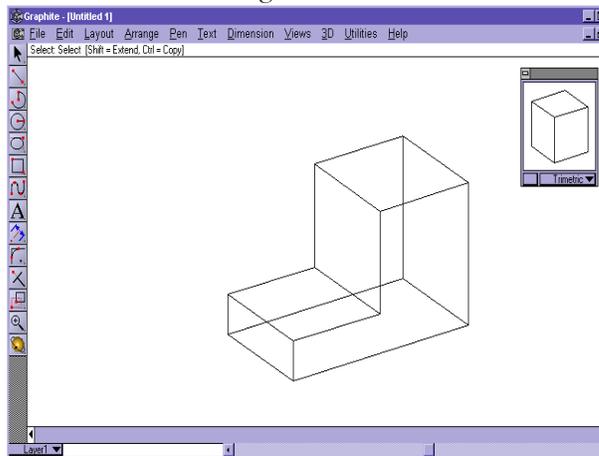
- As before, while the line is still selected, place the pointer at one endpoint, hold down the CTRL (Windows) or the OPTION (Macintosh) key and drag a copy of the line in the y-direction, placing it at the corner of the L. Drag another copy in the z-direction and drop it at the top of the block. Select the long vertical line on the far right side of the part and move a copy of it, as shown. Now, all the lines of the L block should be drawn.



Now you have completed the outline of the Wedge Die. Because of the Drafting Assistant, this process was fast and accurate.

- If you have any doubt if this is truly a 3D object, you can check it with the Trackball.
 - Choose **Views>Show Trackball**.

The Trackball/TrackCube comes up on the left side of the screen. For the purposes of this tutorial, it works best to have it on the right, so grab it by its title bar, and move it over to the right.



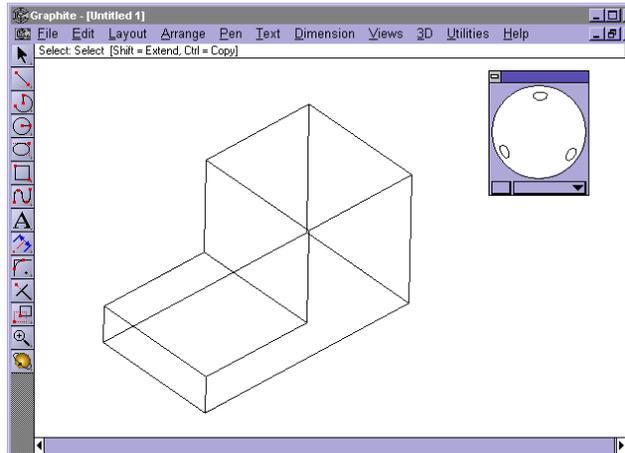
Tip:

You can do the same thing with some of the other lines. You could, if you wanted to, draw these lines using the **Single Line** tool, but sometimes it's easier to copy existing lines.

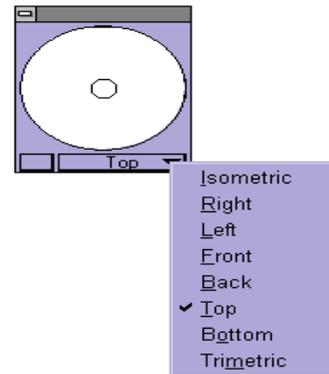
Tip:

With the Trackball/TrackCube you can rotate your model just by clicking and rolling the ball. The model rotates in real time. Keep in mind, however, that you are simply changing your point of view; the geometry itself remains fixed in model space.

- Rotate the model using the Trackball/TrackCube. Notice that the Triad appears at the center of rotation and allows you to see the orientation of the axes as the model is turning.

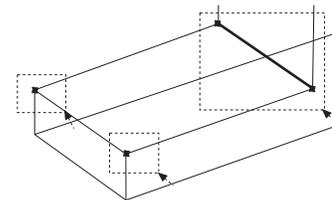


6. Change the view orientation.
 - You can choose any view, such as Right or Top, by selecting each view from the pull-down menu on the bottom panel of the Trackball.

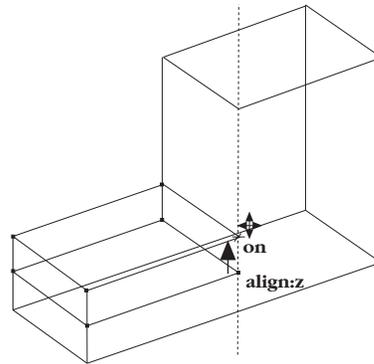


To continue with the tutorial, select Trimetric from the pop-up menu on the Trackball.

7. Modify your design intuitively in 3D.
 - Hold down the SHIFT key and drag three selection fences around the four points shown to select these points. (The one short line in the big fence will also be selected.)
 - Release the SHIFT key. Grab one of the points (the pointer will change to the Four Way Move cursor) and drag it vertically. Make sure you see the words *align:z* at the bottom point as you start to move upward.



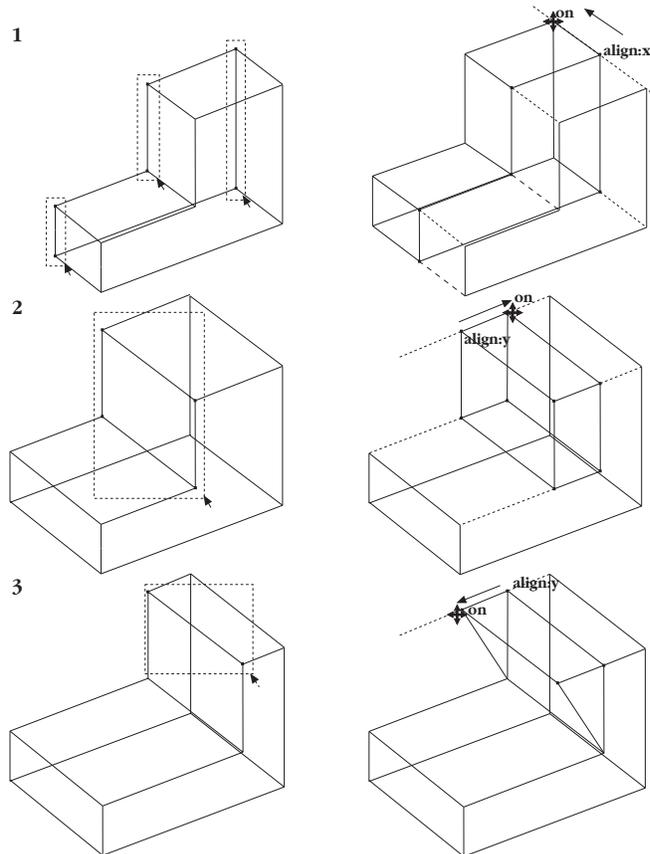
Make the bottom block approximately twice as thick as it was originally.



Tip:

One of the great features of Graphite is the ability to modify your design easily in 3D. For example, you might decide to make this bottom area about twice as thick as it is now. You can select these points, and then move this face straight up until it is where you want it. The Drafting Assistant allows you to constrain this movement along the z-axis.

- In each of the examples below, use the SHIFT key and drag the selection fences shown to select the appropriate points. Then drag one of the points to stretch the part. (You can skip this exercise if you want.)

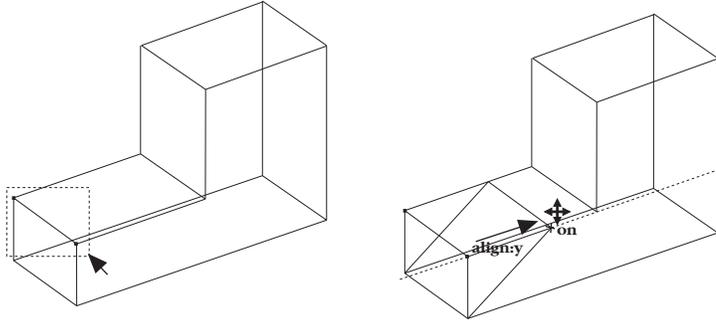


Tech Note:

Graphite has unlimited levels of *Undo*. You can specify the number of Undos when you right mouse click in empty screen space, select *Undo...* and specify the number you wish.

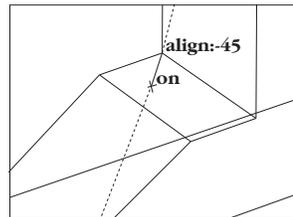
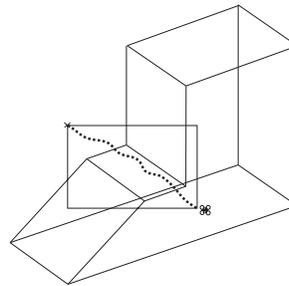
- Choose *Undo* three times to get the part back to how it looked before the stretch exercises on the previous page (skip this if you skipped those exercises). The bottom of the block should be twice as thick as when you originally drew it.
- Regardless of whether you did the steps on the previous page, make sure you do the next one. This step is part of the actual shape of the finished part. Select the two points shown in the graphic and stretch the shelf back along

the y-axis (stretch it two-thirds of the way back as pictured here). As part of the design, you will create a wedge-shaped front.



8. Put a molded face on the front.

- Do a Stroke-Zoom. Hold down the CTRL+SHIFT keys (Windows) or the ⌘ key (Macintosh), and drag the rectangle shown, from upper left to lower right.
- Pick the **Connected Lines** tool and draw the line shown, going about two-thirds of the way across the width of the face. Make sure you see the words *align:-45* and *on* before you click the second point.

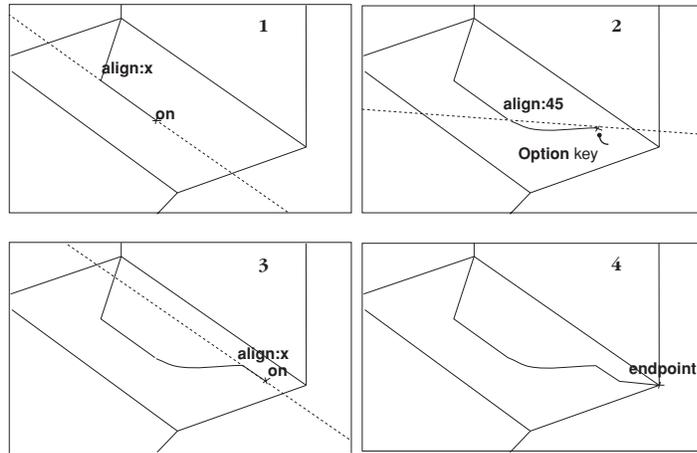


- Finish the contour by drawing the lines and the arc, as shown. In frame 2, hold down the CTRL (Windows) or OPTION (Macintosh) key to temporarily

get the **Arc** tool. Make sure that you see the words *align:45* and *on* before you click the second point.

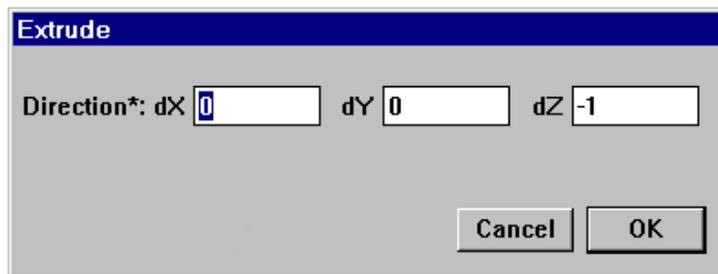
Tip:

By using the Drafting Assistant construction lines, you can draw these lines easily. If you hold the CTRL (Windows) or OPTION (Macintosh) key, you temporarily get the **Arc** tool.



9. Finish the contour by first extruding it up, then by revolving it around the corner of the block, and finally by extruding it along the top face of the block.
 - Select **Arrange>Zoom All**.
 - Select **Views>Hide Trackball**.
 - Select the five lines of the contour that you just drew (if they are not already selected).
 - Select **3D>Extrude**.

The Extrude dialog box appears.



Tip:

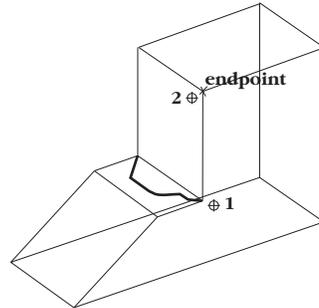
The Extrude dialog box can be moved by grabbing its title bar and dragging it to a new location.

You will now see one of the most important features in Graphite. You have to enter a set of numbers in the three data fields. You know you want to extrude along the vertical line of the face, but you do not know how far it is, and you

do not know what direction it is.

With Graphite, this is easy. Notice the asterisk in the dialog box. Whenever you see an asterisk it means, “you can use the cursor to enter values.”

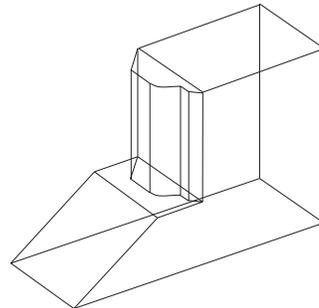
- Put the bulls-eye cursor on the bottom of the vertical line of the face and drag up to the top of the line, then release.



A number equal to the distance of that drag will appear in the dz data field, replacing the -1 that was there.

- Click OK.

The extruded objects appear.

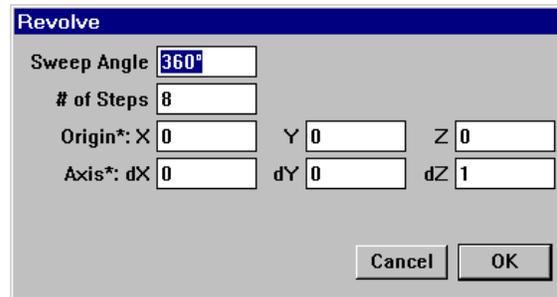


- Now select the same five contour objects, but select the ones at the top of the extrusion. You are about to revolve these objects around the corner of the block.
- Select **3D>Revolve**.

Tech Note:

One of the important features that is unique to Graphite is the ability to drag from one point to another, and have the values appear in the right data fields.

The Revolve dialog box appears.



Now you will take this contour and sweep it 90 degrees, using the corner of the block as the axis of revolution. Knowing this, you can type **90** into the *Sweep Angle* data field. Use 4 divisions for the *# of Steps*. You probably will have no idea how to fill in the remaining six data fields. But there is an asterisk again, which means that, “you can use the cursor to enter values.” All you have to know is that you want to define the axis of revolution to be from point 1 to point 2.

Tech Note:

You indicate how many copies should be placed within the *Sweep Angle* with the *# of Steps*.

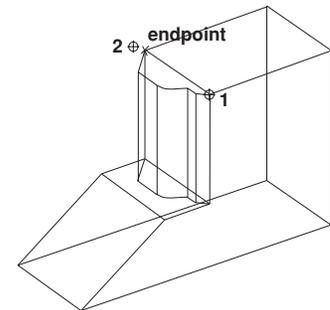
Tip:

You can also click in one of the data fields. Make sure that one of the six lower data fields is active.

- Type **90** in the *Sweep Angle* data field.
- Type **4** in the *# of Steps* data field.
- Press TAB to move the cursor into one of the data fields for the *Origin*.
- Define the Axis of Revolution by dragging from endpoint 1 to endpoint 2 as illustrated below. If you drag in the opposite direction, the contour will sweep in the opposite direction.

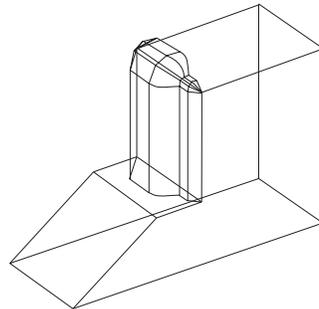
When you release the mouse, the data fields for *Origin* and *Axis* fill with a series of numbers.

Graphite automatically filled in the numbers for the axis and direction of revolution.

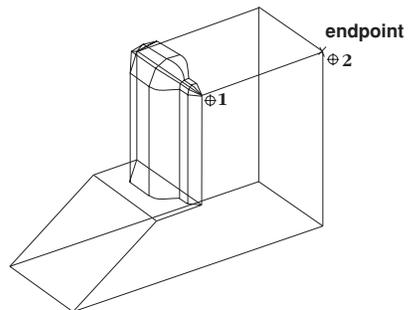


- Click OK.

Graphite builds the sweep.

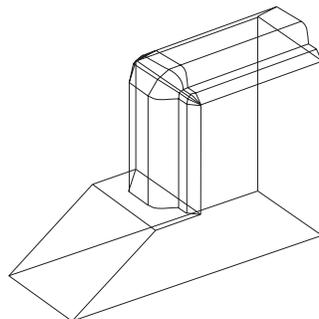


- Select the five contour objects at the end of the sweep.
- Select **3D>Extrude**.
- To specify the length and direction for the extrusion, drag from endpoint 1 to endpoint 2 as shown below.



- Click OK.

The contoured face is finished.

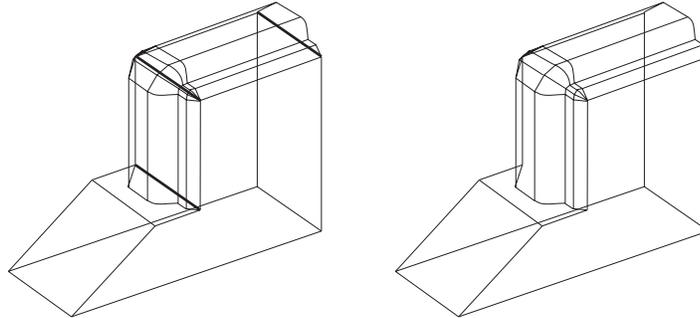


10. The extrude operation has created some lines that you do not need. Select these lines and delete them to clean up your drawing.

Tip:

At this point it might be a good idea to get the Trackball out and rotate the part to see it from various angles.

- Select the three lines shown by SHIFT-Clicking on them, and then delete them.

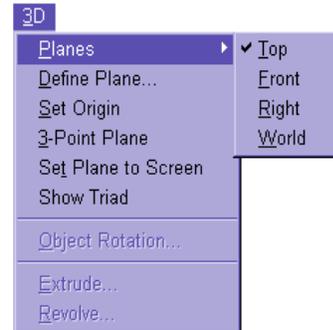


11. The next step is to put a hole on the front face of the wedge and extrude it through the part. By doing so, you will learn about the concept of Work Planes.

Right now, you can see by looking at the Triad that the Work Plane is parallel to the bottom of the Wedge Die. If you were to draw a circle, it would be drawn in that orientation. Try that now. What you want is a circle that stands up along the right face. You must change the work plane to accomplish this.

- Select **3D>Planes>Right**.

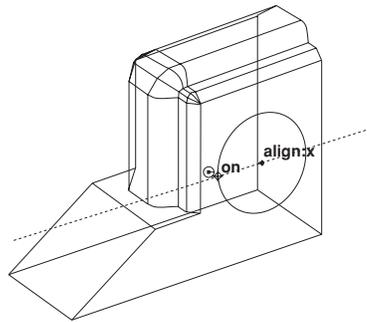
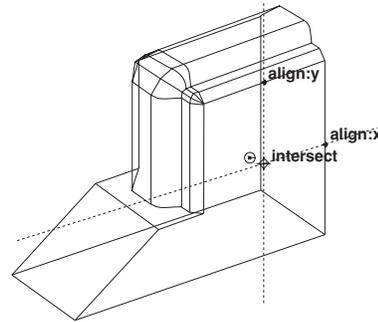
Notice that prior to releasing the mouse button the current Plane was Top or World (indicated by a check mark). Once you release the mouse button, the Triad changes to reflect the new work plane.



- Select the **Center-Point Circle** tool.



- Touch the *midpoint* of both the right side and the top edge of the front face to “wake up” these points. The Drafting Assistant now gives you construction lines off of each point.
- Go to where the Drafting Assistant shows the *intersect* of the two alignments and drag the circle along the face (the center of the circle is placed at the *intersect*). Release the mouse button when the circle is approximately the size shown.



- While the circle is still active, select **Arrange>Divide**.

The Divide dialog box appears.

- Make # of pieces equal to **4**, and deselect the *Show Points* option.
- Click OK.

The circle is now divided into 4 arcs. The circle will flash to let you know that the divide operation executed.

- Select **3D>Extrude**.
- Drag from endpoint 1 to endpoint 2.

Tip:

You can use the Trackball to rotate the part to make it easier to see the *midpoints* of the lines on the right side face.

Tip:

If you want to eventually surface this geometry, there is no need to divide the circle. However, this exercise includes these steps to introduce the *Divide* command.

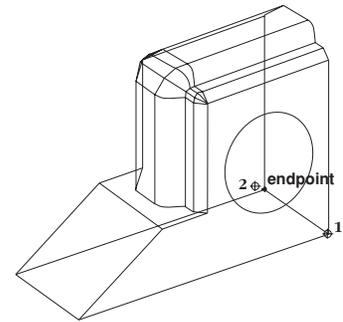
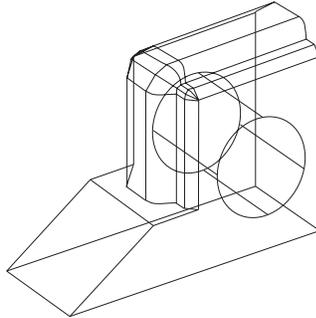
Tech Note:

Dividing the circle results in four extrusion lines (instead of one). The extra extrusion lines make it easier to visualize the extrusion when working in wireframe mode.

This defines the distance and direction you want the circle to be extruded.

- Click OK.

The circle extrudes with four extrusion lines.



12. The next step is to put a hexagon on the sloped face. This is a good exercise because it shows how Graphite deals with non-orthogonal planes. You might guess that in order to draw anything directly on a sloped face, you need to set the work plane to match it. When you go to the **3D>Planes**, none of the choices are appropriate (Top, Front, Right and World). For this operation you want to define a new plane.

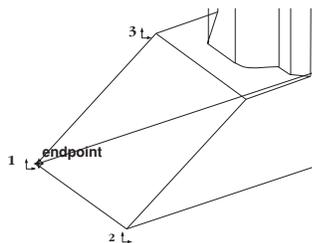
- Choose **3D>3-Point Plane**.

The Message Line asks for three specific things, first the origin, then the positive x direction, finally the positive y-direction.

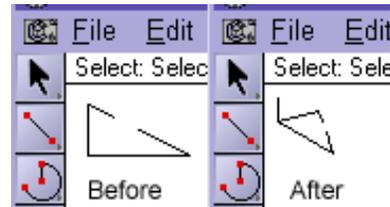
- Click first at endpoint 1 for the origin, then at endpoint 2 for the positive x-direction, and then at endpoint 3 for the positive y-direction as shown in the drawing below.

Tech Note:

You should only use the *Define Plane* command if you want to use that plane orientation more than once.



The Triad changes to indicate that the current work plane now matches the sloped face.

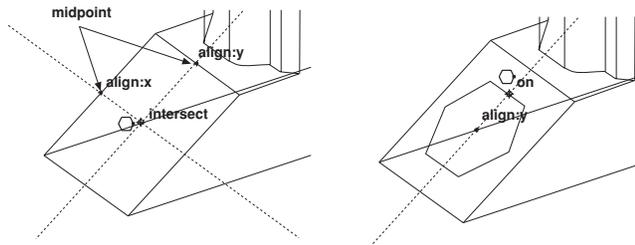


Now that the work plane is the way you want it, you can draw the hexagon.

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



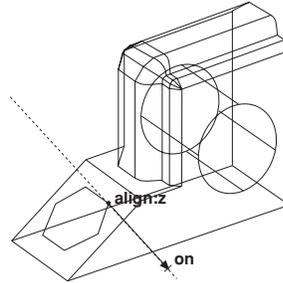
- Touch the two *midpoints* shown to wake them up and let the Drafting Assistant show where the *intersect* is located. This point will be the *center* of the hexagon. Drag the second point straight up, and the hexagon will be placed onto this sloped face. Release the mouse button when the hexagon is similar to the drawing below.



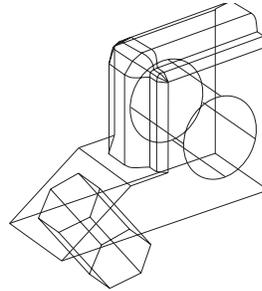
13. Now extrude the hexagon through the part, perpendicular to the face.

- Select **3D>Extrude**.

- Drag one of the points, to below the part along the z-direction. Take the extrusion well below the part and release the mouse button. Make sure you see *align:z* so that the extrusion will be perpendicular to the work plane.



- Click OK.



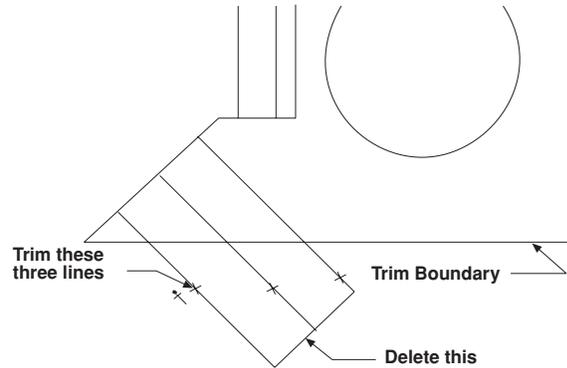
14. You extended the extrusion beyond the base. You will use the **Simple Trim** tool to cut off the excess.
- Change your view to Right (either use the pop-up menu on the Trackball or **Views>Views>Right**).
 - Use the **Selection** tool to select the bottom line of the part. This will act as a trim boundary.
 - Select the **Simple Trim** tool.



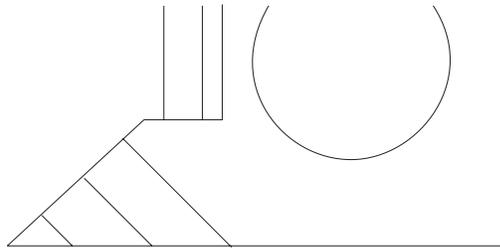
- Click on the extrusion lines that extend below the bottom of the part.

There should be two lines on top of each other in each location, so you will have to click twice on each line.

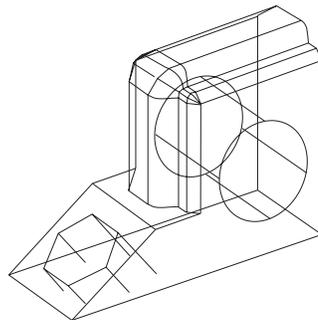
- Select and delete the lower hexagon.



The hexagon trims to the bottom edge of the part.



- Go back to the Trimetric view.



15. Draw the hexagon on the bottom surface of the wedge.

Tech Note:

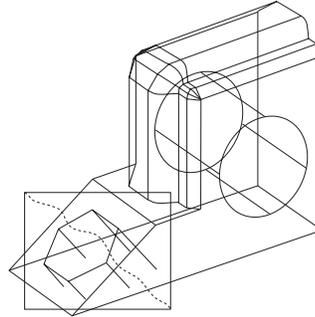
The **Simple Trim** tool works exactly the same as it did in 2D, except now it is working on a 3D part. This is great because it is like trimming to a surface.

Tip:

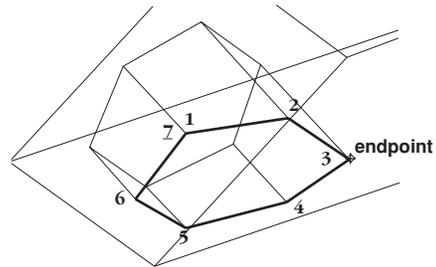
In this case, because of the way the lines came out on the trim, use the Trackball to rotate the model slightly so that you can see all the endpoints more clearly.

Constructing a Wedge Die

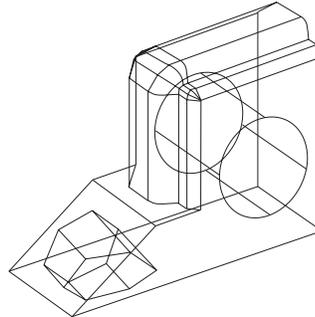
- Execute a Stroke-Zoom. Hold down the CTRL+SHIFT keys (Windows) or the ⌘ key (Macintosh) and drag a diagonal window from upper left to lower right, as shown.



- Using the **Connected Lines** tool, connect each of the six endpoints to finish the hexagon hole along the bottom of the part. Double-click at the last point (point 7) to end the connected line.



- Choose **Arrange>Zoom Previous** to get back to the previous zoom. Hold down the CTRL+SHIFT keys (Windows) or the ⌘ key (Macintosh) and drag a diagonal window from lower right to upper left.



16. Save the view.

- It is a good exercise to save custom views. The 4th view comes up Isometric. Activate it, rotate it slightly and then save the new view. To do this, choose **Views>Define View**. Click New, and then OK. This will create a new view called *View 1*. You can switch back and forth between this and other views. For the purposes of the rest of the tutorial, it is best to switch the work plane back to World and the view back to Trimetric.

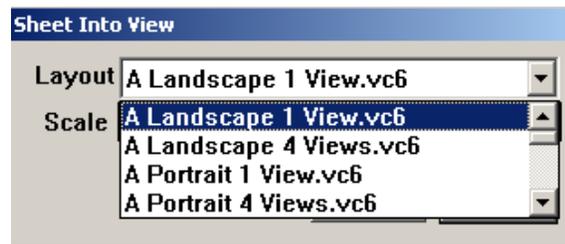
You have finished the Part! But the tutorial is not done yet. So read on.

Creating a 2D Drawing

17. Place this part into a 2D drawing with four views (Top, Front, Right, and Trimetric). In Graphite all it takes is one command.
 - Select **Arrange>Zoom All**. You can leave the part in the Trimetric view.
 - Select **Views>Sheet Into View**.

The following dialog box appears:

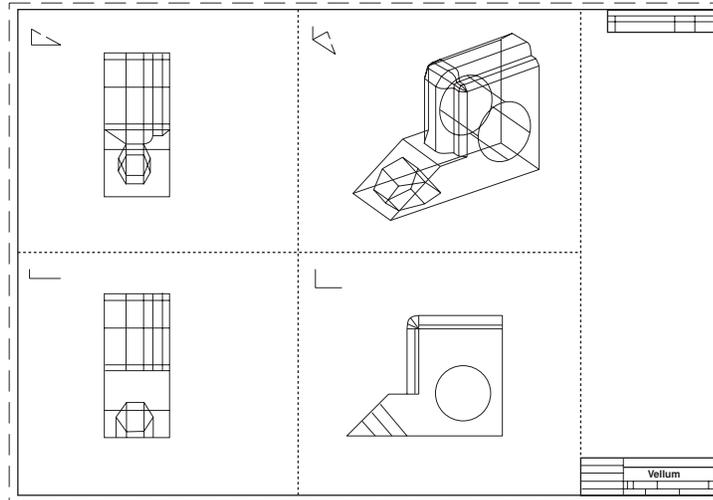
You can specify a layout and viewing scale for the original model. There are many choices in this menu including different variations of view arrangements, surrounded by a border and title block. These choices are user definable.



- If you are working on a 13 or 19 inch monitor, select *A Landscape 4 View.vc6* (Windows and Macintosh). Set the *Scale* to **.75**.
- When you are finished, click OK.
- Select **Arrange>Zoom All**.

With just one command you have gone from a 3D wireframe model to a layout for a 2D drawing. It has the four views, a title block, and a border. It doesn't get

any easier than this.

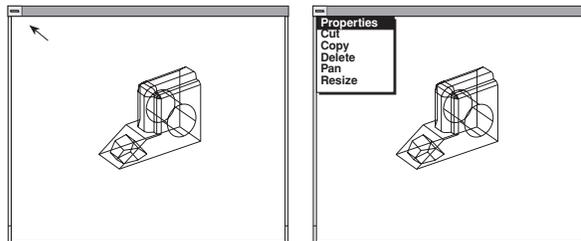


18. Edit the view windows.

If you do not end up with something similar to what is shown in the previous illustration—if you end up with a drawing where the part is too small or too large in each of the views—then you will have to modify the scale of each of the four views.

If your drawing looks fine, then skip the next four steps.

- Activate one of the views by clicking on it.
- Open the pull-down menu from the upper left corner of the view boundary.

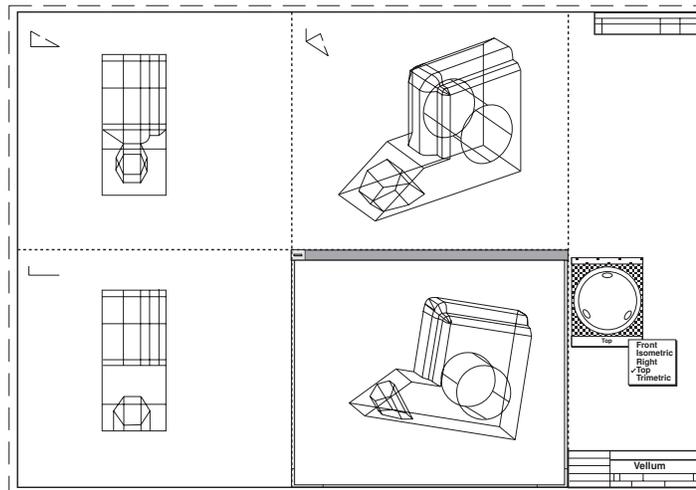


- Select *Properties* from the pull-down menu.
- Modify the scale to make the part show up in the view more clearly. In this case, since the part is too small at .75 scale, perhaps entering a scale of **1.5**

would work better. Whatever number you find that works well, make sure you enter the same value for all four of the views.

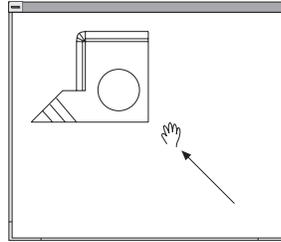
With the Trackball you can rotate any of the views to something nonstandard.

- Select **Views>Show Trackball**.
- Click in the Top view window to make it active. (Its title bar will display.)
- Rotate the model in that view by dragging on the Trackball. You can make any of the four views display any custom view. (Rotate the Right view window next.)

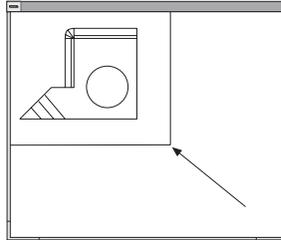


- While the Right view window is still active, pick Right from the Trackball pull-down menu. Click on the Top view window to activate it, and select Top from the Trackball menu to put it back to its orthogonal projection.
- Select **Arrange>Zoom Out** with the SHIFT key (Windows) or CONTROL (Macintosh) pressed down.
- Select *Pan* in the pull-down menu for the Top view window. The cursor changes to a mover hand icon.

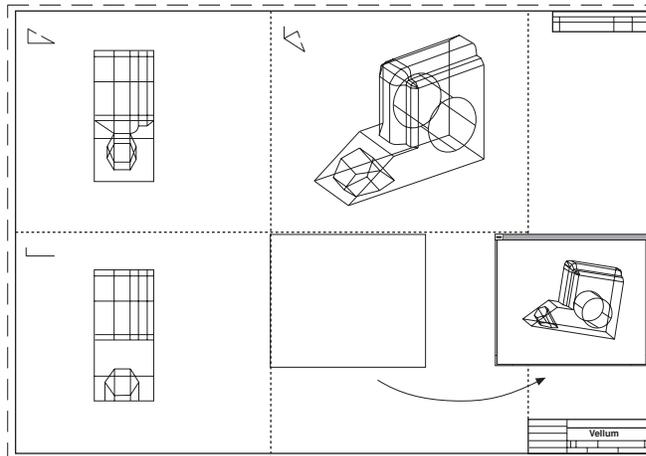
- Drag the wedge to the upper left corner of the view window.



- To finish, drag the lower right corner of the view.
Drag it up, to reduce the size of the view.



- Drag the title bar of the new view and move it into a clean area.



Set the right view to its original size and position.

19. Create an unfolded view.

A common drafting operation is to create a view on the drawing that looks

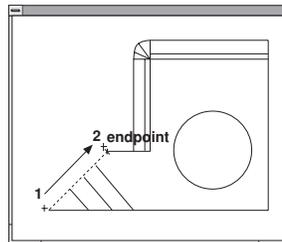
straight at a specified face, much like the Projected View problem from high school drafting class.

In Graphite you simply activate the view, and unfold it.

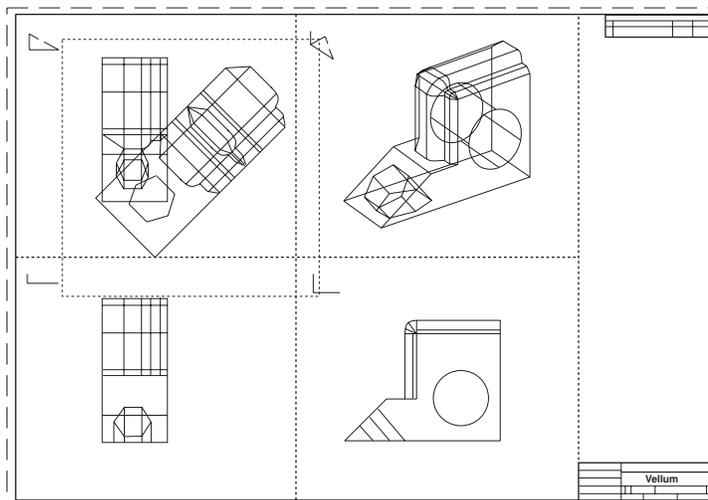
- Click on the Right view window to activate it.
- Choose **Views>Unfold View**.

The message line asks you to, “Pick start of fold line.”

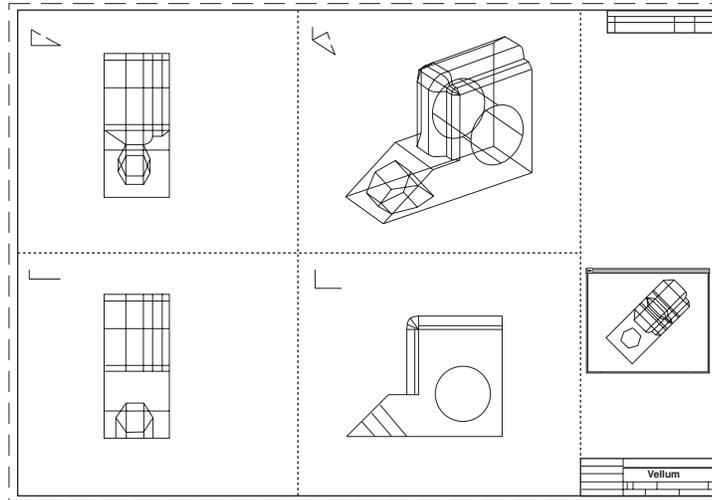
- Drag the mouse button from endpoint 1 to endpoint 2.



- When you release the mouse button, a new view appears on the drawing.
(Note: it may appear below your current drawing, so you may have to scroll down, or *Zoom All* to see it).



- Resize the unfolded view, Drag its title bar and move it to a clean area.



Tech Note:

During dimensioning pay attention to the Triad. The dimensions work only on the current x-y-plane. So a good trick is to pick *Set Plane to Screen* from the 3D menu. You will see that the Triad shows flat in all five views now. All dimensions work as if it were a 2D-drawing.

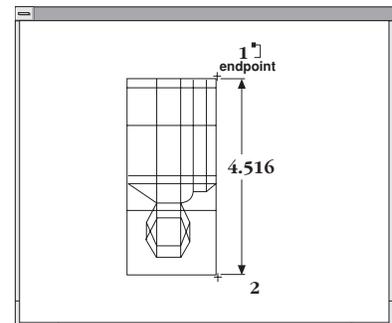
20. Dimension the part.

- Select **Dimension>Show Palette**.

The **Dimension** tool palette appears.

- Select the **Vertical** Dimension tool.
- Click in the Top view window to activate it.
- Click on the right side of the part from top to bottom to make the dimension appear. You can change its font size to make it readable.

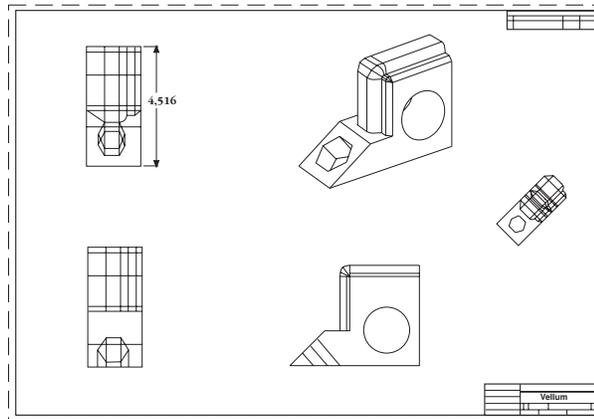
The five views we have here are associative. However, the dimension object we just placed shows up only in the one view, because these views are smart. Dimensions show only in the view in which they are placed. The same is true for Crosshatching and Text.



21. Print the drawing.

- Hide the Trackball, the Triad and the **Dimension** palette.
- Deselect view boundaries (**Views>Draw View Boundaries**).

- Select **File>Print Setup** (Windows) or **Page Setup** (Macintosh) and prepare the page settings for your paper and printer.



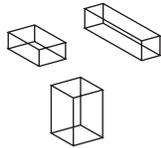
- Use the *Drawing Size* command (if necessary) to scale the drawing to your page size.
- Select **File>Print**.

Accomplishments

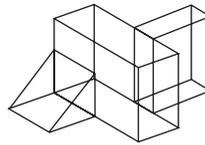
- Creating a 3D Wedge Die
- Creating 2D drawings

More 3D Shapes

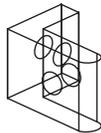
You created a number of objects so far, but only one in 3D. In these exercises you will construct the following models to help you get comfortable creating 3D geometry:



3 ways to construct a box



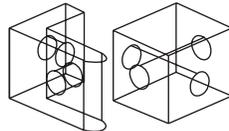
Isometric drawing



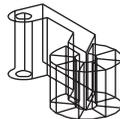
Extruding a bracket



Rotating with the Revolve... command



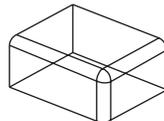
Interlocking parts



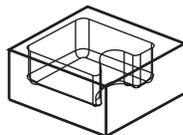
Pipe winch



Cutting an angle



3-corner fillets



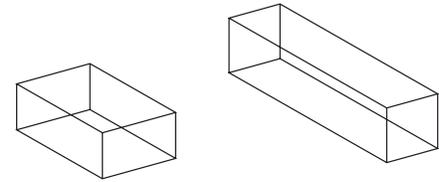
Fillet curves



Compound curves

Exercise 1: Three Ways to Construct a Box

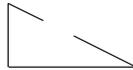
In this exercise, you will use three methods to draw boxes. In each case you will use the Z-Drafting Assistant to draw the figure. 3D drawing is like isometric drafting.



Method I

1. Draw a box using the Z-Drafting Assistant.

- Choose **3D>Show Triad**.



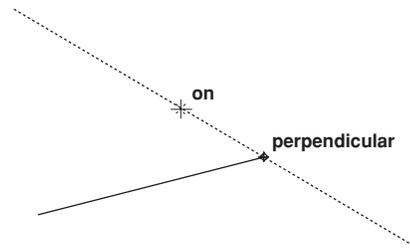
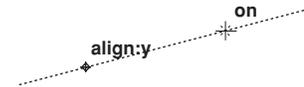
The Triad appears in the upper-left corner of the screen.

- Choose **Views>Views>Trimetric**.

The view rotates to the trimetric orientation.



- In the **Line** tool subpalette, select the **Connected Lines** tool.
- Begin the first line, dragging toward the upper right along the y axis as displayed by the Drafting Assistant.
- Draw the second line segment on the x-axis.



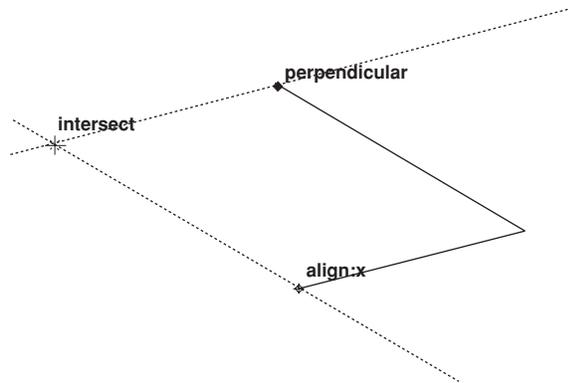
The x-axis displays as *perpendicular* to the y-axis.

Tech Note:

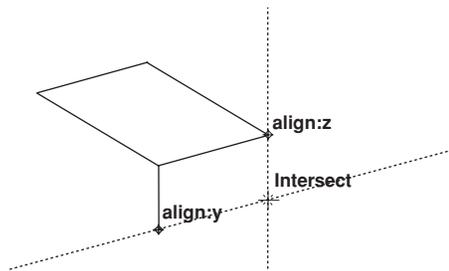
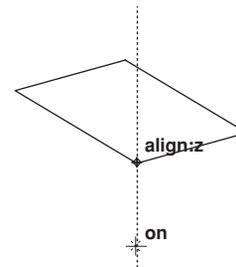
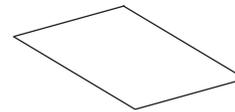
The Triad shows the orientation of the work plane, and since the sheet and the work plane are the same, the Triad shows the orientation you are seeing.

In the Trimetric view with the Top work plane, the Drafting Assistant displays the “vertical” *align:z* notation.

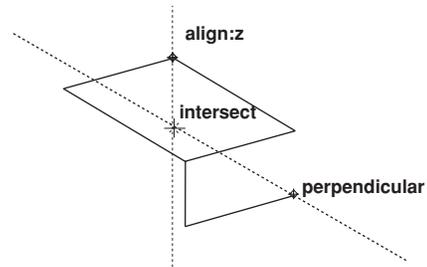
- Draw the next segment to the *intersect* as shown below.



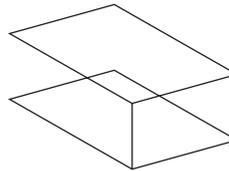
- Complete the rectangle. Don't double-click when the rectangle is complete.
- Extend the rectangle along the *align:z* construction line, clicking the lower corner, as shown.
- Move the pointer to display the *intersect* point as shown, and click.



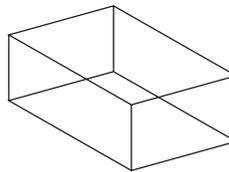
- Click the next *intersect* point as shown.



- Continue drawing, double-clicking when the second rectangle is complete.



- Use the **Single Line** tool to connect the corners of the rectangles.

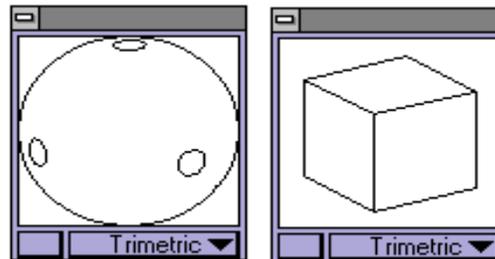


The Trimetric sketch is complete.

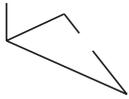
2. Rotate this box to observe that it is a 3D model.

- Choose **Views>Show Trackball**.

The Trackball/TrackCube displays.



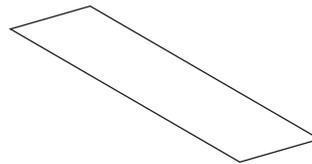
- Drag on the Trackball/TrackCube.



The work plane Triad displays in the center of the screen as you drag the Trackball, and the box you drew rotates as you drag on the Trackball.

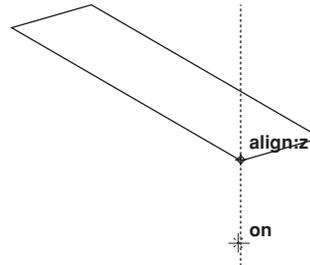
Method 2

1. On the same drawing area, use the **Move** tool to create a box.
 - Rotate the view to Trimetric, by choosing Trimetric from the Trackball pop-up menu.
 - Create another rectangle.
 - If the rectangle is not selected, select it.
 - In the tool palette, choose the **Move** tool. The Message Line reads, *Move: Pick beginning reference point [Shift = Select, Ctrl = Copy (Windows) or Option = Copy (Macintosh)].*



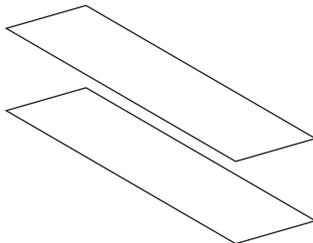
- Hold down the CTRL (Windows) or the OPTION (Macintosh) key.

If you do not hold down the CTRL (Windows) or OPTION (Macintosh) key, the selected object is simply moved, when you do the next two steps.
- Click the front corner of the rectangle for the beginning reference point.
- Indicate the ending reference point by moving the pointer down until the align:z notation appears, and click when the pointer is about an inch from the corner.

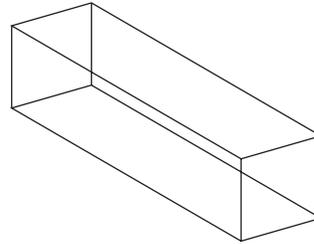


Tech Note:

By holding down the CTRL (Windows) or the OPTION (Macintosh) key during this operation, a copy moves to the new location.

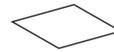


- Use the **Single Line** tool to connect the corners.

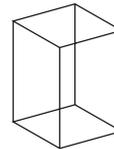


Method 3

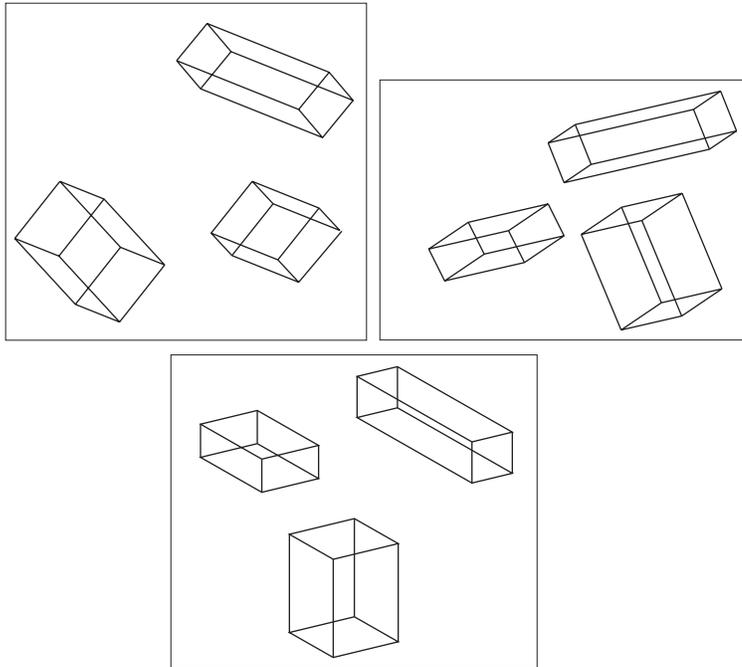
1. On the same drawing area with the view rotated to Trimetric, use the *Extrude* command to create a box.
 - Create another rectangle.



- Select the lines of the rectangle, if they are not already selected.
- Choose **3D>Extrude**.
- Type **-2** in the Z data field.
- Click OK.



2. Manually rotate the view with the Trackball and observe how the models move.



3. Observe the standard views of your models; Isometric, Right, Front, Top and Trimetric.
- Select each of the views from the Trackball pop-up menu, one at a time.
4. Delete all three boxes.
- In the tool palette, double-click the **Selection** tool.
All geometry is selected.
 - Press the BACKSPACE (Windows) or DELETE (Macintosh) key.
All geometry is deleted.

Tip:

The three methods used in this exercise are quite simple. While one method may seem to be more appropriate when constructing a box, as you gain proficiency in 3D modeling, you will find uses for all three methods.

Exercise 2: Isometric Drawing

In this exercise, you will explore the Drafting Assistant working along the z axis further by drawing the figure below. You can observe the snap points and construction lines which aid you in isometric drawings.

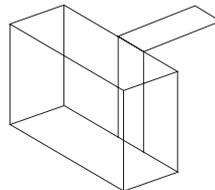
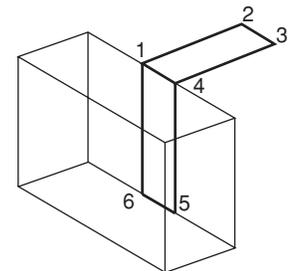
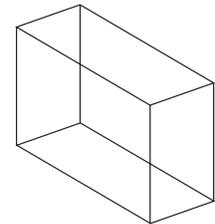
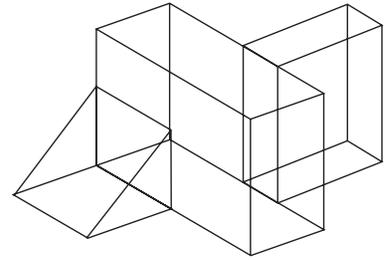
Tech Note:

It is curious that we talk of Isometric drawing and then suggest you use the Trimetric view. The term Isometric, used by drafters, usually refers to any non-orthogonal drawing. In CAD, you can avoid overlapping lines by using the asymmetric Trimetric view.

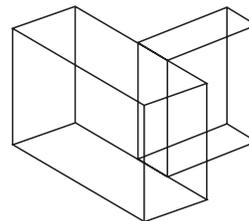
Tech Note:

Remember that you can press the ESC key to backtrack while drawing with the **Connected Lines** tool.

1. Set the view to Trimetric.
2. Draw the rectangular cube shown at the right, using whatever method you wish.
3. Draw the rectangular block adjacent to and aligned at the midpoint of the back face of the first block.
 - Use the **Connected Lines** tool and begin at the *midpoint* of the upper line.
 - Observing the *align:z* and *perpendicular* notations, begin the construction in the order shown below.
 - Double-click at the first location to end this part of the construction.

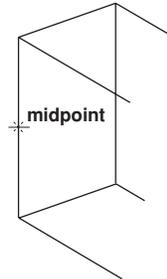


- Complete the block using the **Connected Lines** and the **Single Line** tools.

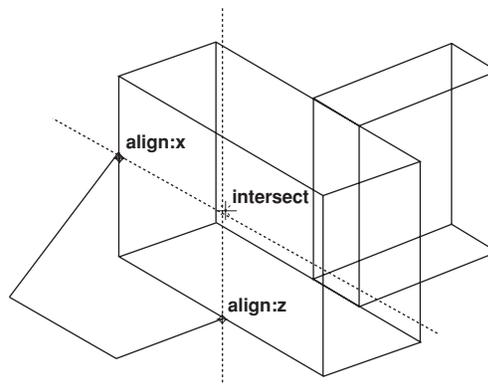
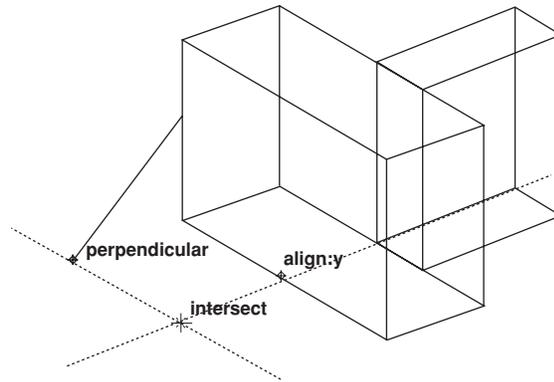


4. Construct a ramp at the *midpoint* of each side of the rectangle at the front of the first block.

- Use the **Connected Lines** tool and begin at the *midpoint* of the left side of the first block.

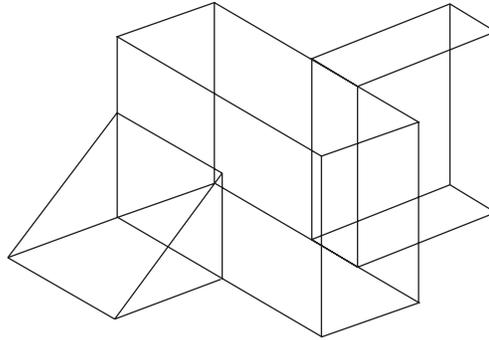


- Draw the lines as shown in the following illustrations.



Tech Note:

Occasionally when you want to snap to a particular point, such as the *midpoint* shown here, the Z-Drafting Assistant snaps back and forth between snap points in close proximity. To snap to the *midpoint*, move the pointer until the notations are snapping and with the mouse button still down, press the *m* key (for *midpoint*) on the keyboard.



If you cannot get the Drafting Assistant to snap to the location you want, it may be because the view orientation puts the geometry too close to other geometry. Try rotating the view slightly with the Trackball.

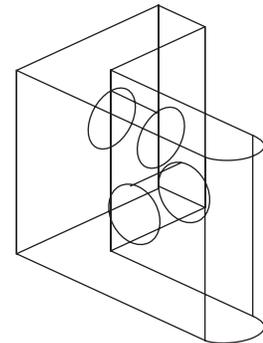
5. Save the drawing, if you like and close the document.
 - Choose **File>Save**.
 - Name the drawing.
 - Choose **File>Close**.

If you didn't save the document, you are asked at this point if you want to save. Answer that question as you wish.

Exercise 3: Extrusion

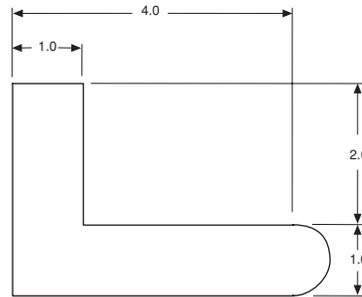
In this exercise, you will use the *Extrude* command to draw the bracket below. Extrusions begin by drawing a figure in 2D on the x,y plane and then extruding it in the z-direction. Once an object is created, you can rotate it; you can also move the work plane to add detail.

1. Open a new document.
2. Draw the 2D outline of the L-bracket, as illustrated, using the measurements provided.



To make the arc segment, hold down the CTRL (Windows) or OPTION (Macintosh) key when you click the location of the endpoint. **Hint:** Start at the upper-

left corner of the bracket.

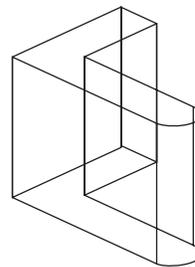


This view of the model is considered the Top view.

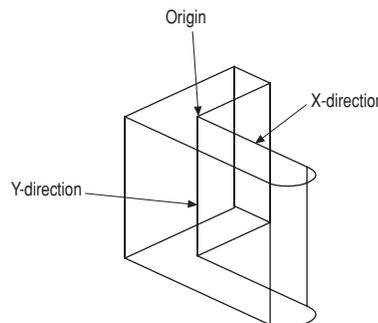
3. Extrude the L-shape into a 3D bracket that is 3 inches wide.
 - Select the geometry if it isn't already selected.
 - Choose **3D>Extrude**.
 - Type **-3** for the Z data field and click OK.

The part extrudes, but you can't see its depth because you are looking straight at the x,y plane.

4. Rotate the view orientation to the Tri-metric view.



5. Move the work plane.
 - Choose **3D>3-Point Plane**.
 - As illustrated on the right, click the origin, the x-direction, and the y-direction. The Message Line displays instructions.
6. Create a one-inch hole at the intersection of the *midpoint* construction



Tech Note:

If the computer screen represents the x,y plane, then the positive z-direction is coming out towards you. Therefore, if you want the apparent depth to go into the monitor, enter a negative z-value.

Tip:

You may want to rotate the view manually so that none of the lines overlap.

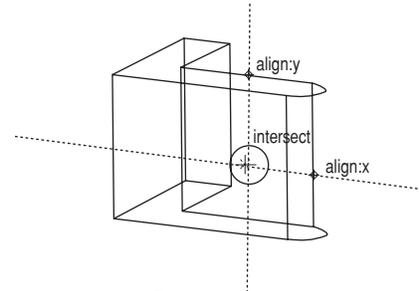
Tech Note:

If you begin drawing without doing Step 5, the geometry would be drawn on the x,y plane of the Top view. You have to move the work plane in order to draw on the front face of the bracket.

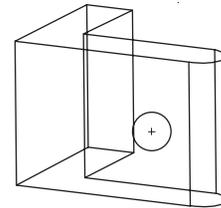
You can display the *Triad* if you want to see the orientation of the work plane.

lines of the front view of the bracket. You may want to rotate the bracket to get access to the lines you want to use.

- Brush over the midpoints of the two lines to wake them up.
- Use the **Center-Point Circle** tool to draw the circle, centered at the middle of the bracket arm.

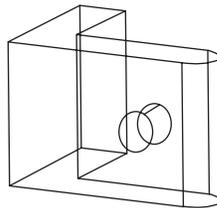
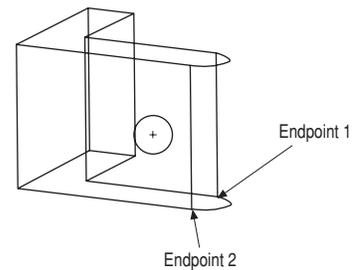


- Type **1.0** and press ENTER (Windows) or RETURN (Macintosh).



7. Extrude the circle through the thickness of the bracket.

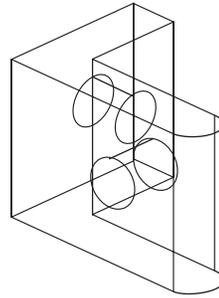
- Choose **3D>Extrude**.
- Drag from endpoint 1 to endpoint 2, as noted below, to indicate the direction and length of the extrusion. When you release the mouse, the values are automatically entered in the data fields.
- Click OK. The hole extrudes.



8. Add a hole in the center of the other arm of the bracket.

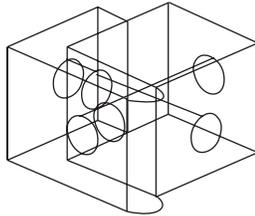
- Choose **3D>Planes>Right** to reset the work plane.

- Draw the circle using the *intersect* point of the *midpoints*.
 - Extrude the circle as you did in the last step, using the thickness of this bracket arm.
9. Save the bracket for use in the next exercise.

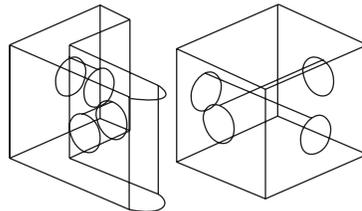


Exercise 4: Interlocking Parts

In this exercise, you will use layers to construct interlocking parts with precise hole alignment.



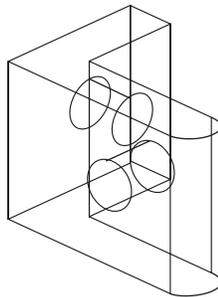
Layer 1 and Layer 2



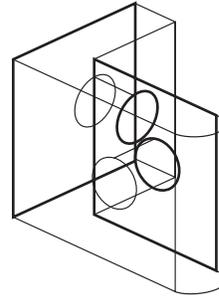
Layer 1

Layer 2

1. If necessary, open the bracket file you saved in the last exercise.



2. Enlarge the bracket to fill the screen and select the lines and circles highlighted bold in the image below. We will use these lines and circles as the basis for building the mating cube.



Tech Note:

This geometry represents the mating surfaces of the two parts.

3. Move copies of the selected lines and circles to a new layer.
 - Choose **Layout>Layers**.
 - Click New.

Layer 2 is added to the list.
 - Click Set Layer.

Layer 2 becomes the work layer.
 - In the tool palette, click the **Move** tool.
 - Hold down the CTRL (Windows) or the OPTION (Macintosh) key and click any *endpoint* on the selected geometry.
 - Click the same *endpoint* again.

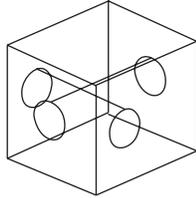
The selected geometry is copied at the same location; two sets of geometry now exist, both on Layer 1, but only one of the copies remains highlighted (selected).
 - Select **Edit>Edit Objects**.
 - Change the Layer entry from Layer 1 to Layer 2.
 - Click Apply.

The duplicate (selected) geometry is now on Layer 2.
4. Hide Layer 1.

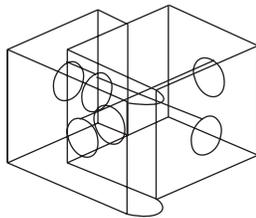
- Click Layer 1 in the list box of the Layers dialog box.
- Click Hide.

The geometry on Layer 1 is hidden, and Layer 2 is still the work layer.

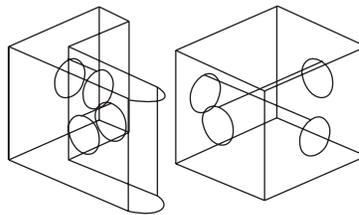
5. Complete the interlocking piece.



6. Use Show and Hide Layers to display these components of the interlocking model.



Layer 1 and Layer 2



Layer 1

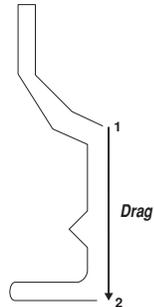
Layer 2

Exercise 5: Revolve

In this exercise you will draw a goblet using the *Revolve* command. You begin by drawing half of a 2D object touching the rotational axis.

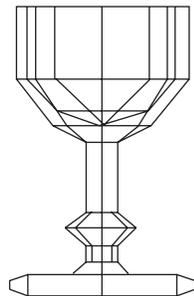
1. Open a New document.
2. Draw half of a goblet along the vertical axis.
 - Choose **Layout>Show Grid** to turn on the grid.

- Drag to indicate the beginning and end of the revolutionary axis.

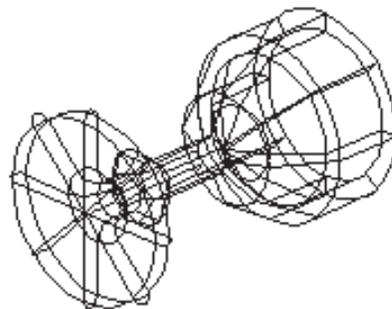


- The default number of steps listed is 8.
- Click OK.

As defined in the Revolve dialog box, the outline revolves 360° through the number of steps entered to make a 3-dimensional goblet.



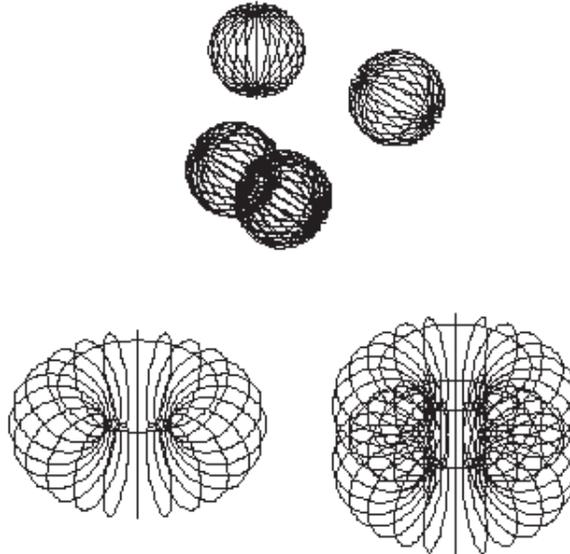
4. Rotate the view to get a better 3D angle.



5. Save the drawing if you want.

Additional Exercise

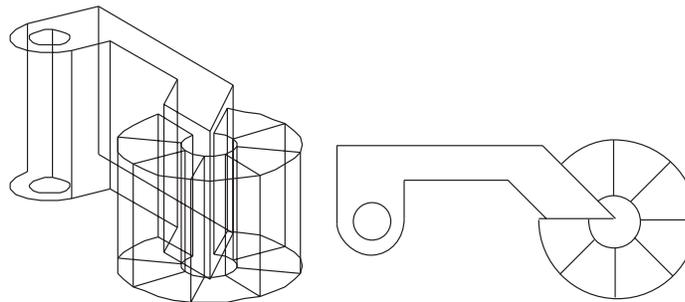
Using circles and the *Revolve* command, create images such as those illustrated below.



Also practice 180° revolutions to understand how the right hand rule is applied while defining the axis of revolution.

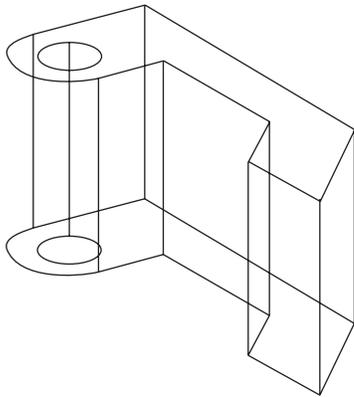
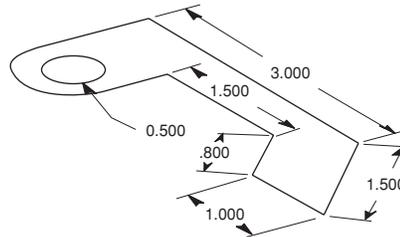
Exercise 6: Pipe Winch

In this exercise, you will put all the 3D modeling methods together to draw a pipe winch for an oil rig.



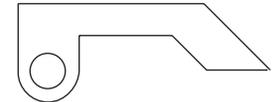
1. Open a New document.

2. Draw the basic shape in the Trimetric view.
 - In the Trackball menu, choose the Trimetric view.
 - Use the **Connected Lines** tool to draw the trimetric figure below using the measurements as illustrated.
3. Extrude the part to a z-depth of **2** inches.



Tech Note:

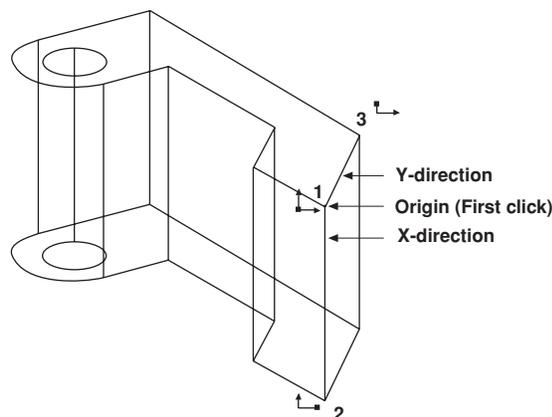
If you have trouble drawing this part in the Trimetric view, you can draw it in the Top view as illustrated below and then change it to the trimetric orientation.



You should then practice drawing in the Trimetric view by imitating the rotated part.

4. Add the roller, using *Revolve* to turn a rectangle on the slanted face.
 - Zoom in on the part.
 - Choose **3D>3-Point Plane** to move the work plane.

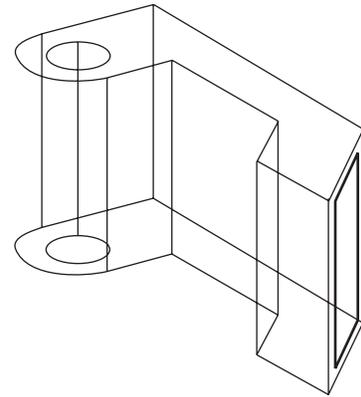
Click at the three points shown to indicate the new orientation of the x and y axes, and therefore the work plane.



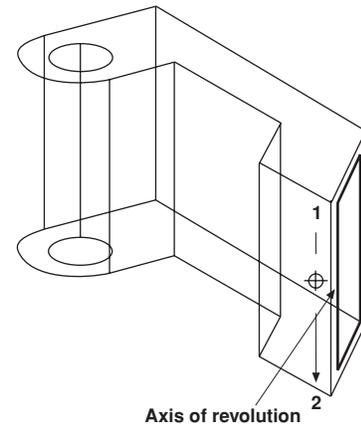
- Draw a rectangle, as shown.

You will notice that you can add this rectangle while you are looking at it from any angle.

- With the new rectangle still selected, choose *Revolve* to revolve the rectangle **315°** (using the default 8 steps)



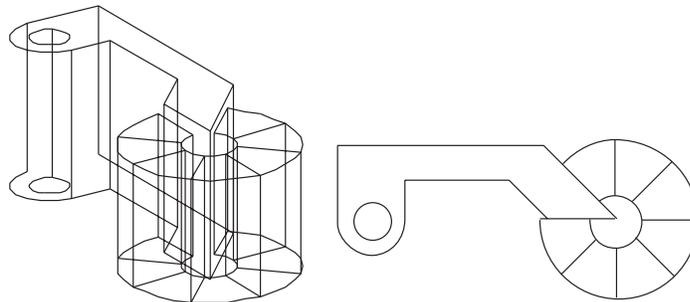
- Enter **315** for the *Sweep Angle* and **8** for the *# of Steps* into the Revolve dialog box. Activate the next data field by hitting the TAB key or clicking in it with the mouse. Then, using the bull's eye cursor drag from point 1 to point 2 and the six data boxes will receive the proper data.



Tech Note:

Be aware of the right hand rule of revolution. You must click the upper point on the axis first to revolve in the proper direction.

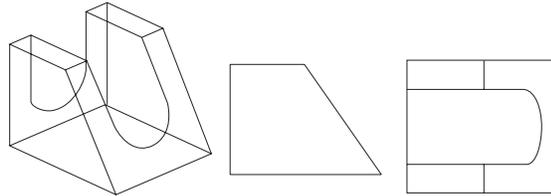
The rectangle revolves.



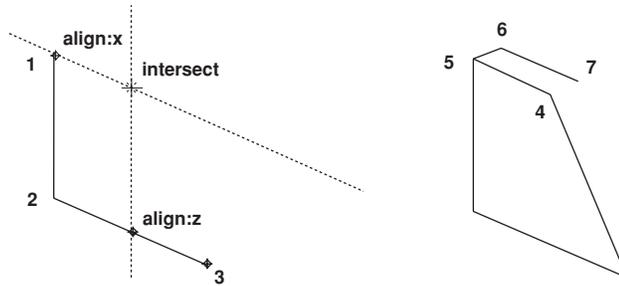
5. Save or delete the part, as you wish.

Exercise 7: Cutting an Angle with Isometric Drawing

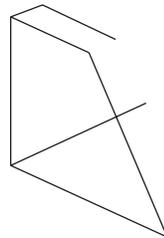
In this exercise, you will explore isometric drawing further, and construct this model.



1. Open a new document and show the Trimetric view.
2. Use the **Connected Lines** tool to follow these steps:



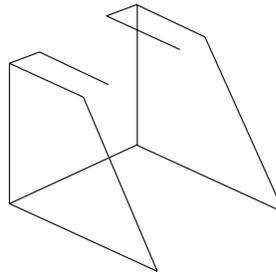
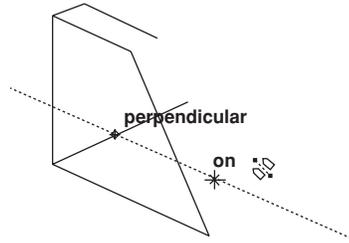
3. Draw the single line representing the width of the part.



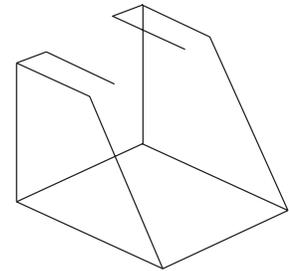
4. With all lines selected, except the last one drawn, use the **Mirror** tool with the *Copy* option (CTRL key - Windows, OPTION key - Macintosh) to make a mirror copy along the mirror line *perpendicular* to the *midpoint* of the width line.

Tech Note:

The **Mirror** tool only works in the x,y plane. In this model, mirroring works correctly, but if you have changed the work plane, mirroring may not work.



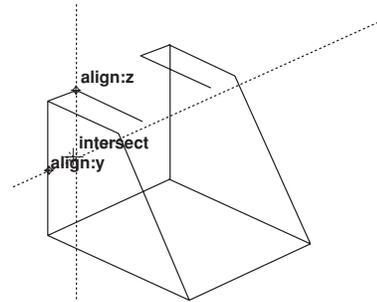
5. Connect the front edges.



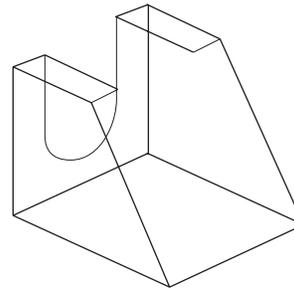
6. Use the **Connected Lines** tool to make the back opening.

Exercise 7: Cutting an Angle with Isometric Drawing

You should change the work plane to *Right* to place the arc correctly.

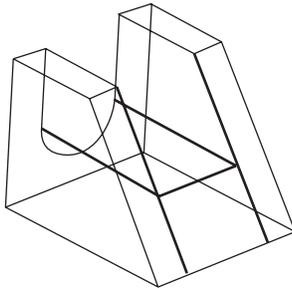


- Continue the construction as shown here:

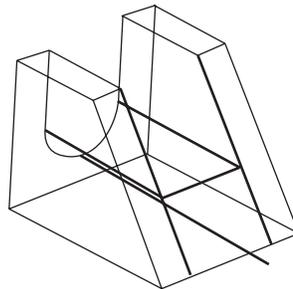


7. Construct the ellipse on the slanted face.

- Add construction lines as shown in bold below.



- Draw another construction line from the *midpoint* of the arc, as shown.



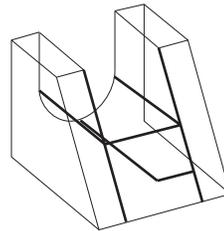
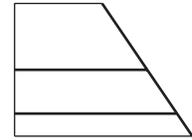
Tip:

Remember when you use tools which require specification of three points, you do not have to move the work plane.

Tech Note:

With these construction lines you can specify the *center* and one point on a 3-point center ellipse; you still need to be able to specify the corner of a box bounding the ellipse.

- From the Front view, trim this line back to the front face.
- Draw a construction line from the *endpoint* of the last line, *perpendicular* to the edge of the slanted face.

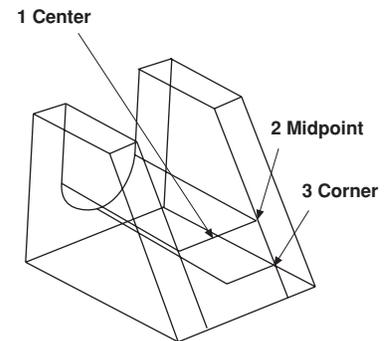


Tip:

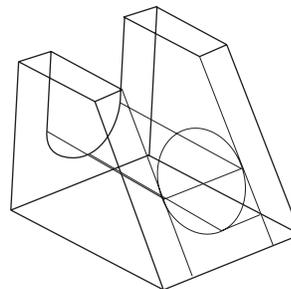
You may have to rotate the part with the Trackball to see the intersections clearly.

- Choose the **3-Point Center Ellipse** tool. The Message Line reads, *3-Point Center Ellipse: Pick center of ellipse [Ctrl = Copy Previous (Windows) or Option = Copy Previous (Macintosh)].*

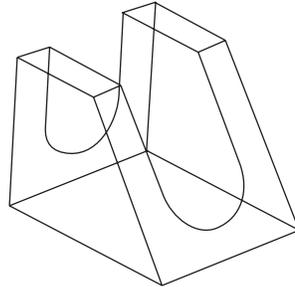
Specify the points as shown.



The ellipse appears.



8. Delete the construction lines and trim the upper half of the ellipse.

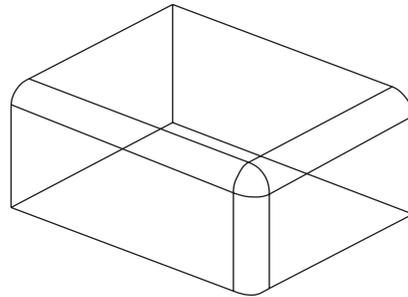


The model is complete.

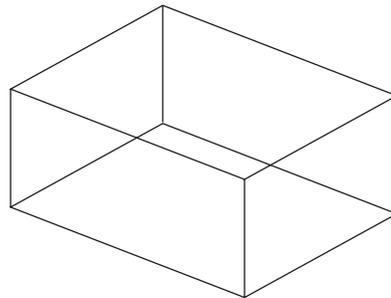
9. Save or discard the part, as you wish.

Exercise 8: Three-Corner Fillets

In this exercise, you will work with one of the trickiest 3D modeling problems, trimming the corner of a box where three fillets intersect.



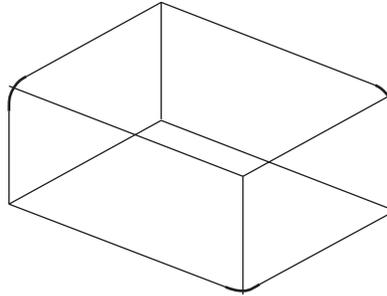
1. Begin by drawing a box.



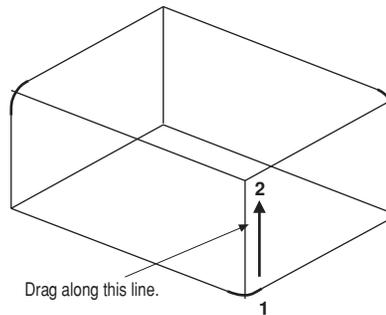
Tip:

With the **2-Entity Fillet** tool you do not have to worry about changing the work plane around. The fillet is automatically placed in the plane based on the two lines you select.

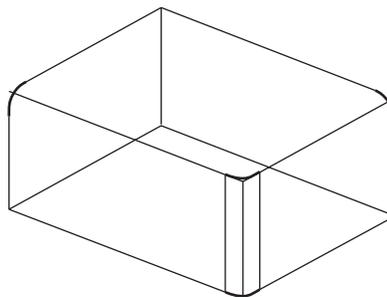
2. Fillet three corners as shown below.



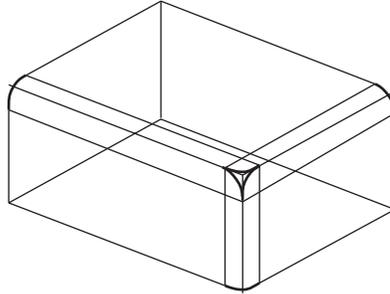
3. Extrude the fillets.
 - Select the bottom fillet.
 - Choose **3D>Extrude**.
 - Drag from one end of the line as shown below to indicate the extrusion distance and click OK.



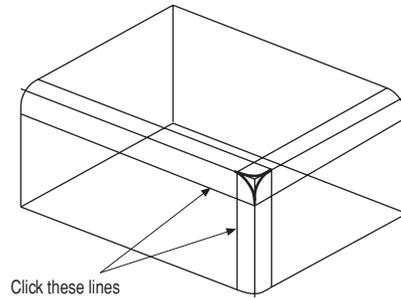
The fillet is extruded.



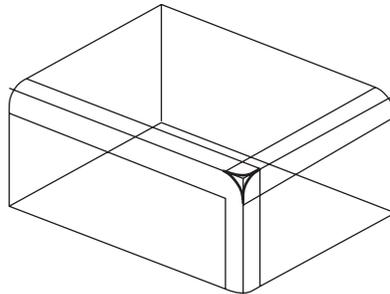
- Extrude the other fillets.



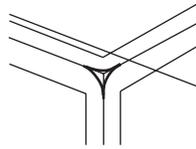
4. Trim the intersecting lines at the corner.
 - Use the **Corner Trim** tool and trim the corner as shown.



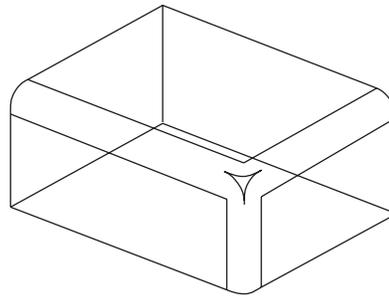
The corner trims.



- Trim the remaining corners, as illustrated.



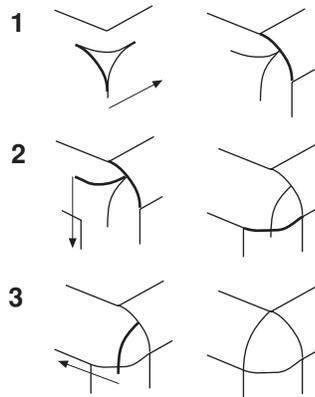
5. Delete the lines which made up the edges of the cube before you added the fillets.



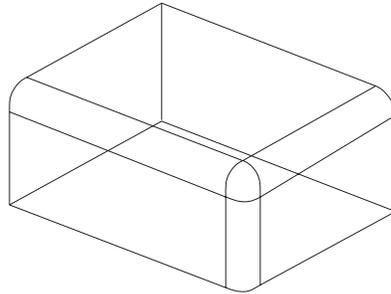
Tip:

You can create a new layer and move those lines to that layer; then hide that layer if you think you may be using them again.

6. Move the remaining arcs into the proper positions.



The cube corners are filleted.

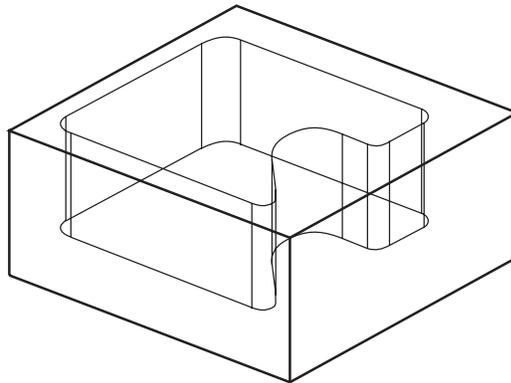


7. Check the Top, Right and Front views to see if they look correct.
8. Save or discard the drawing, as you wish.

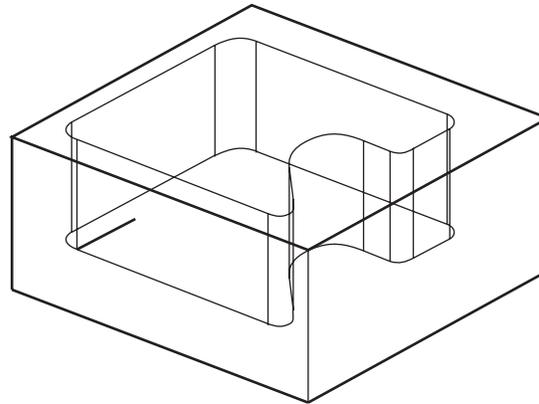
Exercise 9: Filleting Curves

In this exercise, you will create a fillet at the bottom of a pocket.

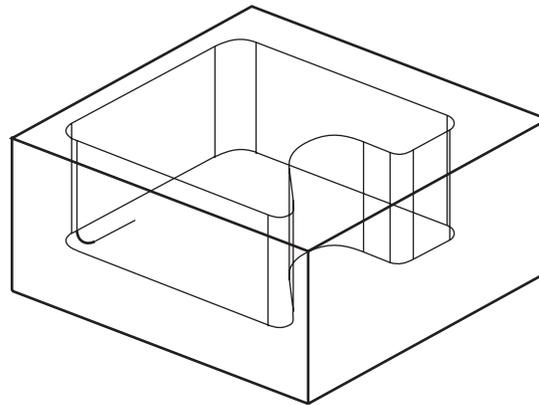
1. Begin by drawing a model similar to the one below.



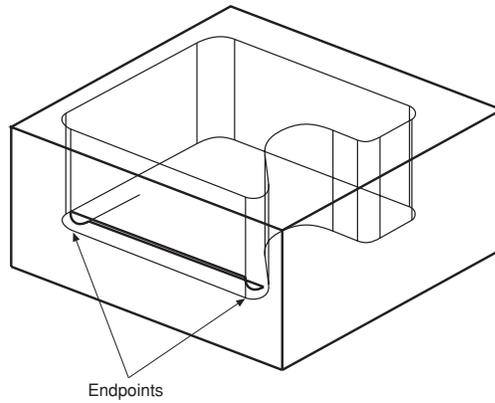
2. Draw a line in the plane of the floor, perpendicular to the pocket side.



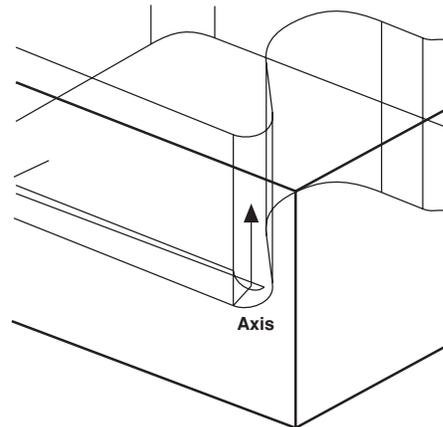
3. Construct a fillet between the line on the floor and the line on the side.



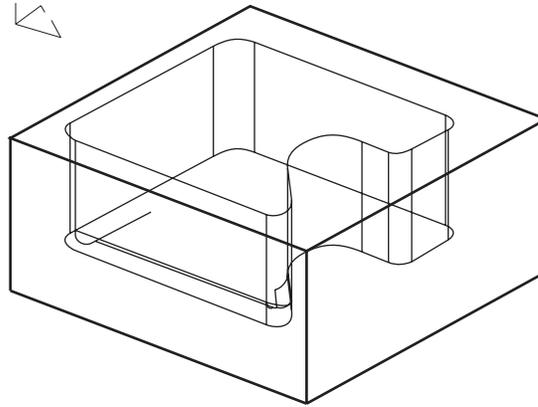
4. Extrude the fillet arc along the length of the side, from *endpoint* to *endpoint*.



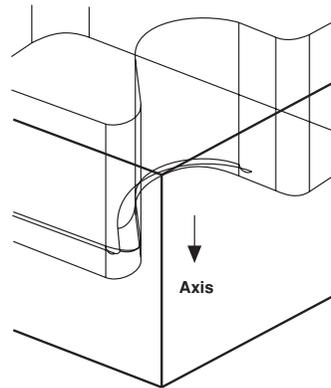
5. Revolve the fillet arc around the corner radius, the number of degrees of the corner angle, using a vector from the corner radius center and 2 steps.



6. Extrude along the next straight line segment.



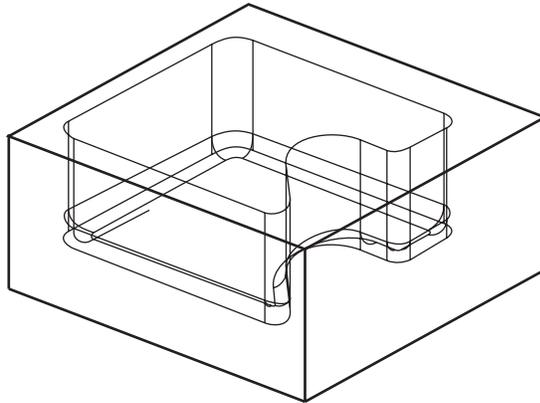
7. Revolve around the curve, by the degrees of the curve angle, using a vector from the radius center.



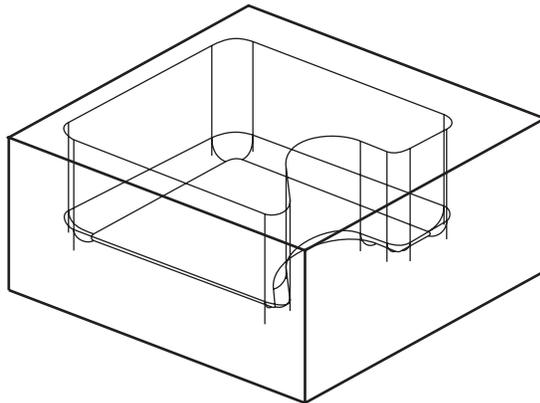
Tech Note:

Use *Edit Objects* to find the value of the angle of the curve.

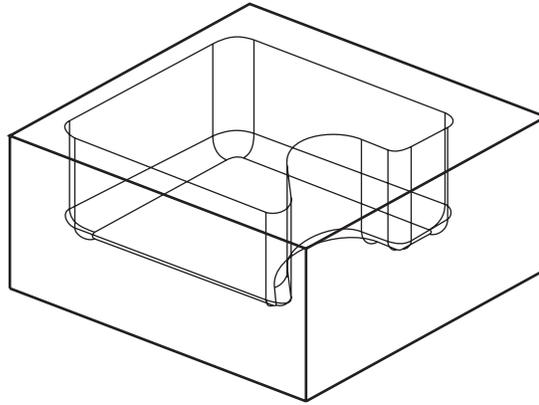
8. Complete the boundary by extruding along the lines and revolving around the arcs.



9. Delete the original boundary and the line you added to create the fillet initially.



10. Trim the vertical lines to the top of the new boundary.

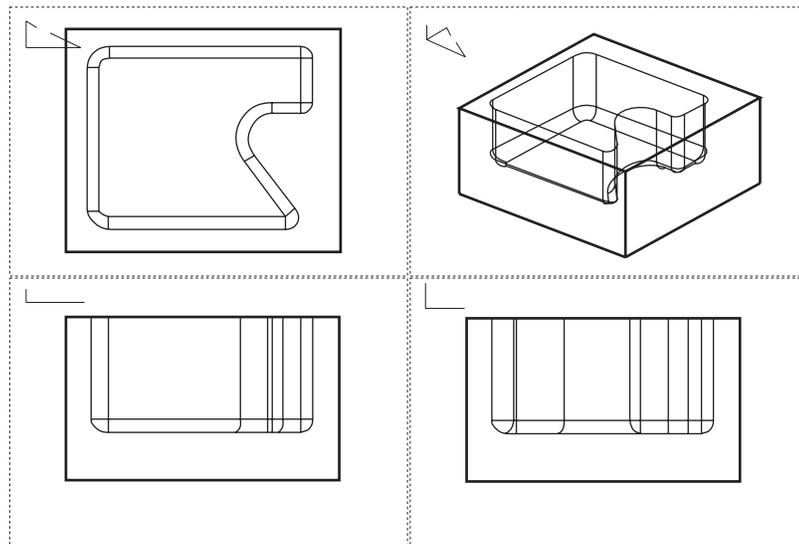


The model is complete.

11. Look at the other views (Front, Top and Right) of the model.

Tech Note:

Remember to hold down the SHIFT (Windows) or the CONTROL (Macintosh) key when you want to zoom in the active view window.



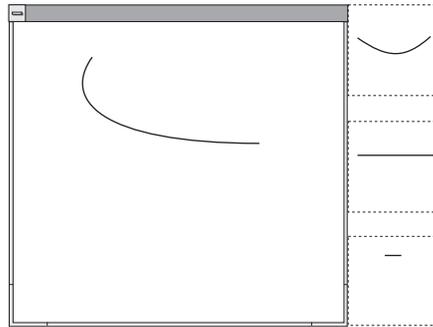
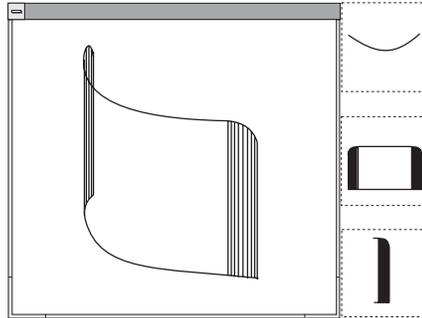
Exercise 10: Compound Curve

In this exercise, you will create a compound curve using multiple views.

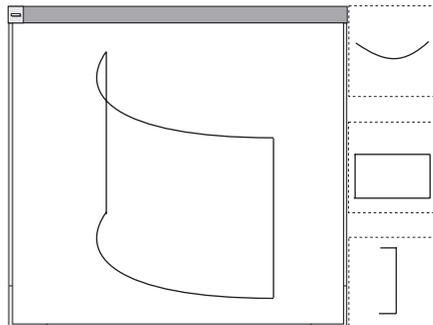
1. Open a new document.
2. Draw an arc so that the endpoints of the arc lie on a horizontal line.
3. Set up design views, so that you can see your work from different view points.
 - Choose **Views>Sheet Into View**.
 - Choose *A Landscape 4 Views.vc6* from the pop-up menu.
 - Click OK.

You may have to zoom to see the arc properly.

The arc displays in the views.



4. Extrude the arc in the z direction.



5. Add lines on the surface of the cylinder.
 - Select the extrusion line at the left end of the arc.
 - Choose **Edit>Polar Duplicate**.

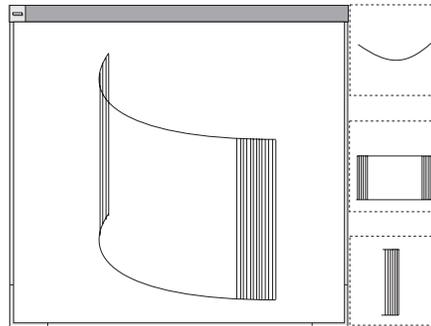
Specify **8** for the number of duplicates.

Activate the *Center X* data field with the TAB key. First touch the arc with the bull's eye cursor to wake up the center point and then click on the center point of the arc.

Make the step angle **2°**.

Click **OK** to finish the operation.

- Repeat the process for the other end of the cylinder, making the step angle **-2°**.

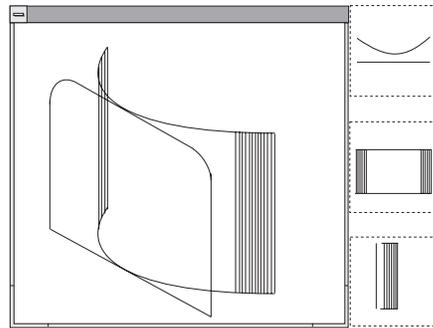
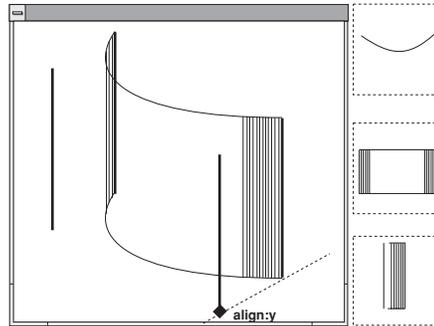


6. Create the profile of the compound curve.

Use the **Selection** tool + CTRL (Windows) or OPTION (Macintosh) key to make copies of the original extrusion lines in front of the surface aligned on the y axis.

 - Select the two original extrusion lines that go between the arcs.

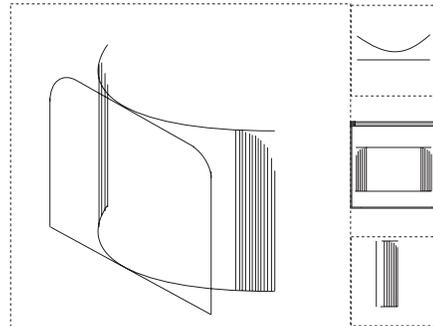
- Move the cursor near an *endpoint* of one of the lines. Hold down the CTRL (Windows) or OPTION (Macintosh) key and drag a copy of the lines out along the y-axis as shown in the next graphic.
- Connect the top and bottom of the lines with the **Single Line** tool to make a rectangle.
- Choose **3D>Planes>Front**.
- Fillet the corners as shown.



Tech Note:

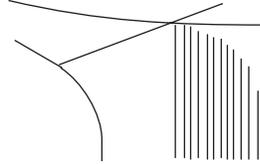
If the drawing looks right in the Top and Front views, then the drawing is right.

- Make sure that the fillets are selected and use them as the boundary of a trim operation. In the Front view, trim off the parts of the surface lines that appear above the fillets.

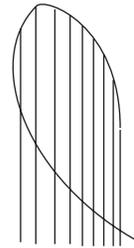


7. Complete the curve.

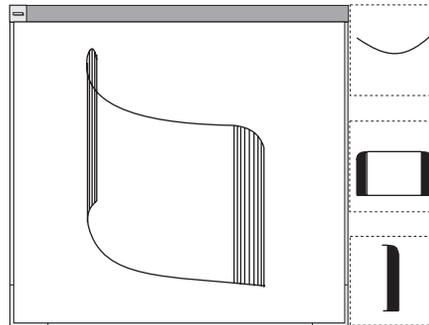
- Draw a line from the beginning of the fillet to the cylinder.



- Trim the cylinder arc.
- Delete the construction line marking the end of the fillet.
- Repeat the process for the other fillet.
- Delete the profile form.
- Use the **Through-Points Spline** tool to connect the endpoints of the cylinder at ends of the surface lines.



8. Save or discard the drawing, as you wish.



Conclusion

These exercises demonstrated many of the techniques you will use for 3D modeling. They showed you how to use the Z-Drafting Assistant for isometric drawing and demonstrated the *Extrude* and *Revolve* commands.

In addition to those basic construction methods, you learned how to rotate the view orientation and move the work plane. Finally, you solved two of the most complex 3D problems, intersecting multiple fillets and driving a fillet around a boundary.

When you combine all these techniques with what you already know using Graphite you can create amazingly complex finished drawings!

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