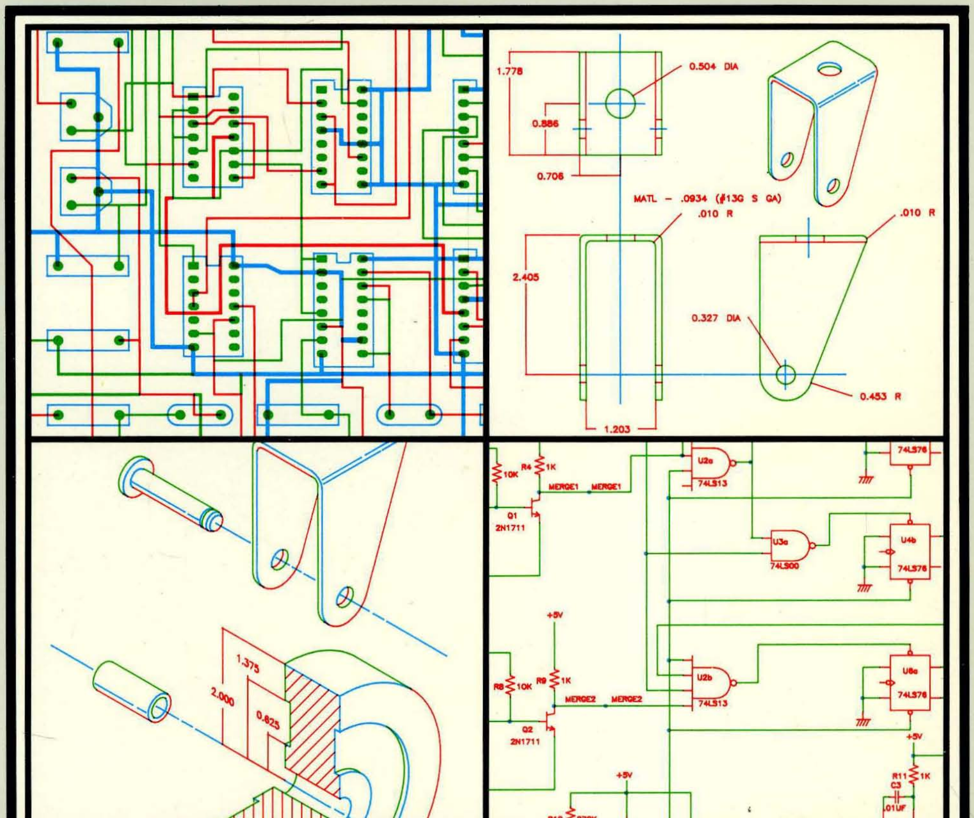
HP 9000 Series 200/300 Computers

HEWLETT PACKARD

Learning HP EGS 2.1



HP EGS Manuals

Installing HP EGS lists the hardware and software needed for an HP EGS system, gives some hints on connecting the hardware, and explains installing the HP EGS software on a hard disc and Shared Resource Manager. Appendices contain information on using a hard disc and Shared Resource Manager, the differences between HP EGS 2.0 and 2.1, and combining HP EGS 2.1 with a complete Pascal 3.1 Operating System.

Learning HP EGS provides tutorials to familiarize you with the HP EGS system and its personalities: General Drawing, Mechanical Drafting, Electrical Schematic Drawing, and Printed Circuit Board Layout. By following the exercises in this manual you will learn the basic concepts of HP EGS, so that you can do more advanced tasks easily.

Understanding HP EGS describes the theory and operation of the HP EGS system. It explains how the Graphics Editor works and how to customize the system. You can learn how to use HP EGS post-processors to create material and connection lists, produce output to run photoplotter and drill machines, and translate drawings to and from the IGES Translator. An application note on using archive files to transport data is also included.

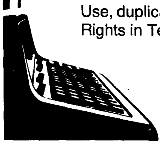
Managing and Editing Files with HP EGS describes file system concepts and provides tutorial exercises to help you learn how to use the text Editor and Filer. A syntax reference further explains the Editor, Filer, and Pascal Command Line commands.

HP EGS Syntax Reference lists all HP EGS commands. Each command is illustrated with a diagram, explained in a table, and shown in use with one or more examples. The introductory sections of the reference explain how to interpret the diagrams and enter the commands. The commands used in archive files are listed at the end of the reference.

Learning HP EGS 2.1

for the HP 9000 Series 200/300 Computers

Manual Set Reorder No. 98305-90008



Hewlett-Packard Company

3404 East Harmony Road, Fort Collins, Colorado 80525

Printing History

New editions of this manual will incorporate all material updated since the previous edition. Update packages may be issued between editions and contain replacement and additional pages to be merged into the manual by the user. Each updated page will be indicated by a revision date at the bottom of the page. A vertical bar in the margin indicates the changes on each page. Note that pages which are rearranged due to changes on a previous page are not considered revised.

The manual printing date and part number indicate its current edition. The printing date changes when a new edition is printed. (Minor corrections and updates which are incorporated at reprint do not cause the date to change.) The manual part number changes when extensive technical changes are incorporated.

July 1985...Edition 1

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Welcome to HP EGS

Chapter 1

Welcome to the Hewlett-Packard Engineering Graphics System. *Learning HP EGS* shows you the basic operations of the system, using examples that you can step through. The manual is divided into four major parts:

- Part I – Getting Started
- Part II – General Drawings
- Part III – Mechanical Engineering Drawings
- Part IV – Electrical Engineering Drawings

Part I introduces you to HP EGS. It includes basic information about the workstation keyboards and about commands, viewports, screens and screen menus, and cursor snapping modes. This Part prepares you for using the General Drawing personality.

In Part II you use the General Drawing personality to step through exercises in which you draw a basic house and then a complex town. You also find out how to easily create isometric drawings and add dimensions to drawings. This Part also provides an exercise on how to save, print, and plot your drawings.

In Part III you use the Mechanical Drafting personality to create an annotated flange drawing and then create an assembly drawing of two of these flanges bolted together.

Part IV introduces two more HP EGS personalities: Electrical Schematic Drawing and Printed Circuit Board Layout.

The Electrical Schematic Drawing personality makes schematic drawings quick and convenient to prepare. By following the examples in this chapter you draw a schematic and make connection and material lists.

The Printed Circuit Board Layout personality chapter provides examples that enable you to draw a printed circuit board. During the exercises, you also generate a rat's nest and compare connection lists. If you want to learn about this personality before reading about the Electrical Schematic personality, you can skip the sections on preparing the post-processor and inputting rat's nest traces.

Note

You must read Part II, General Drawings, before starting any chapters in Part III or IV. Part II contains basic information that you must understand before performing any of the exercises in Parts III or IV.

Learning About Your System

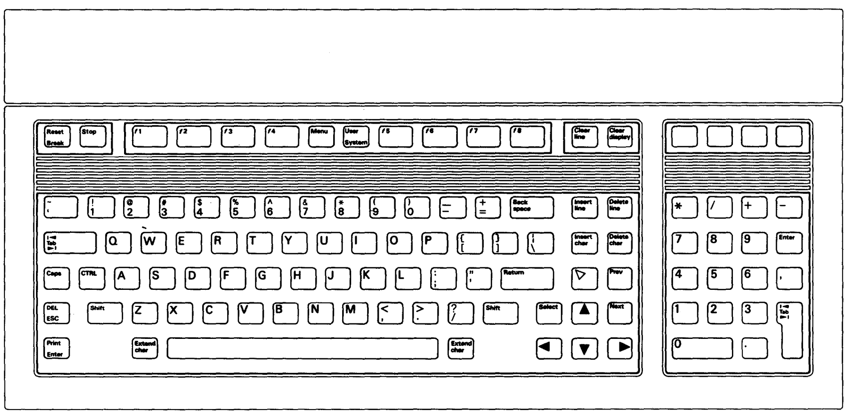
Chapter 2

This chapter explains how to load and run HP EGS on a daily basis. You will also learn some fundamentals of HP EGS operation, such as viewports and operating modes.

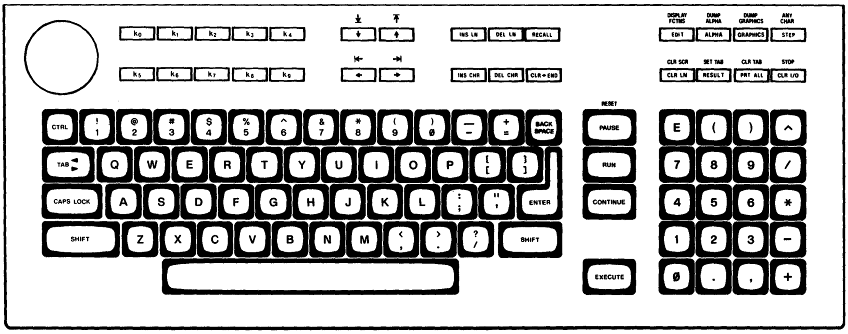
Before you load the system, let's take a quick look at your computer keyboard.

The Keyboard

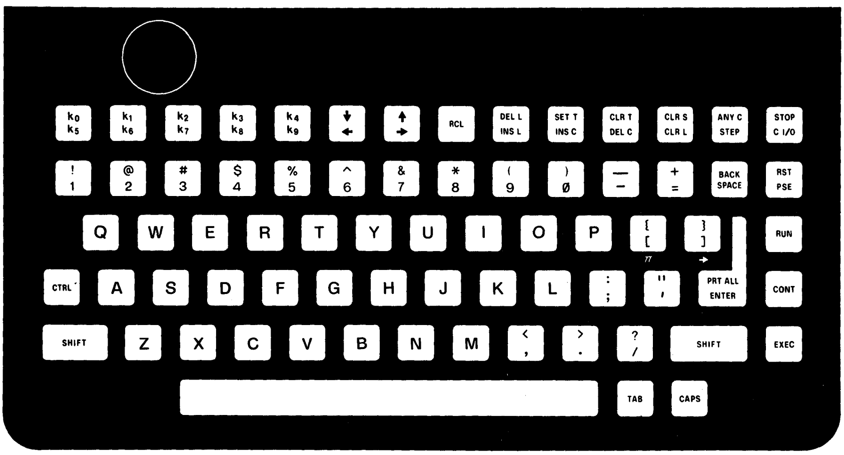
Hewlett-Packard makes several keyboards; each uses different key-labeling conventions. For the sake of simplicity, all examples and procedures in this manual assume you have an HP 46020A keyboard (see below). If you have another keyboard, the following Key Correspondence Table will show you what keys to press in place of those shown in this manual.



HP 46020A Keyboard



HP 98203A Keyboard (Model 216)



HP 98203B Keyboard (Same as Model 226 and 236)

Key Correspondence Table

HP 46020A	HP 98203A	HP 98203B
Return	ENTER	ENTER
Select	EXEC	EXECUTE
Clear line	CLR LN	CLR LN

Entering Data

You must tell the computer when you are done typing in data or commands from the keyboard. To do this press (Return) (or (ENTER) on the HP 98203A or 98203B keyboard). Pressing this key tells HP EGS that you are finished typing, so you can continue input from the graphics input device. Do not use (EXECUTE), (Select), or (CONTINUE) to indicate that you are finished typing.

HP 46020A Keyboard Features

You can perform the following special functions using dedicated keys (f1-f8) on the HP 46020A:

CAUTION

PRESSING (Shift) (Reset) IS THE SAME AS TURNING YOUR COMPUTER OFF THEN ON AGAIN. IF YOU ARE EDITING A DRAWING OR TEXT FILE WHEN YOU PRESS (Shift) (Reset), YOU LOSE ANY CHANGES OR ADDITIONS YOU MADE DURING THE SESSION.

(Backspace)	Moves the cursor to the left one character at a time without erasing the characters, unless the cursor is at the end of a line. When the cursor is at the end of the line, the (Backspace) key erases each character as it moves the cursor to the left.
(◀)	Moves the cursor to the left one character at a time without erasing the characters. (Shift) (◀) jumps the cursor to the beginning of the line.
(▶)	Moves the cursor to the right one character at a time without erasing the characters. (Shift) (▶) jumps the cursor to the end of the line.
(Insert char)	Changes the cursor to a block cursor and enables you to insert characters within the block. If pressed a second time, the cursor returns to normal mode as an underline cursor.
(Delete char)	Deletes the character underlined by the cursor.
(Clear line)	Clears the entire line, leaving the cursor at the beginning of the line. This key also cancels the insert mode if it is set.
(Stop)	Stops the output of the PLOT, ARCHIVE, and GENERATE commands. Pressing (Stop) during the WINDOW and SHOW commands terminates the drawing of the display.

If you are using the type-ahead buffer, which enables you to enter commands while the system is processing the current command, these four keys may be helpful:

Backspace	Prints a BS on the screen and erases the character to its left in the buffer.
CTRL Backspace	Removes the previous character from the buffer and the screen.
Clear line	Inserts a block cursor that clears the type-ahead buffer line.
CTRL Clear line	Removes the line from the type-ahead buffer and the screen.

System Defined Keys

The system defined keys are the definitions given to f1 through f8 when in system mode. Their actual labels are displayed on the screen when Menu or Shift Menu is pressed.

CONT	CONT (f4) is used to continue normal operation after pressing Break.
RECALL	RECALL (f2)is used in the Graphics Editors to recall the last command.
STEP	STEP (f5) is not used in HP EGS.
ANYCHAR	ANYCHAR (Shift f5) is used to generate any US ASCII character. To use it, first press ANYCHAR (ANYCHAR is a shifted system key, so you must press Shift f5 to get ANYCHAR). Then press any three digits from 000 through 255, representing the decimal equivalent of an ASCII character. The corresponding character will be generated; it may or may not be meaningful to the subsystem you are in.
ALPHA GRAPH	The ALPHA (6) and GRAPH (7) keys allow you to turn the alpha and graphics display modes on and off. The ALPHA key turns on the alphanumeric display if you press it once, and turns off the graphics display if you press it a second time. The GRAPH key turns on the graphics display if you press it once, and turns off the alphanumeric display if you press it again.
DMP A	DMP A (Shift f6) key "dumps" whatever is in the alphanumeric display to your printer. Do not use DMP A if you have no printer. It may cause your system to hang for a while, as the computer searches for a non-existent printer.

DMP G

The DMP G (Shift f7) key operates just like DMP A , except that it dumps whatever is in the graphics display to your printer. The key works properly only when you have a graphics printer.

Loading HP EGS

If HP EGS has not been installed on your system, follow the installation procedures in *Installing HP EGS.*

If HP EGS has already been installed, you are ready to run the General Drawing personality:

1. Turn on all attached peripherals before turning on the computer.
2. Turn your computer on.

   The computer loads HP EGS from your local disc or SRM and displays a copyright message ending with:

   ```
   Press <space> to continue.
   ```

3. Press the space bar.

Setting the Date and Time

1. When the system asks you to enter the date, type the date in a DD-MMM-YY form, where DD is the day of the month, MMM the month, and YY the last two digits of the year.

   For example, `27-Jul-85` is the correct format for July 27, 1985. Then press (Return). You need only enter those parts which have changed since you last used the system.
2. Now type the time in an HH:MM:SS form, where HH is the hour based on a 24-hour clock, MM is the minutes, and SS is the seconds.

   For example, `18:21:30` is the correct form for 6:21:30 pm. Press (Return).

   Now the accurate date and time are automatically added to the description of any file you create.
3. Press (Return) to load HP EGS.

You will see words flash on the screen as the computer searches for the control files it needs, opens them, and after about a minute displays this menu of options:

```
Use arrow keys to move cursor to desired function.
Select function with <RETURN> or <ENTER>.
  ->System Utilities
    General Drawing Editor
    Electrical Engineering
    Mechanical Engineering
    IGES Translator
```

Storing Your Drawings

Each HP EGS drawing is stored in a file; groups of files are stored in volumes. Before you load the General Drawing Editor, you need to designate a volume for storage because your drawing files should not be stored in the same volume with HP EGS files.

If your HP EGS is on a Shared Resource Manager (SRM) system, use the storage volume name given to you by the SRM system manager. If your HP EGS is stored on local mass storage, use the following procedure to create a storage volume.

1. Use (▼) and (▲) to move the -> from one menu item to another. Place the -> next to `System Utilities` and press (**Return**). The system displays:

```
HP EGS 2.1                      1-JUL-85                    11:17:22
Use arrow keys to move cursor to desired function
Select function with <RETURN> or <ENTER> key

      ->RETURN To Main Menu
        Media Initialization
        CS80 Tape Backup

        Editor
        Filer
        Pascal Command Line

        Break-up A Large File
        Re-create A large File
        Mag/Paper Tape Utility
        Character code Converter
        Message File Utility

        Install Codeword
        Re-boot System
        Reconfigure System

                                 utils
```

   Most of the utilities available on this menu are not used in this tutorial manual. To learn more about these utilities, refer to *Installing HP EGS* and *Managing and Editing Files with HP EGS.*

2. Place the -> next to Filer and press:

   (**Return**).

   As soon as the System loads the Filer program the following command line appears at the top of the screen:

```
Filer: Change Get Ldir Quit Remove Save Translate Vols What Access Udir ?
```

3. Press (V) to get a list of all the volumes on your system. Your volume list should look similar to this:

```
Volumes on-line
  1   CONSOLE:
  2   SYSTERM:
  6   PRINTER:
 11 * SYSVOL:
 12 # EWSYS:
 13 # EWCODE:
 14 # EWEE:
 15 # EWME
 16 # EWRC:
 17 # EWOPT:
 18 # EWSP:
 19 # V19:
 20 # V20:
```

   The numbers on the left are the volume unit numbers; the names on the right, such as SYSVOL: and V19:, are the volume names. Volumes may be referred to by unit number or by name. For example, the volume numbered 11 may be referred to as #11: or SYSVOL:. In this example, volumes #11: through #18: contain the files needed to run HP EGS.

4. Choose a volume such as V19:. Before you name this volume for storage, verify that it is empty. Press (L) to list the volume contents. The system prompts:

   `List what directory?`

   Type the name of a directory (include the colon in the directory name) such as the one shown below:

   V19: (Return)

   If the volume is empty, a line of text near the bottom of the displayed information should report `Files shown=0 allocated=0`.

   If the volume you selected is empty, go on to the next step. If it is not empty repeat this step until you find an empty volume.

5. To change the volume to a more descriptive name, select:

   (C)

   When the computer prompts:

   `Change what file?`

   Type the name of the empty volume:

   V19: (Return)

When the computer prompts:

`Change to what?`

Type the new volume name:

DRAW: (Return)

6. Press (Q) to return to the System Utilities menu.
7. Move the -> to `RETURN to Main Menu` and press (Return).

Loading General Drawing

Move the -> to `General Drawing Editor` and press (Return).

The computer now loads the Graphics Editor program, gedit.CODE, with the files for the General Drawing personality. This process takes a few minutes.

BREAK TIME

Take a minute to stretch and review what you have done in this section:

- Reviewed keyboard features.
- Learned how to enter information.
- Loaded HP EGS.
- Created a storage volume.
- Loaded General Drawing.

Communicating with HP EGS

This section introduces you to communicating with the HP EGS system. You do this using viewports, menus, alpha and graphics screens, edit key functions, and command entry.

Entering Commands

Commands are key words and parameters which tell HP EGS what you want to do. There are two ways to enter commands into the system:

- Type in the command name and parameters at the computer keyboard.
- Use the graphics input device (tablet or mouse) to select commands and parameters from the screen or tablet menu.

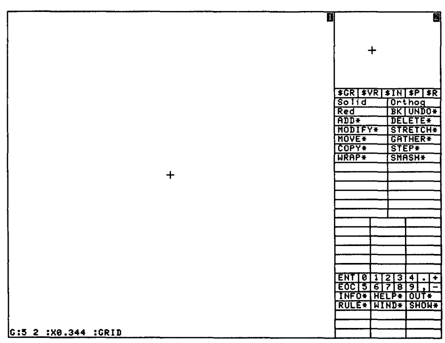
Using the graphics input device is more convenient because you see the effect of the command as it is being entered. To use the graphics input device, position the cursor over the menu command that you want to select. Then, press down on the stylus of the tablet or press the left button on the mouse.

Parameters may be entered with the command either by selecting the appropriate item from a menu or by entering them from the keyboard. Spaces, commas, and (Return) should be used to separate adjacent parameters such as 100,100 or 100 100. Otherwise, an error may occur.

Using the Viewports

A viewport is a "window" through which you can see the drawing you are editing. In the General Drawing personality you have two viewports, as shown in the following figure. One is the large rectangular area with the number 1 in the upper right corner. The other viewport, which has a number 2 in the corner, is the area above the screen menu.

Each viewport can independently display any portion of the drawing you are currently working on. Changing the view in one viewport does not automatically change the view in the other except when you add a component to one viewport and that drawing portion is visible in the other port. If you wish, you can turn off the second viewport so that it is not updated. This procedure is described later.



A common use of the two viewport system is to use viewport 2 as a global view of the entire drawing and viewport 1 as the work space where you can zoom and pan to display the portion you are working on. Remember that the viewports are independent and can have different zoom factors.

The bottom of the main viewport contains a line of information concerning the current system constants.

The values following the `G:` in the lower left corner of viewport 1 tell you how many user units are between each grid point and how many of those grid points are displayed. For example, `G:5,2` means that there are 5 user units between each grid point and that every second grid point is displayed. :GRID indicates that the snapping mode is grid, one of five that you can set. Snapping modes are discussed further at the end of this chapter.

The small cross near the center of each viewport is the origin of your drawing, point (0,0).

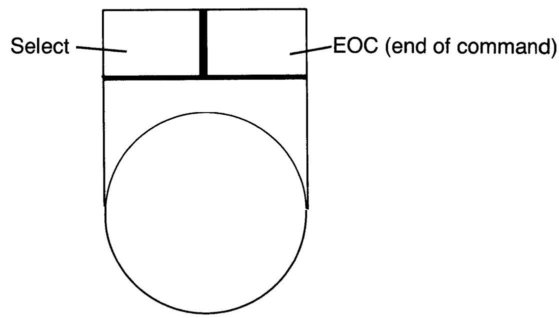
Moving the Cursor

The cursor is the "+" that marks the location on the screen where the next character you enter will appear. Two main devices are available to move the cursor across the screen: a graphics tablet and a mouse. You may also use arrow keys to move the cursor. These items are called graphics input devices.

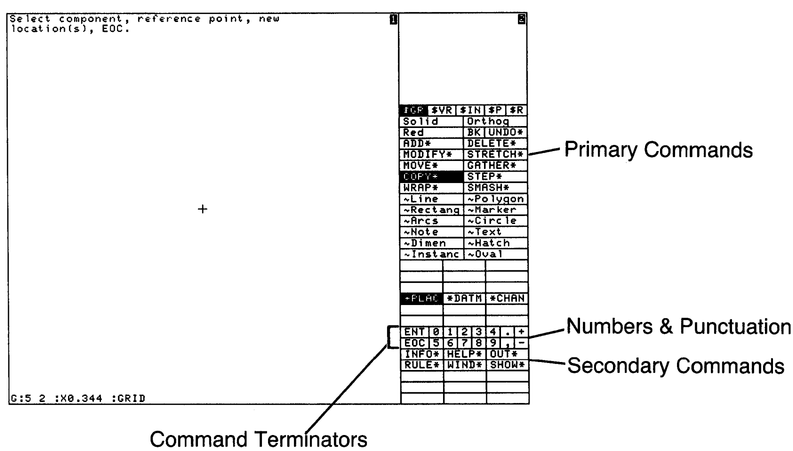
If you have a graphics tablet, pick up its stylus and move the tip across the surface. Notice that only the center portion of the tablet affects cursor movement. This allows you to rest your hand on the tablet while moving the stylus.

If you have an HP 46060 mouse, position the mouse with the cord pointing away from you; then move the mouse across the surface of your desk. Pressing the right button generates an EOC (end of command) producing the same result as typing (;) (**Return**). Pressing the left button on the mouse is the same as pressing (**Select**); use this command to select commands from the graphics menu or to select an xy point on the drawing area.

As you move the cursor in either viewport, the values displayed in the lower right corner of viewport 1 indicate the cursor coordinates, in user units. For General Drawing, user units are in millimetres.



HP 46060A MOUSE



Screen Menus

Screen Menus

The remaining screen area displays the General Drawing screen menu. The menu in each of the other HP EGS personalities is similar.

Elements found on the menus of the four HP EGS personalities are:

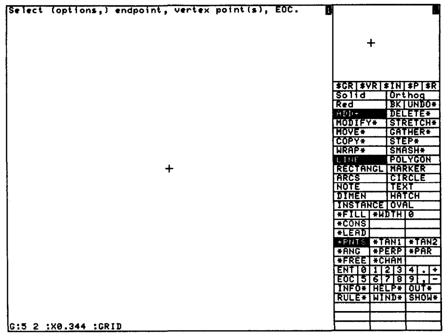
- Modes such as $GR, Solid, and Red. Modes are always at the top of the menu. They are white on color systems.
- Commands such as ADD* and WIND*. Commands are always capitalized. They have a trailing asterisk (*) and are green on color systems. Commands listed in the top portion of the menu are primary commands; those near the bottom of the menu are secondary commands. The differences between primary and secondary commands are shown later.
- Lists of drawing components such as LINE and CIRCLE. Components are always capitalized and are cyan on color systems. Components are listed only when the primary command ADD* is active.
- Lists of drawing component restrictors such as ~Line and ~Circle. The first character is capitalized. Component restrictors are always preceded by a tilde (~), and are cyan on color systems. These are used to restrict the selection of components.
- Lists of command options such as *WDTH (to specify the width of a component) and *FILL (to fill a component). Options are typically capitalized, have a leading asterisk (*), and are yellow on color systems. Only those options appropriate for the selected command are displayed.
- Lists of command methods such as *INBT (for delete in between) or *R&C (for adding a circle with a specified radius at a center point). Methods are typically capitalized, have a leading asterisk (*), and are cyan on color systems. Only those methods appropriate for the selected command are displayed when the command is active.
- Numbers and punctuation for use when "typing" from the screen. The numbers and punctuation are located just above the secondary commands and are cyan on color systems.
- The ENT and EOC command terminators. ENT (enter) acts the same as the (ENTER) or (Return) keys on the keyboard. EOC (end of command) produces the same results as typing (;)(Return) on the keyboard. If you are using the HP 46060 mouse, pressing the right button also generates an EOC. These command terminators are located just above the secondary commands and are yellow on color systems.

The menus change to reflect the available choices as you work on your drawing.

Selecting Menu Items

You can select items from the menu using either a mouse or a tablet. When you select an item, that mode, command, component, option, or method is generally highlighted. The highlighting reminds you what task you are currently performing.

This figure shows ADD*, LINE, and *PNTS highlighted. This combination of selected items means that you can add as many lines as you want when using the *PNTS (points) method. At this stage you may select anything displayed on the menu. You can select a new command or a different component to ADD*, or change the method or option for adding a line.



Alpha and Graphics Displays

Your computer has two modes of display: alpha and graphics. You may also have an external monitor for graphic display.

When you type from the keyboard, your input is displayed on an inverse video line near the bottom of the graphics display. You may type up to 255 characters on this line. As soon as you press (Return) to accept the input, the inverse video line disappears and the entire graphics display is visible.

If you are using an external monitor for your graphics display, the alpha image always appears on the internal CRT.

DISPLAY command is explained in the *HP EGS Syntax Reference*.

Cursor Snapping Modes

Before you begin using HP EGS, you need to know how points are actually selected in the drawing area. In this section you will learn about the cursor snapping modes ($GR, $VR, $IN, $P and $R). You will also learn how to display and change the user grid with WIND*, grON, grOFF, REDRW, RULE*, and grNEW.

To select a point in the drawing area, you must move the cursor to the desired location on the screen with your stylus or mouse and then "digitize" the point. If you are using a graphics tablet, digitize by pressing down on the stylus. If you are using a mouse, digitize by pressing the left button. (Position the mouse with the cord pointing away from you.)

You do not always need to put the cursor directly over each point you select, because the cursor will "snap" to points (when you digitize) as defined by the current snapping mode. The snapping modes are:

$GR	Snaps to the nearest user grid point, whether it is visible or not.
$VR	Snaps to the nearest vertex point. Vertices include endpoints of lines and arcs, center points of arcs and circles and corner or "bend" points of lines, rectangles and polygons.
$IN	Snaps to the nearest intersection point. Intersection points are points where any two components cross or touch, and also include all vertices.
$P	Snaps to the nearest point on the closest component (primitive).
$R	Snaps to the nearest system (raw) point.

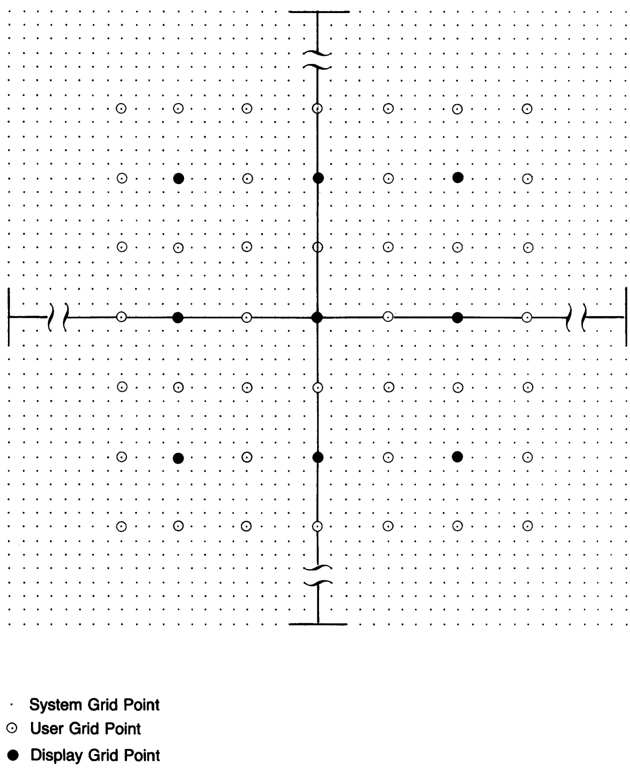
Each snapping mode stays in effect until you change it. The size of your cursor determines the size of the area in which HP EGS will search for points to snap to in the vertex, intersection, and primitive snapping modes. If a valid snapping point is not found in the vertex, intersection, or primitive snapping modes, HP EGS snaps to the nearest user grid point. (Changing the cursor size is explained later.)

You are probably now wondering what we mean by "user grid" and "system grid". First let's understand the system grid.

The system grid defines all of the points which HP EGS can use in describing a drawing. In other words, any vector which HP EGS draws cannot be any shorter than the distance between two system grid points. You can use the UNITS command to set the user units and the system grid. The General Drawing personality uses this command to set the user units to be millimetres and the distance between each system grid point to be 0.1 millimetre. The raw snapping mode can be used to snap to these points.

The user grid defines discrete system grid points which are snapped to in the grid snapping mode (and as defaults to the vertex, intersection, and primitive snapping modes as described above). HP EGS allows you to define the size of the user grid, the alignment of the user grid, and the display of the user grid.

This figure shows the differences between the system grid and the user grid. Displayed user grid points are also shown. By not displaying some of the user grid points you help to keep your drawing area uncluttered. The display grid is a subset of the user grid and is used for display purposes. All user grid points remain active for snapping, whether they are displayed or not.

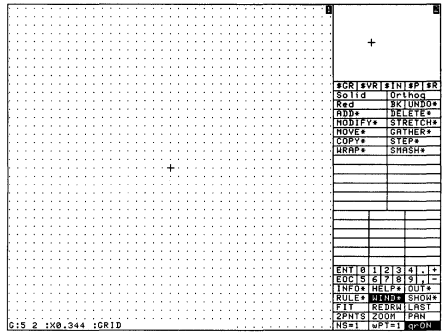


The User Grid and System Grid

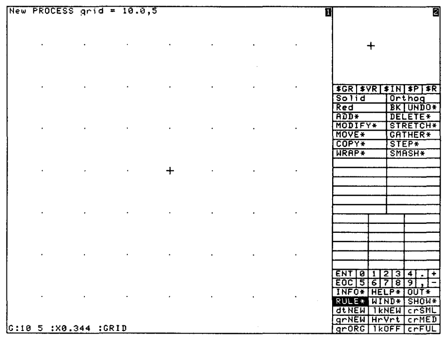
To understand the concepts, try the following:

1. First, let's make the display grid visible. From the secondary command area of the screen menu, select:

   `WIND* grOFF REDRW`



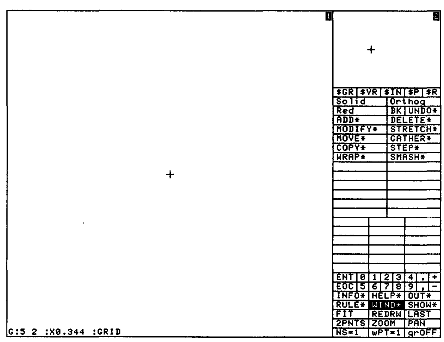
The display grid appears on your screen, as shown. Notice the values of the numbers next to `G:` on the bottom of your display. The `5` next to the `G:` means that there is a user grid point every 5 millimetres. The `2` indicates that every second user grid point is displayed.



2. Next, display every fifth user grid point. Select:

   `RULE* grNEW 10 ENT 5 ENT`

   Notice that the numbers next to `G:` have changed, as shown. Now you have a user grid point every 10 mm and a display grid point every 50 mm.



3. Return the grid to its original values by selecting:

   `grNEW 5 ENT 2 ENT`

4. To remove the grid display from your screen, select:

   `WIND* grON REDRW`

   Your screen should look like this:

5. Return to the menu by typing:

   `QUIT;` **Return**

BREAK TIME

Take a minute to stretch and to review what you have done in this section:

- Learned ways to enter commands.
- Learned about viewports.
- Moved the cursor.
- Learned about screens and screen menus.
- Selected menu items.
- Learned about cursor snapping modes.

General Drawings

Chapter 3: A Drawing Example

Chapter 4: A More Complex Drawing

Chapter 5: Creating an Isometric Drawing

A Drawing Example

Chapter 3

Loading General Drawing

1. If your computer is not displaying the main menu in the HP EGS manager, follow the instructions in Chapter 2, in the *Loading General Drawing* section, to get to the main menu. Your display should look like this:

```
Use arrow Keys to move cursor to desired function.
Select function with <RETURN> or <ENTER>.
  ->System Utilities
    General Drawing Editor
    Electrical Engineering
    Mechanical Engineering
```

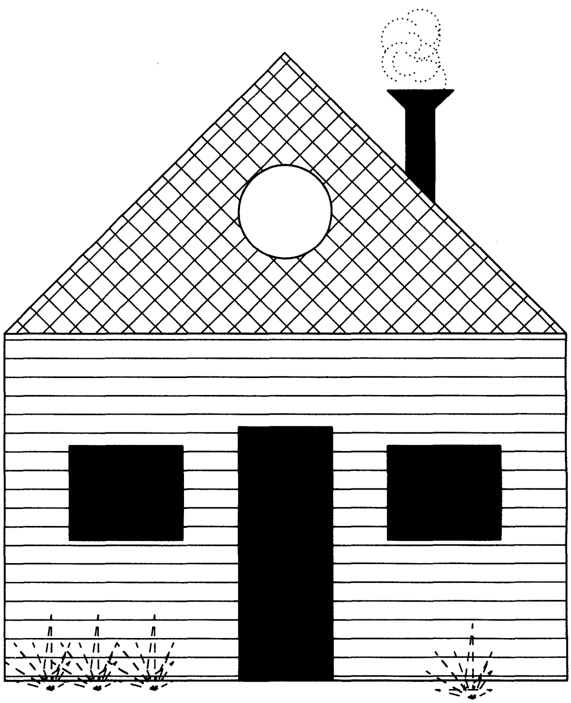
2. Move the cursor to the General Drawing Editor function and press **Return**.
3. Change the default volume from the system volume to your own volume to be sure your drawings are not saved in the HP EGS volumes. The default volume is the first volume the system searches when no volume is specified.

   The tutorial uses a volume named "DRAW:" to store the files created in the following pages. To change the default volume from the system volume to DRAW: type:

```
Prefix DRAW:;
```

Previewing the Drawing

To practice what you learned in Chapter 2, you are going to make a drawing of a house that looks like this:



To see on the screen what your finished house will look like, find the floppy disc labeled EWLRN: and put it in one of your disc drives. Then type:

```
INPUT EWLRN:chap3; (Return)
```

When the complete drawing appears on your screen, the system prompts:

```
Press <space> to continue.
```

Press the space bar. This stops the preview of the completed house and resets the General Drawing personality so you may begin drawing the house step by step.

Note

Once the preview begins do not disturb the computer. It is important to complete the demonstration so that the General Drawing personality is reset to its initial state.

Drawing the Basic House

In this section you will create a new drawing and then add components to it. You will learn to use construction lines and to fill in, delete, copy, and stretch components. You will also learn how to undo, or reverse, a command with UNDO* and how to back up inside some commands.

The first step in drawing the house is setting a clear screen and selecting the primitive you want to use. A primitive is a basic shape: line, rectangle, circle, or polygon.

Enter:

`EDIT;` **(Return)**

A blank drawing is now ready for you to use.

Note

From this point on, "Enter" means to type what follows and then either press **(Return)** or select `ENT` in the menu. "Select" means to use your graphics input device to move the cursor, and then digitize the selection.

If you make a mistake and want to reverse the last command, select:

`UNDO*`

For example, if the previous command was to delete a rectangle, selecting the UNDO* command causes the rectangle to be restored to the drawing. UNDO* can only reverse the effects of the last primary command entered.

Adding a Primitive

You can use the ADD* command to add any primitive (also called a component) to the drawing displayed in the viewports. Different options are available with the ADD* command, depending on the component being added.

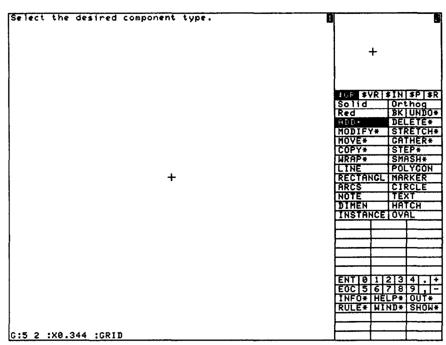
1. The first step in building the house is to set the snapping mode to grid. Select:

   `$GR`

2. Next, select:

   `ADD*`

   The system highlights `ADD*` in inverse video and displays the list of components you can add. Your display should now look like this:



3. Select:

   ```
   RECTANGL
   ```

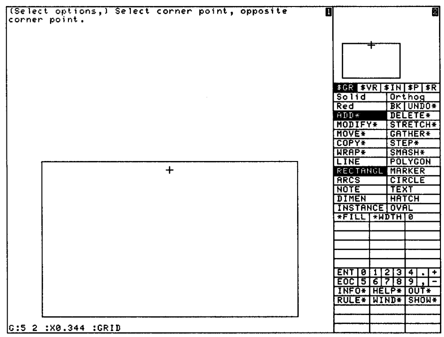
   A rectangle is a parallelogram with right angles. Because of this special property, a rectangle is defined by only two x,y locations. The first point establishes one corner. The opposite corner is tracked until you enter the second point.

   The prompt line at the top of your display tells you what to do for your chosen operation. In this case it tells you how to add a rectangle. Notice that the menu now displays several options.

4. Move the cursor until the numbers in the lower right corner of the drawing area read approximately (−150, −175). Select the point.

   Notice there are now two sets of x,y coordinates in the lower right corner of the graphics display. The top set reports current cursor position; the bottom set reports the last point snapped to.

5. Move the cursor up and to the right until the numbers read approximately (150, 10). Notice how the rectangle expands as you move the cursor. This is called dynamic tracking. Select the points. A rectangle is drawn in both viewports. The rectangle is a solid red line if you have a color system, because the color mode is set to red and the line-style mode is set to solid. Your screen should look like this:



Filling in Items

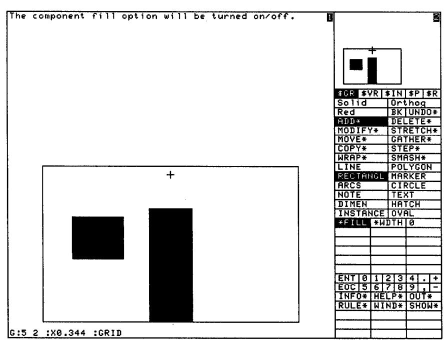
You can fill in selected parts of a drawing with color by using the *FILL option.

1. Select from the menu:

   ```
   *FILL
   ```

2. Add a second rectangle for the door by first moving your cursor to (−25, −175) and selecting the point. Then move the cursor to (25, −40) and press again. As long as you are adding the same type of component, you don't need to select the ADD* command or the *FILL option and RECTANGL component each time.

3. Add a rectangle for a window. Move the cursor to (−115, −100) and select the point. Now place the cursor at (−55, −50) and select the point. A filled-in window appears:



Copying Components

The COPY* command enables you to place a copy of any component located in either viewport at any location in the drawing.

1. To add a second window on the other side of the door, select:

   `COPY*`

   The default copy method is *PLAC, or place copies of components. This is the method you should use. The prompt line at the top of the screen is:

   `Select component, reference point, new location(s), EOC.`

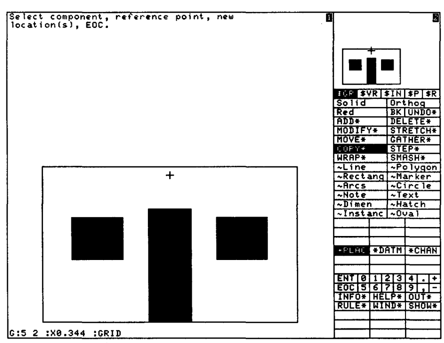
2. Select the component by selecting a point that lies near the the component to be copied. Optionally, you may select a component restrictor to limit selection of the component to be copied. You get to use this feature later. Once you select the component it is highlighted.
3. Select a reference point, also called the origin, so that you can accurately place the copy of the component. To select the origin, choose a point on the window. We recommend the lower left corner. If you consistently pick the same point to be the origin, such as the lower left corner, it is easier to remember later. Once you select the origin a copy of the component will follow your cursor.
4. Finally, select locations where you wish to place copies of the component. You can enter x,y locations by digitizing the position of the cursor, by selecting numbers from the menu, or by entering numbers from the keyboard. The copies are placed such that the origin point of the component is placed exactly on the placement point. Move the cursor to (55, – 100) and select the point. A copy of the window is now drawn at this location.
5. The cursor is still moving a copy of the window shape in case you want another copy. You do not, so move the cursor to the bottom of the menu and select:

   `EOC`

   This signifies the end of the command. It completes copying this component and leaves you ready to copy another component. Commands can be terminated only by entering another command.

   Complete commands are still active; terminated commands are no longer active and must be re-entered to use them again.

   After you copy the window, your house should look like this:



Correcting x,y Location Errors

If you make a mistake while selecting an x,y location, correct your error by pressing:

```
BK
```

The backup command works with some commands that require entering x,y data points. BK enables you to delete, or back over, some x,y points that have been entered. BK works with:

```
ADD* LINE *PNTS
ADD* POLYGON
ADD* HATCH
COPY*
STRETCH* *REPL
GATHER*
WRAP*
WIND* 2PNTS
```

Data points cannot be erased with the backup command once the command has been completed and/or terminated.

The x,y locations are deleted in a last in, first out fashion. For example:

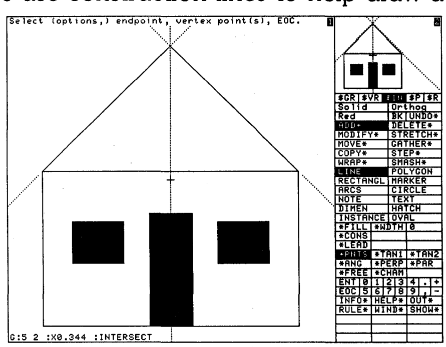
ADD* LINE x_1,y_1 x_2,y_2 x_3,y_3 x_4,y_4 BK BK

The last two points (x_3,y_3 and x_4,y_4) are deleted. Because the command has not been completed yet, you may enter new x,y points. The backup command is, in effect, an electronic eraser that enables you to correct mistakes as they are made, instead of deleting the entry and starting over.

Using Construction Lines

When drawing a figure, you often may need construction lines to help you lay out drawings and set up relationships between components. Let's use construction lines to help draw a roof on the house.

To add the roof, you need two lines that rise from the top corners of the rectangle and meet at an angle of 45 degrees. Here is how the house will look after you add the roof:



The easiest way to draw the lines is to add two 45-degree construction lines at the top corners of the rectangle. Then add the solid roof line once you know the peak intersection.

1. To accurately select all of the vertices and intersections, set the snapping mode to intersect:

   `$IN`

   Notice that `:INTERSECT` appears at the bottom of the viewport and `$IN` is highlighted in the menu.

   You can differentiate between the existing geometry and the construction lines you are about to add by changing the line-style to short dotted. With dotted construction lines you can see the roof line when it is added on top of the construction lines.

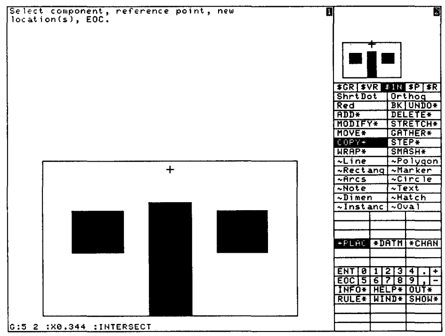
2. To disengage the currently selected line-style, select:

   `Solid`

3. Select the new line-style:

   `*ShrtDot`

   Notice that the menu cell which used to say `Solid` now says `ShrtDot`. This is how the General Drawing personality shows you the current line-style. Your screen should look like this with the same menu items highlighted:



4. To draw the roof construction lines, select:

   `ADD* LINE`

   Several options are now presented which allow different kinds of lines to be added. These options are:

Option	**Description**
*FILL	Turns on and off the fill of lines which have width.
*WDTH	Sets the width of lines. The currently set width is displayed in the menu to the right of the width option.
*CONS	Creates a line with infinite length and is defined by two points.
*LEAD	Automatically places an arrowhead at the endpoint of the line you specify first.

5. Because you want to create construction lines, select:

   `*CONS`

   Several different methods for adding lines appear:

Option	Description
*PNTS	Defines two x,y locations representing endpoints of a line and other discrete x,y locations, representing the junctions of adjacent segments, called vertices.
*TAN1	Defines a point and a tangent point to an arc or a circle.
*TAN2	Defines a line tangent to two arcs or circles.
*ANG	Defines a line by a point, an angle, and an arm length.
*PERP	Defines a line perpendicular to a line.
*PAR	Defines a line parallel to a line.
*FREE	Defines two x,y locations representing the endpoints of a line and draws, in a freehand fashion, all of the vertices in between.
*CHAM	Selects a corner to chamfer, or bevel. The chamfer is defined by an angle and a distance from an edge. Make symmetric chamfers by specifying an angle of 0. The corner must be at the endpoints of two, and only two, line, rectangle, or polygon segments.

The *PNTS (points) method is highlighted because it is the default method. In other words, you can immediately select points for adding a line once you have selected the LINE component.

6. Because you want to add lines at specific points and at specific angles, select:

   `*ANG`

   which is point angle and length. The prompt is:

   `Select (options,) endpoint, ENTer angle, length.`

7. Select the upper left corner of the house. Because the intersection snapping mode is on, you snap to the correct corner if that point is within the rectangle defined by your cursor.
8. You want a 45-degree angle to the roof, so enter:

   `45` (Remember, "enter" means press **Return** or select `ENT`.)

9. Now, from the numbers on the menu, place the second point of the construction line by entering:

    ```
    1
    ```

    The length can be any value for construction lines since they have infinite length. For other lines you may enter specific values.

10. To add the second construction line, select the upper right corner of the house and enter:

    ```
    -45
    ```

11. Then enter:

    ```
    1
    ```

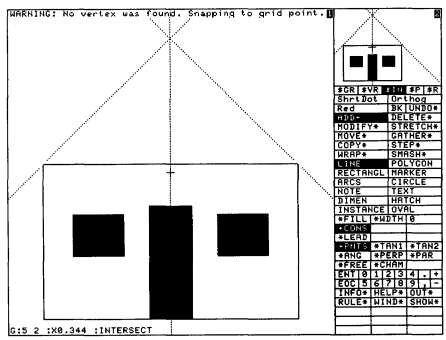
    Because the `*ANG` method is still active you do not have to reselect it.

    Later you will add a circular window to the center of the second story. To add the window you will need a vertical construction line through the center of your house. This construction line is added most easily by the *PNTS method.

12. Select:

    ```
    *PNTS
    ```

13. Select the intersection of the two existing contruction lines defining the peak of the roof. Notice that you are dynamically tracking a construction line, which needs only a second point to complete. Move the cursor until the construction line is vertical (around 0,0) and select the point. Your house should now look like this:



Adding the Roof Line

To add the actual line which represents the roof of the house, you must first change the line-style to solid.

1. Select:

    ```
    ShrtDot *Solid
    ```

2. When you change color, line-style, or axis modes (secondary commands), the General Drawing personality remembers the last command you used and re-executes it. So to add the roof line you need only to select:

    ```
    LINE
    ```

3. Turn off the construction line option by selecting:

   `*CONS`

4. Move the cursor to the left side of the roof (−150,10) and select the point. Then move to the intersection of the construction lines at the peak and select the point. Then move to the right side of the roof and select the point (150,10).
5. Notice that you are still tracking the line. To finish the line you need to complete or terminate the current add line command. You can either select EOC (end-of-command), select another command, or select the last point of your line again. For now, select:

   `EOC`

Adding a Circle

Here's how to add a circular window to the center of the second story of the house:

1. Select:

   `CIRCLE`

   As with `LINE`, some options and methods appear in the menu. You have seen the fill and width options. Unlike a line, a circle without width can be filled.

Many of the methods for adding circles have not yet been discussed:

Method	Description
*C&C	By two x,y locations: the center of the circle and any point on its circumference.
*DIAM	By two points which define the diameter.
*3PTS	By three non-colinear points which define a circle.
*R&C	By a radius and center point.
*TAN1	By a circle center point followed by a point tangent to another primitive.
*TAN2	By a circle tangent to any other two primitives. Enter the radius first. Then select two primitives. Then select an approximate center point.

The default method, *C&C (by a center and circumference point), is highlighted. This is the best method to use for the second-story window.

2. Place the cursor on the construction line about halfway between the roof peak and the top of the first floor rectangle. Select that point. As you move the cursor away from the line, the circumference of the circle is dynamically tracked. Make the circle the size you want and select that point.

Deleting Components

The DELETE* command enables you to delete a component or a section of a component from a drawing.

1. Clean up the drawing of the house by removing the construction lines. Select:

   ```
   DELETE*
   ```

   The default method, *COMP (complete component), is the one you need. The following prompt appears:

   ```
   Select component, EOC.
   ```

   The component you select with the DELETE* command is not deleted until you either select EOC or select the same point again. In this way you can re-select the component to be deleted if you make an error. Optionally, you may select a component restrictor to limit selection of the component.
2. Place the cursor near the vertical construction line and select a point.
3. When the line to be deleted is highlighted, select:

   ```
   EOC
   ```
4. Delete the remaining two construction lines by repeating steps 2 and 3.

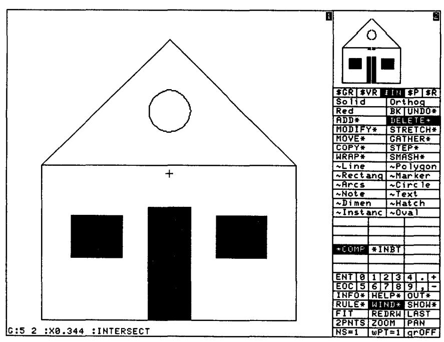
   Notice that the line defining the roof seems to disappear. Actually, when HP EGS un-draws the highlighted construction lines it also un-draws the coincident portions of the roof line.
5. To see that the roof line is still there, redraw the screen. Select:

   ```
   WIND* REDRW
   ```

   Note that you can use the WIND* REDRW sequence at any time to clean up your screen.

   You may delete a section of a primitive by selecting the method *INBT from the menu. Select one point defining the end of the section to be deleted and a second point defining the other end. Then select the part of the component to be deleted. Components which can be partially deleted include: lines, circles, arcs, rectangles and polygons.

   After drawing the circle and deleting the construction lines, your house should look like this:



Adding a Polygon

Next, let's add a chimney to the house.

1. First select the grid snapping mode:

   `$GR`

2. Then select:

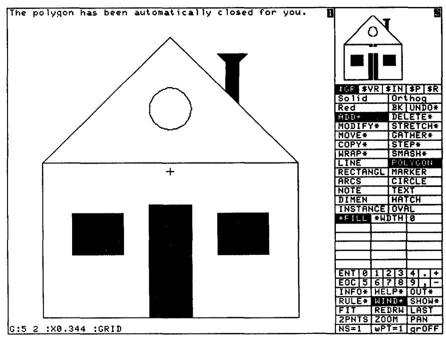
   `ADD* POLYGON`

   The polygon component is a closed plane figure bounded by straight sides. A polygon component is defined by specifying the x,y locations of its vertices in order. It must have at least three vertices. The options for a polygon are the same as for a circle: `*FILL` and `*WIDTH`. There are no alternate methods for polygons.

3. Select:

   `*FILL`

4. Move the cursor to a point on the roof where you want the chimney. Draw the shape you wish, ending the final side near the roof.
5. Select EOC. The system completes the final side of the polygon and fills it. Your house should look similar to this:



Stretching a Shape

After you add the chimney to your house, you can use the STRETCH* command to change its size or shape. Only arcs, circles, lines, polygons, rectangles, and dimensions may be stretched.

To stretch a component using the default method, *NORM (normal), first select the primitive to be stretched. Next, select the part of the primitive you want stretched. You may stretch the endpoints, vertex points, and entire edges of line segments. For polygons and rectangles, you may stretch vertex points and edges. You may stretch either endpoint or the edge of an arc. For circles, you may stretch the radius. For dimensions, you may stretch the endpoints of the leader lines or the endpoints of the extension lines.

Once you have selected the reference point, the system will automatically track the stretched piece of the component with the cursor. This dynamic component tracking allows you to see the results before they are permanent. Once you have moved the cursor to the desired location, select the point.

1. To stretch the chimney, select:

   ```
   STRETCH*
   ```

   You want the default method, *NORM, so you do not need to reselect it.
2. Move the cursor to the top center of the chimney. Select the chimney, then select a point in the center top part of the chimney. Move the cursor and notice the stretching. Select a new point.
3. To reverse the effects of this procedure, select:

   ```
   UNDO*
   ```

   The stretch command has other options which you will explore later.

BREAK TIME

Take another minute to relax and review what you have done in this section:

- Added a primitive.
- Filled in an item.
- Copied components.
- Corrected x,y location errors.
- Used construction lines.
- Changed construction lines to permanent lines.
- Added a circle.
- Deleted components.
- Added a polygon.
- Stretched a shape.

Using the HELP* Command

The HELP* command is a secondary command that provides an online reference for all of the HP EGS personalities. Recall that the secondary commands are located in the lower portion of the menu.

A secondary command is terminated by another command or EOC, unlike a primary command which can be terminated only by entering another primary command.

1. Find the secondary commands and select:

   ```
   HELP*
   ```

   The prompt is:

   ```
   Select or ENTer command/option/method/mode for help, then EOC.
   ```

   You can get help on any menu item just by selecting it. Or, you can enter anything which you have seen in the menu from the keyboard. You must terminate the HELP* command with EOC before you can see any response.

2. To get helpful information about the window command, select:

   ```
   WIND* EOC
   ```

   The following display appears:

```
Secondary Command WIND*

    WHAT = Window the current port.

     HOW = Set the NS=1, wPT=1/wPT=2 and grON/grOFF options appropriately, then
           select the desired window method.

 OPTIONS = NS=1          - Set the nesting level of the current port.
         = wPT=1/wPT=2 - Set the current port for windowing or turn port 2 off.
         = grON/grOFF   - Turn on or off the display of the grid.

 METHODS = FIT   - Fit the entire drawing to the current port as large as
                   possible.
         = REDRW - Redraw the current port.
         = LAST  - Redraw the current port as it was before the last window
                   command.
         = 2PNTS - Define a rectangular area on the drawing to be fit to the
                   current port.
         = ZOOM  - Zoom the current port in or out.
         = PAN   - Define a point on the drawing to be panned (moved) to the
                   center of the current port.
```

All help messages are structured in this fashion:

- The first line contains the type of operation (such as primary command, option, or method) and the name of the operation.
- WHAT describes what the operation does.
- HOW describes how to make the operation work. HOW is a long version of the prompt which the operation will normally present.
- After HOW are other items which this operation presents to you when the menu item is invoked. In the example above these items are OPTIONS and METHODS.
- NOTES provides any miscellaneous information concerning the operation.

Every operation may not have all of these sections. For example, the help for WIND* doesn't have NOTES.

You should recognize the grON/grOFF option and REDRW method you used previously. Read the descriptions in the help message and compare this with what you did (more on WIND* later).

Notice that the last line of the help message says:

```
Press <space> to continue, press <shift-select> to abort.
```

Pressing the space bar displays each page of the help message. Pressing again erases the message and returns to the graphics display.

3. Press the space bar.

   You can also immediately erase the help message and return to the graphics display by pressing (Shift)-(Select).

Zooming In on an Item

If you want to add a line in a very small area, you can zoom in on the small area using the secondary command WIND*, and then continue adding the points.

To try this, let's add a new line to your house as shown in the next figure.

1. Select:

   ```
   ADD* LINE
   ```

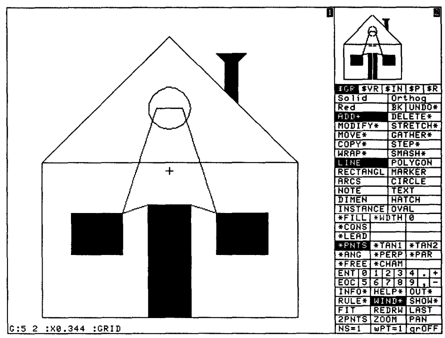
2. Start your line at the top left corner of the door. Select the top right corner of the left window for your second point.

3. It would now be easiest to add the short horizontal segment inside the circular window if the circular window filled the whole display. You can use the WIND* command and the 2PNTS method to zoom in on the window. If `WIND*` is still highlighted you can directly select:

   `2PNTS`

   (If not, select `WIND*` first.)

4. Select points (−35,40) and (35,105) to define a rectangular area around the circular window in the second floor of your house. When the display has redrawn, notice that the circular window fills your screen. Also notice that you are now dynamically tracking the continuation of your line.
5. Select two points inside the circle defining a short horizontal segment.
6. To complete the line you now need to display the house as it was before you used `2PNTS`. Select:



   `LAST`

   This is a `WIND*` method which redraws the screen as it was before the last change. Again notice that you are dynamically tracking the line. Draw the line to the top left corner of the right window and then to the top right corner of the door. Remember that to complete your line you must select `EOC`.

7. Now get rid of the spurious line you just added. Select:

   `UNDO*`

Saving Your Drawing

Now that your house is cleaned up you can save it on your disc.

1. Select:

   `INFO* VOL?`

   The screen displays all of the numbers and names of the volumes your system recognizes. This tells you what volumes you can store a file or drawing on.

Notice that the numbers all begin with a number sign (#) and that the names are characters followed by a colon (:). A volume name can be up to 6 characters, either uppercase or lowercase. If you are using a Shared Resource Management (SRM) system, ask your system manager for the conventions used to name volumes.

2. Select:

   `CAT?`

3. Enter:

   `EWSYS:`

   The system lists all files on that volume, like all the chapters in a book. You can then pick a name for your drawing that is not a duplicate of those already stored. Use `CAT?` to list the current files of any volume.

4. From the secondary command menu, select:

   `OUT* SAVE*`

   The SAVE* command has two methods. The first method, OLD, saves a drawing in the file it came from. You can use this only if your drawing is already named. The second method, NAME, saves a drawing in a new file (and also names your drawing if it is un-named).

5. Because your drawing does not yet have a name, select:

   `NAME`

   The prompt is:

   `ENTer the name of the file in which to save the drawing.`

6. Enter:

   `house`

   A copy of your drawing is saved on the default volume for General Drawing, EWSYS, as EWSYS:house_d. The prompt on your screen tells you that the drawing "house_d" has been saved. HP EGS automatically appends the "_d" to drawing names so that it can find them easily.

Printing and Plotting Your Drawing

If you want a quick copy of your drawing and a graphics printer is attached directly to your computer, select:

```
DUMP*
```

The DUMP* command copies the graphics as it appears on the CRT to the system printer.

To get a plot of your drawing, and you have a plotter attached directly to your computer, select:

```
PLOT*
```

The prompt is:

```
(Select methods, then) EOC.
```

This means that you may select from the following methods before selecting `EOC`.

Method	**Description**
CENTR	Both scaled and unscaled plots centered on the plotter. If unscaled, as large as possible on the paper.
SCALE	Set the absolute scale of the plot.
FILL	Fill option which causes primitives with width to be filled solid as they are plotted.
VLCTY	Set the pen speed with which the drawing is to be plotted.
SPOOL	Send plot information to a file, designating a paper size and plotter type to be used.
HPTW	Send plot information to a file, ready to be pre-processed by HP TechWriter.

If you do not specify any methods for plotting, only the portion of the drawing in the large window on your screen will be plotted. The CENTR method is typically used for plotting because it plots the entire drawing as large as possible centered on the plotter.

Select:

```
CENTR EOC
```

Plots may be scaled or not scaled. If a plot is not scaled, the system plots the portion of the drawing displayed in the large viewport on the plotter. The lower left corner of the window is mapped to the lower left corner of the plotter. (P1 on HP plotters.) The drawing is plotted so that it occupies the largest possible area on the plotter without distorting the drawing. (This area is defined by P2 on HP plotters.)

If a plot is scaled, the drawing is plotted, multiplied by the scale factor supplied. (Remember that the user units in General Drawing are millimetres.)

For example, suppose that the units of a drawing are in feet and the drawing ranges from -20 feet to +20 feet in user units. If the drawing is plotted with the scale factor 2, the system attempts to plot the drawing from -40 feet to +40 feet, an obvious limitation for an 11 x 17 inch plotter, such as the HP 7475.

The personalities provided with HP EGS do not use all of the capabilities of the PLOT command. If you have specific plotting needs see the *HP EGS Syntax Reference* for more details. (An example of another plotting capability is pen width specification. This becomes important when filling components.)

The PLOT* command can be aborted by pressing (Stop). Only those portions of the drawing in active layers that have been turned on with the SHOW* and WIND* commands are plotted (more about layers and SHOW* later).

BREAK TIME

Once again, take a few minutes to stretch, relax, and review what you have done in this section:

- Used the HELP* command.
- Zoomed in on an item.
- Saved your drawing.
- Learned how to print and plot your drawing.

Adding the Trimmings

In this section you can practice changing the color and the line-style, adding lines, and copying. You will also add arcs and produce hatching. Finally, you will use the SHOW* command and learn how to end your editing session with the QUIT command.

Adding Arcs

First, let's use arcs to draw smoke coming out of the chimney. An arc is a section of a circle.

1. Change the color to blue and the line-style to short dotted. Select:

   ```
   Red *Blue
   ```

   ```
   Solid *ShrtDot
   ```

2. You should still be in the ADD* command, so select:

   ```
   ARCS
   ```

   As with LINE and CIRCLE, some options and methods now appear in the menu. You will recognize the fill and width options. But note that, like lines, arcs without width cannot be filled.

   Many of the methods of adding arcs have not yet been discussed:

Method	Description
*3PTS	By two x,y locations, called the endpoints of the arc, and a third point, which lies on the arc.
*ANGS	By a radius; two angles, one starting and one finishing; and a center point.
*FILT	By entering a radius value followed by the location of the corner where you want to place the arc. This corner must be at the endpoints of two and only two line, rectangle, or polygon segments.
*SEMI	By defining two points which represent a half circle.
*TAN2	By entering a radius value and then selecting two primitives to which the arc will be tangent and an approximate center.

3. Select the method:

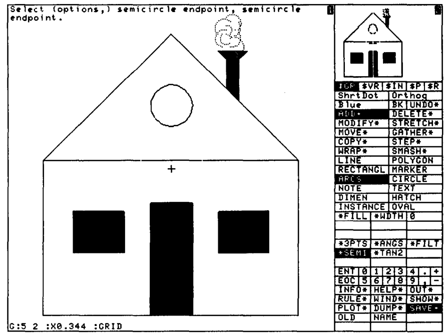
   ```
   *SEMI
   ```

   Note that HP EGS draws arcs in a counter-clockwise direction from the first point specified.

4. Select:

   ```
   $GR
   ```

   if it is not already set, to snap to grid points.
5. Then add a few arcs above the chimney to represent smoke. Your house should look similar to this house:



Using Hatching

Hatching is a separate component from those which define the hatch boundary. Be sure to follow the hatching directions exactly or you may get incorrect results.

The process for defining a hatch involves:

- Defining the hatch angle, hatch style, and distance between hatch lines.
- Defining any holes to be left unhatched.
- Defining the outside boundary of the area to be hatched.

The options that hatch presents to help you with this process are:

Option	Description
*HOLE	Specifies what follows to be a single hole in the hatching area.
*ANGL	Sets the angle of the hatch.
*STYL	Sets the style of the hatch. The value 1 specifies a solid fill, 2 a single hatch, and 3 a cross hatch.
*SPAC	Sets the spacing between single and cross hatch lines.

1. First, change the color to yellow and the line-type to solid. Select:

   ```
   Blue *Yellow

   ShrtDot *Solid

   HATCH
   ```

2. For the siding on the first floor you need a 0 degree, single hatch with a 10 mm distance between hatch lines. Because single hatch and 10 mm spacing are the default values, you need only to set the *ANGL value. Select:

   ```
   *ANGL 0 ENT
   ```

   In the next step you will define any area within the hatching area which you do not want hatched. For siding on the first floor you need to specify both windows and the door. To define the boundary of the holes, either select an existing closed component (rectangle, polygon, or circle) while in the primitive ($P) snapping mode, or define the area to be hatched point by point.

3. Because the windows and the door are single components, as opposed to shapes made up of 2 or more components, select:

   ```
   $P *HOLE
   ```

   Now select a point on the outline of one of the windows. The holes you select are highlighted and the system will not hatch over them.

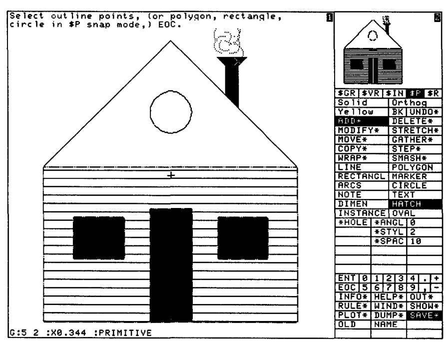
4. Again select:

   ```
   *HOLE
   ```

   and select a point on the outline of the other window. Repeat this procedure for the door. You must select *HOLE each time you identify a hole in the hatching area. If you do not select *HOLE each time you define a hole, HP EGS assumes that you are defining the outside boundary of the hatched area.

5. Define the outside boundary of the hatched area by either selecting an existing closed component (rectangle, polygon, or circle) to be hatched, or by defining the area to be hatched point by point. Because you are still in the $P (primitive) snapping mode, you need to pick only a single point on the rectangle representing the first floor of the house.

   The outline is highlighted and the yellow hatching added to the rectangle. Your house should now look like this:



6. To hatch the second floor of the house with a hatch rotation of 45 degrees, a cross hatch style, and a hatching width of 10 mm, select:

   `*ANGL 45 ENT *STYL CROSS`

   Note that the General Drawing personality displays `SOLID`, `SINGL`, and `CROSS` when you select *STYL. This keeps you from having to remember which hatch styles go with the numbers 1, 2, and 3.

7. Specify the circular window by selecting the hole option and a point on the circle.

   The boundary hatch for the second floor is defined by a line and a portion of a rectangle, not a complete component. So you must specify the hatch boundary by individual points this time.

8. To snap to intersections, select:

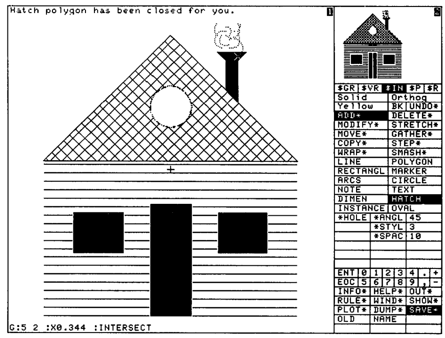
   `$IN`

9. Then select the bottom left, top center, and lower right corners of the triangular area you want to hatch.

10. To close the hatch boundary, select:

    `EOC`

    Your house should now look like this:



Adding Multivertex Lines

Now let's add some shrubbery to the front yard. You can add a bush by specifying a bunch of vertices for a multi-vertex line. This is the default method for adding a line, so no method selection is necessary. Approximate the shape of a bush using the next figure as a guide.

1. To change the color to green and the line-style to dashed, select:

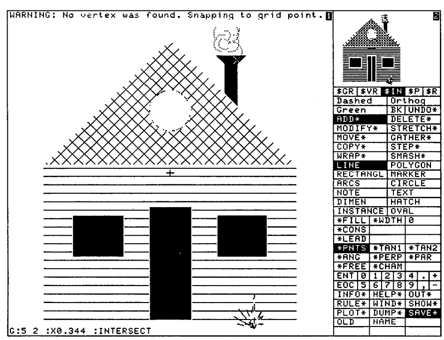
   `Yellow *Green Solid *Dashed`

   `LINE`

2. Draw a bush. When you are happy with the bush, select:

   `EOC`

   The green bush is added to your drawing.



Using the SHOW* Command

Before taking another break, let's experiment with the `SHOW*` command.

1. First, select the secondary command:

   ```
   SHOW*
   ```

Two options are available:

Option	Description
shON/shOFF	Sets the show command to turn components on or off.
sPT = 1/sPT = 2	Sets the show command to affect components in port 1 (the large port) or port 2 (the small port).

The methods are:

Method	Description
COLOR	Turns on or off components with the current color.
LNTYP	Turns on or off components with the current line-style.
BOTH	Turns on or off components with both the current line-style and color.
ALL	Turns on or off all components.

The SHOW* command can be used to temporarily remove unnecessary components from the display of the drawing in either viewport. The components are not lost or destroyed, and can be redisplayed at any time by turning them back on with the SHOW* command. It is also important to note that the effects of the SHOW* command are cumulative.

2. Select:

   ```
   ALL
   ```

   Since the shON/shOFF option is currently set to shOFF, all components on your display will be turned off.

3. Select:

   ```
   SHOW* shOFF COLOR
   ```

   The shON/shOFF option is now set to shON, so everything in the currently defined color (green) is turned on. Now you see the bush with no house.

4. Select:

   ALL

   Now you see the whole drawing again.

Copying Restricted Components

1. To add additional bushes, select:

   COPY*

2. Select the component restrictor:

   ~Line

3. Select the bush.
4. Select the origin of the bush, perhaps the point where it goes into the ground. Move the copy of the bush to the desired location and select the point.

   When you use a component restrictor, you may select only a component of that type. This makes it easier to pick components that overlap.

5. Repeat these steps until you are content with the number of bushes in your drawing.
6. To complete the *PLACE method of the *COPY command, select:

   EOC

Saving a Previously Saved Drawing

Now you will save the latest version of your house, including all of the changes you have made in this lesson.

1. Because it is already named, select:

   OUT* SAVE* OLD

   HP EGS asks you if you want to remove the old drawing. This is just a reminder that the old version will be lost if you save this one with the same name.

2. Press:

   Y

   The most recent version of your house is now saved in the old file.

You are now done with this lesson. To leave the General Drawing personality and return to the HP EGS manager, enter:

```
QUIT;
```

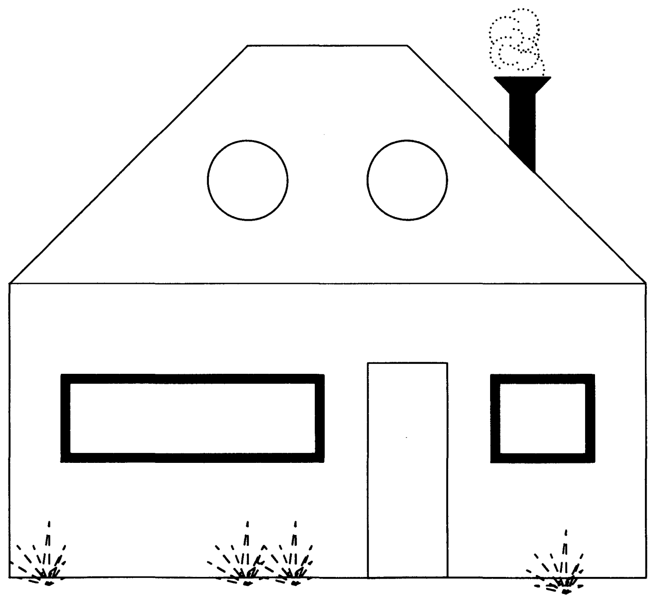
This ends the current editing session. You can now safely turn off your computer or continue with the next lesson. In this session you:

- Added arcs and multivertex lines.
- Used hatching.
- Used the SHOW* command.
- Copied restricted components.
- Saved a previously saved drawing.

A More Complex Drawing

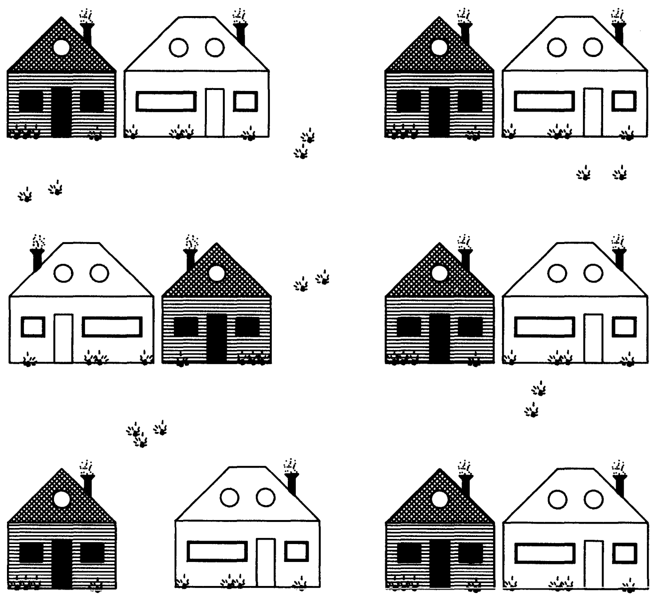
Chapter 4

This lesson shows you how to use more complex drawing commands. First, we'll show you how to create a second house by modifying the one you just finished. The complete new house will look like this:



Modified House

Later in this lesson you will use these two houses to create a town like this:



Finished Town

Now that you have learned the basic drawing commands and options, this lesson assumes you do not need as many specific instructions. Remember to use the prompts at the top of the screen. The `HELP*` command is always available if you need more detailed instructions.

You terminated the last editing session with `QUIT`, which returned you to the HP EGS manager. You now need to get back into the General Drawing personality. If you have forgotten how, refer back to Chapter 2.

After entering General Drawing, preview this lesson by entering:

```
INPUT EWLRN:chap4;
```

Remember that you must let the demonstration complete and then press the space bar so that the General Drawing personality is reset.

Modifying the House

To create a different house by modifying the one you already made, you will use MODIFY to change the width, and fill and color existing components. You will also use the *VRTX (vertex) method of STRETCH* and the *MOVE option of GATHER* to change the shape of your house, and DELETE* to delete the siding. While using these primary commands you will be introduced to many more secondary commands found in WIND*, INFO*, and RULE*.

1. Prefix to DRAW: by entering:

   PRE DRAW:;

2. To get the house you drew in the last lesson, enter:

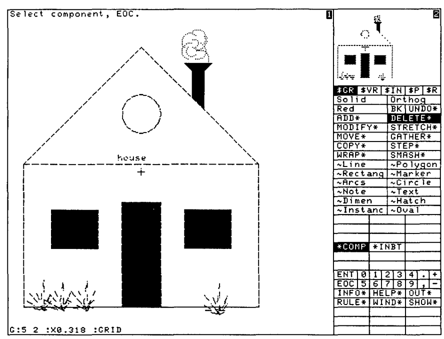
   EDIT house;

   Note that the semicolon (;) is used as an end-of-command indicator. This is exactly the same as selecting EOC from the menu. In this case it's just easier to type the semicolon from the keyboard.

3. To set cursor snapping to the user grid, select:

   $GR

4. To remove the siding from the ground floor and the cross-hatching from the second floor select DELETE* and use the default method *COMP. Your house should now look like this:



Identifying a Component

The first change to your house is to modify the width of the windows and change their color to cyan. But first, find out what HP EGS knows about the component which defines the window. You can use IDEN? to identify any component.

1. Select:

   INFO* IDEN?

2. Select the right window. The display now reads:

   Rectangle 1 :WO :FILL

   and the window is highlighted. The first item in the display line tells you that the component identified is a rectangle. The second item tells you that the rectangle is on layer 1 (more on layers in a moment).

The remaining information shows the state of any options that the component may have. In this case the width (:W) is 0 mm and fill is turned on (:FILL). The cryptic notation that you see is in the form of low-level HP EGS syntax. Usually you can make a good guess at what the options are, but if you have any questions, refer to the *HP EGS Syntax Reference* for the component you identified. In this case, refer to ADD RECTANGLE.

You can continue to select other components for IDEN? until entering EOC.

3. To terminate `IDEN?` so you can continue with `MODIFY*`, select:

   ```
   EOC
   ```

Unfilling an Item

1. The next change to your house is to turn off the fill of the door. Select:

   ```
   MODIFY*
   ```

2. Select the door of your house and a reference point on the door. (In this example, select the top left corner so you don't inadvertently select the outside of the first floor.)
3. You can now select:

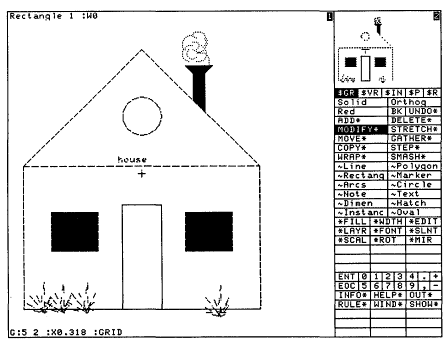
   ```
   *FILL
   ```

   The rectangle defining your door is now un-filled.

4. To finalize the modification, select:

   ```
   EOC
   ```

   The reason the reference point was required to turn off filling is because HP EGS allows you to select as many options as you desire while modifying a component. Some of these options, such as `*SCAL` (scale) and `ROT*` (rotate), require that a reference point be previously selected. Your house should look like this:



Modifying a Window

In this section you will change the width and color of the right window.

1. To change the color to cyan, select:

   ```
   Red *Cyan
   ```

2. Select the right window and then a reference point on it (you could choose the lower left corner). Note that the display again shows:

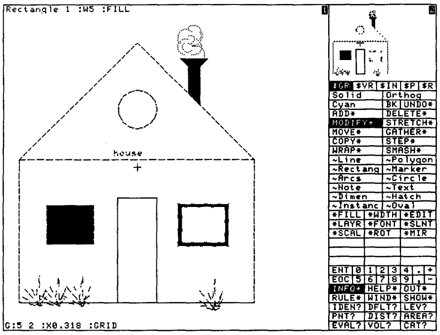
   ```
   Rectangle 1 :W0 :FILL
   ```

3. Select:

   ```
   *WDTH
   ```

   This option enables you to modify the width of a component. Your display now reads:

   ```
   ENTer the new width value.
   ```



4. Enter:

   ```
   5
   ```

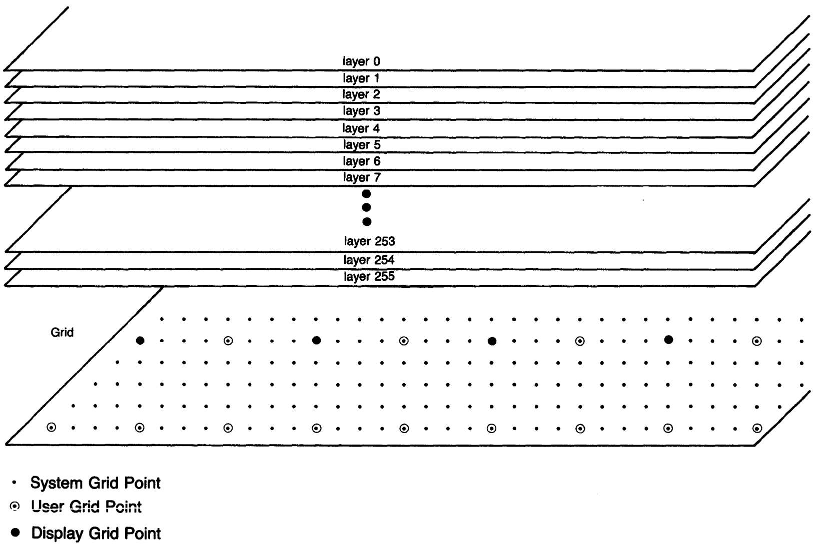
   The window should now look like the window on the right:

Also notice that the display shows that the width has changed to 5 mm:

```
Rectangle 1 :W5 :FILL
```

5. Before you change the color of the window, you need to understand more about layers.

   The drawing area can be thought of as a stack of 256 clear plastic sheets, called layers, upon which the drawing is created:



256 Layers

Each numbered layer includes attributes for line-type, display color, and plotter pen color. Any component added to a particular layer is displayed and plotted according to that layer's definition. You set the current layer by selecting the color and line-type modes.

Here are the layer definitions used in General Drawing:

LAYER DEFINITIONS

LINE-TYPE	COLOR					
	Red	Yellow	Green	Cyan	Blue	Magenta
Solid	1	2	3	4	5	6
Dashed	7	8	9	10	11	12
LngDash	13	14	15	16	17	18
DotCent	19	20	21	22	23	24
DashCnt	25	26	27	28	29	30
Phantom	31	32	33	34	35	36
Dotted	37	38	39	40	41	42
ShrtDot	43	44	45	46	47	48

LAYER NUMBER

The SHOW* command enables you to look at individual layers or combinations of layers as you did in the previous chapter.

From the table of layers you can see that layer 1 is defined to have a red color and a solid line-style.

6. To change the color of the rectangle, select:

   *LAYR

   Your display briefly flashes this message:

   The color and line-style will be changed to what is currently set.

   The display then reads:

   Rectangle 4 :W5 :FILL

   This shows that the layer of the rectangle has been changed to 4. The preceding table shows that layer 4 is defined to have a cyan color and a solid line-style.

Determining the Area of a Window

1. Just for fun, use the AREA? method in the secondary command INFO* to determine the areas of your windows. Because INFO* is still highlighted, you need only to select:

   AREA?

2. Select the window on the right. The display now reads:

   AREA = 1100 SQ MM ; CUM = 1100

   This tells you that the enclosed area of the rectangle is 1100 square millimetres. Because the rectangle has width, the enclosed area includes only what you see filled.

3. Select the window on the left. The display now reads:

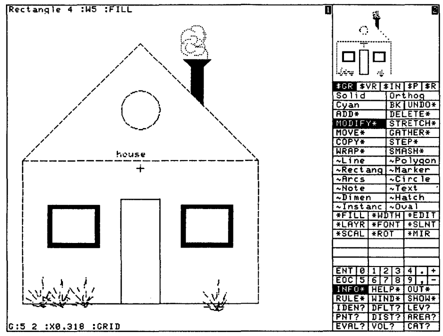
   AREA = 3000 SQ MM ; CUM = 4100

   This tells you that the enclosed area of this rectangle is 3000 square millimetres. Because the rectangle does not have width, the enclosed area is much larger. CUM = 4100 indicates that the cumulative area of both rectangles is 4100 square millimetres. The cumulative figure continues to be updated until AREA? is terminated with an end-of-command indicator.

4. To terminate AREA? so you can continue with MODIFY*, select:

   EOC

5. Modify the left window just as you did the right one. Your house should now look like this:



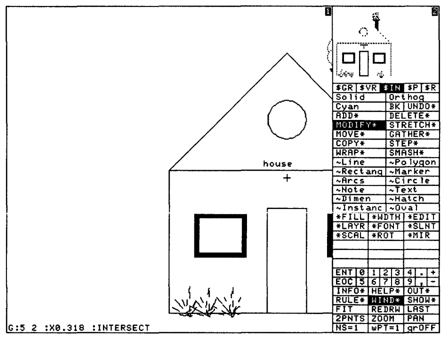
Modifying Multiple Components Simultaneously

You are now going to expand the left side of the house.

1. To prepare for this, first change the snapping mode to $IN (snap to intersections).
2. Next you need some room to work with in your drawing area. Select:

   WIND* PAN

3. Select the upper left corner of the first floor of your house.



Notice that your drawing area has "panned" to the left side of your house. The PAN method of WIND* allows you to select a point in your drawing to be moved to the center of the drawing area.

Note

Now you are going to use a powerful command in HP EGS, GATHER*, to expand the size of your house. It is important that you follow these instructions carefully because GATHER* can damage your drawing just as easily as it can improve your drawing. Remember that UNDO* is a handy way to get yourself out of trouble.

GATHER* enables you to stretch, move, and modify multiple components all at once. To do this you first select all of the components and pieces of components to be changed. This is called making a collection. Once the collection is made it can be modified (or transformed).

Gathered collections can be moved, scaled, rotated, and/or mirrored. Once you have completed the modifications you should destroy (or un-collect) the gathered collection.

4. Select:

    GATHER*

    You are now presented with the main options:

Option	Description
*MAKE	Makes a collection.
*MDFY	Modifies the collection just made.
*DSRY	Destroys the collection just modified.

You must use *MAKE before using *MDFY or *DSRY. Because the order of use is important, the General Drawing personality automatically selects *MAKE, which presents you with the following methods to aid in creating collections:

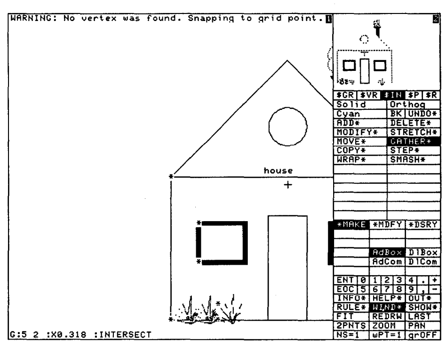
Method	Description
AdBox	Selects a rectangular area in which all components are to be added to the collection.
DlBox	Selects a rectangular area in which all components are to be deleted from the collection.
AdCom	Selects an individual component to add to the collection.
DlCom	Selects an individual component to delete from the collection.

5. In this case, you will use the default method for creating collections, AdBox.

    In creating the gathered collection for your house, you should include the left portion of the first floor, the left portion of the second floor, and the left portion of the left window.

6. Move your cursor to (−220, −190) and select the first point of the AdBox rectangle. Then move to (−90,70) and select the second point of the rectangle.

    You should now notice stars (*) highlighting vertices of components in your collection. If you have bushes on the left side of your house notice that some of them are completely highlighted and that others only have vertices highlighted.



7. To avoid changing the shape of your bushes, select:

   `DlCom ~LINE`

   (delete components)
8. Select the bushes that have only stars in them. You should now have only completely highlighted, or completely un-highlighted bushes (no stars in them).
9. Select:

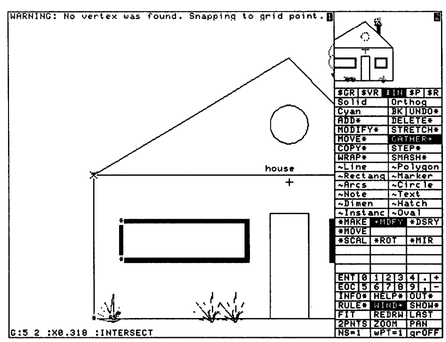
   `*MDFY`
10. Select the upper left corner of the first floor (-150,10) as the reference point for modifying the gathered collection.

    You are now presented with additional options that allow you to specify how the collection should be modified. These are:

Option	Description
`*MOVE`	Moves the collection.
`*SCAL`	Scales the collection with respect to the reference point.
`*ROT`	Rotates the collection about the reference point.
`*MIR`	Mirrors the collection about the x-axis, y-axis, or both, through the reference point.

11. Select:

    `*MOVE`



    As you move the cursor across the drawing area, notice that you are dynamically moving components completely enclosed in your collection, and stretching the components which cross the boundaries of the collection.
12. Select a new point for the gather which maintains the rectangular shape of the first floor, but which also expands the house to the left $(-250,10)$. Your house should now be larger than before:

13. At this point you could *MOVE the collection again, or use the other options to perform further modifications. For now, just select:

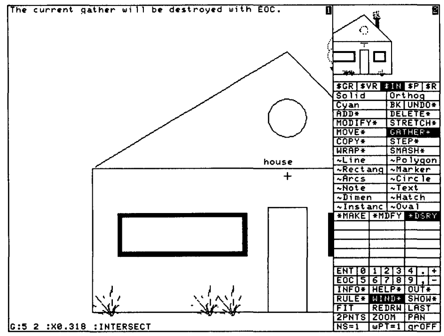
    *DSRY

    to destroy the collection. This removes the modification options from the menu. The *DSRY prompt requests an end-of-command (EOC) to go ahead and destroy the collection.

14. Select:

    EOC

    Notice that all of the highlighting, including the stars, disappears. Your screen should look like this:



Note

While using AdCom and DlCom to create a collection, the $VR (vertex snapping) mode will only add or delete individual vertices of a component.

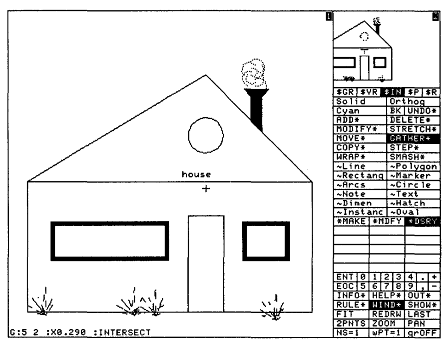
Preparing to Add Another Roof Peak

Now that the house is bigger than the current drawing area, use a new WIND* method, FIT, to fit the entire house in the large port. Then you will redraw the roof using vertical construction lines.

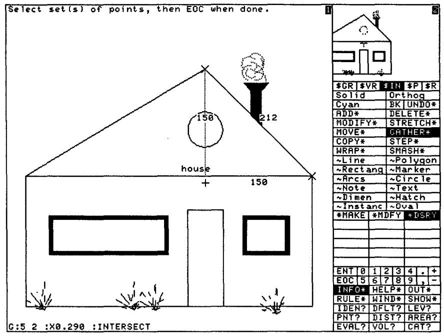
1. Select:

    FIT

    Your screen should look like this:



Now that the house has been expanded the roof line of the house looks lopsided. You can add a second peak to the roof to make it symmetrical. First you need to know the size of the right portion of the roof.



2. Select:

   `INFO* DIST?`

3. Select the top right corner of the first floor of the house, then the peak of the roof.

Now you can see the horizontal, vertical, and direct distances between these two points. The distances which have been drawn are temporary and will disappear the next time the port is redrawn.

4. To terminate `DIST?`, select:

   `EOC`

5. Change the line-style to `*ShrtDot` so you can add construction lines to help you place the second peak.
6. To add a vertical construction line along the left side of the house, select:

   `ADD* LINE *CONS`

7. Then select bottom and top left corners of first floor. Remember that the snapping mode is still `$IN`, so you need only be within the cursor area of the corners.
8. From the `DIST?` calculation you made earlier you found that the peak of the roof is currently 150 mm from the right side of the house. To place a vertical construction line 150 mm from the left side of the house, select:

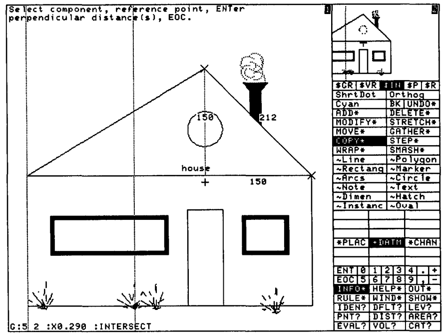
   `COPY* *DATM`

   The `*DATM` (datum) method of `COPY*` enables you to specify discrete distances to place copies away from the original. Each distance is measured perpendicular to the original, from the selected reference point on the original. In this case you need a copy of the construction line you just added 150 mm to the right.

9. Select the construction line, then select a point on the construction line (the upper left corner of the first floor). Enter:

   `150`

10. Another vertical construction line appears. Terminate `COPY*` with `EOC`. The original line is now un-highlighted. Your house should look like this:



There is one last method of `COPY*` that you have not used: `CHAN*` (chain). You will not use the `CHAN` method in this example. `*CHAN` is very similar to `*DATM` except that each distance is measured from the last copy instead of from the selected reference point on the original.

Adding Horizontal Construction Lines

To find the actual peak of the roof and the center point for a second circular window, you will add a horizontal construction line at the peak of the roof and another through the center of the existing circular window. You could use `*ANG` to add these lines, but let's try another method.

1. Select the secondary command:

   `RULE*`

   You are presented with several methods. Three are useful for putting further restrictions on cursor snapping. Each deals with setting and enforcing the lock angle.

Method	**Description**
`lkNEW`	Sets the value of the lock angle. (If the lock angle is set to 0 degrees, lock angle enforcement is automatically turned off.)
`HrVrt`	Sets the lock angle to 90 degrees and turns on enforcement.
`lkON/lkOFF`	Turns lock angle enforcement on or off. `lkOFF` in the menu indicates that the lock angle is currently off.

   When you set the lock angle, HP EGS checks to see that straight segments of components (lines and polygons) are added at increments of the lock angle. If the lock angle is enforced, the points you select will snap to the lock angle. Otherwise, HP EGS will beep and display a harmless warning message that you broke the lock angle rule.

2. Select:

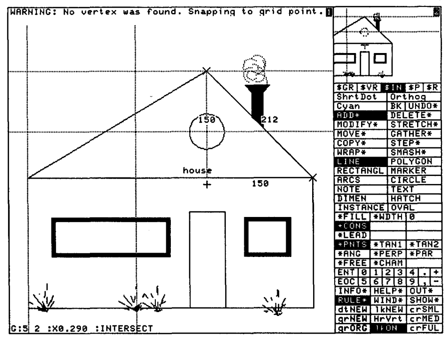
   `HrVrt`

   Notice that the method `lkOFF` is automatically changed to `lkON` and is highlighted, indicating that the lock angle is being enforced.

3. Select `ADD* LINE` again and place the first point of a construction line at the peak of the roof. Move the cursor around this point in a circle and notice how the dynamic tracking of the line bounces to increments of 90 degrees.

4. Move the cursor to the left of the peak and select a point when the construction line is almost horizontal.

5. Add another construction line at the center of the circular window in the same way. Note that HP EGS considers the center point of a circle to be a vertex. Because you are still in the `$IN` snapping mode you will automatically snap to this center point if it is within the boundary of your cursor. Your lines should look like this:



Adding a New Vertex to the Roof

Now that you have added the horizontal construction lines, you need to turn the lock angle enforcement off.

1. Select:

   `lkNEW`

2. Enter:

   `0`

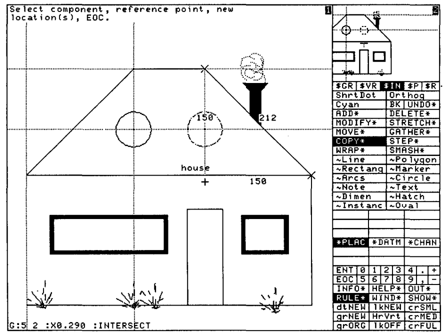
   Notice that the method `lkON` is automatically changed to `lkOFF`, indicating that the lock angle enforcement has been turned off. `lkNEW` automatically turns off lock angle enforcement when 0 degrees is entered, because a 0 degree lock angle cannot be enforced.

3. To put the second peak in the roof, select:

   `STRETCH* *VRTX`

   This method of `STRETCH*` enables you to add a new vertex to a line, rectangle, or polygon. To use this you must select the component you wish to stretch (using a component restrictor if necessary), and then select the segment of the component where you want to add the vertex. Then you can select the location of the new vertex.

4. Select the roof line and then the middle of its left segment.
5. Now select the intersection of the two construction lines defining the point for the second peak of the roof (about −100,160).
6. Select COPY* and use the default method *PLACE to make a copy of the circular window. Be careful when selecting the circle that you don't get the construction line component by mistake. Because the intersection of two construction lines defines the center point for the copy of the circle, the best reference point to select is the center of the circle. Place the new window at the intersection of the construction lines below the new peak. Your house should look like this:



7. All of the changes have now been made to the original house. You need only to clean up the drawing and save it. Use DELETE* to remove the four construction lines from your house.
8. Select WIND* REDRW and note that the displayed distances are gone after the screen is redrawn.

   Now that your new house is completed it should be saved under a new name. Do not save it under the old name because you will loose the original, which you will need to use again in the next section.
9. Select SAVE* NAME and enter the new name:

   ```
   newhouse
   ```

   A name can be up to 8 characters long.

BREAK TIME

Stretch, relax, and review what you have done in this section:

- Identified a component.
- Unfilled an item.
- Modified the width of a component.
- Learned about layers and how to modify them.
- Expanded the size of a component.
- Added vertical and horizontal construction lines.
- Added a vertex.

Making a Town

In this section you will create a town using the houses you made in the last two exercises. You will add the houses to a new drawing and then create a new instance (or component) out of these two houses. You will also copy the instance into a matrix form, move these copies around, and break the instances back down into their component parts. Finally, you will use the `NS=1` and `wPT=1` options of the `WIND*` secondary command.

Removing a File from Computer Memory

Before you begin to create the town you need to clean up one item. In the last section you changed the original `house` and saved it under the name `newhouse`. The disc contains two different drawing files: `house_d` and `newhouse_d`. But in your computer memory, the version of `house` looks just like `newhouse` because you changed it while making `newhouse`. You should now remove the copy of `house` from the computer memory so that HP EGS will get a new copy from your disc the next time you use `house`.

To remove a file from memory, use the `REMOVE` command as follows.

Note

`REMOVE` selectively removes a drawing from both the disc and the computer memory, so be careful using this command.

---

1. To exit from the house drawing so it can be removed, enter:

   ```
   Edit;
   ```

2. Enter:

   ```
   REMOVE house_d;
   ```

   The display now shows:

   ```
   Remove "house_d" from the disc? (Y/N).
   ```

3. Press (N) because you want to leave the copy on the disc intact. The display now shows:

   ```
   Remove "house_d" from memory? (Y/N).
   ```

4. Press (Y).

   When using the `REMOVE` command you must enter the complete file name, including any suffixes which HP EGS automatically adds (for example, `house_d`).

Adding Instances

To start making your town, you need a blank drawing.

1. Enter:

   ```
   EDIT town;
   ```

   Your town drawing is now pre-named, unlike the first house you drew, which you did not name until it was saved. Note that the name of your drawing now appears in inverse video next to the (0,0) origin on your drawing. This indicates that the drawing is empty.

2. Press **SPACE**. Notice that the drawing name also appears to the left of the entry line carrot:

   ```
   town> _
   ```

   This is a handy way to see the name of the current drawing. Press **Return** to return to the graphics display.

3. Set the snapping mode to `$GR` to prepare for the rest of this section.

   In HP EGS an instance component is a component made up of other components. An instance can even include other instance components. The house drawings which you created are made up of several components. We will now create an instance which contains all of the components. Once created, the instance can be moved, copied, and so on, just like a single component.

4. Select:

   ```
   ADD* INSTANCE
   ```

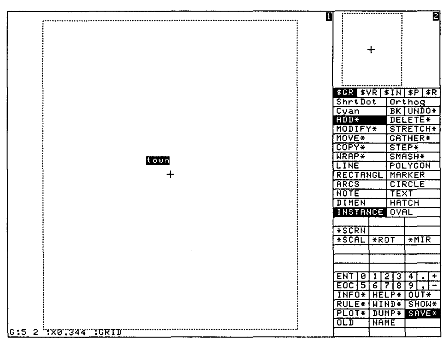
5. Enter:

   ```
   house
   ```

   The instance will be called `house`. You will be dynamically tracking the boundary of the `house` drawing. The cursor is located at the origin (0,0) of the `house` instance.

6. Move the cursor to the origin of your `town` drawing (0,0) and select the point.

   When you add an instance to your drawing you do not see the components contained in the instance. Rather, you see only the instance boundary. That is why you can't see the house. (More on how to see inside instances shortly.) Your drawing should look like this:



Before you add the `newhouse` instance you need to make more room for it in the drawing area.

7. Select:

   `WIND* ZOOM`

   The `ZOOM` method of `WIND*` enables you to specify a magnification factor for the drawing. Plus (+) values zoom the port in (move into the screen area), and minus (−) values zoom the port out (move away from the screen area). Because you need more room you must `ZOOM` the port out.

8. Enter:

   `-3`

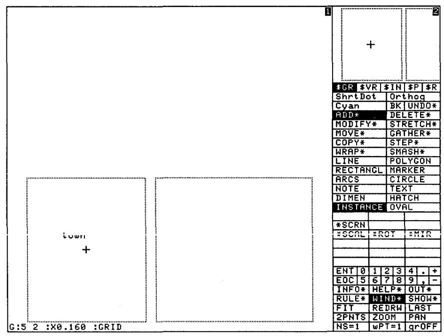
9. Now that there is enough room in your drawing to add the second house you made, enter:

   `newhouse`

   You have just made `newhouse` an instance. You will be tracking the boundary of the `newhouse` drawing.

10. Place the second house to the right of the first house (about 425,0).

11. Select `FIT`. Notice the instance boundaries of both houses filling the large port:



You have learned that by making a drawing you create an instance that can be used in other drawings. Let's examine another way to create instances.

1. Select:

   `WRAP*`

   This command collects components together to create a new instance. Before beginning the wrap, decide whether to name the instance it creates. If you name it, the instance will be saved separately when the drawing is saved. If you do not name it, the instance will be contained within the drawing when saved. If an instance is saved separately you can edit it and add it to other drawings as you could any other instance.

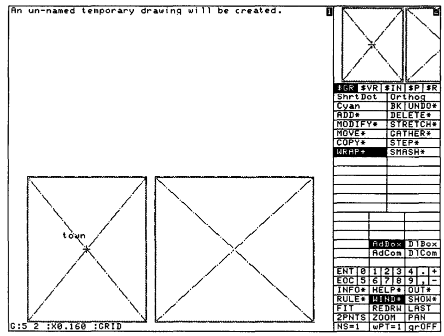
   After you enter the name, or decide not to, select the origin point for the new instance. This point will become the (0,0) location of the new instance.

Finally, collect the components (the two instances, in this case) to be included in the new instance. Notice that WRAP* uses the same methods for collecting components as do GATHER*, AdBox, DlBox, AdCom, and DlCom. The major difference between wrapping components in WRAP* is that only complete components are included. You can't WRAP* portions of, or individual vertices of, components, like you can with GATHER*.

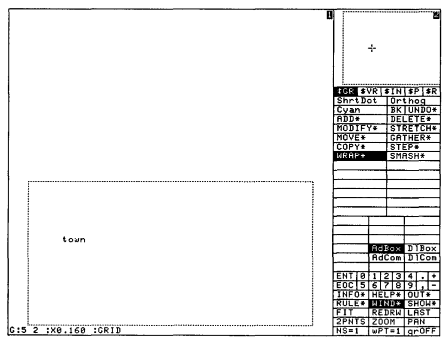
In this example you are using WRAP* to combine the two instances in your drawing into a new one so that you can make some copies. You do not need to name the instance.

2. To wrap the two instances, first select an origin point. (0,0) is good. Using the default collection method, AdBox, select two points which define a box that surrounds both the house and newhouse instance components.

   Your drawing should now look like this:



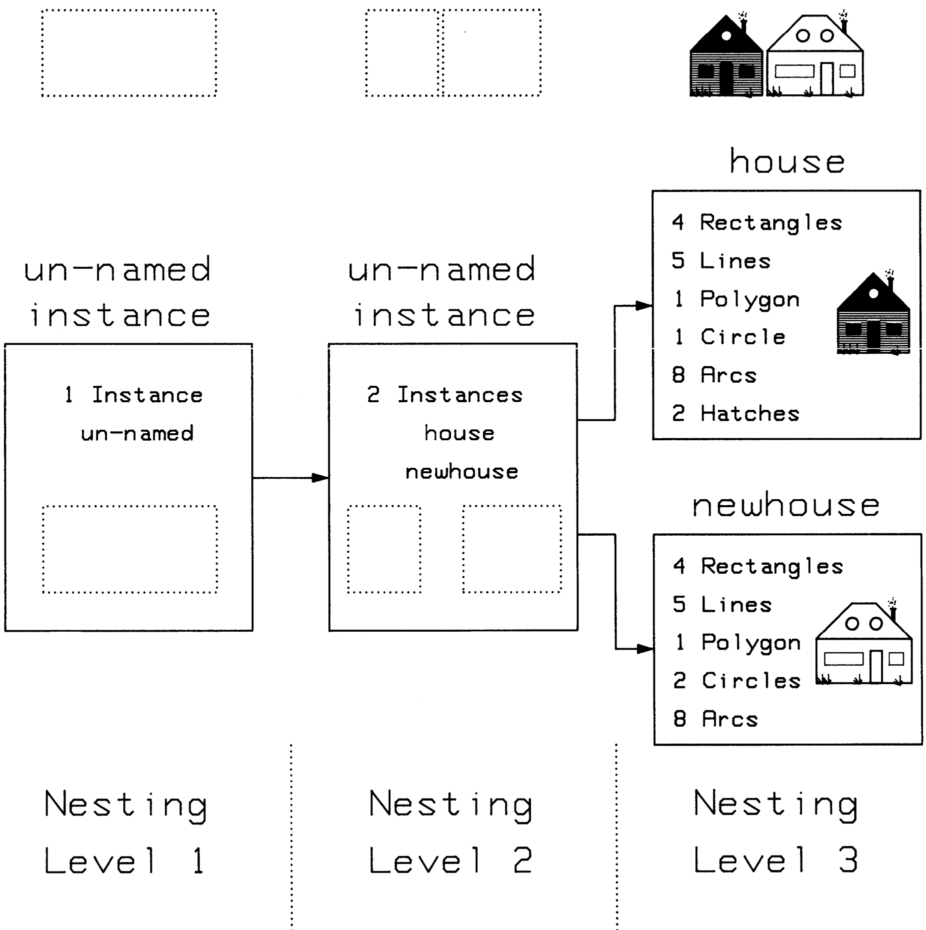
3. Select EOC to complete the wrap. Your drawing should now look like the this picture. Remember that you can't currently see inside of instances.



Nesting Levels

To see inside of instances you must first understand nesting levels. Nesting represents the hierarchy of instances in your current drawing. The drawing now on your display is nested.

The first level of nesting, which is what you can see, contains the unnamed instance you just created with `WRAP*`. Inside of this instance are two instances, `house` and `newhouse`. These instances are in the second level of nesting. Inside the `house` and `newhouse` instances are components which are on the third level of nesting. The following figure illustrates this nesting.



Nesting Levels

To display the nested components you must set the nesting level in the `WIND*` secondary command.

1. Select the `WIND*` option:

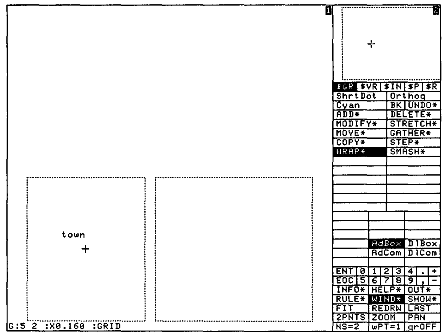
   `NS=1`

2. Enter:

   `2`

   Notice that `NS=1` changed to `NS=2`.

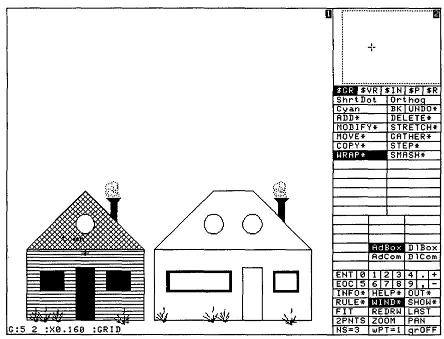
3. Select `REDRW` to redraw the display. You should now see the two instances that are on the second nesting level:



4. Select:

   `NS=2 3 ENT REDRW`

   to change the nesting level to 3 and to redraw the display. All components contained in the two houses appear:



Setting the nesting level gives you control of the detail to which your drawing is displayed. When you set the nesting level you see all components on all equal or lower-numbered levels. Therefore, you could have skipped setting the nesting level to 2 and had the same result.

Copying Instances

Next you will make five more copies of the instance. First you need make to some room again.

1. Use WIND* ZOOM to zoom out by a factor of eight (enter -8).
2. Select:

   STEP*

   The STEP* command enables you to make copies of components in rectangular matrices and polar arrays. You will use the default method, *MTRX, to create a rectangular matrix of your new instance. The *MTRX method requires that you first select the component you wish to step.
3. Select the instance in your drawing. Select a reference point ($-175, -160$) and then for the number of columns enter:

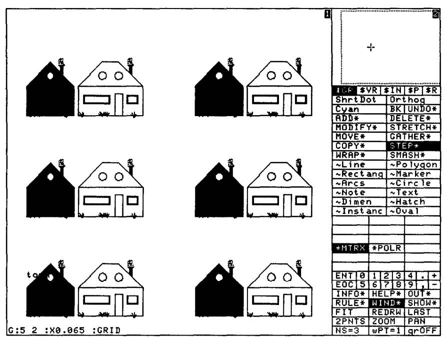
   2
4. For the number of rows enter:

   3

   Notice the dynamic tracking which shows you the outside boundary of the matrix and the boundary of the component in the second column and row.
5. Place the step above and to the right of the original copy (875,470).
6. Select:

   FIT

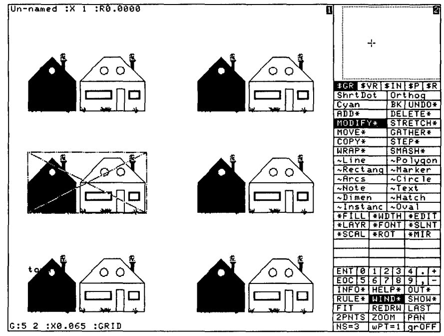
   This picture appears:



It may look like you are using six times the amount of drawing space you originally had. This is not the case, because an instance occupies only one location in your computer's memory. When you add or copy an instance you are really only adding a link (or pointer) to the original drawing. You are not adding all of the components within the instance.

It is important to know that you can only manipulate (DELETE*, MODIFY*, STRETCH*, MOVE*, GATHER*, COPY*, STEP* and WRAP*) components that are on the first nesting level. Right now this limits you to the six copies of the same instance currently in your drawing.

7. Select `MODIFY*` and then select the pair of houses in the first column of the second row of your matrix (about 250,570). Your screen should look like this:



Notice that the entire instance is now highlighted.

8. Select a reference point in the center of the instance. Then select:

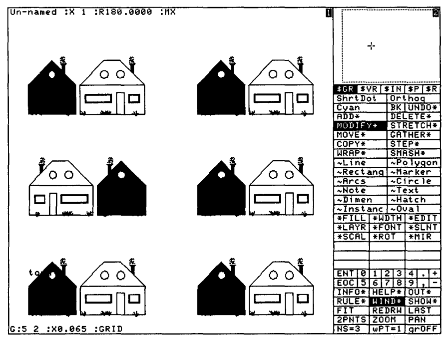
```
*MIR
```

from the modify options. You are presented with a choice of axis to mirror the instance about.

9. Select:

```
YAXIS EOC
```

Your instance will be mirrored about the y-axis, as shown here:



Smashing an Instance

If you want to manipulate the components inside of an instance you must destroy the first level of hierarchy. This moves the contents of the second nesting level to the first, the third level to the second, and so on. This process is called smashing.

1. Before you smash anything, select:

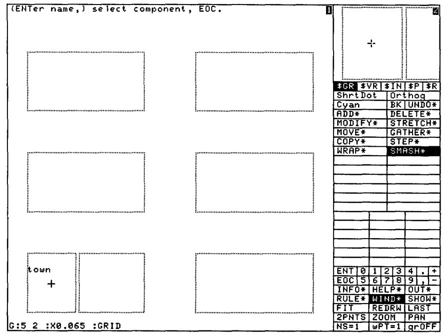
```
WIND* NS=3 1 ENT REDRW
```

to change the nesting level back to 1.

2. Select:

```
SMASH*
```

3. Select the lower left instance, and then EOC. The selected instance is replaced by the two components it contained. These components have been moved from nesting level 3 to nesting level 2. Your screen should look like this:



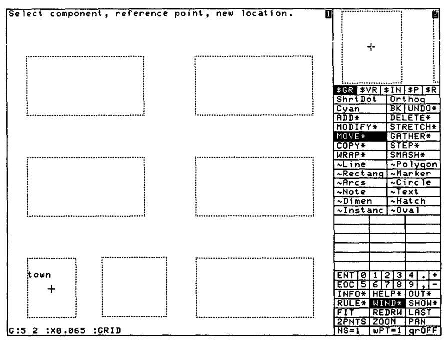
Moving Components

1. Select:

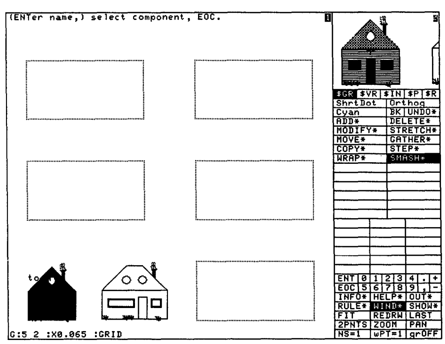
   MOVE*

   This command enables you to move components. MOVE* requires that you select a component (optionally using a component restrictor), a reference point, and then the new position for the component.

2. Select the right instance (newhouse) which resulted from the previous SMASH*. Select a reference point, in the middle of the instance, and then you should be tracking the instance. Move the cursor until you have centered the instance between the house on the left and the instance to the right. Your screen should look like this:



3. SMASH* the instance you just moved (newhouse) and the instance to its left (house). The last level of hierarchy is removed from these instances and their contained components are included in the town drawing. Your screen should look like this:



Using Both Viewports

Above the menu is a second viewport, or port, into your drawing. This port is probably displaying a small portion of your drawing. All of the commands you have used to manipulate the large port are valid in the small one. You need only to change the current port from port 1, the large one, to port 2, the small one.

Two ports are useful because you can select points *from either port at any time.*

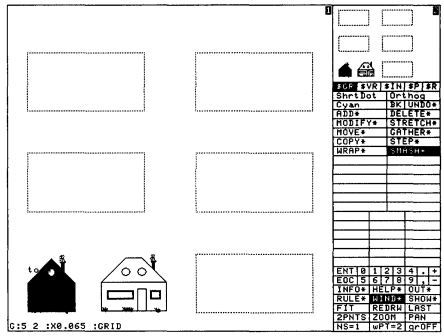
1. Select the `WIND*` option:

   `wPT=1`

2. Enter:

   `2`

   You have just made port 2 the current port for `WIND*` methods. Select `FIT` and notice that your drawing fits in the small port, as shown:



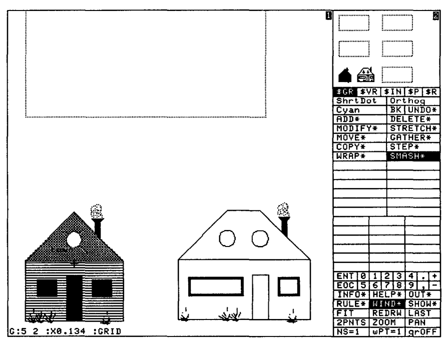
3. Set the current port back to port 1:

   `wPT=2 1 ENT`

   Further `WIND*` methods will now affect port 1.

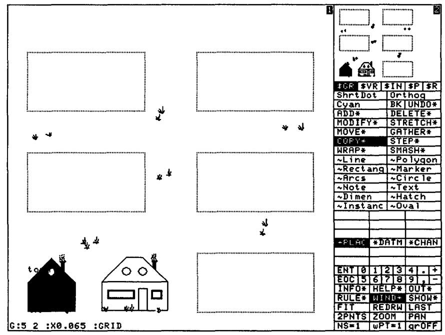
   Next you will zoom in on the two houses in the lower left of port 1, using a new way to select the points.

4. Select `2PNTS`. Then select the lower left corner of the box (about −200, −220) in port 1. Move the cursor to port 2 and select the other corner (780,310). Your screen should look like this:



5. COPY* a bush from one of the smashed houses and make several copies around your drawing. Move the cursor into the second port at any time to place the bushes at locations not available in the large port.

6. When you are satisfied that your town has enough greenery, select FIT to have a final look. Then save your drawing. Remember that you can use the OLD method of SAVE* because the drawing is already named. Your completed town should look like this:



BREAK TIME

Stretch, relax, and review what you have done in this section:

- Removed a file from computer memory.
- Added instances.
- Learned about nesting levels.
- Copied instances.
- Smashed a layer.
- Moved components.
- Used both viewports.

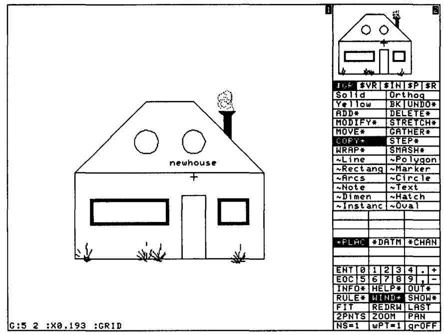
Selling the Houses

In this section you will change the `newhouse` drawing and learn how to use `ADD* NOTE` and `ADD* TEXT`.

1. Start changing the drawing by adding a FOR SALE sign to the `newhouse` drawing. Enter:

   ```
   EDIT newhouse;
   ```

2. Change the color to `*Yellow` and the line-style to `*Solid`. Use `WIND* ZOOM` to zoom out the large port by a factor of 1.5 (−1.5). Your screen should look like this:



3. Select:

   ```
   ADD* TEXT
   ```

4. Enter:

   ```
   'FOR SALE'
   ```

   Be sure to include the single quotation marks. The prompt on your display reminds you to include the quotation marks by enclosing the word `string` in single quotation marks.

The TEXT component presents you with the following options:

Option	Description
*FILL	Turns on or off the filling of text.
*FONT	Sets the font height of text. This also implies a change in the font width.
*SLNT	Sets the slant angle for the characters in the text string. The slant angle is measured clockwise from vertical.
*ROT	Sets the rotation angle for the text string. The rotation angle is measured counter-clockwise from horizontal.
*JUST	Sets the justification positon for the text string. This is the location in the boundary rectangle your cursor will dynamically track and to which the snapping mode will apply. The justification positions and their numeric identifiers are shown in the following diagram.

3	6	9
2	5	8
1	4	7

Place the text string to the right of the house. Notice that your cursor appears on the left side of, but centered on, the text string. This is justification position 2, as displayed in your menu, to the right of *JUST.

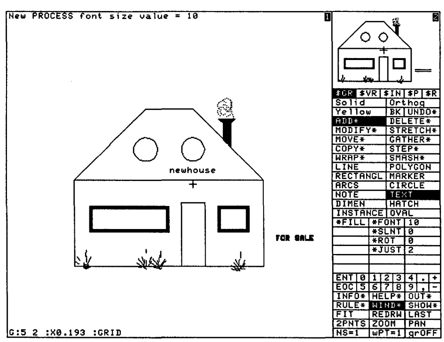
5. Select:

   *FONT

6. Enter:

   6

   You have now changed the current font size from 10mm (the default) to 6 mm. (Note the change of the value in the menu.)



7. Select:

   NOTE

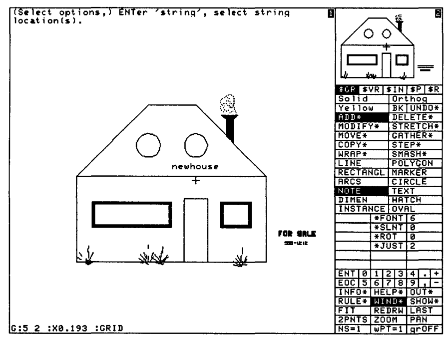
   The options for NOTE are the same as TEXT except that *FILL is missing because notes cannot be filled. The value for the *FONT height still reflects the change you made previously.

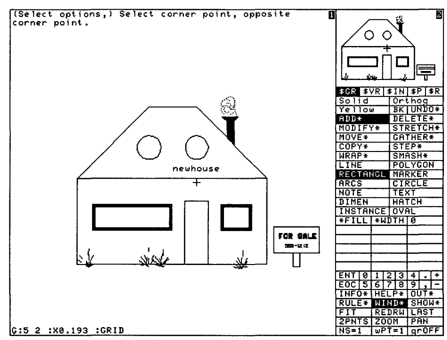
8. Enter:

   '555-1212'

   Place this string directly below the FOR SALE string.

   Notice a difference between the styles of the note and text characters. A note is a stick letter representation of characters and text is a block letter representation of characters. Because the block nature of text gives it width, text can be filled. Notes cannot. Therefore, the *FILL option does not appear when NOTE is selected. Your picture should look like this:





9. Select ADD* RECTANGL and add a rectangular border around the note and text strings you just added.
10. Finish your sign by adding another rectangle to make the signpost. Your picture should look like this:

11. Select the OLD method of OUT* SAVE* to save your drawing. HP EGS asks if you want to remove the old copy of newhouse from the disc. Answer yes by pressing (Y).

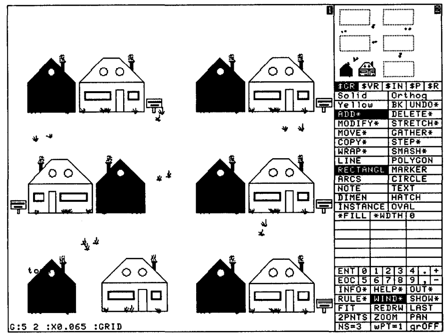
    Now that you have changed the newhouse drawing you have also changed any other drawings (instances) where newhouse resides as an instance. Let's see what this means.

12. Enter:

    `EDIT town;`

    So far nothing appears to have changed. The two houses in the lower left were not affected because they are not instances, but components in the `town` drawing. (They were smashed.)

13. Change the nesting level to 3 and redraw the large port (`WIND* NS=1 3 ENT REDRW`).



You can now see that all of the `newhouse` instances have FOR SALE signs. Also, notice that the FOR SALE sign in the houses which you mirrored earlier is also mirrored.

BREAK TIME

Once again relax and review what you have just done:

- Added text to a drawing.
- Added a border around the text.

Other Commands

This section includes useful information about other HP EGS commands not described in the tutorials. For more information on all HP EGS commands see the *HP EGS Syntax Reference.*

The ARCHIVE and GENERATE Commands

There are ways besides using the SAVE command to save drawings for specific purposes. If you want to see a textual listing of your drawing (such as for post processing) you can ARCHIVE or GENERATE it.

The ARCHIVE command differs from the SAVE command in that ARCHIVE saves your drawing into an ASCII file, and can include the definitions of all instances in the drawing. SAVE puts your drawing into a machine-dependent binary file and has links to only the included library parts. The inclusion of library parts can be important to you when storing drawings for a long period of time. For an example, see the *Using the Archive Reference* section of the *HP EGS Syntax Reference.*

The ARCHIVE command creates an ASCII file which contains a list of the low-level Graphics Editor commands used to create that drawing. If the drawing contains instances, these instances are preserved in the archive file to the nesting level specified. Because the information in the archive file is in ASCII format it can be accessed by the text Editor or other programs.

The GENERATE command creates an ASCII file which, like an archive file, contains a list of the Graphics Editor commands used to create that drawing. However, GENERATE smashes all instances and their original components. Just like an archive file, the information in the generate file is in ASCII format and so can be accessed by the text Editor or other programs.

The RETRIEVE Command

The RETRIEVE command recreates a drawing from an archive file. When you enter the command, the system reads the contents of the archive file and reconstructs component by component the drawing from which the archive file was created. If the ARCHIVE file contains instances, these instances are also reconstructed.

Be careful when using the RETRIEVE command because you can lose drawing files. When retrieving an archive file, HP EGS first removes any drawing files on the destination volume whose names match the name of the archived drawing or the instances it contains. The archived drawing, and its instances, are then saved in the destination volume with their original names. So the safest way to use this command is to always specify an empty volume as the destination.

The INPUT Command

The INPUT command is used to create a drawing from the contents of a generate file. When you enter this command, the system reads the contents of the generate file and reconstructs component by component the drawing from which the generate file was created. Remember that the reconstructed drawing will not contain any instances.

The INPUT command can also be used to "stream" any HP EGS commands from a file. Commands that INPUT reads from stream files are treated as if they are being typed from the keyboard.

The LIST Command

The LIST command enables you to get information about the files currently loaded into the HP EGS graphics editor. LIST also provides detailed information about the process file and system defaults, macro commands, and detailed information about the drawings currently loaded in memory.

Enter:

```
LIST;
```

Your display now has a list of the files used to create the General Drawing personality. Press **SPACE** and your display shows you a list of the drawings currently in the computer memory.

Press **SPACE** until your graphics display reappears. Enter:

`LIST town;` **SPACE**

Your display now has a list of all the components and instances used in the `town` drawing.

Any listings (such as from `LIST, VOL?, or CAT?`) are displayed on the CRT page by page. To output an individual page to the local printer, press:

Shift **ALPHA**

on a 236 or

Shift **PRINT**

on a 217 or 237. You can have the entire listing printed on a printer or written to a spool file on your SRM.

The DFLT? method of the INFO* command does a LIST of the process file. This gives you the current values of the HP EGS defaults and a listing of the currently defined layers.

Stopping an Operation

Some HP EGS operations, such as redrawing a port and plotting or archiving a drawing, may take a long time to perform. To abort these operations you can press (Stop). HP EGS responds by asking you if you really want to abort the current operation. Respond by pressing (Y) or (N) as you desire. This terminates the operation but does not undo it. Commands which you can stop include:

- `WIND*` methods: When a method of `WIND*` is stopped, the port is left partially redrawn. When this happens HP EGS allows you to use the port as if the `WIND*` method had completed normally.
- `PLOT*`: When `PLOT*` is stopped, it finishes plotting the current component before the command is terminated.
- `RETRIEVE`: When `RETRIEVE` is stopped, the drawing will contain all of the components added to the stopping point. You may then work on the drawing as you would normally.
- `INPUT`: When `INPUT` is stopped, the input file is held ready to be continued. You may work on the drawing as you would normally and then continue inputting later. To continue the input process, enter:

```
INPUT;.
```

Any other input command will automatically close an input file which was left open (ready to continue).

File Operations

The `PREFIX` command enables you to set the default volume. If many of your drawings are on a particular mass storage volume, you can avoid typing the volume designator by using the prefix command. This tells the system that you always want the data stored on the default volume unless you type in a specific volume.

You can set the prefix volume to any volume. When a file is being loaded into the system and no volume is given, the default volume is searched first. If you use the SAVE command to store data without giving a volume, it is saved on the default volume.

You can also make copies of drawings, delete drawings, change the names of drawings, and pack your disk to consolidate unused space. Examples of how to perform these operations follow.

- PREFIX: To set the default volume to the local volume EWMINE, enter:

  ```
  PREFIX EWMINE:;
  ```

- FILECOPY: To make a copy of the file EWSYS:town_d, enter:

  ```
  FILECOPY EWSYS:town_d,newtown_d;
  ```

  The drawing would then exist in two different files, town_d and newtown_d.

- CHANGE: To change the name of the file EWSYS:newtown_d, enter:

  ```
  CHANGE EWSYS:newtown_d,nowhere_d;
  ```

  You may also change the name of a volume with this command. For example:

  ```
  CHANGE EWYOU:,EWMINE:;
  ```

- REMOVE: To remove nowhere you would enter:

  ```
  REMOVE EWSYS:nowhere_d;
  ```

  If the file name is also a drawing in memory, the system will ask if the copy in memory is to be removed.

- PACK: To pack the local volume EWMINE you would enter:

  ```
  PACK EWMINE:;
  ```

  The PACK command reorganizes the specified volume to eliminate gaps between existing files on the volume. This enables you to use all available space on any given volume. However, you cannot pack SRM volumes.

Creating an Isometric Drawing

Chapter 5

This lesson shows how HP EGS can help you easily create isometric drawings and add dimensions to drawings. If you want to create mechanical drawings, you need to read this chapter. However, if you want only to draw schematics and printed circuit boards, at least skim this chapter.

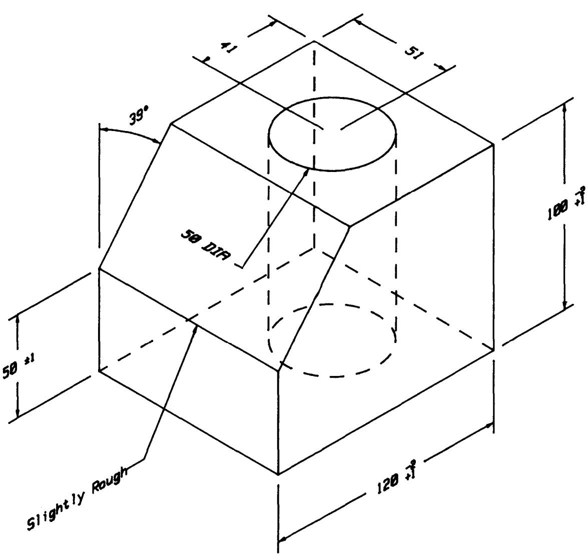
If you quit the General Drawing personality after the last lesson, start it back up. If not, enter:

```
edit;
```

To preview this lesson, enter:

```
INPUT EWLRN:chap5;
```

The following isometric drawing appears. Remember to let the demonstration complete and then press the space bar so that the General Drawing personality will be reset.



Isometric Block

The Basic Block

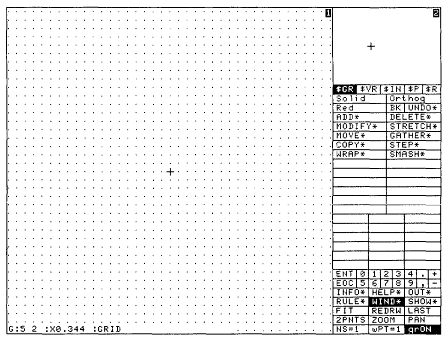
In this section you will change the user grid to simplify the creation of isometric drawings. You will also use the crMED method of RULE* to change the cursor size. Finally, you will learn what happens to components when they are added isometrically.

For a better understanding of isometric drawings, refer to Chapter 2 of *Understanding HP EGS.*

Drawing an Isometric Rectangle

1. Set the snapping mode to $GR, the color to Red, and the line-style to Solid if they are not already set this way.
2. To edit an empty drawing, enter:

   EDIT;



3. Now turn on the user grid by selecting:

   WIND* grOFF REDRW

   The grid that you now see is called an orthogonal grid, as shown. In an orthogonal grid the x-axis is horizontal and the y-axis is vertical, or perpendicular to the x-axis.

4. Select the mode:

   Orthog

   The following four choices of grid types appear:

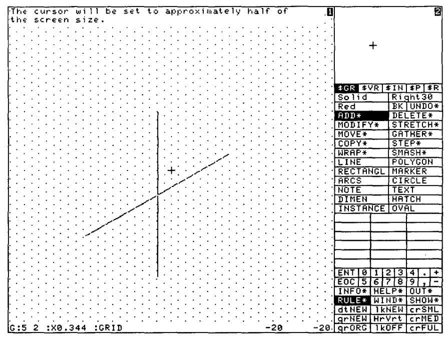
Grid	Description
*Orthog	The x-axis is at 0 degrees and the y-axis is at 90 degrees.
*Top30	The x-axis is at -30 degrees and the y-axis is at 30 degrees. This represents the top side of a 30-30 isometric system.
*Right30	The x-axis is at 30 degrees and the y-axis is at 90 degrees. This represents the right side of a 30-30 isometric system.
*Front30	The x-axis is at -30 degrees and the y-axis is at 90 degrees. This represents the left side of a 30-30 isometric system.

5. Select:

   `*Right30`

   The grid is redrawn, the x-axis of the grid is at a 30-degree angle, and the cursor changes shape.
6. Move the cursor to the (0,0) point of the large port.
7. Move the cursor along the new x-axis. To do this, look at the cursor values in the lower right corner of the large port. You will be moving along the x-axis when the y-value (x,y) remains at 0 in the lower right corner of the large port. Notice that you are moving the cursor at an angle of 30 degrees up across your screen to keep the y-value at 0.

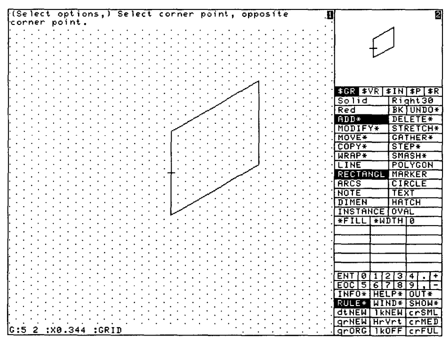
   Look closely at the cursor and see that the horizontal line is also skewed at an angle of 30 degrees.



8. Select:

   `RULE* crMED`

   This changes the size of the cursor to about one-half of the large port. You can now easily see the isometric nature of the cursor.



9. Select `ADD* RECTANGL` and then select the point (0,-50).

   Move your cursor around the screen and notice that the dynamic tracking is relative to the right 30 isometric user grid.
10. Complete the rectangle by selecting the point (120,50).

Adding an Isometric Rectangle

1. Use the secondary command `INFO* IDEN?` to identify the component you just added. The display now reads:

   `Polygon 1 :W0`

   What you added as a "rectangle" was converted to a polygon because when you changed the user grid to an isometric state the system grid did not change. HP EGS only stores data with respect to the system grid. When the "rectangle" you added is projected onto the system grid (which is always orthogonal) it must be turned into a polygon.

   Therefore, when the user grid is not orthogonal, components are automatically transformed when they are added. The transformations are:

- Lines remain lines.
- Polygons remain polygons.
- Markers remain markers.
- Hatch remains hatch.
- Instances are unaffected.
- Notes remain notes, but are slanted and rotated to match the grid skew.
- Text remains text, but is slanted and rotated to match the grid skew.
- Dimensions remain dimensions, but the dimension is skewed and the values are slanted and rotated to match the grid skew.
- Ovals remain ovals but are transformed to match the grid skew.
- Rectangles become polygons.
- Arcs become ovals.
- Circles become ovals.

It is important to remember that because of the 2-dimensional nature of HP EGS, every component must be stored on the system grid. When you set the user grid to isometric, the system automatically translates commands and points into the appropriate commands and points for the system grid. This enables you to use all of the commands as you understand them when creating isometric drawings.

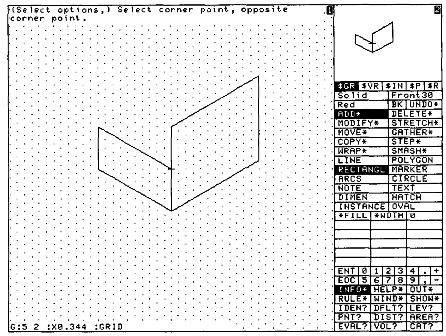
2. Select `Right30` and change the user grid to `*Front30`.

   The x-axis of the user grid is now at an angle of -30 degrees. Try moving the cursor along the x-axis.

3. Select RECTANGL and the point (0, −50).

   Move your cursor around the screen and notice that the dynamic tracking is relative to the isometric front user grid.

4. Complete the rectangle by selecting the point (−100,0). Your complete basic block should look like this:



5. Save the drawing as the new file named:

   `block`

BREAK TIME

Relax and take some time to review this section carefully before you go on. If you are still unsure about what is happening when you make the user grid isometric, refer to *Understanding HP EGS.*

Completing the Block

In this section you will use some of the drawing commands you have already learned with an isometric user grid set. You will also learn a few new methods for these commands, such as *INBT in DELETE*.

Adding the Top

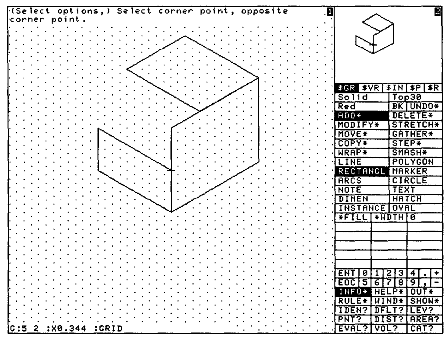
1. Change the user grid to the top of a 30-30 isometric system by selecting:

   Front30 *Top30

2. Select ADD* RECTANGL and the point (−50,90).

   Move your cursor around the screen and notice that the dynamic tracking is relative to the isometric top user grid.

3. Complete the rectangle by selecting the point (−150,170). Your screen should look like this:



4. Select RULE* crSML to get the small, or default, cursor back.

   You could also use the crFUL method of RULE* for setting the cursor size. crFUL sets the cursor to a full screen size.

Adding a Beveled Face

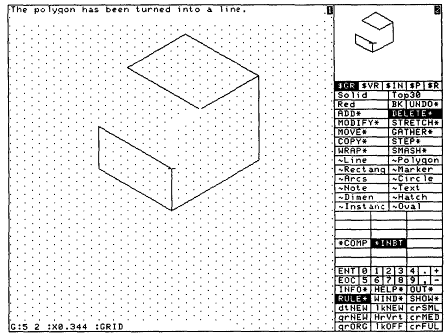
The next task is to add lines between the top and left rectangles of your block to create a beveled face. But first you must delete that portion of the right rectangle which is there now.

1. Select:

   DELETE* *INBT

   To delete a portion of a component between two points you must first select the endpoints of the portion you want to delete. Then you select the portion of the component to delete.

2. Select the grid point that coincides with the lower right corner of the top rectangle (− 50,90). Then select the grid point which coincides with the upper right corner of the left rectangle (0,0).
3. Now that the endpoints of the deletion have been determined, select the upper left corner of the right face (− 50,50) to indicate that this is the portion of the component to delete.



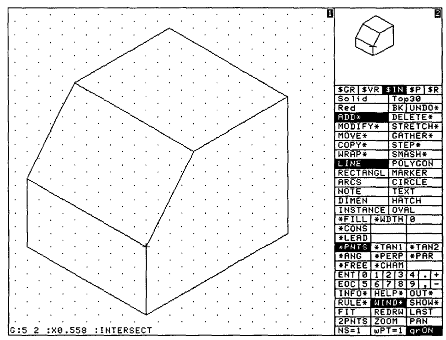
4. Change the snapping mode to intersection snapping:

    `$IN`

5. Select:

    `ADD* LINE`

6. Add two lines between the adjacent corners of the top and left rectangles. Instead of terminating the individual lines with `EOC` this time, try terminating them by selecting the last point twice. An example would be to select the points (− 50,90)(0,0)(0,0).
7. `FIT` the drawing to the large port when you have completed the lines. Your block should look like this:

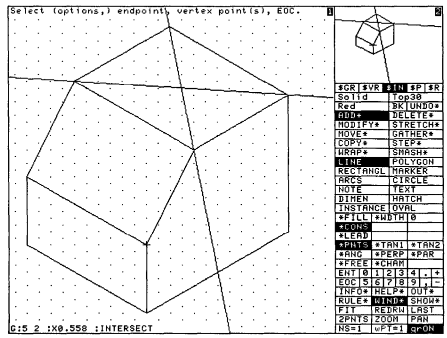


Adding a Hole

Your next task is to put a hole that goes through the center of the top of the block and out through the bottom.

1. To find the center of the top face, turn on the *CONS option and place two lines through opposite corners of the top face. Because you are in the $IN snapping mode, you can snap to the corner points easily.

   Your block should look like this:



2. Select:

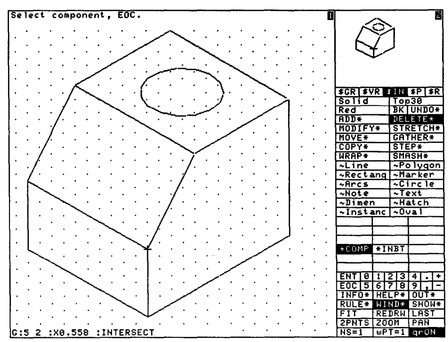
   ```
   CIRCLE *R&C
   ```

   The *R&C method allows you to add a circle of a given radius at its center point. Enter:

   ```
   25
   ```

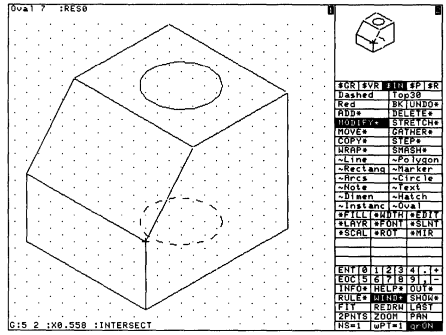
   for the radius and place the circle at the intersection of the two construction lines. Note that the circle is being dynamically tracked as the appropriate oval for this isometric grid setting.

3. Use DELETE* to delete the two construction lines. Your screen should look like this:



4. To indicate a hole in the block you need this same circle in the bottom face but in a dashed line-style.
5. Select COPY* and copy the circle to the bottom face. To copy the circle correctly, select it, then select the upper right corner of the right face (−50,170) as the reference point, then the lower right corner of the right face (50,70) as the destination. End the COPY* command with EOC.

6. Change the line-style from Solid to *Dashed.
7. Select MODIFY*, then the lower circle, and then a reference point. Then select *LAYR to change the circle's layer. Don't forget EOC to finalize the change. Your screen should look like this:



Adding Hidden Lines

Next you will complete the block by adding the hidden lines defining the rear three faces and the hole in the block.

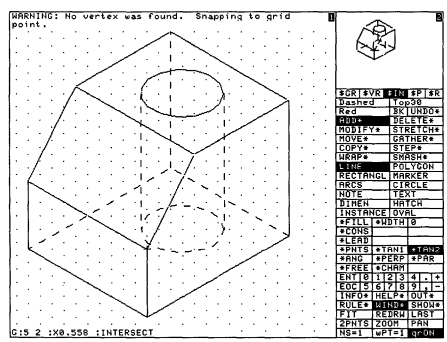
1. Select ADD* LINE *CONS to turn off construction lines, and select the lower right corner of the right face. Use the grid to help you carefully place the bottom rear corner of the block. Now select the lower left corner of the front face to finish the bottom face. End the line.
2. Now select the bottom and top rear corners to add the line defining the rear edge of the block. End the line.
3. Select:

   *TAN2

   The *TAN2 method enables you to add a line tangent to two arcs and/or circles. Select the right side of the top circle and then the right side of the bottom circle.

   Tangencies to arcs and circles in the isometric mode are found only if they were created in the same isometric mode. Remember that isometric arcs and circles are really ovals. Tangencies to ovals are found only if HP EGS can internally convert them back into arcs or circles, based on the grid mode set.

4. Once the right side of the hole has been added, select the left side of the top and bottom circle to finish the hole. Your screen should look like this:



You can now save the completed block drawing using the OUT*SAVE*OLD method.

BREAK TIME

Relax, stretch, and take some time to review this section.

Annotating the Block

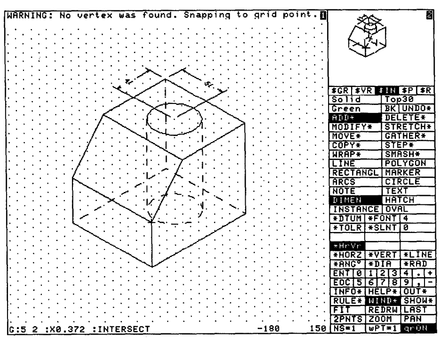
In this section you will learn how to use ADD* DIMEN and many of its methods. You will also learn how to use the *LEAD option of ADD* LINE.

Dimensioning a Circle Location

1. Before you begin adding dimensions to the block, ZOOM the large port out by one and a half times (–1.5). Then change the color to *Green and the line-style to *Solid.
2. Select:

   DIMEN

   The default method, *HrVr, enables you to add horizontal or vertical line dimensions between two points. All of the line dimensioning methods, *HrVr, *HORZ, *VERT, and *LINE, require that the endpoints of the extension lines be selected first. Then the placement point for the value is selected.
3. Select the center point of the circle on the top of the block, then the rear corner of the top of the block. Next, move your cursor past the left rear boundary of the block and then past the right rear boundary of the block. Note the change of dynamic tracking of the dimension lines, from vertical to horizontal. Select a point past the right rear of your block at which to place the horizontal dimension value.
4. Select the center point of the circle, and the rear corner of the block again. Move out past the left rear of the block and select a point at which to place the vertical dimension value. Your screen should look like this:



Dimensioning the Diameter of a Circle

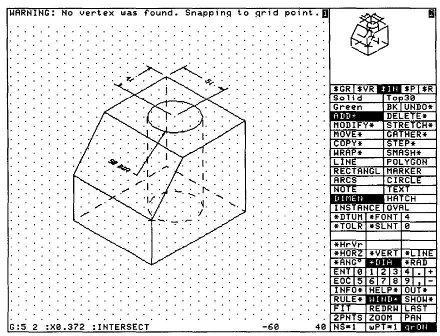
1. Select:

   `*DIA`

   `*DIA` enables you to automatically dimension the diameter of an arc or circle. To use the `*DIA` method, first select the arc or circle to dimension. Then select the placement point for the value. When the dimension is drawn, HP EGS automatically appends `DIA` to the numeric value.

2. Select the circle in the top of the block, then move the cursor into the center of the block and then outside of the circle. The dynamic tracking of the dimension shows you how your dimension will look, depending where you place the value. Select a point in the middle of the beveled face to place the dimension value.

The `*RAD` method of `DIMEN` enables you to dimension the radius of an arc or circle. This method works the same way as `*DIA`, except that when the radius dimension is drawn, HP EGS automatically appends `R` to the dimension value.



Dimensioning the Face of a Block

1. To dimension the right faces of the block, first change the user grid from `Top30` to `*Right30` so that the dimensions will be calculated and skewed properly.

2. Select the option:

   `*TOLR`

   The `*TOLR` option enables you to specify the type of tolerancing you want, and to turn off tolerancing if it is on. The tolerancing options are:

Option	**What Occurs**
`*SNGL`	A single tolerance value, preceded by `+/-`, is placed to the right of the dimension value.
`*PLMI`	Dual tolerance values, a plus and minus value, are placed to the right of the dimension value.
`*HILO`	The dimension value is replaced by two values, one above the other. The upper value is the sum of the dimension value and the plus tolerance value. The lower value is the dimension value less the minus tolerance value.

3. Select:

   `*PLMI`

4. Enter:

   `1`

   for the plus tolerancing value and enter:

   `2`

   for the minus tolerancing value. The menu now displays the selected tolerancing option, `*PLMI`, as well as the plus and minus values to be used with this option.

5. Select the line dimensioning method:

   `*HORZ`

6. Select the lower left corner of the front face and then the lower right corner.

   Move the cursor around the screen and notice that you are restricted to only horizontal dimensioning. Also notice that you can now place the dimension value outside the bounds of the extension lines. This is the major difference between `*HrVr` and `*HORZ`.

7. Now move the cursor below the right face of the block and select a point at which to place the horizontal dimension value.

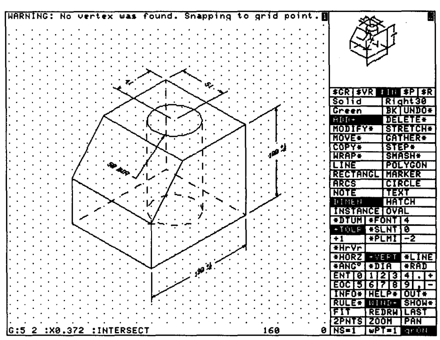
   Notice that plus and minus tolerancing values now appear in the dimension.

8. Select the line dimensioning method:

   `*VERT`

9. Select the lower right corner of the right face and then the upper right corner. Move the cursor around the screen and notice the similarities with the `*HORZ` method.

10. Now move the cursor to the right of the right face and select a point at which to place the vertical dimension value.



Changing the Tolerancing Option

1. To change the tolerancing option, first turn off dimension tolerancing by selecting:

    `*TOLR`

    Then select:

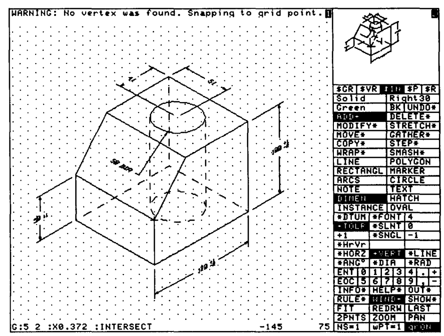
    `*TOLR *SNGL`

2. For the single tolerancing value, enter:

    `1`

    The menu now reflects this selection.

3. The method is still `*VERT`, so select the lower left corner of the front face and then the upper left corner (where the bevel begins). Move the cursor to the left and place the dimension value at that point.



Note the difference in how the tolerance value is added.

Last, you will dimension the angle of the bevel. Because angular dimensioning, using `*ANG°`, can only be done between segments of lines, rectangles, and polygons, you must first add a vertical extension line as a reference for the angular dimension.

4. Select `LINE` and then select a grid point above the upper left corner of the front face. Do not let any part of the cursor touch the corner or you will snap to it because snapping mode is currently $IN. Select a point vertically above the last, but higher than the top of the block (about −100,170). End the line.

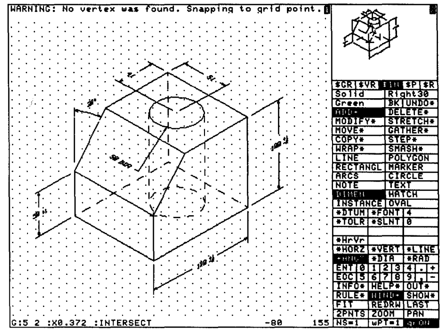
5. Select:

    `DIMEN *ANG°`

    The `*ANG°` method requires you to select the segments of a line, rectangle, or polygon for which the angle between is desired. Then you must select the location where the angular value is to be placed.

6. Before you add the dimension, turn off tolerancing by selecting `*TOLR`.

7. Select the extension line you just added. Then select the left line of the beveled face (again avoiding the corner points). Move the cursor between these lines and then outside of these lines. If you place the value between the lines, the angular value will read near 30 degrees. If you place the value outside the lines, the value will read near 330 degrees. Select a point for the value near the top of the extension line but between both selected lines. Your screen should look like this:



Adding a Note

The final addition to the block is a note with a leader line pointing to the beveled face.

1. Select LINE and then turn on the leader option by selecting:

   *LEAD

   The *LEAD option places an arrowhead at the first point of the line.
2. Change the snapping mode to $P (primitive or component snapping) and select a point near the center of the line defining the lower edge of the bevel.
3. Change the snapping mode to $GR and move to left of the front face and finish the leader line by adding a short segment from (−75, −5) to (−85, −5). Terminate the leader line. Notice the arrowhead that appears at the end near the bevel.
4. Select NOTE and enter the string:

   'Slightly Rough'

   Don't forget the quotation marks.
5. Move the cursor into the large port. Notice that the note string has been appropriately rotated and slanted for the current user grid setting.
6. To make the note the same size as the dimension values, select:

   *FONT
7. Enter:

   4

   The justification position is currently 2, or the center of the left side of the string. It would be easier to place the string next to the leader line if the justification position was 8, or the center of the right side of the string.

8. Select:

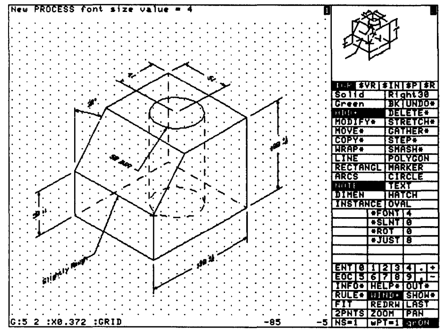
   `*JUST`

9. You are now presented with some aids. `jLEFT` enters 2 automatically, `jCENT` enters 5 automatically, and `jRGHT` enters 8 automatically. Select:

   `jRGHT`

   The value next to `*JUST` is changed to 8 and the cursor is tracking the center of the right side of the note string.

10. Select a point to the left of the leader line at which to place the string. Your picture with a note should look like this:



You have now completed the drawing of your block. You may save the drawing if you want.

To plot the block to a local plotter, make sure the plotter is switched on and has a piece of paper. Then select:

```
OUT* PLOT* CENTR EOC
```

This will plot the block as large as possible, centered on the sheet of paper in your plotter.

Where To Go From Here

Congratulations on your hard work and thanks for being such a good student.

You now have a basic understanding of HP EGS, especially of the General Drawing personality. The other three personalities, Mechanical Drafting, Schematic Drawing, and PC Board Layout, are very similar to General Drawing. In fact, many of the commands are identical. But you will find that each personality has specific differences which make it more appropriate for the task it was designed for.

Proceed to the chapter of your choice to learn to use HP EGS in your application. We encourage you to go through all of the chapters to learn more about the diverse capabilities of HP EGS. You could also learn techniques for using HP EGS that may be appropriate for your particular application.

Mechanical Engineering Drawings

Chapter 6: Creating a Flange Drawing

Chapter 7: Making a Flange Assembly Drawing

Creating a Flange Drawing

Chapter 6

Introduction

As discussed in earlier chapters, HP EGS is a very customizable drawing system. With HP EGS, Hewlett-Packard has provided a pre-configured customization of the basic graphics editor for Mechanical Engineering.

The Mechanical Drafting personality makes mechanical drawings quick and convenient to prepare by providing menus, macros, process files, and some commonly used mechanical parts and geometric tolerancing symbols. The personality operates the same as the General Drawing Personality, with some changes appropriate for the creation of mechanical drawings with library parts and symbols.

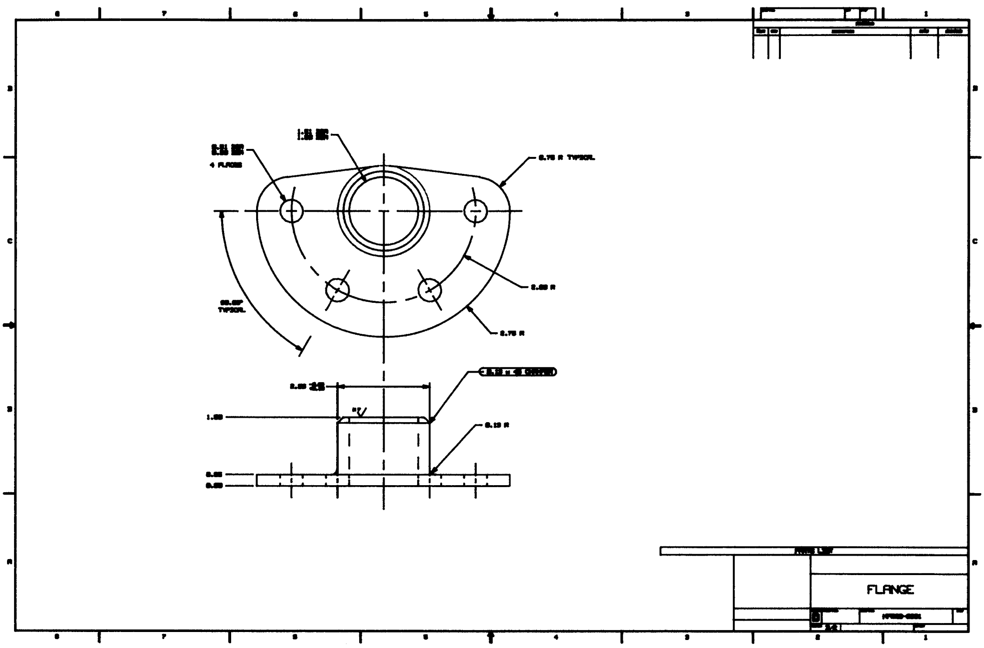
This personality also provides post-processors that enable you to generate material lists (bills of materials) of the parts that you use to create a mechanical drawing. HP EGS can output these lists to a printer, and can format them to be included in the original drawing.

Note

Before you begin the lessons in this chapter, be sure you have read Part II of this manual. It contains necessary basic information that is not repeated in this chapter.

In this chapter you will learn how to use the Mechanical Engineering capabilities of HP EGS by creating an annotated drawing of a simple flange. In Chapter 7 you will create an assembly drawing of two of these flanges bolted together. Included in this assembly drawing is an automatically generated parts list.

You will also learn the differences between the Mechanical Drafting personality and the General Drawing personality. Then you will use the Mechanical Drafting personality to create the annotated flange drawing shown in the following figure.



Understanding Mechanical Drafting

To use the Mechanical Drafting personality you need to understand the basic differences between this personality and the General Drawing personality, which you should already understand. We will approach these differences by first examining how the process file has been changed and then how the macro file has been changed.

The Process File

The HP EGS process file defines the system units, the default parameters, and the layer definitions for the data in your drawing. For this personality, the basic unit of measurement is the inch, with 10,000 system points per inch. This means that the location of every point in your drawing can be specified to 0.0001 inches. By changing the user grid setting and snapping modes, you may force points to fall on some multiple of the basic resolution.

Several defaults are different due to this change in units. For example, the default font size is 0.125 inches, where it was 10 millimetres in General Drawing.

As you learned by using the General Drawing personality, all of the components in your drawing are placed on discrete layers. Recall that each layer is like a transparent sheet which exactly overlays the other layers. Each layer specifies the color and line-type of the component as well as the pen it will be plotted with.

The Mechanical Drafting personality takes advantage of this layering capability by separating the process file into two sections. The first section contains the components used to annotate your drawings, and the second section contains the components used to create the basic geometry. The annotation layers are defined as follows:

Layer	Use	Color	Pen #
0	Instances boundaries	White	1
$$Annotations			
1	Solid annotation geometry	Green	7
2	Dashed center-line annotation geometry	Green	7
3	Phantom line annotation geometry	Green	7
4	Dashed line annotation geometry	Green	7
6	Note and text components	Green	8
7	Dimension components	Green	8
8	Item information for the generation of parts list data	Green	8
9	Extra annotation geometry	Green	8

The first geometry layer of interest is used to contain all construction geometry. It is defined as follows:

Layer	Use	Color	Pen #
5	Contains all construction geometry as short dotted lines	Blue	0

This layer has a color of blue and a line-type of short dotted so that construction geometry does not overpower other geometry in your drawing. It also has a pen number of 0, which means that it will not be plotted.

The balance of the geometry layers are defined in sets of ten. Each set of layers contains a reserved layer (white color and unplotted), a layer for each line-type, and an extra layer which has a solid line-type. The colors and pen numbers for these sets of layers are:

Layer Set	Color	Pen #
10-19	Red	1
20-29	Yellow	2
30-39	Green	3
40-49	Cyan	4
50-59	Blue	5
60-69	Magenta	6

Following is the definition for the set of red layers. The other sets are defined in a similar format.

Layer	Use
$$Geo	1
10	Reserved (special use, white, short dotted lines)
11	Solid line geometry
12	Long dashed center-line geometry
13	Phantom line geometry
14	Dashed line geometry
15	Dotted center-line geometry
16	Long dashed line geometry
17	Dotted line geometry
18	Short dotted line geometry
19	Extra geometry (solid line)

This format for the layer definition enables you to easily select color and line-type using the mode commands provided in the Mechanical Drafting personality. This does not include the reserved layers, which are used for special purposes internally in the personality.

You can change the process file parameters using such commands as UNITS and EQUATE. For more information on changing the process file, see the Customizing chapter of *Understanding HP EGS.*

The Macro File

The macro file contains the definitions of all the items that appear in the screen menu. Because the macro file for the Mechanical Drafting personality is based on the macro file for the General Drawing personality, you should be familiar with its operation. Where appropriate the macro file has been changed. Some of the important changes follow.

Changes to the Modes

A major difference to the General Drawing personality is the presence of two overall drawing modes, Geometry (create geometry) and Annotatn (create annotations). These modes are defined as follows:

- Geometry is used to create geometry. When using this mode you are restricted to the geometry layers and to those components which can be placed in them. That is, you cannot add components such as DIMEN and NOTES.
- Annotatn is used to create annotations. When using this mode you are restricted to the annotation layers and to the components that can be placed in them.

The isometric modes (Front30, Right30, and Top30), which set the angles of the isometric x and y axes automatically, are present while the drawing mode is set to either Geometry or Annotatn. These modes behave the same as in General Drawing. You can also set the isometric axis angles to any value using the X_Axis and Y_Axis mode commands.

While using either of the drawing modes, a line-type mode such as Solid is available. The state of this mode is stored in each drawing mode. For example, if you set the line-type mode to Phantom in Geometry and then switch to Annotatn, the line-type changes to Solid. When you return to the Geometry mode, the line-type changes back to Phantom.

While Geometry is set you have another mode for the color. This enables you to access a specific set of geometry layers as defined in the process file. This mode is not available while Annotatn is set.

While Annotatn is set you have another mode, Defaults, which enables you to manipulate the dimensioning defaults. The defaults you can set are:

- D_DIRECT: Sets the dimensioning value orientation. The choices are *HOR for horizontal values and *INLN for in-line values (preset to *HOR).
- D_DECIM: Sets the number of decimal places to be included in dimensions (preset to 3).
- D_SCALE: Sets the scaling factor for dimension values. This enables you to accurately dimension scaled parts (preset to 1).
- D_UNIT: Sets the units to be used in dimensions. The choices are DCMLS for decimal values; IN*FR for inches and fractions of inches; and FT*FR for feet, inches, and fractions of inches (preset to DCMLS).
- D_EXTENS: Sets the distance between the ends of extension lines and objects (preset to 0.125 inches).
- TOL_FONT: Sets the font size of dimension tolerance values (preset to 0.0938 inches).

Note that only those dimensions which are added *after* defaults are changed are affected by those changes, except for D_EXTENS which affects *all* dimensions that are on the drawing. The Default mode is not available while Geometry is set.

Changes to the Primary Commands

The following additional options and methods have been added to the primary commands to make the creation of mechanical drawings easier.

- For ADD* LINE: The new method *CNTR enables you to select an arc or circle in which you want precise center lines. Then 6 lines are automatically added to create the center lines.
- For ADD* OVAL: Three new methods are available. *C&Rs enables you to select the locations of the center and the two endpoints of the major and minor radii of an oval. *RADs enables you to enter the major and minor radii and the rotation angle, and then to select the locations of the oval. *RATO enables you to enter the major radius, the ratio of the minor to the major radius, and the rotation angle, and then to select the locations of the oval.
- For DELETE*: The new method *ALL enables you to select a component type for which you want to delete all. NOTE: Be careful when using this method, because UNDO* does not bring back components deleted in this way.
- For MODIFY*: Two new options are available. in>mm enables you to convert a value in inches to one in millimetres, and mm>in enables you to convert a value in millimetres to one in inches. Any dimension, note, or text string may be converted as long as the value is the first set of characters in the string.

- For GATHER*: The new method *DIMN enables you to select a dimension on your drawing to specify the direction and distance to move a gathered collection.
- For STEP*: The new method *POL2 enables you to execute a polar step by selecting the polar center, and then entering the number of copies and the position of the first copy.
- ZONE_OFF: Turns off the display of the zone markings in C, D, and E size drawing borders.
- ZONE_ON: Turns on the display of the zone markings in C, D, and E size drawing borders.
- LIST_MAT: Compiles a list of the items (parts) used in a mechanical drawing. This list can then be post-processed to produce a formal parts list.

Changes to the Components

This personality includes some "special" components that have been created using macros. These are:

- BUBBLE: This component is actually two lines and two arcs that define a closed shape. There are two methods for adding bubbles: *RECT and *CMPT. *RECT enables you to define a rectangular area to enclose in a bubble, and *CMPT enables you to select any component to enclose in a bubble.
- ITEM: This component is actually a circle and 2 to 4 associated notes. It tracks item numbers, quantities, identification numbers, and zone information for processing to create parts lists.

  There are two methods for adding items: *DATA and *BLNK. *DATA adds an item with an associated item number (tag 1051), an associated quantity (tag 1052), an associated identification number (tag 1053), and an associated zone (tag 1055). *BLNK adds an item with an associated item number (tag 1051) and an associated quantity (tag 1052). NOTE: At least one occurrence of an item with a given item number must be added using the *DATA method.

Changes to the Secondary Commands

The only change made to the secondary commands is the addition of more classifications of components that can be turned on and off with SHOW*. These classifications are:

- ANNOS: All components in the annotation layers are affected.
- CONST: All construction geometry is affected.
- ITEMS: The undisplayed part identification and zone information in items are affected.

Loading Mechanical Drafting

1. If your computer is not displaying the main menu in the HP EGS manager, follow the instructions in Chapter 2, in the *Loading General Drawing* section, to get to the main menu. Your display should look like this:

```
Use arrow keys to move cursor to desired function
Select function with <RETURN> or <ENTER>.

  ->System Utilities
    General Drawing Editor
    Electrical Engineering
    Mechanical Engineering
    IGES Translator
```

2. Move the cursor to the Mechanical Engineering function and press (Return). The display should look like this:

```
Use arrow keys to move cursor to desired function
Select function with <RETURN> or <ENTER>.

  ->RETURN To Main Menu
    Mechanical Drafting Editor
    ME Parts File Editor
    ME Material Lister
```

3. Select:

```
Mechanical Drafting Editor
```

The Mechanical Drafting personality will now load automatically.

In the next section of this chapter you begin the drawing example in which you learn to create a flange using the Mechanical Drafting personality. To preview this lesson, enter:

```
INPUT EWLRN:chap6;
```

Note

Before you begin to draw the flange, use the PREFIX command to prefix HP EGS to your personal mass storage volume. This keeps you from editing or destroying someone else's data as you edit and save drawings.

Starting the Flange

In this section you will begin to create the flange by drawing its top view.

Preparing the Screen

1. Prefix to your storage volume by entering:

   ```
   PREFIX DRAW:;
   ```

2. To edit a new drawing called flange, enter:

   ```
   EDIT flange;
   ```

3. To set the drawing mode to geometry creation, enter:

   ```
   Geometry
   ```

4. Prepare the drawing area for the addition of components by turning the grid on and zooming in on a smaller portion of the large port. Select:

   ```
   WIND* grOFF ZOOM -20
   ```

Adding Construction Lines

You are ready to add the basic construction lines for the flange.

1. Select:

   ```
   ADD* LINE *CONS
   ```

   and then select (0.0,0.0) and (0.0,1.5) to add a vertical construction line. Add a horizontal construction line by selecting (0.0,0.0) and (−1.5,0.0).

2. To place two more vertical construction lines 2 inches on either side of the existing one, use the datum method of copy. Select:

   ```
   COPY* *DATM
   ```

   then select the vertical construction line (0.0,1.0), and a reference point on it (0.0,0.0).

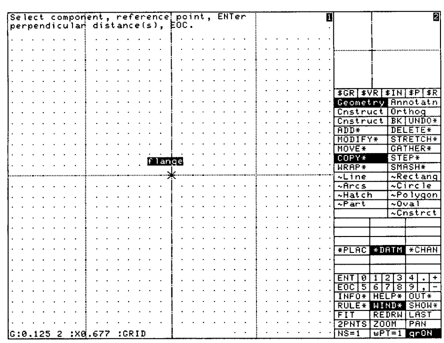
3. Enter:

   ```
   2
   ```

   and

   ```
   -2
   ```

   Your screen should look like this:



Adding Construction Circles

You can now add the 2-inch construction circle defining the central diameter and the 1.5-inch construction circles defining the ends of the flange.

1. Set the snapping mode to intersection snapping ($IN).
2. Prepare to add circles by their center points and radii by selecting:

   `ADD* CIRCLE *R&C`

3. Note that the construction option is still set (*CONS is highlighted). Place the center circle by entering:

   `1`

   and then selecting the intersection of central horizontal and vertical construction lines (0.0,0.0).

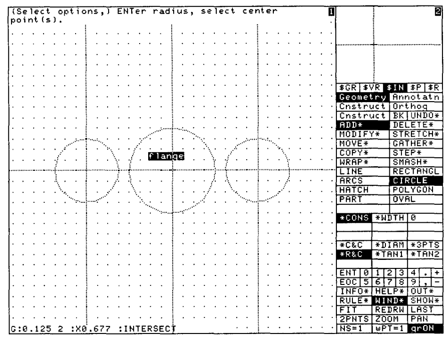
4. Place the end circles by selecting:

   `*R&C`

   and then entering:

   `.75`

   and then selecting the intersections of the horizontal and remaining vertical construction lines (−2.0,0.0 and 2.0,0.0). Your screen should look like this:



Adding Semicircle and Tangent Lines

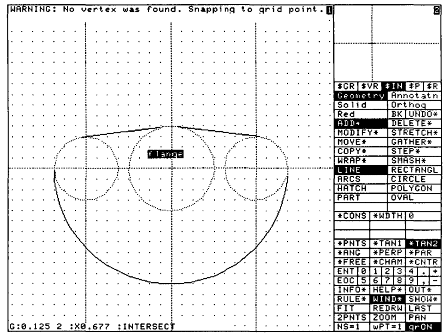
1. To add the solid geometry that outlines the flange, first add the semicircular arc which defines the forward portion. Select:

   `ARCS *CONS *SEMI`

2. Note that *CONS must be selected to turn off the construction geometry option. Since semicircular arcs are drawn counter-clockwise, select the left intersection of the left construction circle and the horizontal construction line (−2.75,0.0), and then the right intersection of the right construction circle and the horizontal construction line (2.75,0.0).
3. Add the tangent lines at the rear of the flange. Because these lines are tangent to two construction circles, select:

   `LINE *TAN2`

4. To add the left tangent line, select the rear side of the left construction circle (−1.7,0.7), and then the rear of the center construction circle (−0.5,0.8). To add the right tangent line select the rear side of the center construction circle (0.5,0.8), and then the rear of the right construction circle (1.7,0.7). Your screen should look like this:



Adding Arcs

There are many ways to add the arcs defining the rear corners of the flange. Here is a way that uses the *3PTS method of arcs.

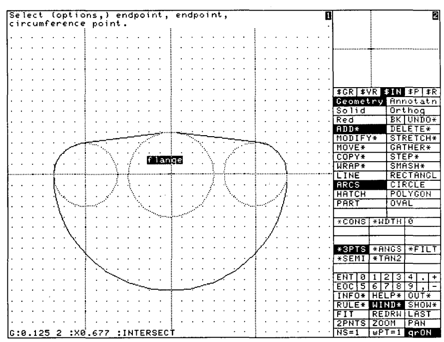
1. Select:

   ```
   ARCS
   ```

2. To add the left arc, select the left end of the semicircle (−2.75,0.0), then select the left end of the left tangent line (−2.125,0.75). To choose the third point, first change the snapping mode to primitive snapping, then select a point on the left construction circle, between the first two selected points (−2.5,0.6).

3. To add the right arc, set the snapping mode back to intersection snapping. Then select the right end of the right tangent line (2.125,0.75), then the right end of the semicircle (2.75,0.0). Again, set the snapping mode to primitive snapping, and then select a point on the right construction circle (2.5,0.6).

4. Before continuing with this lesson, set the snapping mode back to intersection snapping. Your screen should look like this:



Adding Diameter Holes

Four 0.5-diameter holes now need to be added, equally spaced, in a semicircular pattern.

1. To add the first hole, select:

   `CIRCLE *R&C`

2. For the radius value, enter:

   `.25`

3. Select the intersection of the horizontal and leftmost vertical construction lines as the circle's center point (–2.0,0.0). The final three circles will be copied from the one you just added using the *POLR method of STEP*. Select:

   `STEP* *POLR`

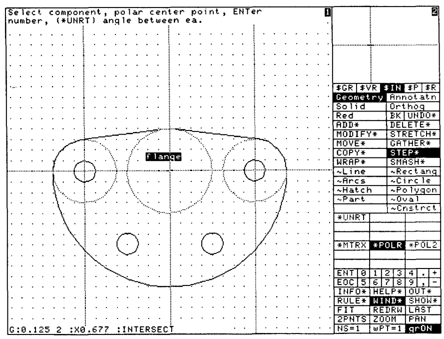
4. Select the circle you just added (–1.9,0.2), then select the intersection of the horizontal and central vertical construction lines as the polar center point (0.0,0.0). For the number of copies, enter:

   `3`

5. Because you need three more circles in the next 180 degrees, enter the following expression as the angle between each copy:

   `(180/3)`

   Your screen should look like this:



Adding Center Lines

1. To add the center lines for the holes, first change the line-type to Center by selecting `Solid *Center`. Next, add a circle that passes through all of the holes you just added:

   `ADD* CIRCLE`

2. Select the center of the flange (0.0,0.0), and then the center of the left hole (–2.0,0.0).

3. To remove the rear portion of this circle, set the snapping mode to primitive snapping, and then select:

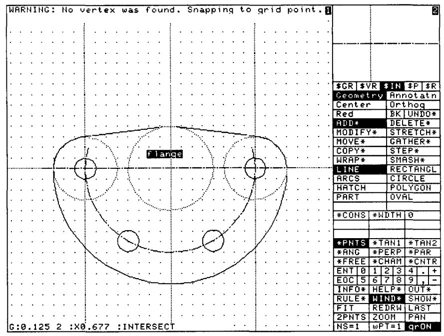
   `DELETE* *INBT`

4. Select a point on the circle you just added, counter-clockwise of the right hole (1.9,0.5). Then select another point on the circle, just clockwise from the left hole (– 1.9,0.5). Finally, select a point on the circle, to the rear of the flange, which indicates the portion of the circle to delete (– 0.4,2.0).

5. Set the snapping mode back to intersection snapping, and then select:

   ADD* LINE

6. Add a horizontal center line from the left side of the flange (– 3.0,0.0) to the right side of the flange (3.0,0.0). Your screen should look like this:



Adding Center Lines at an Angle

1. To add the short, radial, center lines to the remaining two holes, first create two short, horizontal center lines. Select the points (– 1.5, – 1.25) and (– 0.5, – 1.25), then EOC for the first line. Then select the points (0.5, – 1.25) and (1.5, – 1.25), then EOC for the second line.
2. Now you will rotate and move these lines to the appropriate positions. Select:

   MODIFY*

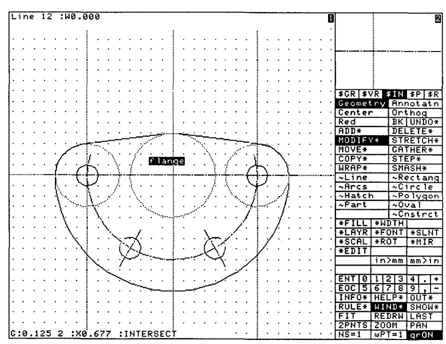
3. Select the right line (1.0, – 1.25) and then the center of the line as the reference point (1.0, – 1,25). Rotate the line counter-clockwise 60 degrees by selecting:

   *ROT -60 ENT

4. Select the center of the lower right hole as the new location (1.0, – 1.75), then EOC.
5. To modify the left center line, select it at (– 1.0, – 1.25) and then select its center as the reference point (– 1.0, – 1.25). Rotate the line clockwise 60 degrees by selecting:

   *ROT 60 ENT

6. Select the center of the lower left hole as the new location (– 1.0, – 1.75), then EOC. Your screen should look like this:



Drawing the Cylindrical Hub

The final task in finishing the geometry is to draw the cylindrical hub.

1. Set the line-type to Solid by selecting:

   `Center *Solid`

   The cylinder has an outside diameter of 2 inches and an inside diameter of 1.5 inches, and has a 0.125 x 45 degree chamfer on the outside edge. Because the central construction circle has the appropriate diameter, you only need to modify it to the current line-type and color. (That is, to the current layer.)

2. Select the construction circle ($-0.5, -0.5$) and a reference point ($-0.5, -0.5$). Then select the modification option to change a components layer to the current layer:

   `*LAYR`

3. Terminate the modification with EOC.
4. To add the remaining two circles, select:

   `ADD* CIRCLE *R&C`

5. As the radius for the inside circle, enter:

   `.75`

   and place it at the center of the flange (0.0,0.0).

6. To change the radius, select:

   `*R&C`

7. As the radius of the chamfer break, enter:

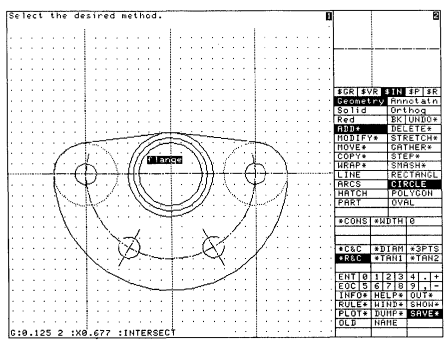
   `.875`

   and place it at the center of the flange (0.0,0.0).

8. As a precaution, save the flange. Because the flange drawing is already named, you need only to select:

   `OUT* SAVE* OLD`

   Your screen should look like this:



BREAK TIME

Take a short break now. When you return you will draw the front view of the flange. In this section you:

- Learned about the process file and macro file.
- Loaded the Mechanical Drafting personality.
- Added construction lines and circles to the flange.
- Added semicircle and tangent lines.
- Added arcs, diameter holes, and short center lines.
- Drew a cylindrical hub.

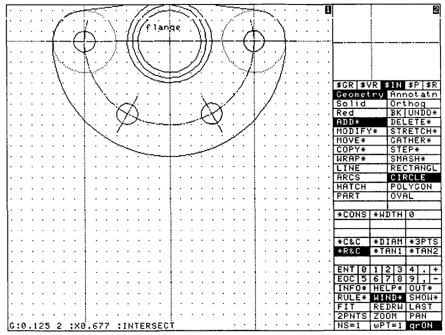
Creating the Front View of the Flange

In this section you will complete the geometric description of the flange by drawing the front view.

Before starting the front view, you need more room to draw in the large port. Select:

```
WIND* PAN
```

and then select (0.0,-3.0). This provides enough room to create the front view and also leaves a portion of the top view in the port for reference. Your screen should look like this:



Adding Horizontal and Vertical Construction Lines

1. To begin the front view, add the horizontal construction lines by selecting:

   ```
   ADD* LINE *CONS
   ```

   and then selecting $(-3.0, -6.0)$ and $(2.5, -6.0)$.
2. Add the two construction lines which determine the height of the hub and the thickness of the flange plate. Select:

   ```
   COPY* *DATM
   ```

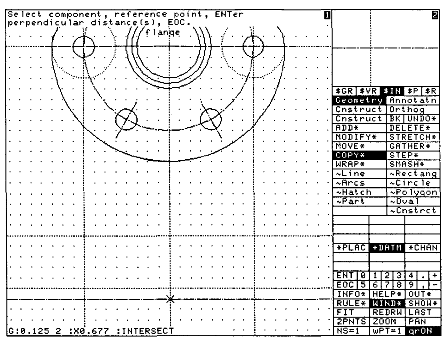
3. Select the horizontal construction line just added $(1.0, -6.0)$ and a reference point on it $(0.0, -6.0)$. For the thickness of the plate, enter:

   ```
   .25
   ```

4. For the height of the hub, enter:

   ```
   1.5
   ```

   Your construction lines should look like this:

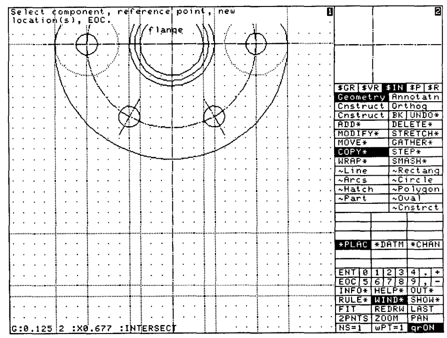


5. Now you will add vertical construction lines, which project the width of the flange and the inside and outside diameters of the hub. You can do this easily by placing copies of an existing vertical construction line at the points mentioned above. Select:

```
*PLAC
```

6. Select the central, vertical, construction line (0.0,-3.5) and a reference point on it (0.0,-4.5). Then place copies at the following points:

```
(-2.75,0.0)
(-1.0,0.0)
(-0.75,0.0)
(0.75,0.0)
(1.0,0.0)
(2.75,0.0)
```



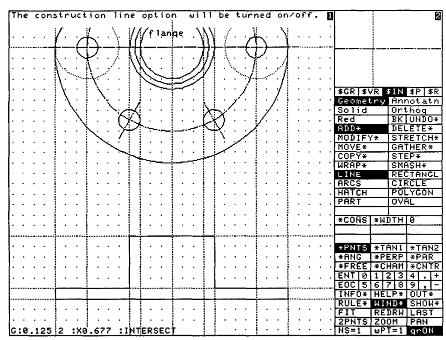
Remember that the current snapping mode is set to $IN, so the placement points for the construction lines will snap to the appropriate intersections. Your vertical construction lines should look like this:

7. Now you can add the outline of the front view of the flange by adding a solid line to the appropriate vertices. *CONS must be selected to turn off the construction geometry option. Select:

```
ADD* LINE *CONS
```

8. Select the following points (intersections of construction lines) for the outline:

```
(-2.75,-6.0)
(-2.75,-5.75)
(-1.0,-5.75)
(-1.0,-4.5)
(1.0,-4.5)
(1.0,-5.75)
(2.75,-5.75)
(2.75,-6.0)
(-2.75,-6.0)
```



and terminate the line with EOC. Your screen should look like this:

Detailing the Hub

1. To easily detail the cylindrical hub portion of the flange, first zoom in on this section of the flange. Select the window method:

   ```
   2PNTS
   ```

   and then select (−1.5, −6.5) and (1.5, −4.5).
2. Add the fillets at the base of the hub by selecting:

   ```
   ARCS *FILT
   ```

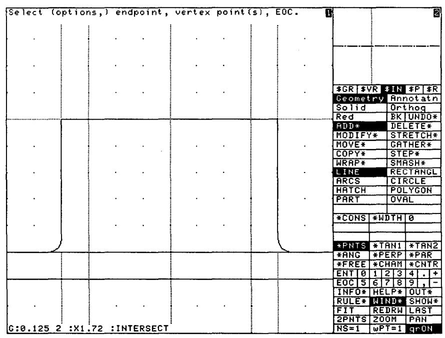
   Enter the radius value:

   ```
   .125
   ```

   and select the two base corners (1.0, −5.7) and (−1.0, −5.7).
3. Select:

   ```
   LINE
   ```

   and add a line from the left end of the left fillet (−1.1, −5.75) to the right end of the right fillet (1.1,-5.75). Your screen should look like this:



4. To chamfer the corners at the top of the hub, select:

   ```
   *CHAM
   ```
5. Enter the chamfer edge distance and chamfer angle, respectively:

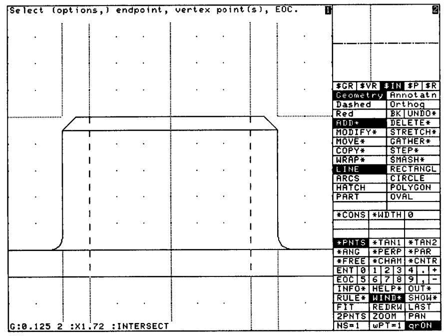
   ```
   .125
   0
   ```

   A zero chamfer angle instructs HP EGS to symmetrically break the corner, regardless of the angle of the corner.
6. Choose the corners to be chamfered by selecting (−1.0, −4.6) and (1.0, −4.6).
7. Select:

   ```
   *PNTS
   ```

   and add a line from the base of the left chamfer (−1.0,4.6) to the base of the right chamfer (1.0,4.6).

8. To complete the detail of the hub, add the hidden lines for the internal hole. Set the line-type to `Dashed`. Select `LINE`. Then add the first hidden line from (0.75, −4.5) to (0.75, −6.0), and add the second one from (−0.75, −4.5) to (−0.75, −6.0). Your screen should look like this:



Deleting Excess Construction Lines

The next stage is to add the center lines and hidden lines for the four holes in the base of the flange. But first, delete the unnecessary construction lines.

1. Return to the previous state of the large port by selecting:

   `LAST`

2. Select:

   `DELETE*`

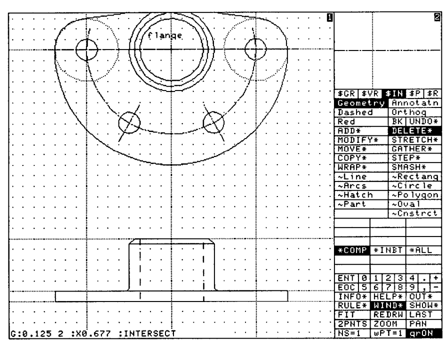
   and delete the construction lines at the following points (remember to use EOC to complete each deletion):

   ```
   (-2.75,-3.5)
   (-1.0,-3.75)
   (-0.75,-3.75)
   (0.75,-3.75)
   (1.0,-3.75)
   (2.75,-3.75)
   ```

3. You may notice that while deleting the construction lines, some of the geometry seemed to disappear. To make it reappear, select:

   `REDRW`

   Your screen should look like this:



Creating More Construction Lines

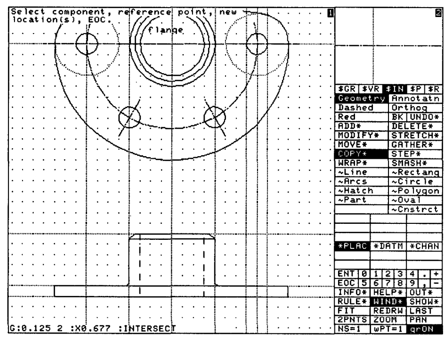
To make the creation of the hole geometry in the front view easier, add more vertical construction lines to project the width of the right hole and the centers of the middle two holes into the front view. Again, you can do this by placing copies of an existing vertical construction line at the points mentioned.

1. Select:

   ```
   COPY*
   ```

2. Select the central, vertical, construction line (0.0, −3.5) and a reference point on it (0.0, −4.5). Place copies at the following points:

   ```
   (2.25,0.0)
   (1.75,0.0)
   (1.0,-1.75)
   (-1.0,-1.75)
   ```

   Remember that the placement points for the construction lines will snap to the appropriate intersections because the current snapping mode is set to $IN.

Creating the Holes

To create the hole geometry for the four holes, you will first add the two hidden lines and the center line to the right hole. You will then wrap the three lines into an instance and copy this instance to the locations for the other three holes. Finally, you will smash the four instances.

1. Add the two dashed lines on the construction lines projected from the top view by selecting:

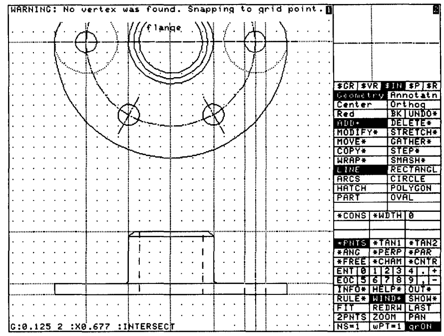
   ```
   ADD* LINE
   ```

   and placing the first one from (1.75, −6.0) to (1.75, −5.75) and the second one from (2.25, −6.0) to (2.25, −5.75).

2. Set the line-type to Center. Then select:

   ```
   LINE
   ```

   and add the center line from (2.0, −6.25) to (2.0, −5.5). Your screen should look like this:



3. To wrap these three lines into an instance, select:

   ```
   WRAP*
   ```

4. Select a reference point on the center line (2.0, −6.0) and enclose the three lines in a box. The points (1.5, −6.5) and (2.5, −5.5) work well.
5. To make the three copies of this instance, select:

   ```
   COPY*
   ```

   and select the instance (2.0, −6.25).
6. It is important to select a reference point that lies on a known point so that the copies can be placed accurately. Therefore, select the intersection of the center line of the hole and the lower surface of the flange plate (2.0, −6.0). Now place the three copies at the intersections of the construction lines projected from the center of the remaining holes and the lower surface of the flange plate. Approximate locations are:

   ```
   (1.0,-6.0)
   (-1.0,-6.0)
   (-2.0,-6.0)
   ```

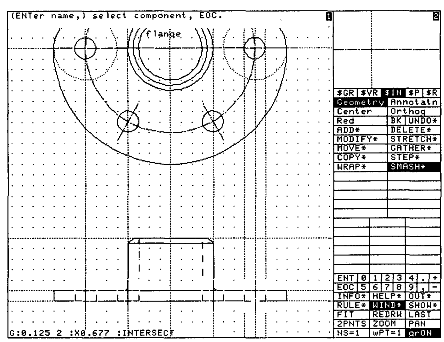
7. To smash these instances, select:

   ```
   SMASH*
   ```

   and select the four instances. Remember to terminate each smash operation with EOC. Approximate locations are:

   ```
   (2.0 -6.0)
   (1.0,-6.0)
   (-1.0,-6.0)
   (-2.0,-6.0)
   ```

   Your screen should look like this:



Adding a Center Line and Removing Construction Geometry

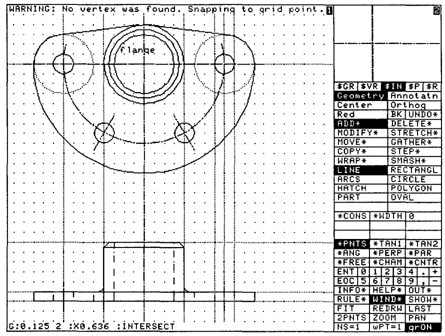
1. To add the vertical center line from a point below the front view to a point above the top view, first get both parts in the large port by selecting:

   `FIT`

2. To add the line, select:

   `ADD* LINE`

   and select two points on the center construction line, (0.0, −6.5) and (0.0,1.25). Your screen should look like this:



3. Next, you will remove all construction geometry from the drawing. You could use the component method of delete, but it is tedious. An easier and faster method is to use *ALL, which deletes as many specified components as can be found. Save your drawing as a precaution before using the *ALL method of delete. Select:

   `OUT* *SAVE OLD`

4. Select:

   `DELETE* *ALL`

   The following prompt appears:

   `Select the component type to delete all of.`

5. Use care when making your component selection. Select:

   `~Cnstrct`

   The display reads:

   `Delete selection of the current line-type and color (Y does it)?`

6. This prompts you for confirmation that the selected components on the current layer are to be deleted. Enter:

```
Y
```

All construction geometry is deleted. Note that you must respond with an uppercase 'Y', or the system assumes you mean 'no'.

WARNING

Be careful when you use the *ALL method of DELETE* because UNDO* does not bring back components deleted in this way. Normally it is a good practice to save your drawing just before using this method.

---

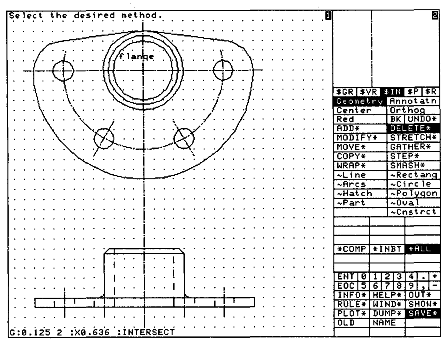
7. Display the undrawn portion of the remaining components:

```
WIND* REDRW
```

8. Save this final version of the flange by selecting:

```
OUT* *SAVE OLD
```

Your screen should look like this:



BREAK TIME

Relax, take a break, and review how you created the front view of the flange:

- Added horizontal and vertical construction lines.
- Detailed the hub.
- Deleted unnecessary construction lines.
- Created new construction lines.
- Created a hole geometry.
- Added a center line.
- Removed all construction geometry.

Annotating the Flange Drawing

In this section you will create the annotated drawing of the flange you completed in the last section. The annotations include dimensions, notes, symbols, and bubbles.

Initializing a Drawing

1. Begin drawing the annotated flange by editing a new drawing called `fldwg`. Enter:

```
EDIT fldwg;
```

2. To set the drawing mode to annotation creation, select:

```
Annotatn
```

3. Add a horizontal D-size drawing border by selecting:

```
ADD* BORDER D_Hor
```

   Notice that the large and small ports are automatically fit to the size of the border and that the drawing origin (point 0,0) is in the center of the border.

4. To make this lesson easier, move the border into the upper right quadrant so that all x and y values will be positive. Enter:

```
MOVE*  -14.0,-8.5  0.0,0.0  17.5,10.5
```

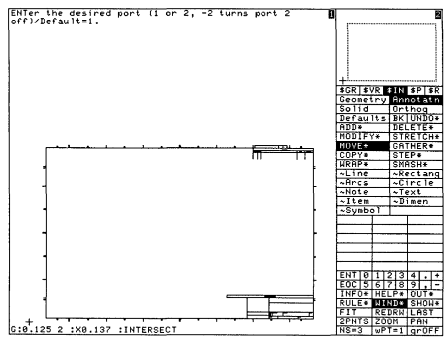
5. To see inside this 'library part', set the nesting level to 3 and fit the drawing to the large port. Select:

```
WIND* NS=1 3 ENT FIT
```

6. You also need to fit the drawing to the small port. To do this, select:

```
wPT=1 2 ENT FIT wPT=2 1 ENT
```

   Your screen should look like this:



Adding a Flange

Now that the border has been selected, the flange drawing you created in the last section can be added. Recall from the discussions about instances in the General Drawing part of this manual that any drawing can be an instance in any other drawing. In the Annotation mode of Mechanical Engineering, most instances you add to drawings are 'symbols'. This is why SYMBOLS is in the screen menu. (*SCRN will also access a list of symbols.) Therefore, you may add any instance when SYMBOLS is selected.

1. To add the flange drawing (instance), select:

   `ADD* SYMBOLS`

2. Enter:

   `flange`

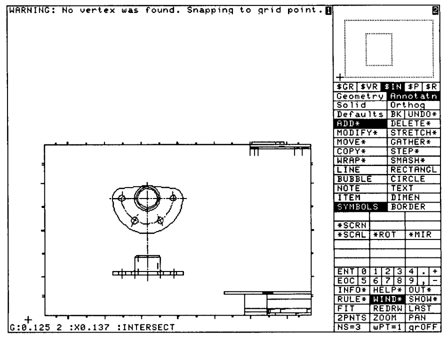
3. For this example you need to scale the flange by a factor of 1.5. Select:

   `*SCAL`

4. Enter:

   `1.5`

   and place the flange at (14.0,14.25). Your screen should look like this:



Preparing to Add Dimensions

Note

You can change any of the dimensioning defaults at any time. Only those dimensions added after the change will be affected by the new default (except for D_EXTENS).

---

1. Turn on the display grid and zoom in on the top view of the flange by selecting:

   `grOFF 2PNTS`

   and then selecting the points (8.0,8.0) and (20.0,18.0).

2. Before adding dimensions, you need to change the dimension scale default to match the scale of the flange. Otherwise, the dimension values will be incorrect. Select:

   `Defaults D_SCALE`

3. Enter:

   `1.5`

   This sets the scale to exactly match that of the flange. If you ever doubt the scale of an instance, use the INFO* method IDEN? to identify it. HP EGS provides you with the scale and the name of the instance.

4. Select:

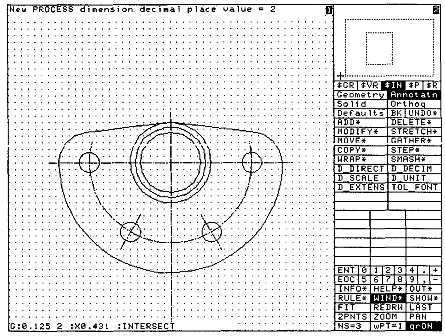
   `D_DECIM`

5. Enter:

    ```
    2
    ```

    This changes the number of decimal places in dimensions from the default of 3 to 2.

Your screen should look like this:



Adding an Angular Dimension

1. The first dimension to add is the one that defines the angle between the upper left hole and the lower left hole. Select:

    ```
    ADD* DIMEN *ANG°
    ```

    and then select the short center line that passes through the lower left hole (12.4,11.5).

2. Select the horizontal center line near the upper left hole (11.6,14.25), and place the value for the dimension at (9.5,11.5).

3. Extension lines are not automatically added for angular dimensions because these angles are often placed directly on object lines. In this example, you need to add extension lines for the angular dimension, so select:

    ```
    LINE
    ```

    and add a line from (9.25,14.25) to (8.5,14.25).

4. To add the second extension line you will make a copy of the first one on top of itself and then rotate this copy 60 degrees about the center of the flange. Select:

    ```
    COPY*
    ```

    and select the first line (9.25,14.25), select a reference point on the line (8.75,14.25), and then place a copy on top of it (8.75,14.25).

5. To rotate the copy, select:

    ```
    MODIFY*
    ```

    then select the line (9.0,14.25), change the snapping mode to intersection snapping, and select the center of the flange as the reference point (14.0,14.25).

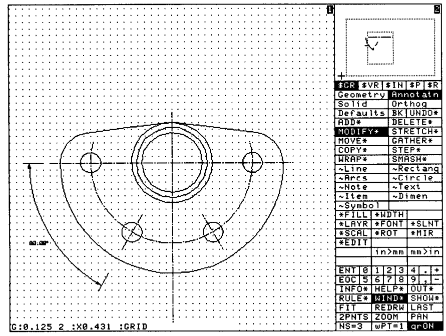
6. Select:

   `*ROT`

   and then enter:

   `60`

7. Terminate the modification with EOC. Redraw the screen by selecting `REDRW` to see that the original extension line is still present.
8. Set the snapping mode back to grid snapping. Your screen should look like this:



Adding Radial Dimensions

Next you will add the radial dimensions which define the major part radius, the radius of the arc on which the holes lie, and the radius of the rear corners of the flange.

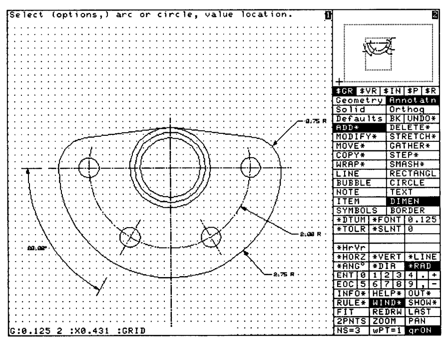
1. Select:

   `ADD* DIMEN *RAD`

2. Select the following points for the arcs to dimension and the value location for each arc.

Arc	Arc Location	Value Location
Hole arc	16.75,13.0	18.5,11.75
Major radius	11.8,11.2	17.5,10.25
Rear corner	17.8,14.9	18.75,16.0

Your screen should look like this:



Adding Diametral Dimensions

Next you will add the diametral dimensions for the upper left hole and the inside diameter of the hub.

1. Because these dimensions should be in the form maximum value above minimum value, you must first turn on the tolerancing option and set it to *HILO with the appropriate values. Select:

   `*TOLR *HILO`

2. For the high, or maximum, tolerance, enter:

   `.01`

3. For the low, or minimum, tolerance, enter:

   `0`

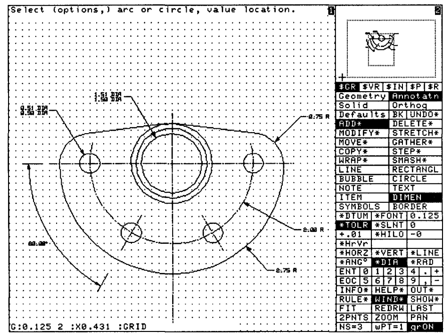
4. To add the two dimensions, select:

   `*DIA`

5. Select the following points for the circles to dimension and the value location for each circle.

Circle	Circle Location	Value Location
Center	13.4,15.1	12.5,16.75
Upper left	10.7,14.5	9.75,16.25

Your screen should look like this:



Adding Dimension Notes

You need to add notes which indicate that some of the dimensions just added are typical of other dimensions that are not included on the drawing.

1. Select:

   `NOTE`

2. Enter:

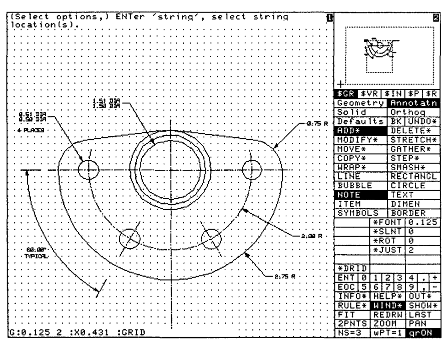
   `'4 PLACES'`

   and place this note just below the dimension for the upper left hole (8.4,15.75).

3. Enter:

   `'TYPICAL'`

   and place this note just to the right of the radius of the right rear corner (20.0,16.0). Also place this same note just below the dimension for the 60 degree angle (8.625,11.0). Your dimension notes should look like this:



Adding Front View Dimensions

1. Get the front view into the large port by selecting:

   `WIND* PAN`

2. Select the point (14.0,8.5) to bring to the center of the large port.
3. The first dimension you add will be the outside width of the hub. When adding linear dimension, the cursor must snap to the appropriate intersection points. Therefore, set the snapping mode to intersection snapping.
4. Because you want a plus/minus dimension tolerance, turn off the previous tolerance option by selecting:

   `DIMEN *TOLR`

5. To turn on the tolerancing option again and set it to *PLMI, select:

   `*TOLR *PLMI`

6. For the plus, or positive, tolerance, enter:

   `.10`

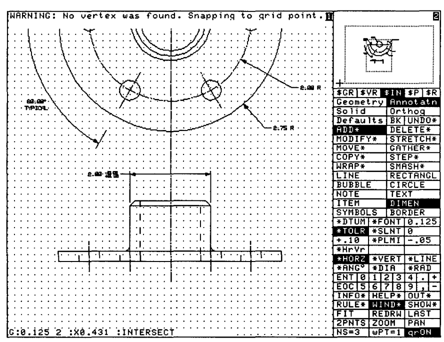
7. For the minus, or negative, tolerance, enter:

   `.05`

8. To add the horizontal dimension, select:

   `*HORZ`

   and then select the top right corner of the hub (15.5,7.25) and the top left corner of the hub (12.5,7.25), and place the dimension value to the left of the hub (11.5,8.5). Your screen should look like this:



Adding Vertical Dimensions

Next you will dimension the vertical dimensions of the hub using the datum option.

1. Turn off the tolerancing option by selecting:

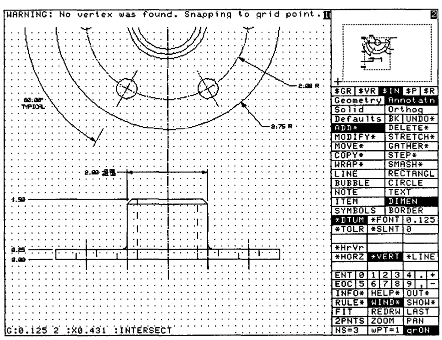
   ```
   *TOLR
   ```

2. Turn on the datum dimensioning option by selecting:

   ```
   *DTUM
   ```

   Add datum dimensions the same way you add normal dimensions, except that the point you select first (of the three: first endpoint, second endpoint, value location) in each dimension must be the 'zero' datum point. To dimension the zero point you must add a zero width dimension (where the first and second endpoints you select are the same).

3. Add the zero datum point by selecting the lower left corner of the base (10.0,5.25) as the first endpoint, and the same point (10.0,5.25) as the second endpoint. Place the value directly to the left of the lower corner of the base (8.5,5.25).
4. Dimension the thickness of the flange base by selecting the zero point (10.0,5.25) as the first endpoint and the upper left corner of the base (10.0,5.625) as the second endpoint. Place the value directly to the left of the upper corner of the base (8.5,5.625).
5. Finally, dimension the total height of the flange by selecting the zero point (10.0,5.25) as the first endpoint and the upper left corner of the hub (12.75,7.5) as the second endpoint. Place the value directly to the left of the upper corner of the hub (8.5,7.5). Your screen should look like this:



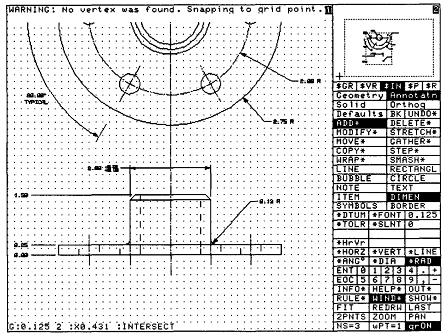
Dimensioning the Fillet Radius

Now you need to dimension the fillet radius, which is at the bottom of the circular hub.

1. Turn off the datum dimensioning option and select the radius dimensioning option by selecting:

   ```
   *DTUM *RAD
   ```

2. Select the fillet arc at the lower right corner of the hub (15.6,5.75), and then place the dimension value above and to the right of the point (17.0,7.25). Your screen should look like this:



Dimensioning the Chamfer

To dimension the chamfer you must add a leader line and the appropriate note, and then place a bubble around the note.

1. To turn on the leader option for lines, select:

   ```
   LINE *LEAD
   ```

2. Because the arrowhead is drawn at the first point of a leader line, select the upper right corner of the hub (15.5,7.25) as the first point. Complete the leader line above and to the right of the flange by placing a vertex at (16.75,9.0) and the last point at (17.25,9.0).
3. To add the chamfer note, select:

   ```
   NOTE
   ```

4. Enter:

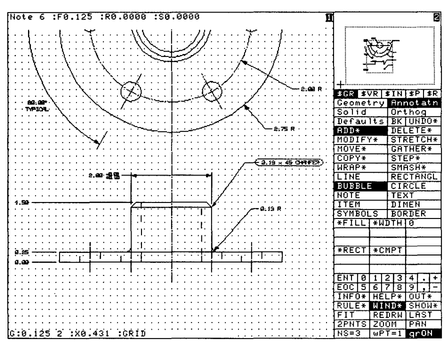
   ```
   ' 0.13 x 45 CHAMFER'
   ```

   and place the note to the right of the leader line (17.25,9.0). The leading space in the note enables you to snap the note to the end of the leader line and still have an adequate gap between the end of the line and the first character in the note.

5. Encircle the note in a bubble by selecting:

   ```
   BUBBLE *CMPT
   ```

6. Select the note you just added (17.25,9.0). The *CMPT method of BUBBLE will automatically find the size of the note and encircle it with two line and two arc components. Your screen should look like this:



Adding a Surface Finish Symbol

You are now going to add a surface finish symbol to the top of the flange to describe its allowable roughness.

1. Change the snapping mode to grid snapping.
2. To add the symbol, select:

   `SYMBOLS`

3. You can now enter the name of the symbol to add, but because you don't know the actual file names for the symbol to add, select the option:

   `*SCRN`

   Your screen menu is redrawn, containing a list of the symbols provided with the HP EGS system.

4. For the standard surface finish symbol, select:

   `surface1 ENT`

   and place it on the upper surface of the hub (13.25,7.5).

5. To return to the Mechanical Drafting command menu, select:

   `*RETN`

6. The note 'N 7' now needs to be added to the symbol to specify this as a "Good Machined Finish". Select:

   `NOTE`

7. Enter:

   `'N 7'`

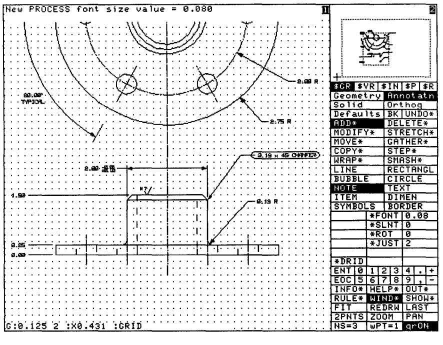
8. If you move this note close to the symbol you will see that the note is relatively large. To reduce the note size, select:

   `*FONT`

9. Enter:

   `.08`

   and then place the the note just above the 'check' portion of the symbol (13.0,7.75). Your screen should look like this:



Updating the Title Block

The final task in this chapter is to put the name of the drawing, the drawing number, and the drawing scale into the title block.

1. To make the title block fill the large port, select:

   `2PNTS`

   and then select two points encompassing the title block area from the small port. (27.0,0.0 and 34.0,4.0 work well.)

2. Enter the drawing name:

   `'FLANGE'`

   Notice that the font size is now too small.

3. Select:

   `*FONT`

4. Enter the font size:

   `.25`

5. Place the drawing name at (29.75,1.875).

6. The font size for the scale should be 0.125, so select:

   `*FONT`

   and enter:

   `.125`

7. Enter the drawing scale note:

   `'3/2'`

   and place it in the scale box (28.375,0.625).

8. To add the drawing number, select the note option:

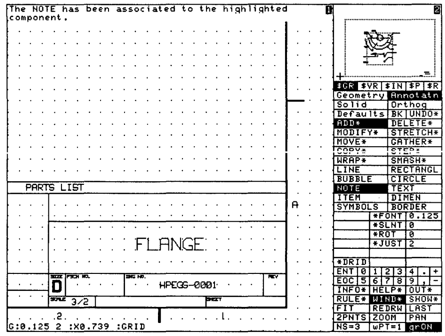
   `'*DRID'`

   and enter the drawing number:

   `'HPEGS-0001'`

   and place it in the drawing number box (30.25,1.0).

Notice that the drawing number flashes a few times. This is because the *DRID option automatically adds some associated text so that the drawing number will be automatically passed on to the material list post-processor if this drawing was to be material listed with the LIST_MAT command.



9. Before you save and plot the flange drawing, fit the entire drawing to the large port by selecting:

   `FIT`

10. Save the drawing under its present name by selecting:

    `OUT* SAVE* OLD`

11. To plot this drawing at its actual size, make sure you have a plotter on-line with a D-size sheet of paper loaded. Then select:

    `PLOT* SCALE 1 ENT CENTR EOC`

BREAK TIME

Relax, stretch, and take a break now that you are done with the single-part drawing example. Through this example you:

- Initialized a drawing.
- Added a flange.
- Added angular, radial, and diametral dimensions.
- Added dimension notes.
- Added front view and vertical dimensions.
- Dimensioned the fillet radius and the chamfer.
- Added a surface finish symbol.
- Updated the title block.

Making a Flange Assembly Drawing

Chapter 7

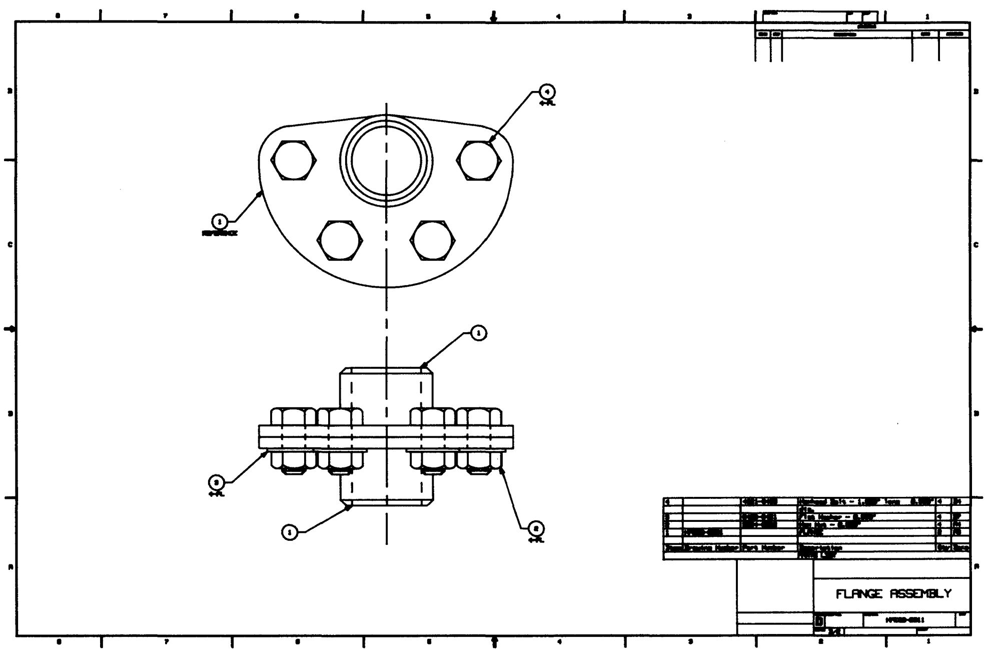
In this chapter you will bolt together the two flanges you made in the last lesson to create an assembly drawing. This annotated drawing will also include a material list automatically generated from the data in the assembly drawing.

The first section of this chapter assumes you are already inside the Mechanical Drafting personality. If you are not, refer to the *Loading Mechanical Drafting* section of Chapter 6 to access the Mechanical Drafting personality. Remember to prefix to your storage volume, DRAW:.

To preview this lesson, enter:

```
INPUT EWLRN:chap7;
```

Your drawing should look like this:



Assembling the Parts for the Front View

In this section you will start to create the assembly drawing by adding a border and the original flange geometry to a new drawing. Then you will modify the front view of the geometry so that it contains a second flange and the necessary bolts, nuts, and washers to fasten the two flanges together.

Initializing the Drawing

1. Start the assembly drawing by editing a new drawing called `flassy`. Enter:

   ```
   EDIT flassy;
   ```

2. Add a D-size border to this new drawing by setting the drawing mode to annotation. Select:

   ```
   Annotatn
   ```

3. Add the border by selecting:

   ```
   ADD* BORDER D_Hor
   ```

4. Notice that the drawing origin (point 0,0) is in the center of the border. To make this lesson easier, move the border into the upper right quadrant so that all x and y values will be positive. Enter the following information from your keyboard:

   ```
   MOVE* -14.0,-8.5  0.0,0.0  17.5,10.5
   ```

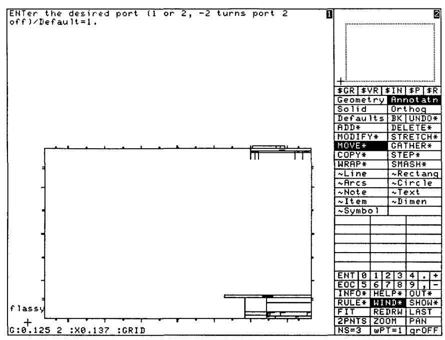
5. Set the nesting level to 3 to see inside the border, and fit the drawing to the large port. Select:

   ```
   WIND* NS=1 3 ENT FIT
   ```

6. Fit the drawing to the small port by selecting:

   ```
   wPT=1 2 ENT FIT wPT=2 1 ENT
   ```

   Your screen should look like this:



Adding a Flange Part

You are now ready to add the original flange drawing.

1. Change the drawing mode to geometry by selecting:

   ```
   Geometry
   ```

2. Select:

```
ADD* PART
```

and enter the part name of the flange you drew initially:

```
flange
```

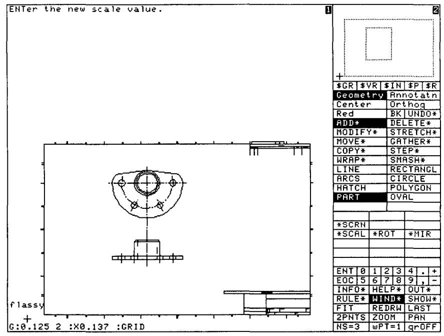
3. Again, scale this part by a factor of 1.5. Select:

```
*SCAL
```

and enter:

```
1.5
```

4. Place the flange at (14.0,16.0). Your screen should look like this:



Copying a View

Now you will make a copy of the front view of this part and add it, mirrored, to the bottom of the front view to indicate the position of the second flange.

1. Zoom in on the front view and turn on the display grid by selecting:

```
grOFF 2PNTS
```

and then select (9.0,2.0) as the first point and (20.0,10.0) as the second point for the new port setting.

2. Before you can copy the front view of the flange, you must smash the part. Select:

```
SMASH*
```

and select the flange part (9.5,8.5), then terminate the smash command with EOC.

3. Wrap just the front view so that you can make another copy. Select:

```
WRAP*
```

4. Select a wrap origin (14.0,7.0), then enclose the entire front view in a box by selecting (9.0,5.0) and (20.0,10.0). The four short center lines should not be included in this wrap, so delete them by selecting the delete component method:

```
DlCom
```

and then select each of the center lines. Approximate locations are:

(17.0,6.5)
(15.5,6.5)
(12.5,6.5)
(11.0,6.5)

5. Terminate the wrap operation with EOC.
6. It is important to keep the copy of the front view lined up with the original, so set the snapping mode to intersection snapping.
7. Copy the front view. Select:

```
COPY*
```

and select the front view (9.75,7.0). Select a reference point where the base line crosses the flange center line (14.0,7.0), and place the copy directly on top of the original (14.0,7.0).

8. Mirror the new copy of the front view so that it lies below the original. Select:

```
MODIFY*
```

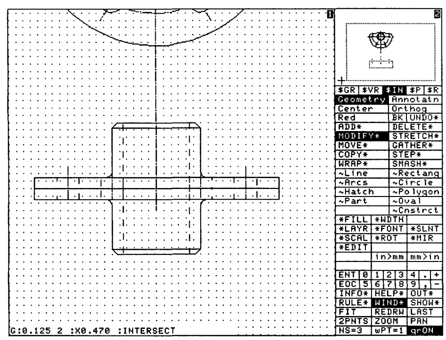
and select the copy (9.75,7.0) and a reference point on the base line of the original where it intersects the center line (14.0,7.0). To mirror this instance about the x axis, select:

```
*MIR XAXIS
```

and then select EOC to terminate the modification. To re-display the original, select:

```
REDRW
```

Your screen should look like this:



Adding a Bolt

The next task is to add the four sets of bolts, nuts, and washers to the front view. You will do this by adding one set, wrapping this set together, and then copying the resulting instance to the remaining three locations.

1. Zoom in on the right side of the joined flanges. Select:

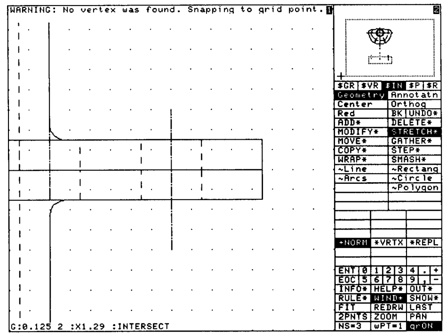
```
2PNTS
```

and select (15.0,5.0) and (19.0,9.0).

2. To insure that the fasteners line up properly, stretch the lower end of the right center line down 3/4 of an inch. Select:

   `STRETCH*`

   then select the right center line (17.0,7.2) and its lower end (17.0,6.7), and then stretch the end to (17.0,6.0). The stretched center line should look like this:



3. To add the fasteners, select:

   `ADD* PART`

4. You can now enter the name of the part to add, but because you may not know the names, select this option:

   `*SCRN`

   Your screen menu is redrawn with the names of the parts provided with HP EGS.

5. To add the side view of a bolt, select:

   `boltsd`

   (not `bolthd`, which is the top view). This library part is a macro instance, which is different from a normal instance in that you must input values which determine its actual size when it is added. In this way you can make one library part that represents an entire family of parts. In this case you can define the bolt to have any diameter, overall length, and thread length.

6. When prompted for the diameter of the bolt, enter:

   `.5`

7. When prompted for the length of the bolt, enter:

   `1`

8. When prompted for the thread length of the bolt, enter:

   `.75`

9. If you move the bolt into the large port you will notice that it seems small and incorrectly rotated. It is small because its scale does not yet match the scale of the drawing. To set the scale to 1.5, select:

   ```
   *SCAL
   ```

   and enter:

   ```
   1.5
   ```

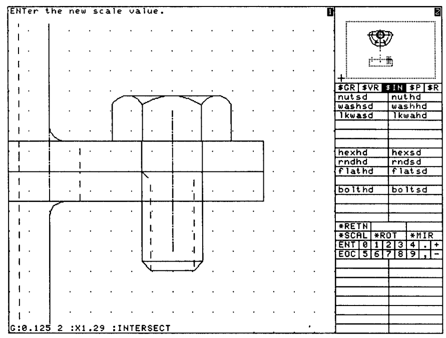
10. Rotate the bolt into a vertical position by selecting:

    ```
    *ROT
    ```

    and then entering:

    ```
    -90
    ```

11. The point you are tracking in this part is the base of the head. Place this point at the intersection of the right center line and the top surface of the upper flange (17.0,7.4). Your screen should look like this:



Adding a Washer and Nut

You are now going to add the washer and nut for the bolt. Both of these parts are macro instances.

1. To add the side view of the washer, select:

   ```
   washsd
   ```

2. When prompted for the washer's diameter, enter:

   ```
   .5
   ```

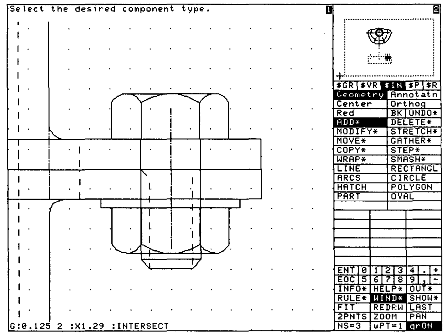
3. Because the rotation and scale of this part reflect the values entered earlier, you only need to add it to the drawing. Place the washer at the intersection of the right center line and the bottom surface of the flange assembly (17.0,6.625).
4. To add the side view of the hex nut, select:

   ```
   nutsd
   ```

5. The rotation and scale are already set, so place the nut at the intersection of the center line and the bottom surface of the washer you just added. Then return to the Mechanical Drafting personality by selecting:

   `*RETN`

   Your screen should look like this:



Deleting Hidden Portions

Look closely at the bolt you just added and notice that a major portion of it should be hidden from view because it is inside the two flanges, the washer, and the nut. Now you must smash it so that you can delete those hidden portions.

1. Select:

   `SMASH*`

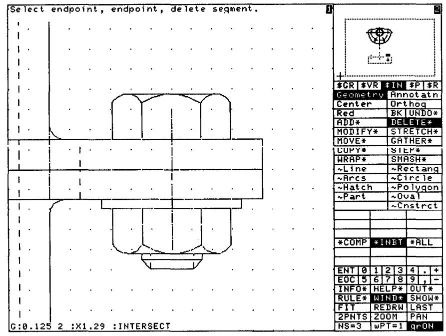
2. Select the top of the bolt (17.7,7.8), and then terminate the smash command with EOC.
3. Now that the bolt is smashed you can use the in-between option of delete to remove the hidden portions. Select:

   `DELETE* *INBT`

   The following table shows the approximate locations of the points on the lines to delete. Remember that the proper sequence for deleting in-between is to select the two endpoints of the segment and then a point on the segment.

Line	1st Endpoint	2nd Endpoint	Point on Segment
Right dashed	17.36,6.99	17.29,5.95	17.26,6.21
Left dashed	16.64,6.98	16.73,5.95	16.75,6.82
Right solid	17.39,5.95	17.39,7.34	17.41,6.83
Left solid	16.59,5.94	16.62,7.37	16.61,6.83

Your screen should look like this:



Copying the Set

1. To make the remaining three copies of this bolt, nut, and washer set, first create an instance by wrapping them together. Select:

   WRAP*

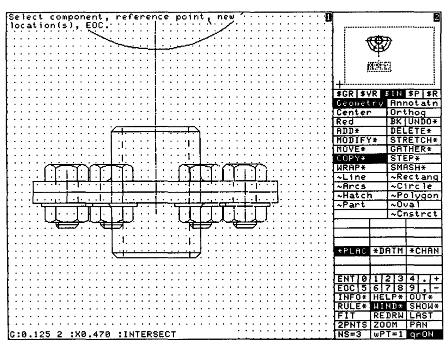
   and select a point in the bolt as the origin (17.0,7.0), then collect the components defining the three fasteners by enclosing them in a box (select 16.0,5.5 and 18.0,8.5).
2. The center line should not be included in this new instance, so delete it from the collection by selecting the option:

   DlCom

   and then selecting the center line at (17.0,7.5). Terminate the wrap option with EOC.
3. Return the large port to its previous state by selecting:

   LAST
4. Then select:

   COPY*
5. Select the instance you just created (17.0,8.0). To place the copies accurately, select the intersection of the right center line and the top of the flange plate as the reference point (17.0,7.4). Place the three copies at the intersections of the remaining three, short, center lines and the top of the flange (11.0,7.4 and 12.5,7.4 and 15.5,7.4). Terminate the copy operation with EOC. Your screen should look like this:



Deleting Center Lines and Fillets

1. The inner two fastener sets you just added to the drawing now overlay portions of the center hub. To delete those portions of the center hub on the right, zoom in on that area to see it better. Select:

   `2PNTS`

   and then select (14.0,5.0) and (18.5,8.5).

2. Both front views of the flanges are instances now, so to delete components contained in them you must first smash them. Select:

   `SMASH*`

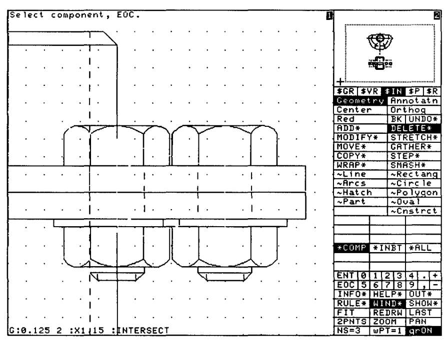
3. Select the upper flange (18.0,8.5) and EOC. Then select the lower flange (18.0,5.5) and EOC.

4. Delete the two center lines for the bolts, because they are unnecessary, and the fillet arcs at the base of each flange. Select:

   `DELETE*`

   and then select the components described above. The table below lists the approximate locations of the components:

Component	Location
Far right center line	17.0,7.2
Near right center line	15.5,7.2
Upper fillet	15.55,7.52
Lower fillet	15.55,6.59



Deleting Hidden Lines and Fillets

1. Now you need to delete those portions of the solid and dashed lines defining the hub which are behind the fastener assembly. Again, use the in-between option of delete for this. Select:

   `*INBT`

The following table indicates the approximate locations of the points on the lines to delete:

Line	1st Endpoint	2nd Endpoint	Point on Segment
Upper solid	15.50,7.95	15.50,7.56	15.50,7.74
Lower solid	15.52,6.44	15.52,5.76	15.48,6.22
Upper dashed	15.15,7.96	15.13,7.01	15.06,7.65
Lower dashed	15.13,7.01	15.10,5.95	15.09,6.39

2. Before you can delete the portion of the washer on the right that is behind the washer on the left, you must smash the wrapped fastener instance and then the washer itself. Select:

   SMASH*

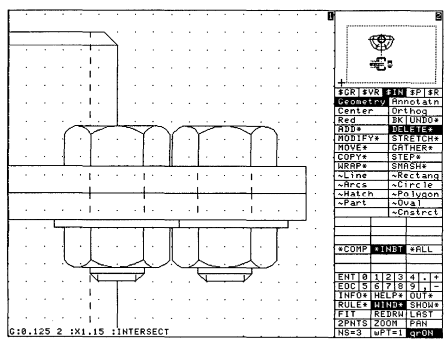
3. Select the fastener instance (17.26,6.56) and EOC, then the washer (17.86,6.51) and EOC.
4. Delete the hidden portion of the washer. Select:

   DELETE* *INBT

   Approximate locations of the points on the washer are:

Line	1st Endpoint	2nd Endpoint	Point on Segment
Washer	16.37,6.52	16.37,6.63	16.15,6.58

5. Redraw the large port to see how the drawing looks after all of these deletions. Your screen should look like this:



6. The deletions you performed on the right side of the flange assembly must now be done on the left. Return the large port to its previous state, then zoom in on the left side of the flange front view. Select:

   LAST 2PNTS

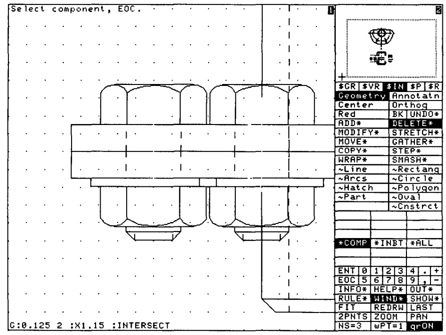
   and then select (9.0,4.5) and (13.5,8.5).

7. Delete the two center lines for the bolts because they are unnecessary, and the fillet arcs at the base of each flange. Select:

```
*COMP
```

and then select the components described above. The table below lists the approximate locations of the components:

Component	Location
Far left center line	11.0,7.2
Near left center line	12.5,7.1
Upper fillet	12.47,7.50
Lower fillet	12.43,6.59



8. Again, you need to delete the portions of the solid and dashed lines defining the hub that are behind the fastener assembly. Select:

```
*INBT
```

The following table indicates the approximate locations of the points on the lines to delete:

Line	1st Endpoint	2nd Endpoint	Point on Segment
Upper solid	12.50,7.95	12.53,7.62	12.50,7.75
Lower solid	12.49,6.43	12.50,5.76	12.49,6.14
Upper dashed	12.87,7.95	12.87,7.01	12.94,7.56
Lower dashed	12.89,5.94	12.87,7.01	12.94,6.35

9. Before you can delete the portion of the washer on the left that is behind the washer on the left, you must smash the wrapped fastener instance and then the washer itself. Select:

```
SMASH*
```

10. Select the fastener instance (10.95,6.35) and EOC, then select the washer (10.18,6.52) and EOC.

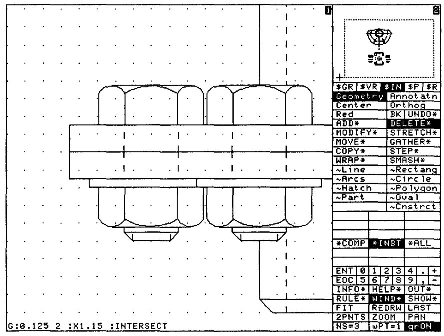
11. Delete the hidden portion of the washer. Select:

    DELETE* *INBT

    Approximate locations of the points on the washer are:

Line	1st Endpoint	2nd Endpoint	Point on Segment
Washer	11.63,6.51	11.63,6.62	11.85,6.58

12. Redraw the large port to see how the drawing looks after all of these deletions. Your screen should look like this:



Stretching the Center Line

1. The final task in this section is to stretch the vertical center line below the lower flange. First return to the previous state of the large port by selecting:

    LAST

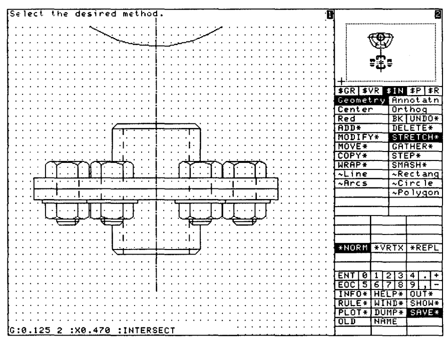
2. To stretch the vertical center line, select:

    STRETCH*

    then select the center line (14.0,6.25) and the lower end of the center line (14.0,6.25), and then place the end below the bottom of the lower flange (14.0,3.5).

3. Save the flange assembly drawing as a precaution. Because the drawing is already named, select:

    OUT* SAVE* OLD



BREAK TIME

Take a short break. When you return you will complete the geometric modifications on the top view of the flange assembly. In this section you:

- Added a flange part.
- Copied a view.
- Added a bolt, washer, and nut.
- Deleted hidden portions.
- Copied the set.
- Deleted center lines.
- Deleted hidden lines and fillets.
- Stretched the center line.

Modifying the Top View of the Assembly

In this section you will modify the top view of the flange to reflect the addition of the bolts in the assembly.

Adding the Right Bolt Head

1. Before beginning the modifications to the top view, you must 'move' it into the large port. Select:

   ```
   WIND* PAN
   ```

   and then select (14.0,12.5). This places the top view in the port and also keeps some of the front view in it for reference.
2. The strategy for modifying the top view is to replace the circles representing the holes in the flanges with the head, or top, view of the bolt. To do this you will delete the four circles and replace them with bolt heads. To delete the holes, select:

   ```
   DELETE*
   ```

   then delete the bolt holes. Approximate locations are:

Component	Location
Far right circle	16.74,16.32
Near right circle	15.83,13.34
Near left circle	12.65,13.00
Far left circle	11.36,16.37

3. Add the fasteners by accessing the screen parts menu. Select:

   ```
   ADD* PART *SCRN
   ```
4. To get the top, or head, view of the bolt, select:

   ```
   bolthd
   ```
5. When prompted for the diameter of the bolt, enter:

   ```
   .5
   ```
6. As in the previous section, you must scale this part by a factor of 1.5. Select:

   ```
   *SCAL
   ```

   and enter:

   ```
   1.5
   ```

7. Notice that the bolt head is oriented vertically. To match the front view it must be oriented horizontally. To rotate the bolt head, select:

   ```
   *ROT
   ```

   and enter:

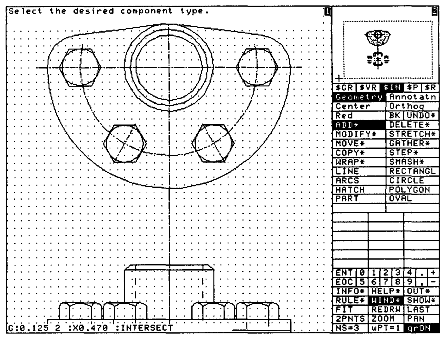
   ```
   90
   ```

8. Place the bolt head where the holes used to be. Because you are tracking the center of the bolt head, select the intersection points of the center lines: (17.0,16.0), (15.5,13.4), (12.5,13.4), and (11.0,16.0).

9. Return to the Mechanical Drafting command menu by selecting:

   ```
   *RETN
   ```

   Your screen should look like this:



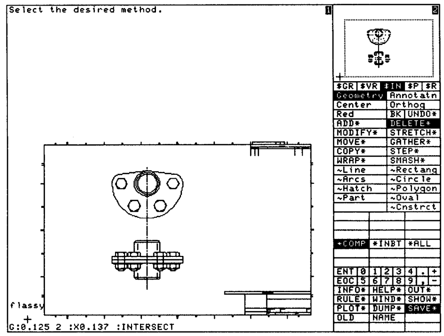
Deleting the Center Lines

1. The final task in this section is to delete all remaining center lines, except for the vertical one in the center of the flange assembly. Select:

   ```
   DELETE*
   ```

   and then delete the center lines. Approximate locations of these center lines are:

Component	Location
Semircircular arc	16.71,14.61
Horizontal line	18.56,15.99
Short right line	15.47,13.34
Short left line	12.39,13.21



2. Fit the entire drawing to the large port to see all of the geometric changes you have made:

   FIT

3. Save the flange assembly drawing as a precautionary measure. Select:

   OUT* SAVE* OLD

BREAK TIME

Relax now, take a break, and review what you have done to create the geometry for the flange assembly drawing:

- Added the bolt heads.
- Deleted center lines.

Annotating the Assembly Drawing

In this section you will annotate the assembly drawing you just created in preparation for creating a material list.

Collecting Part Information

As a preface to this section, let's discuss how part information is collected from your drawing. One method is to count the library parts you used in the actual drawing (this is how Electrical Engineering material listing operates). The problems with using this method for mechanical drawings are:

- The drawing may contain multiple views, such as top and side views, of the same part.
- The actual parts may have been smashed so that hidden geometry could be removed. Once the part has been smashed, HP EGS no longer recognizes it.
- If the part is a macro instance, it could represent any of a number of different parts.

For these reasons a new method of determining the number of parts in a drawing has been created. In this new method the item number references are collected from the drawing. The Mechanical Engineering Material List post-processor then compiles the material list from this information.

To implement this method, a new component named ITEM has been added to the Annotation mode. An ITEM is actually a circle and two to four associated notes. This set of components is what tracks item numbers, quantities, identification numbers, and zone information for processing to create parts lists. The two methods for adding items are:

- *DATA, which adds the item with the current item number and the current quantity value, and then prompts you for the item's identification number and zone.
- *BLNK, which adds the item with the current item number and current quantity.

Each item (such as a 0.25-inch-long 10-32 screw) on your drawing must have one item reference which was added using the *DATA method. The remaining item references for this component can be added with the *BLNK method, which is easier because it does not prompt you for the identification number or zone.

There are also the four following options for adding items:

- *FONT, which sets the font height for the note strings in the item.
- *SLNT, which sets the slant for the note strings in the item.
- *NUMB, which sets the current number for the item (character strings such as A, 7A, or DF can be used as well as numbers).
- *QTY, which sets the quantity for the item. This must be a contiguous string (no spaces), as long as the first characters in the string represent the quantity value. Examples are 4, 4-PL, and 4-PLACES. If the characters REF are contained in the quantity string the item will not be counted.

HP EGS enables you to add associated notes either displayed or undisplayed. The macro command that adds item components has been designed to add the item number and quantity as displayed notes, and the identification number and zone as undisplayed notes. This means that, with two exceptions, you cannot see the identification number or zone notes on your drawing.

The first exception is that when an item is added, the identification number and zone strings will be displayed so that you can verify that they are correct. These notes will remain displayed until you use one of the WIND* methods to redraw the port.

The second exception is that you can explicitly turn on or off the display of all identification number and zone notes with the ITEMS method of the SHOW* command.

When you enter data for items, never include quotes unless you want them in your data. This is the only case in HP EGS where you add note strings without quotes. Also, the strings you enter must be contiguous; that is, they cannot contain spaces or commas.

Adding Items for the Flange Parts

1. To illustrate the use of items, you will annotate the flange assembly drawing using this new component. First, prepare the large port for the task by setting the drawing mode to annotation:

   ```
   Annotatn
   ```

2. Zoom in on the front view of the assembly:

   ```
   WIND* 2PNTS
   ```

   and then select (8.0,3.0) and (20.0,11.5) as the two corners for the new port.
3. For ease of adding the items, set the snapping mode to grid snapping.

4. Recall that one of the fields required in the *DATA method of adding items is the zone. This is the location with respect to the drawing border where the item is placed. To make it easier to determine the zone, redraw your display with the zones marked by entering:

   ```
   ZONE_ON
   ```

   Later you will turn off the zone markings with ZONE_OFF.

5. Add the item information for the lower of the two flanges. Select:

   ```
   ADD* ITEM *DATA
   ```

   For the identification number (as you specified in the previous lesson), enter:

   ```
   HPEGS-0001
   ```

   For the zone, which is where the item will be placed, enter:

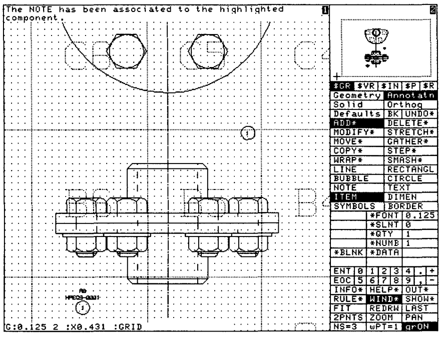
   ```
   A6
   ```

   and place the item at (10.875,3.875). Notice that the item number is added as 1, since this is the current setting for *NUMB. Also notice that a quantity string was not added to the drawing, since the current setting for *QTY is 1.

6. To add the item reference for the upper flange, select:

   ```
   *BLNK
   ```

   since the data for this part was already added. Place the item at (17.0,10.375). Your screen should look like this:



Adding Nut and Washer Items

1. To add an item that calls out all four hex nuts, select:

   ```
   *QTY
   ```

   and set the current quantity to 4:

   ```
   4-PL
   ```

2. Change the item number to 2 by selecting:

   `*NUMB`

   and then entering:

   `2`

3. Now add the item by selecting:

   `*DATA`

4. For the identification number, enter:

   `5234-0392`

5. For the zone, enter:

   `A4`

6. Place this item at (18.875,4.0).
7. Add the item for the four washers shown in this view by first changing the item number to 3. Select:

   `*NUMB`

   and then enter:

   `3`

8. Select:

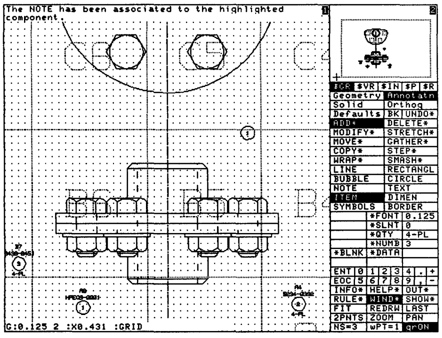
   `*DATA`

9. Enter the identification number and zone:

   `8438-8451`

   `B7`

10. Place this item at (8.5,5.5). Note you did not have to change the quantity, because 4-PL is correct. Your screen should look like this:



Adding Leader Lines

Now that all of the items have been added for the front view of the assembly, you need to add the leader lines from the items to the appropriate points in the drawing. But first, look at what happens to the identification number and zone notes when the large port is redrawn.

1. Select:

   ```
   REDRW
   ```

   Notice that they disappear, as explained earlier, because they were added in an undisplayed manner.

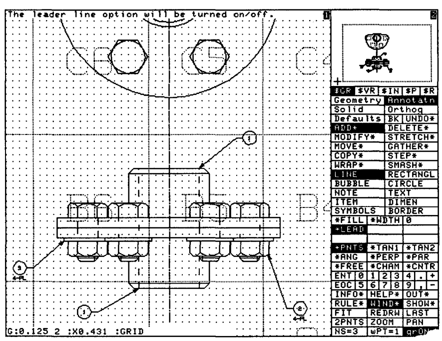
2. To turn on the leader line option, select:

   ```
   LINE *LEAD
   ```

   Remember that the arrowhead on a leader line is always placed at the point that is added first.

3. To add the leader lines for the items on the screen, select the following points. Note that snapping mode changes should be made as indicated to ensure that the arrowhead end of the leader snaps to the appropriate point.

	15.125,9.25		16.5,10.375	16.75,10.375	EOC
$P	17.7,6.0	$GR	18.375,4.0	18.625,4.0	EOC
	12.875,4.75		11.375,3.875	11.125,3.875	EOC
$IN	10.125,6.5	$GR	9.0,5.5	8.75,5.5	EOC



Adding the Remaining Items and Leader Lines

1. Now the items for the top view need to be added. Get the top view into the large port by selecting:

   ```
   PAN
   ```

   and selecting the point (14.0,14.5).
2. Add the item for the four bolts by first changing the item number to 4. Select:

   ```
   ITEM *NUMB
   ```

   and then enter:

   ```
   4
   ```
3. Select:

   ```
   *DATA
   ```

   and enter the identification number:

   ```
   4321-8463
   ```
4. Enter the zone:

   ```
   D4
   ```
5. Place this item at (19.25,18.25). You did not have to change the quantity, because 4-PL is still correct.
6. Add another item for the flange in this view. The item number for this part is 1, so select:

   ```
   *NUMB
   ```

   and enter:

   ```
   1
   ```
7. Change the quantity to REFERENCE by selecting:

   ```
   *QTY
   ```

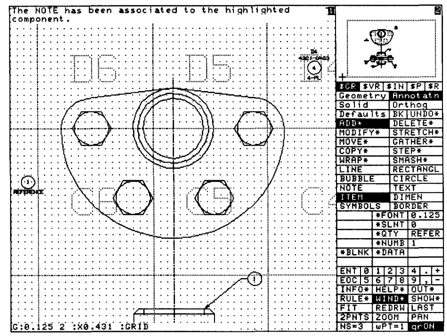
   and then entering:

   ```
   REFERENCE
   ```
8. Select:

   ```
   *BLNK
   ```

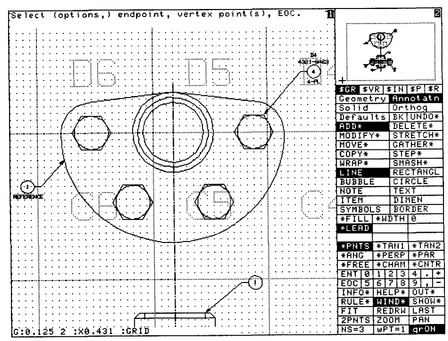
   and place this item at (8.625,14.0).

As discussed at the beginning of this chapter, the Mechanical Engineering Material List post-processor will recognize that this item does not count in the final material list since the quantity field has the characters REF in it.



9. Now that all of the items have been added for the top view of the assembly, it is time to add the leader lines from the items to the appropriate points in the drawing. Select:

```
LINE
```



10. Note that the leader option is still turned on. To add the leader lines for the items on the screen, select the following points.

$VR	17.4,16.6	$GR	18.75,18.25	19.0,18.25	EOC
$P	10.0,15.0	$GR	9.125,14.0	8.875,14.0	EOC

Updating the Title Block

1. Now that all items have been added, turn off the display of the zone grid by entering:

```
ZONE_OFF
```

2. The final task in this section is to put the name of the drawing, the drawing number, and the drawing scale into the title block. To fill the large port with the title block, select:

```
2PNTS
```

and then select two points encompassing the title block area from the small port; (25.0,0.0) and (33.5,4.5) work well.

3. To add the drawing name, select:

   ```
   NOTE
   ```

   and then select:

   ```
   *FONT
   ```

4. To set the font size to 0.25 inches as you did in the flange drawing, enter:

   ```
   .25
   ```

5. Enter:

   ```
   'FLANGE ASSEMBLY'
   ```

   and then place the drawing name at (28.625,1.875).

6. The font size for the scale should be 0.125, so select:

   ```
   *FONT
   ```

   and enter:

   ```
   .125
   ```

7. Enter the drawing scale note:

   ```
   '3/2'
   ```

   and place this in the scale box (28.375,0.625).

8. To add the drawing number, select the note option:

   ```
   '*DRID'
   ```

9. Enter the drawing number:

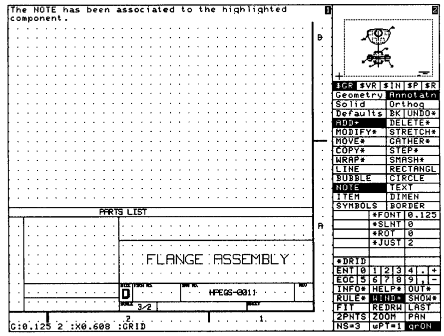
   ```
   'HPEGS-0011'
   ```

   and place it in the drawing number box (30.25,1.0).

   The drawing number flashes a few times once you have placed it. This is because the *DRID option automatically adds some associated text so that the drawing number will automatically pass on to the material list post-processor.

10. Fit the entire drawing to the large port to see all of the changes you have made.
11. Save the flange assembly drawing as a precautionary measure:

```
OUT* SAVE* OLD
```



BREAK TIME

Relax, take a break, and go over what you did to prepare this drawing for material listing:

- Learned how part information is collected.
- Added items for the flange parts.
- Added nut and bolt items.
- Added leader lines.
- Updated the title block.

Material Listing the Assembly Drawing

In this section you will prepare a material list from the assembly drawing you just completed. Then you will re-edit the flange assembly drawing and add the data generated.

Creating Material List Data

1. To produce a material list, enter:

```
LIST_MAT
```

The display reads:

```
ENTer volume and file name for the source drawing file.
```

2. Enter the name of the current drawing:

```
flassy
```

The display reads:

```
ENTer destination volume, PRINTER: or CONSOLE:, then EOC.
```

3. Decide where to put the material list. The system automatically assigns a file name; if you enter a volume name, the material list is put there with a file name of flassy_m (note the suffix of _m). If you enter PRINTER:, the material list is listed to a local printer. If you enter CONSOLE:, it is listed to your display. If you don't enter anything, the list is put in the prefixed volume with a file name of flassy_m. For this exercise, the prefixed volume is a good place, so select:

```
EOC
```

The material list file appears as follows (you can look at your file later with the text Editor if you want):

```
N    |drawid|HPEGS-0011||||
C    | 1 | REFERENCE ||||
C    | 4 | 4-PL |4321-8463||D4|
C    | 3 | 4-PL |8438-8451|| B7|
C    | 2 | 4-PL |5234-0392|| A4|
C    | 1 | ||||
C    | 1 | |HPEGS-0001|| A6|
```

This file contains a list of each item you added to the drawing. It also contains a special line (the first line, in this case) that contains the drawing identification number. Each line represents a separate item where the first entry is the item number; the next entry is the quantity, if there is one (no entry indicates a quantity of 1); following this is the part identification number. The next field is for size information. For more details about the size field, see *Understanding HP EGS*. The last entry is the zone where the part is located.

Changing the Parts File

1. To process this preliminary material list file to create a true parts list, you must first exit the Mechanical Drafting personality and return to the Mechanical Engineering menu in the manager. Enter:

```
QUIT;
```

The menu looks like this:

```
Use arrow keys to move cursor to desired function.
Select function with <RETURN> or <ENTER>.
    RETURN To Main Menu
  ->Mechanical Drafting Editor
    ME Parts File Editor
    ME Material Lister
```

2. Next you must add the original flange part (number HPEGS-0001) to the parts file so that it will be adequately documented in the final parts list. Select:

```
ME Parts File Editor
```

and wait for the following display to appear:

```
Should the hpegs volumes be searched for parts files? (y/n)
```

3. Answer this prompt by pressing (Y). The part file editor searches all HP EGS volumes for parts files. Wait for the following prompt to appear:

```
Enter the name of another volume to be searched for parts files. (<enter> if no more)
```

4. You don't need to search any other volumes, so press (Return).

Next you will select the part file to change, and then add the new part to it. The following menu should now be displayed:

```
Use arrow keys to move marker.
Select with <RETURN> or <ENTER>.
->QUIT
  CREATE A New Parts File
  EWEE:eepart_pf
  EWME:mepart_pf
```

5. So that you can add the flange to this file, select:

   ```
   EWME:mepart_pf
   ```

   The following menu is displayed:

   ```
   Use arrow keys to move marker.
   Select with <RETURN> or <ENTER>.
   ->QUIT
     ADD A New Part
     CHANGE An Existing Part
     DELETE An Existing Part
     LIST The Existing Parts
     SELECT A New Parts File To Edit
   ```

6. Select:

   ```
   ADD A New Part
   ```

   HP EGS prompts you for all of the data fields that need to be added for the new part. These data fields are defined as follows:

   - Part Number: The part number for the part.
   - Drawing Number: The drawing number for the part.
   - Description: Any description you want included in parts lists that contain this part.
   - Specifications: Any data you want included in parts lists that contain this part.

7. Answer each of the prompts as follows:

   ```
   Enter the Part Number
    Max. length is 14:
   Enter the Drawing Number
    Max. length is 14: HPEGS-0001
   Enter the Description
    Max. length is 40: FLANGE
   Enter the Specifications
    Max. length is 40:
   ```

   Note that in this case the part number and specification fields were left empty.

8. You can now quit the parts file editor. After you enter the specifications, HP EGS returns you to the previous menu as follows:

```
Use arrow keys to move marker.
Select with <RETURN> or <ENTER>.
->QUIT
  ADD A New Part
  CHANGE An Existing Part
  DELETE An Existing Part
  LIST The Existing Parts
  SELECT A New Parts File To Edit
```

9. Quit the parts file editor:

```
QUIT
```

The parts file you changed, EWME:meparts_pf, is automatically saved for you, and HP EGS returns you to the Mechanical Engineering menu.

Creating the Parts List

Now you are ready to create the final parts list. The Mechanical Engineering menu should look like this:

```
Use arrow keys to move cursor to desired function.
Select function with <RETURN> or <ENTER>.
    RETURN To Main Menu
    Mechanical Drafting Editor
  ->ME Parts File Editor
    ME Material Lister
```

1. Select:

```
ME Material Lister
```

and wait for the following prompt to appear:

```
Should the hpegs volumes be searched for parts files? (y/n)
```

2. Answer this prompt yes by pressing (Y). The part file editor then searches all of the HP EGS volumes for parts files. Wait for the following prompt to appear:

```
Enter the name of another volume to be searched for parts files. (<enter> if no more)
```

You don't need to search any other volumes, so press (**Return**).

Next you will produce two parts lists for your flange assembly drawing. The first will be a printable version and the second will be one that you can easily add to your drawing. The following menu should now be displayed:

```
Use arrow keys to move the marker
Select with <RETURN> or <ENTER>
  -> USE ALL Parts Files
     EWEE:eepart_pf
     EWME:eepart_pf
     RE-START Selecting Parts Files
     DONE Selecting Parts Files
```

3. So that your material list will be based on this parts file, select:

   ```
   EWME:mepart_pf
   ```

4. A star (*) should now appear next to this file name to indicate it will be used. Select:

   ```
   DONE Selecting Parts Files
   ```

   The following prompt appears:

   ```
   Which listing format do you want to use?
   ```

5. For the pre-defined printable format, enter:

   ```
   p
   ```

   The following prompt appears:

   ```
   Enter the name of the output file:
   ```

6. Decide where you want to list the file, and enter the volume/file name string. (You can send it to CONSOLE: and PRINTER: as well as to a particular file.) This prompt appears:

   ```
   Enter the number of lines per page: (0 for no pagination)
   ```

7. Respond by entering 55 or a value appropriate for your selected file. The next prompt is for the file to be processed:

   ```
   Enter the name of the list mat file:
   ```

   This is the name of the file you created while you were using the Mechanical Drafting personality. Remember to include the volume you were prefixed to (substitute your volume name for EWYOU:).

   ```
   EWYOU:flassy
   ```

   (the _m is automatically appended). The next prompt is:

   ```
   Should the output be Forward or Backward (f/b)
   ```

8. This prompt enables you to specify whether the first item is to be at the top or the bottom of the parts list. For a printed version select the Forward option by pressing (F). The final prompt is:

```
Enter the optional one line header:
```

9. Enter:

```
FLANGE ASSEMBLY
```

The post-processor will create a material list and send it to the file you specified. It should look like this when it is complete:

```
                                                   PAGE 1
HPEGS-0011
FLANGE ASSEMBLY
Item Drawing Number Part Number    Description                          Qty Zone
---- -------------- ------------- ------------------------------------ --- ----
1    HPEGS-0001                    FLANGE                               2   A6
2                   5234-0392      Hex Nut - 0.500"                     4   A4
3                   8438-8451      Flat Washer - 0.500"                 4   B7
4                   4321-8463      Hexhead Bolt - 1.000" long   0.500"  4   D4
                                   dia.
```

10. When the list has been generated, the following prompt appears:

```
Do you want to do another material list? (y/n)
```

To make another version of this list to easily add to your drawing, respond to this prompt by pressing (Y). You are prompted with the same prompts as before, but answer them as follows:

```
Use arrow keys to move the marker
Select with <RETURN> or <ENTER>
  -> USE ALL Parts Files
     EWEE:eepart_pf
     EWME:mepart_pf
     RE-START Selecting Parts Files
     DONE Selecting Parts Files
```

11. Select:

```
   EWME:mepart_pf

DONE Selecting Parts Files
```

12. The following prompt appears:

```
Which listing format do you want to use?
```

For the pre-defined drawing (or graphics) format, enter:

```
G
```

13. The following prompt appears:

    ```
    Enter the name of the output file:
    ```

    Enter:

    ```
    EWYOU:flassy
    ```

14. The following prompt appears:

    ```
    Enter the number of lines per page: (0 for no pagination)
    ```

    Enter:

    ```
    0
    ```

15. The next prompt appears:

    ```
    Enter the name of the list mat file:
    ```

    Respond by entering:

    ```
    EWYOU:flassy
    ```

16. Because drawing versions of parts lists start with the first item at the bottom of the list, press (B) in response to:

    ```
    Should the output be Forward or Backward (f/b)
    ```

17. The next prompt appears:

    ```
    Enter the optional one line header:
    ```

    Just press (Return), because the header is not neccessary for drawing versions of material lists.

18. When the list has been generated, this prompt appears:

    ```
    Do you want to do another material list? (y/n)
    ```

    Press (N). HP EGS returns you to the Mechanical Engineering menu.

Adding the Material List

Now you will add to the flange assembly drawing the graphical version of the material list you just created.

The Mechanical Engineering menu should be displayed:

```
Use arrow keys to move cursor to desired function.
Select function with <RETURN> or <ENTER>.
    RETURN To Main Menu
    Mechanical Drafting Editor
    ME Parts File Editor
  ->ME Material Lister
```

1. Return to the Mechanical Drafting personality by selecting:

   `Mechanical Drafting Editor`

   When the personality is loaded, prefix to your personal mass storage volume and then edit the original flange assembly drawing by entering:

   `EDIT flassy;`

2. Set the drawing mode to annotation and the current snapping mode to grid snapping by selecting:

   `Annotatn`

   `$GR`

3. Turn the zones off and set the nesting level to 3 and redraw the large port. Enter:

   `ZONE_OFF`

   Select:

   `WIND* NS=1 3 ENT REDRAW`

4. To accurately input the parts list file and then move it, you must set the user grid to 0.625,2. Select:

   `RULE* grNEW`

   When prompted for the distance between points, enter:

   `.0625`

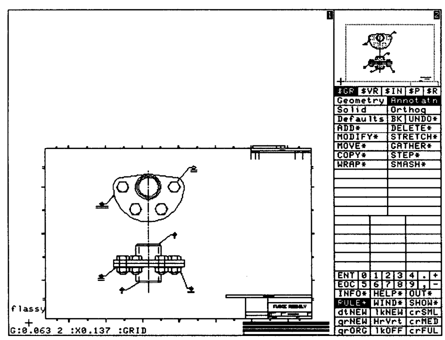
   When prompted for the skip factor, enter:

   `2`

5. Input the graphical version of the parts list by entering:

   `INPUT flassy;`

   The material list will appear just below the right corner of your border. Normally it will appear far to the right of the drawing border and you will have to use FIT to see it, but in this case you moved the border.



Moving the Parts List

1. To place the parts list in the appropriate position, first zoom in on the parts list area. Select:

   ```
   WIND* PAN
   ```

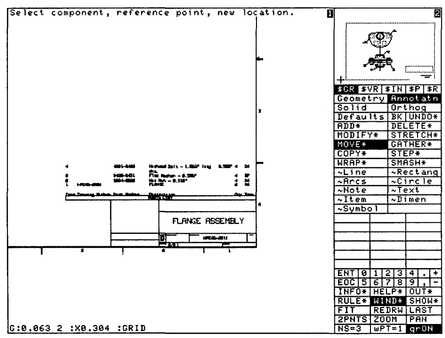
   and then select (28.0,0.0) to bring the parts list to the center of the large port.

2. Select:

   ```
   2PNTS
   ```

   and then select (20.0, −4.0) and (37.0,8.0) as the new boundary for the large port.

   When the Mechanical Material List post-processor builds the graphical version of the parts list, it outputs HP EGS add note commands. It also outputs a wrap command at the very end. This is convenient because the parts list is automatically wrapped into an instance, which makes moving it as a whole very easy.



3. Move the parts list close to its final position by selecting:

   ```
   MOVE*
   ```

   and selecting the parts list (25.0, −1.5), then selecting a reference point at (25.0, −1.5), and then placing it at (23.0,3.25).

Placing the Parts List

1. To complete the final adjustments and additions to the parts list, you must turn on the display grid and zoom in on the left side of the parts list. Select:

   ```
   grOFF 2PNTS
   ```

   and then select (22.5,2.5) and (26.0,6.0) as the new boundary for the large port.

2. A special library part is available so that you can easily add the parts list border lines. To add this part, select:

   ```
   ADD* SYMBOLS
   ```

3. Enter:

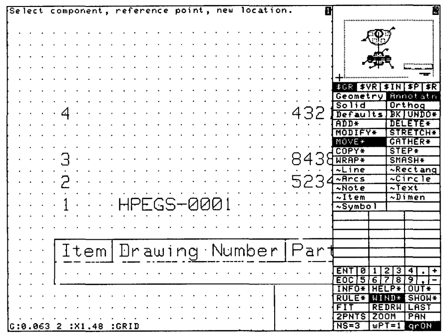
   ```
   ml_bord
   ```

   and then place this part at the upper left corner of the parts list rectangle in the title block (23.0,3.25). You can now see that the material list is not exactly placed yet, because the headings are not centered in their appropriate cells. Move the material list again so that the headings are centered.

4. Select:

   ```
   MOVE*
   ```

   and select the parts list (23.0,3.75) and then the left end of the ITEM heading as the reference point (23.0,3.3125). Then move this point up and to the right one user grid location (23.0625,3.375).



Stepping the Parts List Border

1. To complete the modifications to the parts list, you must make a copy of the parts list border for each line in the parts list. Select:

   ```
   STEP*
   ```

   and carefully select the parts list border (23.0,3.25) and then the lower left corner of the parts list border as the reference point (23.0,3.25).

2. Since you are using the matrix method of step, enter the following values for the number of columns and rows, respectively:

   ```
   1
   ```

   ```
   7
   ```

   (The rows correspond to the total number of lines in the parts list.)

3. Place the first copy of the step matrix at the upper left corner of the original (23.0,3.5).

4. Before you save and plot the flange assembly drawing, fit the entire drawing to the large port. Select:

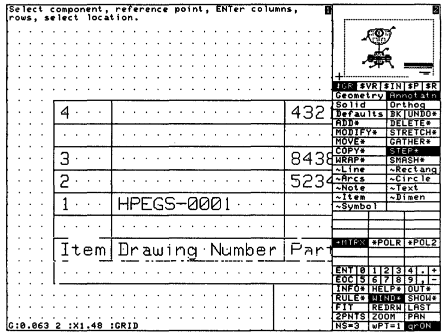
   ```
   FIT
   ```

5. Save the drawing under its present name by selecting:

   ```
   OUT* SAVE* OLD
   ```

6. To plot this drawing at its actual size, make sure you have a plotter on line with a D-size sheet of paper loaded. Then select:

   ```
   PLOT* SCALE 1 ENT CENTR EOC
   ```



BREAK TIME

Relax, stretch, and take a break. You have now completed the flange assembly drawing example. In this section you:

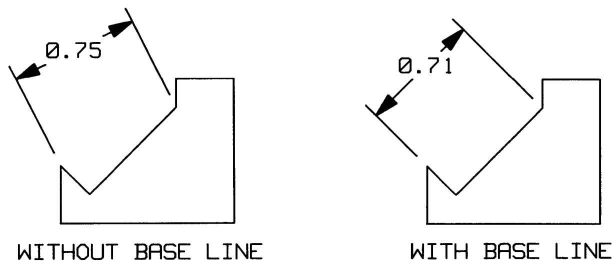
- Made a material list data file from the assembly drawing.
- Changed the parts file.
- Created a printed and graphical parts list.
- Added the graphical list.
- Moved the parts list.
- Stepped the parts list border.

Adding Linear Dimensions Using Baselines

The *LINE method of adding dimensions enables you to add dimensions that provide you the actual, straight-line distance between two points. The dimension in the upper left corner of the following figure is an example of using the *LINE method.

You may sometimes want to use the *LINE method, but with the distances measured parallel to a base line instead. The dimension in the lower right corner of the following figure is an example of this. To add a linear dimension this way you must first set the base line.

To set the base line, select the option *BASE and then select two points defining the base line. All linear dimensions you add will be measured parallel to this base line until you either select *BASE again or select some other dimensioning method.

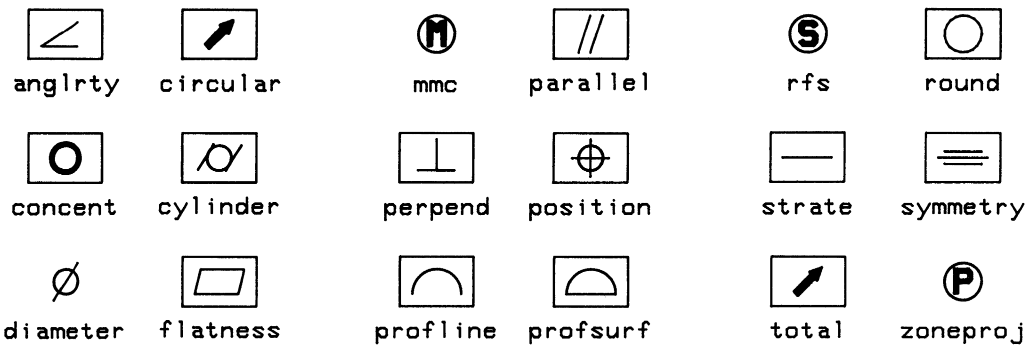


Effects of Using a Baseline

Mechanical Library Parts

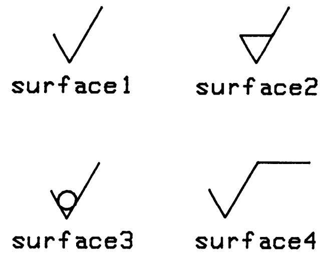
To help you get started creating mechanical drawings, a set of symbols, fasteners, and drawing borders are provided with this personality. These library parts are stored in the default volume for this personality, EWME.

The symbols provided can be grouped into two general categories: geometric tolerancing and surface finish symbols. The geometry for these symbols was created in the Annotation mode of Mechanical Drafting with the line-type set to Extra. The members of each group of symbols are listed below.



Geometric Tolerancing Symbols

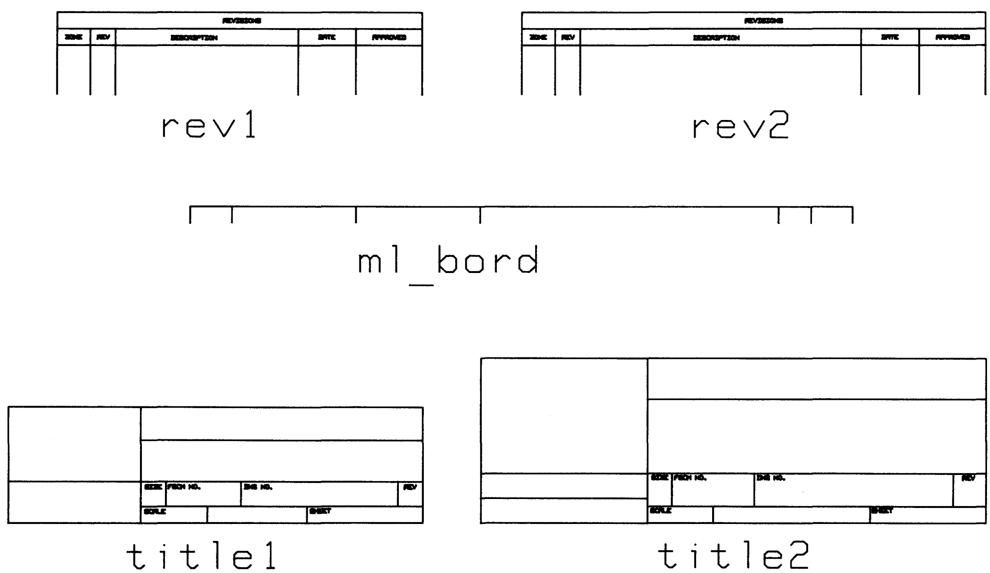
Symbol Name	Description	
anglrty	Form	- Angularity
circular	Runout	- Circular
concent	Location	- Concentricity
cylinder	Form	- Cylindricity
diameter	Other	- Diameter
flatness	Form	- Flatness
mmc	Other	- Maximum Material Condition
parallel	Form	- Parallelism
perpend	Form	- Perpendicularity (Squareness)
position	Location	- Positional
profline	Form	- Profile of a Line
profsurf	Form	- Profile of a Surface
rfs	Other	- Regardless of Feature Size
round	Form	- Roundness
strate	Form	- Straightness
symmetry	Location	- Symmetry
total	Runout	- Total
zoneproj	Other	- Projected Tolerance Zone



Surface Finish Symbols

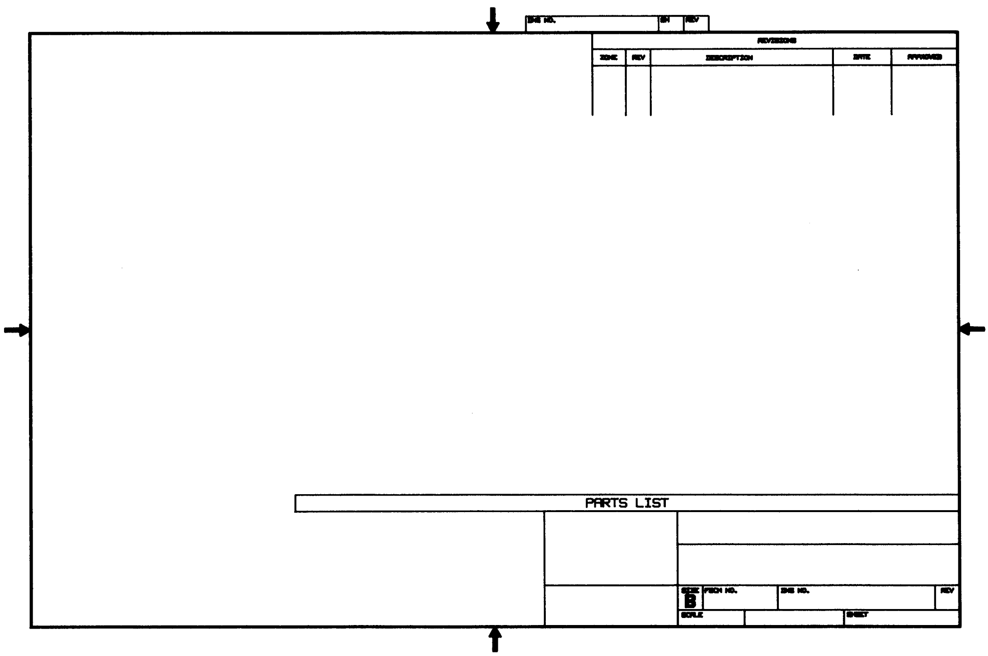
Symbol Name	Description
surface1	Basic surface texture symbol
surface2	Material removal by machining is required
surface3	Material removal is prohibited
surface4	Surface texture symbol

Library parts are also provided, which define the borders for A through E-size sheets. Also provided as library parts are the revision and title blocks used in these borders, which conform as closely as possible to the ANSI standard borders. The only difference is that the overall size of each border has been sufficiently reduced so that it can be plotted on the corresponding paper using an HP 7580, HP 7585, or HP 7586 plotter.



Drawing Border Parts

Symbol Name	Description
revision1	A, B, and C size revision block
revision2	D and E size revision block
title1	A, B, and C size title block
title2	D and E size title block
ml_bord	Border for one line of a parts list

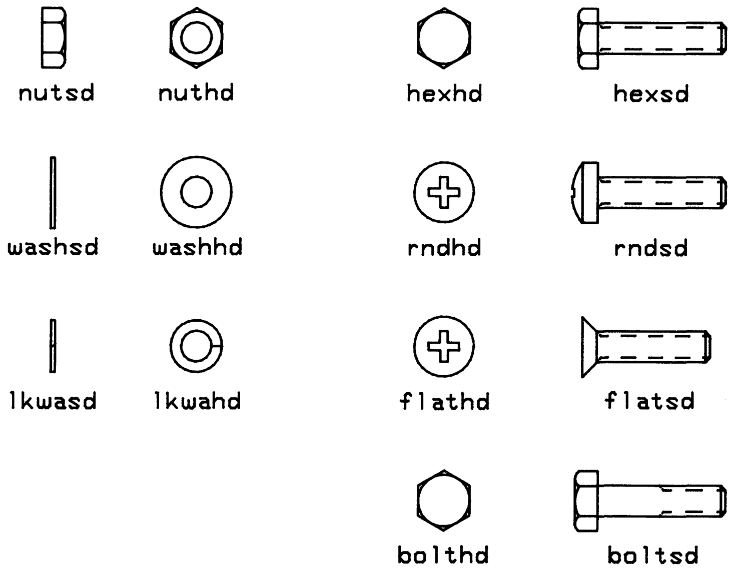


Sample Border (B-Size)

Symbol Name	Description
asize	Horizontal A size border
bsize	Horizontal B size border
csize	Horizontal C size border
dsize	Horizontal D size border
esize	Horizontal E size border

The parts provided are simple fasteners that have all been created as macro instances. This means that when you add one to your drawing, HP EGS prompts you for parameters needed to determine the actual size of the part.

For example, when you add the side view of the hex screw (hexsd), you will be prompted for the length and diameter of the screw. Also, two views of each part are provided. The side views have the suffix 'sd', and the head, or top, views have the suffix 'hd'. When adding these parts, you will find that the solid geometry is always added to layer 11 and the dashed geometry to layer 14. This corresponds to having the color mode set to Red and the line-type mode set to Solid or Dashed.

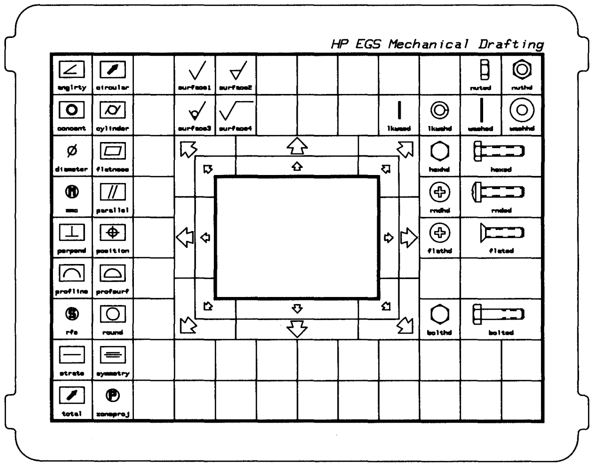


Fasteners

Part Name	Description
nuthd	Top view of a hex nut
nutsd	Side view of a hex nut
washhd	Top view of a washer
washsd	Side view of a washer
lkwahd	Top view of a split lock washer
lkwasd	Side view of a split lock washer
hexhd	Top view of a hex screw
hexsd	Side view of a hex screw
rndhd	Top view of a round head screw
rndsd	Side view of a round head screw
flathd	Top view of a flat head screw
flatsd	Side view of a flat head screw
bolthd	Top view of a hex head bolt
boltsd	Side view of a hex head bolt

The Mechanical Drafting Tablet Menu

The tablet menu provided for use with the Mechanical Drafting personality is shown in the following figure:



Mechanical Drafting Tablet Menu

This menu is set up for the tablet rather than the screen so that you can select symbols and library parts without having to use the *SCRN option in the ADD* SYMBOLS and ADD* PART commands. You may also find this tablet menu convenient since it gives you a picture of the symbols and parts as well as the name.

Also notice that the tablet menu has some port manipulation commands around the screen tracking block. These commands enable you to pan the large port in the direction of the arrows with a single stroke. The large arrows represent full window pans while the small arrows represent half window pans.

See Appendix A in this manual to plot this tablet menu, or Appendix C to add a new part to this tablet menu.

BREAK TIME

Take a long break now that you are done with this lesson.

Electrical Engineering Drawings

Chapter 8: Electrical Schematic Drawings

Chapter 9: Printed Circuit Board Layout

Electrical Schematic Drawings

Chapter 8

Introduction

As discussed in earlier chapters, HP EGS is a very customizable drawing system. With HP EGS, Hewlett-Packard has provided pre-configured customizations of the basic graphics editor for Electrical Engineering. These customizations address the drawing of electrical schematics and printed circuit board artwork. The two personalities included with the Electrical Engineering Drawings are the Electrical Schematic Drawing personality and the Printed Circuit Board Layout personality.

- Electrical Schematic Drawing personality: The Electrical Schematic Drawing personality makes schematic drawings quick and convenient to prepare by providing menus, macros, process files, and some commonly used schematic symbols, stored as library parts. The personality operates the same as the General Drawing personality, with some changes appropriate for the creation of schematic drawings and schematic library parts or symbols.
- Printed Circuit Board Layout personality: The Printed Circuit Board Layout personality provides the tools to lay out printed circuit boards. A set of library parts and some easily used drawing conventions are provided for your help. This personality operates the same as the General Drawing Personality, with some changes appropriate for the creation of printed circuit board artwork and the associated library parts. The Printed Circuit Board Layout personality is described in the next chapter.

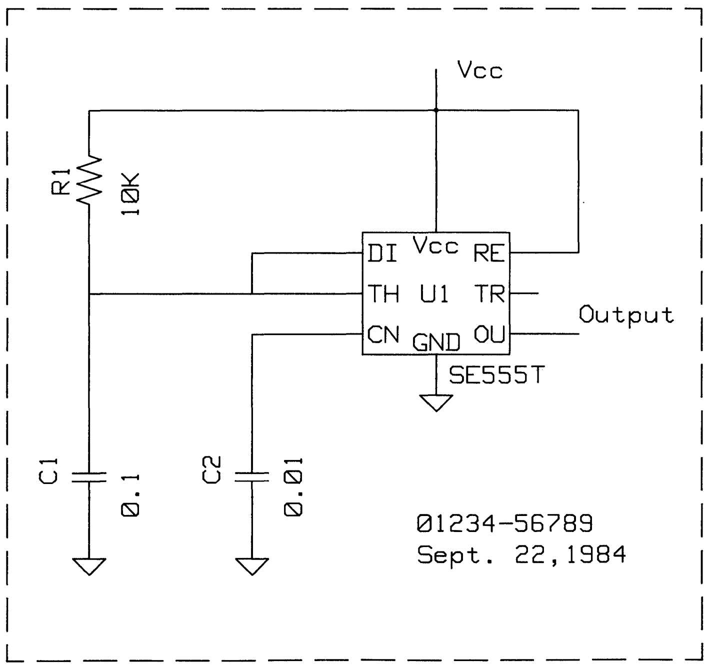
Note

Before you begin the lessons in this chapter, be sure you have read Chapters 2, 3, and 4 of this manual. They contain necessary basic information that is not repeated in this chapter.

In this chapter you will learn the differences between the Electrical Schematic Drawing personality and the General Drawing personality. Then you will use the Electrical Schematic Drawing personality to begin creating the schematic for a one-shot. During this process you will create a new library part representing a 555 timer. When the schematic is complete you will generate a connection list and a material list from the drawing.

In Chapter 9 you will create the artwork for a small circuit board which implements the circuit.

The following figure shows the finished drawing from the one-shot pulse generator example.



Completed Schematic

Understanding Electrical Schematic Drawing

To use the Electrical Schematic Drawing personality you need to understand the basic differences between this personality and the General Drawing personality, which you should already understand. We will approach these differences by first examining how the process file has been changed and then how the macro file has been changed.

The Process File

The HP EGS process file defines the system units, the default parameters, and the layer definitions for the data in your drawing. For this personality, the basic unit of measurement is the centimetre, with 100 system points per centimetre. This means that the location of every point in your schematic can be specified to 0.01 centimetre. By changing the user grid setting and snapping modes, you may force points to fall on some multiple of the basic resolution.

Several defaults are different due to this change in units. For example, the default font size is 0.20 centimetres, where it was 10 millimetres in General Drawing.

The changes to two default values, LOCK and RESOLUTION, pose more interesting differences. LOCK, which is the default lock angle, is set to 45 degrees in this personality. (LOCK is 0 degrees in General Drawing.) This means that when you are adding straight segments of lines and polygons, HP EGS informs you if they are not at intervals of 45 degrees.

RESOLUTION, which is the default resolution angle for arcs and circles, is set to 15 degrees. (RESOLUTION is 0 degrees in General Drawing.) Circles and arcs appear rougher on your display. For example, a 360-degree circle appears to have 24 sides. For more information on the effects of these commands, see the *HP EGS Syntax Reference*.

As you learned by using the General Drawing personality, all of the components in your drawing are placed on discrete layers. Recall that each layer is like a transparent sheet which exactly overlays the other layers. Each layer specifies the color and line-type of the component as well as the pen it will be plotted with. The Electrical Schematic Drawing personality takes advantage of this layering capability by separating various aspects of the drawing on different layers. The definition of the layers is different than in General Drawing and is as follows:

Layer	Use	Color	Line Type	Pen #
0	Instances	White	Short Dotted	1
1	Part Geometry	Red	Solid	1
2	Part Notes and Text	Red	Solid	2
3	Connections	Green	Solid	3
4	Buslines	Cyan	Solid	4
5	Junctions	Yellow	Solid	5
6	Schematic Notes and Text	Magenta	Solid	6
7	Outlines	Yellow	Broken	7
50	Logical Ports	Yellow	Solid	1
51	Physical Ports	Yellow	Solid	1
52	Part Types	Yellow	Solid	1
53	Reference Designators	Yellow	Solid	1
54	Net Names	Yellow	Solid	1
255	Errors	White	Solid	1

You can change these parameters using such commands as UNITS and EQUATE. For more information on changing the process file, see the Customizing chapter of *Understanding HP EGS.*

The Macro File

The macro file contains the definitions of all the items that appear in the screen menu. Because the macro file for the Schematic Drawing personality is based on the macro file for the General Drawing personality, you should be familiar with its operation. Where appropriate the macro file has been changed. Some of the important changes follow.

Changes to the Modes

A major difference to the General Drawing personality is the presence of two overall drawing modes, Edit_Sch (edit a schematic) and Make_Prt (make a part). These modes are defined as follows:

- Edit_Sch is used when actually drawing a schematic.
- Make_Prt is used to interrupt the drawing of a schematic to create a new part. Some functions in this mode are changed from Edit_Sch to make the creation of parts easier. Upon completing the part, the system automatically returns to the schematic that was interrupted.

The General Drawing personality has color and line-type modes (such as Solid and Red) that you set to determine the layer on which to place components. The Electrical Schematic Drawing personality has four distinct modes, Geometry, Connect, Busline, and Outline, which determine the layers for placing geometric components.

- Geometry: Specifies geometric components to be added to the part geometry layer (layer 1).
- Connect: Specifies geometric components to be placed on the connection layer (layer 3).
- Busline: Specifies geometric components to be placed on the busline layer (layer 4). This is the only mode in which the components can have width (the default is 0.1 centimeter).
- Outline: Specifies geometric components to be placed on the outline layer (layer 7). Note: an outline does not represent any kind of logical connection in the schematic you are creating.

NOTE and TEXT components are always placed on layers determined by the current drawing mode. They are automatically placed on layer 6 in the Edit_Sch mode and layer 2 in the Make_Prt mode.

Changes to the Primary Commands
Additional primary commands have been constructed, using macros, to make the creation of schematics easier. These commands are:

- ROUTE*: Provides a convenient way to re-route an existing connection. This is similar to STRETCH* *REPL.
- LEVEL*: Provides a convenient way to set the logical level of a part (logical levels are explained later).
- MOD_NOTE*: Provides a convenient way to modify notes. It is most often used to change the part type, reference designator, and part name defaults defined with library parts to the actual values.
- LIST_MAT: Compiles a list of the library parts used in a schematic. This list can then be post-processed to produce a formal bill of materials.

LIST_MAT never appears in the screen menu. Therefore, you must enter this command from the keyboard when you want to use it.

Changes to the Components

Some of the components presented in General Drawing (such as HATCH and DIMEN) are not useful for creating schematics and therefore are not included in the Schematic Drawing personality. But other "special" components are useful. These components were created using macros and are included in the Electrical Schematic Drawing personality. These are:

- JUNCTION: This component forces the logical connection of two crossing connection lines. A junction is a marker that is automatically placed on layer 5.
- NET_NAME: This component specifies the name of a net. It is a NOTE component that you associate to a specific connection or bus line. This NOTE is automatically placed on layer 54 and given a tag value of 1002.
- PART_TYP: This component specifies the identifying type of a library part. It is a NOTE component that you associate to a part when it is created. This NOTE is automatically placed on layer 52 and given a tag value of 1004.
- REF_DESI: This component specifies the reference designator of a part. A reference designator is a unique name used to distinguish between multiple occurrences of the same library part. It is a NOTE component that you associate to a part when it is created, and later modify when the part is added to a schematic. This NOTE is automatically placed on layer 53 and given a tag value of 1001.
- LOG_PORT: This component specifies the name and location of a logical port. It is a NOTE component that you associate to a part when it is created. When you add this component you must also specify the location of the port (or connection point) in the part which it names. This NOTE is automatically placed on layer 50 and given a tag value of 1003.
- PHY_PORT: This component specifies the name and location of a physical port. Physical port names are used to reflect the name of the port in the corresponding printed circuit board for the schematic. It is a NOTE component that is added just like a logical port, but is modified when the printed circuit board is created. This NOTE is automatically placed on layer 51 and given a tag value of 1003.

The tag values mentioned above are used by the material and connection listing processors to differentiate between the different notes in your drawing.

Changes to the Secondary Commands

The only change made to the secondary commands is in the operation of SHOW*. The shON/shOFF and sPT=1/sPT=2 options and the ALL method work the same as in the General Drawing personality. Due to the differences in the layer structure, the COLOR, LNTYP, and BOTH methods have been replaced with NETNA, LOGPT, PHYPT, NOTES, REFDE, and PRTYP.

Loading Electrical Engineering

1. If your computer is not displaying the main menu in the HP EGS manager, follow the instructions in Chapter 2, in the *Loading General Drawing* section, to get to the main menu. Your computer display should look like this:

```
Use arrow keys to move cursor to desired function.
Select function with <RETURN> or <ENTER>.
  ->System Utilities
    General Drawing Editor
    Electrical Engineering
    Mechanical Engineering
```

2. Move the cursor to the Electrical Engineering function and press (Return). The display should look like this:

```
Use arrow keys to move cursor to desired function.
Select function with <RETURN> or <ENTER>.
  ->RETURN To Main Menu
    Printed Circuit Board Layout Editor
    Electrical Schematic Drawing Editor

    EE Parts File Editor
    EE Material Lister
    Convert Old EE Parts Files

    Printed Circuit Board Connection Lister
    Electrical Schematic Connection Lister
    Compare Connection Lists
    Merge Connection Lists
    Generate Rat's Nest

    PHOTOPLOT/DRILL Post-processor
```

3. Select:

```
Electrical Schematic Drawing Editor
```

The Electrical Schematic Drawing personality will now load automatically.

In the next section of this chapter you begin the drawing example in which you learn to create a schematic using the Electrical Schematic Drawing personality. To preview this lesson, enter:

```
INPUT EWLRN:chap8;
```

Note

Before you begin to draw the schematic, use the PREFIX command to prefix HP EGS to your personal mass storage volume. This keeps you from editing or destroying someone else's data as you edit and save drawings.

Starting the Schematic

In this section you will begin to create a one-shot schematic by adding the necessary library parts. This includes the creation of a new library part which represents an SE555 monolithic timer.

Preparing the Screen

1. Prefix to your storage volume by entering:

   `PREFIX DRAW:;`

2. To set the drawing mode to editing schematics, select:

   `EditSch`

   The display prompts you for the name of the schematic to edit.

3. Enter:

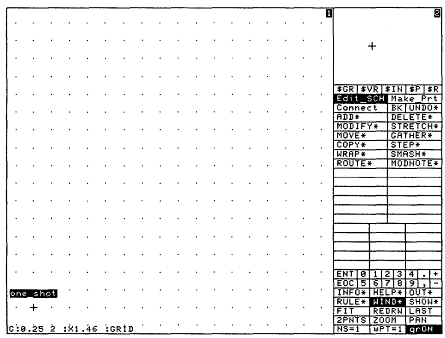
   `one_shot`

   Because the one_shot drawing does not exist yet, you will be editing an empty drawing. It is important to set the drawing mode, however, so that the Schematic Drawing personality can store the name of the current schematic for later use.

4. Prepare the drawing area for the addition of components by turning the grid on and zooming in on a smaller portion of the large port. Select:

   `WIND* grOFF 2PNTS`

5. Select the two points (−0.75, −0.75) and (8.25,8.25). Your screen should look like this:



Adding Capacitors and Resistors

Now you are ready to add the passive parts, two capacitors and one resistor, to the drawing.

1. Select:

   `ADD* PART`

   Enter the names of the parts you want to add.

2. Because you don't know the actual file names for resistors or capacitors, select the option:

    `*SCRN`

    Your screen menu is redrawn, and it displays a list of the parts provided with HP EGS.

3. From the upper left corner of the menu, select:

    `CAP`

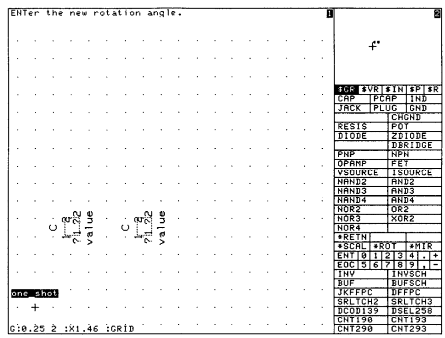
4. Move the cursor into the drawing area. Note that you are tracking the boundary of the capacitor, which is horizontal. As shown in the first figure of this chapter, the capacitor needs to be rotated to a vertical orientation before it is placed. From the middle of the menu, select the option:

    `*ROT`

5. Enter:

    `90`

    for a 90-degree rotation. Now place the capacitor at (1.00,2.00) and (3.00,2.00). Your screen should look like this:



6. Add the resistor to the schematic by selecting:

    `RESIS`

    and placing it at (1.00,5.50). Notice that the resistor is rotated 90 degrees.

7. Add the two ground symbols by selecting:

    `GND`

8. You do not want to rotate these parts, so select:

    `*ROT`

9. To reset the rotation to 0 degrees, enter:

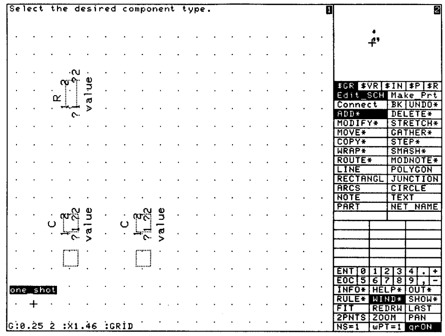
    `0`

10. Place the ground parts at (1.00,1.50) and (3.00,1.50).

11. Now that all of the discrete parts have been added, return to the original menu by selecting the option:

    `*RETN`

    Your screen should look like this:



Viewing the Instances

1. To set the nesting level to 3 and to redraw the screen so that you can see inside the instances you have just added, select:

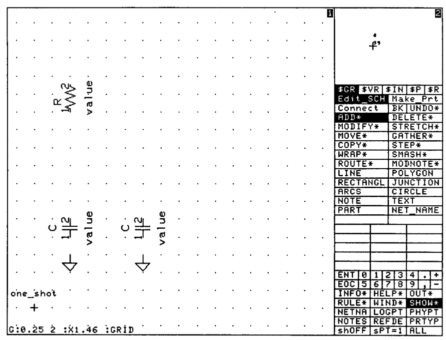
    `NS=1 3`

    `ENT REDRW`

2. Reduce the clutter in your drawing by using the SHOW* command to turn off the display of the physical port names. Select:

    `SHOW* PHYPT`

    (Remember that if shON/shOFF is set to shOFF, components are turned off.) Your screen should look like this:



Connecting Passive Parts

To connect the passive parts you must add lines, or other geometric components, between the appropriate ports. To start, insure that the current layering mode is set to Connect.

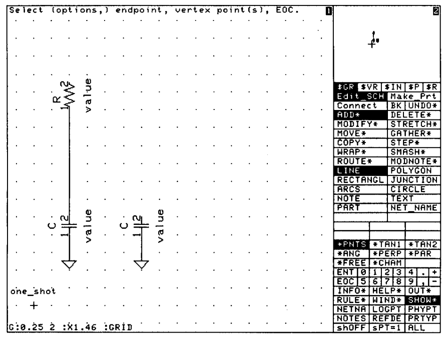
1. Select:

   ```
   ADD* LINE
   ```

   Add lines between the following points:

   (1.00,5.50) (1.00,2.50) EOC
   (1.00,2.00) (1.00,1.50) EOC
   (3.00,2.00) (3.00,1.50) EOC

   Your screen should look like this:



Creating a Schematic Library Part

1. Select the drawing mode:

   ```
   Make_Prt
   ```

2. When prompted for the drawing name, enter:

   ```
   tim555
   ```

3. Again, you need to prepare the part drawing area for the addition of components. This time you will set the grid to 0.05 centimetre spacing so that your part is consistent with the others in the library. Select:

   ```
   RULE* grNEW 0.05 ENT 2 ENT
   ```

4. Turn the grid on and zoom in on a smaller portion of the large port by selecting:

   ```
   WIND* grOFF 2PNTS
   ```

5. Select the two points (−0.75, −0.75) and (3.00,3.00). Note that this is smaller than before, because you are making a single part.

Defining Part Geometry

The first step in defining a schematic library part is to draw the part geometry.

1. The 555 timer is a digital device which is best represented as a block with short lines for the input and output ports. To draw the block, select:

   ```
   ADD* RECTANGL
   ```

2. Select the points (0,0) and (1.80,1.50). This size is consistent with the other parts provided with HP EGS.

3. To add the ports, select:

   `LINE`

4. Add a line from (1.80,0.25) to (2.15,0.25). This position and length is consistent with other parts.

   The length of the port must be 0.35 centimetres because all of the ports must terminate on 0.25 centimetre grid positions when the part is completed. Once ports have been placed on each side of the block, the overall length will be (0.35 + 1.80 + 0.35), or 2.50 centimetres, an interval of the 0.25 grid. You will soon see how all of the ports are finally aligned with the 0.25 centimetre grid.

5. To add the balance of the horizontal ports, select:

   `COPY*`

6. Select the existing port, then the left end (or 1.80,0.25) as the reference point. Select the following points for the positions of the copies:

   (1.80,1.25)
   (1.80,0.75)
   (−0.35,1.25)
   (−0.35,0.75)
   (−0.35,0.25)

7. Terminate copying with EOC.
8. Add the vertical ports for the supply voltage and the ground by selecting:

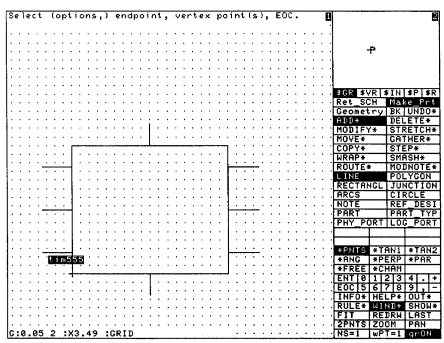
   `ADD* LINE`

9. Select the following sets of points:

   (0.90,0.00) (0.90, −0.25) EOC
   (0.90,1.50) (0.90,1.75) EOC

   Again note that the overall height of the part is (0.25 + 1.50 + 0.25), or 2.00 centimetres, which is an interval of the 0.25 grid. Also notice that the top of the vertical port is 0.50 centimetres above the uppermost horizontal port. Therefore, once any endpoint of a port is aligned with a 0.25 centimetre grid, all other endpoints of ports will be aligned as well.

   Your screen should look like this:



Aligning a Part

Before aligning the part, consider how you want to add it to a schematic. Remember that the cursor will track the (0,0) position of the part, so a good choice would be to place one of the port endpoints at (0,0).

It is generally not important which port is placed at (0,0), as long as it is always the same one. Because the parts in the provided library all have the upper left port placed at (0,0), so will the parts in this example.

1. To align the part, select:

   `WRAP*`

2. Select the wrap origin, such as (0,0), and wrap all of the components on the screen by selecting (–0.65, –0.65) and (2.60,2.20). Select EOC to terminate the wrap.
3. To see inside the instance, change the nesting level to 2 and redraw the port by selecting:

   `NS=1 2 ENT REDRW`

4. Move the instance so that the upper left port is properly aligned with (0,0). Select:

   `MOVE*`

5. Select the instance, then select the left endpoint of the upper left port as the reference point (–0.35,1.25), and then place the instance at (0,0).

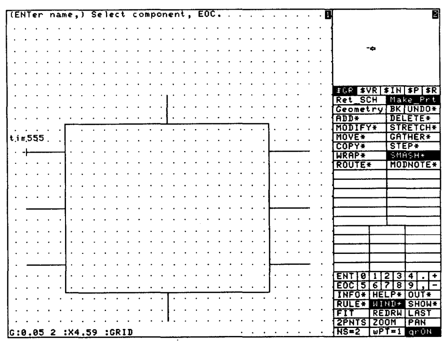
6. To fit the part to the screen, select:

   `FIT`

7. Select:

   `SMASH*`

8. Select the instance so it will be smashed back to the component level. Your screen should look like this:



Adding the Reference Designator and Part Type

You are now ready to annotate the part with the reference designator and part type. These notes must be in your library part so that you can create valid material and connection lists from the final schematic.

The reference designator is the name by which a single instance of a part is referred to, such as U37 or R54. When building a library part you need to add a default designator (i.e., U or R), which you will modify later in the actual schematic.

1. To add the default designator, select:

   ```
   ADD* REF_DESI
   ```

2. Enter:

   ```
   'U'
   ```

   (Don't forget the quotation marks.)

3. Select the center of the rectangle (1.05, –0.50) as the location of the note string.

4. The part type represents the value or type of your part. If your part has an exact part type in the part file you will use for material listing (such as SE555T), add it here.

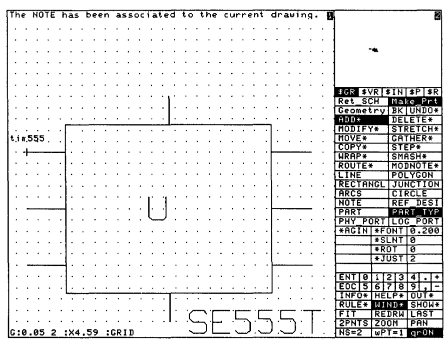
   For parts which do not have exact types (such as resistors), add the word 'value' or 'type' here. You would modify this word later when the part is added to a schematic (that is, change 'value' to '10K' for a resistor). To add the part type for this example, select:

   ```
   PART_TYP
   ```

5. Enter:

   ```
   'SE555T'
   ```

6. Select (1.40, –1.50) as the location of the note string. Your screen should look like this:



Adding Logical Ports

The next annotation task is to assign the logical names of the input and output ports for this part. Like reference designators and part types, port names are associated notes. An important difference is that port names also point to a logical location on the part. This logical location is the point on the part where the connections in a schematic drawing will be made. Therefore, the sequence for adding a port name is to first select the logical location, then enter the port name, and then place the port name string.

1. Add the logical port name for the supply voltage port by selecting:

    ```
    LOG_PORT
    ```

2. Select the top end of the port on the top of the part (1.25,50), and enter:

    ```
    'Vcc'
    ```

3. Place this string just inside the part body, below the port (0.95,0.10).
4. To add the next logical port, select:

    ```
    *AGIN
    ```

    This resets the adding of associated notes so you can again select a logical location.
5. Select the bottom end of the bottom port (1.25, −1.50), and enter:

    ```
    'GND'
    ```

6. Place this string at (0.95, −1.10).
7. Add the remaining logical ports by repeating steps 4 through 6 for the following values:

Port Location	Name	Name Location
2.50,0.00	'RE'	1.70,0.00
2.50, −0.50	'TR'	1.70, −0.50
2.50, −1.00	'OU'	1.70, −1.00
0.00,0.00	'DI'	0.40,0.00
0.00, −0.50	'TH'	0.40, −0.50
0.00, −1.00	'CN'	0.40, −1.00

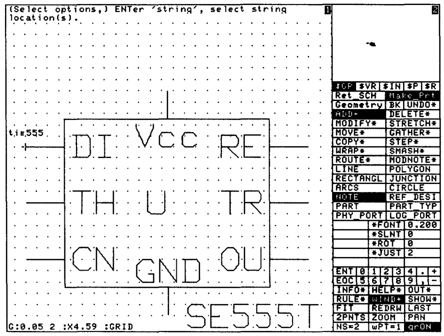
8. In most parts, such as capacitors and resistors, you will want to turn off the display of the logical ports. However, because you want to display complete chips just like the part geometry, you should re-add these logical port names as geometry, directly over the existing names. Select:

    ```
    NOTE
    ```

9. Re-add the logical port names. Here are the names and locations:

Name	Name Location
'Vcc'	0.95,0.10
'GND'	0.95, −1.10
'RE'	1.70,0.00
'TR'	1.70, −0.50
'OU'	1.70, −1.00
'DI'	0.40,0.00
'TH'	0.40, −0.50
'CN'	0.40, −1.00

Your screen should look like this:



Adding Physical Ports

The final annotation task is to assign the physical names of the input and output ports for this part. Physical port names are a second set of port names that represent the names of the ports in a printed circuit board part. These names are used when comparing connection lists between schematics and printed circuit boards, and when generating a rat's nest for a printed circuit board.

The sequence for adding a port name is to first select the logical location, then enter the port name, and then place the port name string.

1. To add the first physical port name, select:

   ```
   PHY_PORT
   ```

2. Select the top end of the port on the top of the part (1.25,0.50), then enter:

   ```
   '8'
   ```

3. Place this string to the left of the port (1.00,0.40).

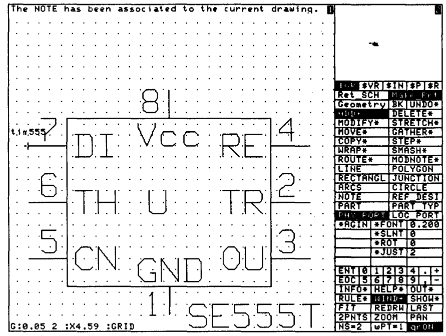
4. Select:

   ```
   *AGIN
   ```

   Add the remaining physical port names for the following values:

Port Location	Name	Name Location
0.00,0.00	'7'	0.10,0.15
0.00, −0.50	'6'	0.10, −0.35
0.00, −1.00	'5'	0.10, −0.85
1.25, −1.50	'1'	1.00, −1.40
2.50,0.00	'4'	2.20,0.15
2.50, −0.50	'2'	2.20, −0.35
2.50, −1.00	'3'	2.20, −0.85

   Your screen should look like this:



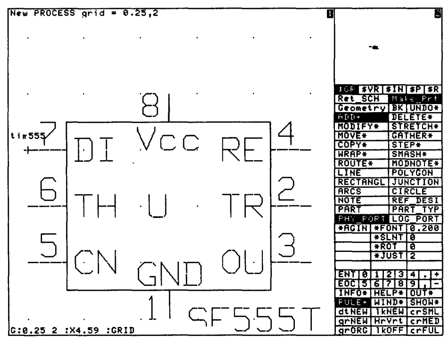
Verifying the Part

Next you will check this part to make sure it has been built correctly.

1. Verify that the part is properly aligned with the grid that will be used in the schematic drawing by selecting:

   ```
   RULE* grNEW 0.25 ENT 2 ENT
   ```

   Your screen should look like this:



2. Examine this part to make sure that all port endpoints lie exactly on a grid point (exactly on a displayed grid point, or exactly halfway between displayed grid points).
3. Make sure that all logical locations of the ports are correct by selecting:

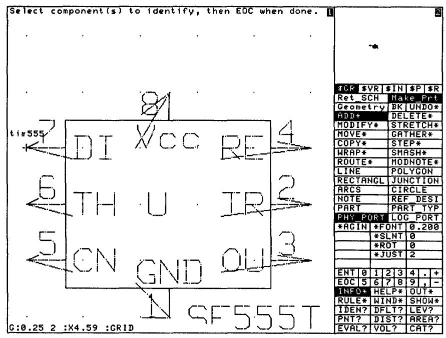
   ```
   INFO* IDEN?
   ```

4. Enter the tag value for port names (discussed at the beginning of this chapter):

   `:T1003`

   All port names, logical and physical, should now be highlighted, and a small line is drawn from the lower left corner of each port name string to its corresponding logical location. If the logical location of one of the port names does not point to the end of the proper port line, reset the grid to 0.05,2, and then delete and re-add the port properly.

   Your screen should look like this:



Understanding Logical Levels

The part is now complete. To return to the original schematic, select:

`Retn_Sch`

Notice that two commands, Continue and LEVEL*, appear at the bottom of the command menu. The display reads:

`Remember to assign level, save the part, then select Continue.`

This prompt reminds you to do two more tasks before the part is ready for use: set the logical level and save the part. Before you set the logical level, you need to know a little more about logical levels.

Logical levels are similar to nesting levels, except that they define logical rather than physical relationships. The logical hierarchy is created with a user assignment of a logical level number to each part or instance.

Unlike nesting levels, higher positive numbers mean a higher logical position in the hierarchy. When adding an instance to a drawing (or to another instance), the instance must have a logical level less than or equal to the logical level of the drawing (or instance) being drawn.

If you look in the process file for each personality you will find that the default logical level is set to 1 (LEVEL 1;). Therefore, any drawing you create will have a logical level of 1 unless you reset it. The logical level can be changed only by editing the instance as an individual drawing. If you create an instance with the WRAP or GROUP commands, you must edit the instance so you can execute the LEVEL command on it explicitly.

Because all library parts furnished with the system are at a logical level of 0, the :D0 option can be specified in the LIST_MAT macro command. This command will then process only "base level" parts in a drawing, and ignore groups and wraps (which would be at a logical level of 1 or higher).

Through the use of logical levels, you can specify a hierarchy of instances within a drawing. Setting the logical level can also be useful for the ARCHIVE and GENERATE commands, since unwanted instances can be eliminated from the output file.

1. By convention the post-processors furnished with your system must have the schematic and printed circuit board parts set to a logical level of 0 or lower. To set the logical level of your part to 0, select:

   ```
   LEVEL*
   ```

2. Enter:

   ```
   0
   ```

3. You are now ready to save your part on the default volume under the name tim555, which you entered earlier. Select:

   ```
   OUT* SAVE* OLD
   ```

   Now that you have completed the final two tasks, return to the original schematic by selecting:

   ```
   Continue
   ```

   Answer No to the prompt to input the "one_shot" from disc, since the version in memory is the one currently being edited.

BREAK TIME

Take a short break now. When you return you will complete the schematic. Review what you have done in this section:

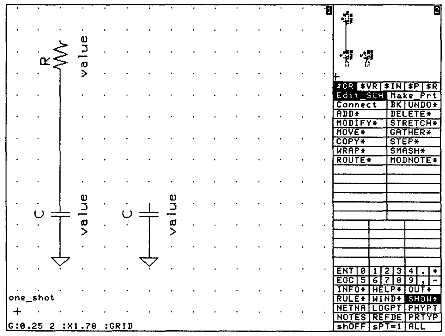
- Learned about the process file and the macro file.
- Loaded the Electrical Schematic Drawing personality.
- Added capacitors and resistors.
- Viewed the instances.
- Connected passive parts.
- Defined a schematic library part.
- Aligned a part.
- Added the reference designator and part type.
- Added logical ports and physical ports.
- Verified and saved the part you created.

Completing the Schematic

In this section you will complete the one-shot schematic by adding the final connections and modifying the appropriate annotations in the drawing.

Adding Connections

Your screen should now be displaying the original schematic. Notice, however, that the nesting level is set to 1 and all of the layers are shown. This is because the one_shot schematic was automatically brought back using the EDIT command.



1. Reset the nesting level to 3, turn on the grid, and turn off the display of the physical ports by selecting:

```
WIND* grOFF NS=1 3 ENT REDRW SHOW*
PHYPT LOGPT
```

2. Add the tim555 library part which you just created. Select:

```
ADD* PART
```

3. Enter:

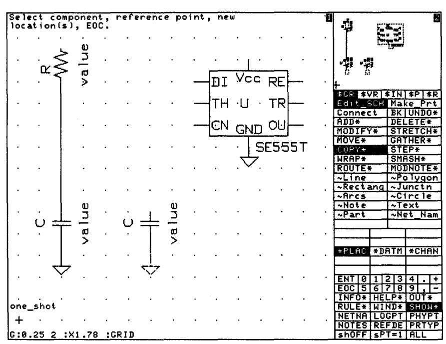
```
tim555
```

4. Place the part at (4.00,5.50).

   The logical port names are still visible in the tim555 ports because you added them as geometry as well as logical port names.

5. To connect the GND port of the tim555 part to ground, copy one of the existing grounds and place it such that its vertical lead touches the GND port. Select:

```
COPY*
```



6. Select the ground on the right capacitor, then select the end of the vertical lead as the reference point (3.00,1.50), and place it at the endpoint of the GND port (5.25,4.00). Your screen should look like this:

7. Note that the layering mode is set to Connect, so connections can be added by selecting:

   ```
   ADD* LINE
   ```

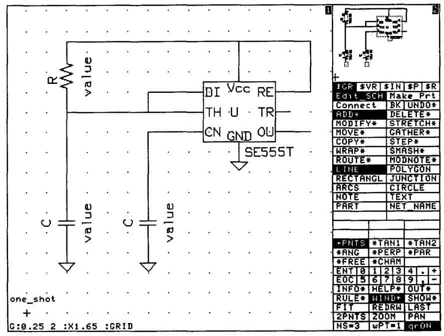
   and add connection lines between the following points:

   (3.00,2.50) (3.00,4.50) (4.00,4.50) EOC
   (1.00,5.00) (4.00,5.00) EOC
   (4.00,5.50) (3.00,5.50) (3.00,5.00) EOC
   (5.25,6.00) (5.25,6.75) EOC
   (1.00,6.25) (1.00,6.75) (7.00,6.75) (7.00,5.50) (6.50,5.50) EOC
   (6.50,4.50) (7.00,4.50) EOC

8. To fit the one-shot schematic to the large port, select:

   ```
   WIND* FIT
   ```

   Your screen should look like this:



Moving a Connected Part

In Chapter 4 of this manual, you used the GATHER* command to stretch the entire end of a house. In this chapter you will use GATHER* to move a portion of a logically connected drawing without losing the connections.

1. Select:

   ```
   GATHER*
   ```

2. Collect the components around the tim555 part by selecting the points (2.00,3.00) and (7.25,6.25). When this collection is moved, the direction will be down. To avoid "bending" the connection to the 'TH' port, include the entire connection line to 'TH' in the collection.
3. Select:

   ```
   AdCom
   ```

4. Select the connection line (2.00,5.00).

5. Move this collection by selecting:

   `*MDFY`

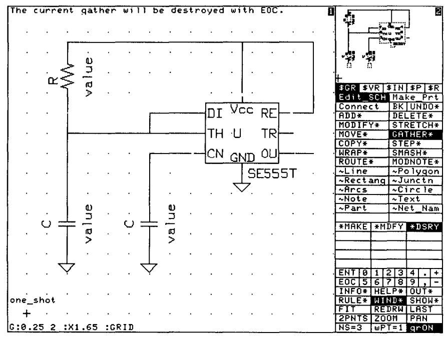
   Then select the point (5.25,5.00) as the reference point.
6. Select:

   `*MOVE`

   You will now be dynamically tracking the collection. Notice how all of the connections that cross the collection boundary are being stretched.
7. Select the point (5.25,4.50) as the new location of the collection.
8. To complete the gather operation by destroying, or un-gathering, the collected components, select:

   `*DSRY EOC`

   Your screen should look like this:



Adding Source Voltage Input

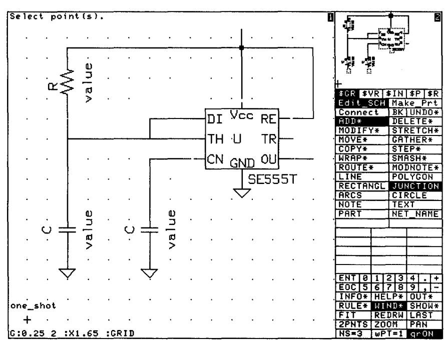
1. To provide the source voltage with input from the outside world, select:

   `STRETCH*`

2. Select the connection line which leads into "Vcc" (5.25,6.00). Then select the top endpoint of this line (5.25,6.75) and stretch it above the intersecting connection, to (5.25,7.25).
3. By the rules of logical connections, crossing lines do not make a connection. To enforce a connection at the intersection above the "Vcc" port, you must add a JUNCTION. Select:

   `ADD* JUNCTION`

   and place a junction at the intersection (5.25,6.75). Your screen should look like this:



Naming Nets

You now need to name the input and output nets. Net names are added similar to the way port names were added. The difference is that instead of selecting a logical location for the port, you must select any component in the net to associate the net name to.

1. Select:

   ```
   NET_NAME
   ```

2. To add the input net name, select the connection line you just stretched (5.25,7.00). Enter:

   ```
   'Vcc'
   ```

   Then place the net name string to the right of the connection line (5.50,7.25).

3. Add the output net name by selecting:

   ```
   *AGIN
   ```

4. Select the connection line which is hooked up to the "OU" port (7.00,4.25). Enter:

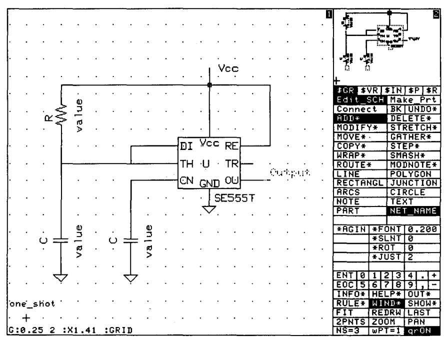
   ```
   'Output'
   ```

   and place the net name at (7.00,4.25).

5. Notice that the string you just added ('Output') is not entirely contained in the port, so select:

   ```
   FIT
   ```

   Your screen should look like this:



Adding an Outline

1. It would be convenient to include the drawing number and date of creation in the material list that you generate for this part. To have EGS do this automatically, select:

   ```
   PART
   ```

2. Enter the part name:

   ```
   id
   ```

   and place the part at (5.00,1.50). Two associated notes, 'drawing no.' and 'date', are automatically added to your drawing.

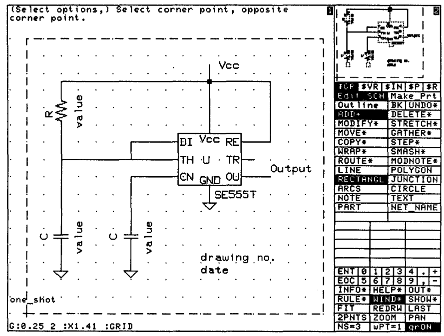
3. Before you modify these notes, first add an outline around the schematic. Change the layer mode to Outline by selecting:

   `Connect *Outline`

4. Then add a rectangle around the drawing by selecting:

   `RECTANGL`

   and selecting two points (such as 0.00,0.00 and 8.50,8.00). Your outline should look like this:



Modifying a Note String

Your next task is to modify the default reference designators, the passive component values, the drawing number, and the date strings, to reflect the actual values for this schematic.

1. Select:

   `MODNOTE*`

2. Select the reference designator for the left capacitor, 'C', at (0.50,2.25). The 'C' appears on your alpha screen. Use the editing softkeys to change the 'C' to 'C1'. Press (Return).
3. To change the reference designator on the other capacitor, select:

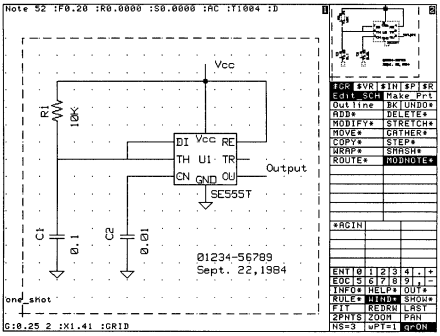
   `*AGIN`

   and then select the 'C' for it at (2.50,2.25). The first 'C1' you changed is un-highlighted and the second 'C' appears on the alpha screen. Change this one to 'C2' as you did before.

4. Repeat this process for the rest of the strings as defined in the following table:

Default String	Approx. Location	New String
'R'	0.75,5.75	'R1'
'U'	5.00,4.50	'U1'
'value'	1.50,5.75	'10K'
'value'	1.50,2.00	'0.1'
'value'	3.50,2.00	'0.01'
'drawing no.'	6.25,1.75	'01234-56789'
'date'	5.50,1.25	'Sep. 22,1984'

Your screen should look like this:



You have now completed the schematic for the one-shot. Before you continue with this example, save your drawing by selecting:

```
OUT* SAVE* OLD
```

You may also plot this schematic if you want, by using the PLOT* command as you did earlier.

BREAK TIME

Relax, take a break, and review how to create a schematic. In this section you:

- Added connections.
- Moved a connection.
- Named the nets.
- Added an outline.
- Modified a note string.

Making Connection and Material Lists

In this section you will prepare and create a connection list and a material list from the schematic you just created. In preparing the material list you will add the tim555 part to the parts file.

Preparing a Connection List File

Before you create the connection list file, review some rules for logical connection listing:

- Connections touching a port are connected.
- Endpoints of a connection touching another connection are connected.
- Crossing connections are not connected unless a JUNCTION is placed at the intersection.
- Connection nets with the same associated net name are connected.

An example of the last case are the grounds. Each ground contains an associated note 'GND', which is tagged with the appropriate value to tell the connection lister it is a net name. In this way the connection lister determines that all of the ground symbols are connected. You cannot see these net names in your drawing because they were added to the part in an undisplayed manner.

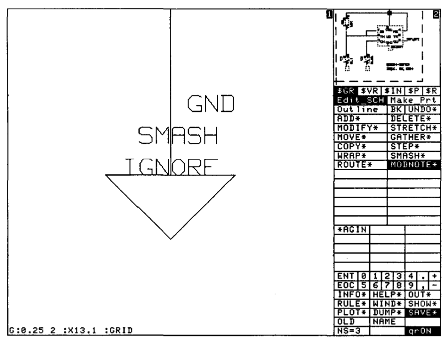
1. To see the net names, enter:

   ```
   WINDOW :A;
   ```

   You can now see several strings of text around the ground parts.

2. Use 2PNTS to zoom in on one of the grounds (we suggest using the points 2.50,0.75 and 3.50,1.75).

   Your screen should look like this:



You can now see the 'GND' net name string. 'IGNORE' is a reference designator string that tells the material list post-processor to ignore this part. 'SMASH' is another reference designator that tells the connection lister to smash an extra level of hierarchy in this part. See the connection listing chapter of *Understanding HP EGS* for more details about these conventions.

3. Turn off these note strings by entering:

   ```
   WINDOW :A;
   ```

4. Fit the drawing to the large port by entering:

   ```
   FIT
   ```

Archiving the Schematic

Your drawing must be archived using the :P and :M options before the Connection Lister can process it.

1. Enter:

```
ARCHIVE :P :M one_shot;
```

2. To create the logical connection list, you must first exit the Schematic Drawing personality. Enter:

```
Quit;
```

to return to the Electrical Engineering menu in the manager, which now looks like this:

```
Use arrow keys to move cursor to desired function.
Select function with <RETURN> or <ENTER>.
    RETURN To Main Menu
    Printed Circuit Board Layout Editor
  ->Electrical Schematic Drawing Editor

    EE Parts File Editor
    EE Material Lister
    Convert Old EE Parts Files

    Printed Circuit Board Connection Lister
    Electrical Schematic Connection Lister
    Compare Connection Lists
    Merge Connection Lists
    Generate Rat's Nest

    PHOTOPLOT/DRILL Post-processor
```

Making a Logical Port Connection List

1. To enter the Electrical Schematic Connection Lister, select:

```
Electrical Schematic Connection Lister
```

2. The following text appears briefly on the screen:

```
Loading 'EWRC:verify.CODE'
Enter MESSAGE FILE name? EWRC:con
Initialization in progress
```

The first prompt asks for the name of the message file that tells the system how to create a connection list. As the previous example illustrates, the default message file, EWRC:con, is provided by the system.

Next the Connection Lister prompts for the name of the file containing default values to be used. HP EGS provides the file name, EWRC:schphy.

```
Enter DEFAULT FILE name? EWRC:schphy
```

3. The Connection Lister prompts for the name of the archived drawing file that you want to process:

```
Enter ARCHIVE FILE name?
```

When you enter the name of the archive file, it is not necessary to include the suffix `_a`. If you are using an SRM, type the full pathname of the file. If you are using local mass storage, type:

```
DRAW:one_shot
```

4. The Connection Lister prompts for the destination of the connection list.

```
Enter OUTPUT DESTINATION?
```

If you press **Return** the Connection Lister output is placed in the same volume as the input file (in this case DRAW:). If you want the output in another volume, you can enter the volume name now. You want the output in DRAW:, so press **Return**.

5. The system presents a menu listing the available Connection Lister options:

```
Use the Arrow Keys to position cursor.
Press RETURN or ENTER to edit a field.

  ->QUIT
    Re-set to Default File          = EWRC:schphy.ASC
    Physical or Logical Rules       = Logical
    Flat or Hierarchical List       = Hierarchical, 32767
    Reference Designator Paths      = External only
    Smash to Logical Level          = 0
    Components to Include           = #E -E7 -E51
    Net Properties Tag List         =
    Port Properties Tag List        =
    Instance Properties Tag List    = 1004
    Net or Instance List Order      = Net
    Output Destination              = DRAW:
    Archive File to Process         = DRAW:one_shot_a
    RUN Connection Lister
```

On the right side of the menu the Connection Lister displays the values found in the default file (EWRC:schphy) for each option and the names of the output volume and the archive file you entered in steps 3 and 4.

The default file for schematic connection lists, EWRC:schphy.ASC, is used to produce connection lists for physical ports; to generate a logical port connection list, you must change the default file to EWRC:schlog.ASC.

6. Change to the correct default file by positioning the -> at `Re-set to Default File` and pressing **Return**. Use the cursor control keys to position the cursor at the p in schphy. Type log over phy and press **Return**.

Notice that the values for `Components to Include` has changed:

```
Use the Arrow Keys to position cursor.
Press RETURN or ENTER to edit a field.

 ->QUIT
   Re-set to Default File          = EWRC:schlog.ASC
   Physical or Logical Rules       = Logical
   Flat or Hierarchical List       = Hierarchical, 32767
   Reference Designator Paths      = External only
   Smash to Logical Level          = 0
   Components to Include           = #E -E7 -E50
   Net Properties Tag List         =
   Port Properties Tag List        =
   Instance Properties Tag List    = 1004
   Net or Instance List Order      = Net
   Output Destination              = DRAW:
   Archive File to Process         = DRAW:one_shot_a
   RUN Connection Lister
```

Because you can use the values supplied by the default file for the remainder of the menu items, you are ready to run the Connection Lister.

7. Move the -> to `RUN Connection Lister` and press **Return**.

The Connection Lister begins by displaying the modified option menu to verify the options you have selected. If you wanted to make another change, you could press **Stop** to return to the menu. In this case, let the Connection Lister continue.

The Connection Lister will output dots on the screen to indicate that it is processing the archive file.

When the connection list is complete the following message is displayed (your values for memory available and used may be different):

```
Connection list generated.  2 errors

Starting memory available           = 750798
Total memory used                   = 99318
Overhead memory used                = 58968
Drawing memory used                 = 20994
Processing memory used              = 19356

All processing complete.

Press <space> to continue.
```

Two files have been created. The first, one_shot_c, is the connection list. The second, one_shot_e, is an error file.

The connection list file appears as follows. (You can look at your file later using the text Editor.)

```
one_shot 1
ERROR 1 Part has no ports.

1
      tim555|U1|DI|SE555T|
      tim555|U1|TH|SE555T|
      resis|R1|1|10K|
      cap|C1|2|0.1|

2
      tim555|U1|CN|SE555T|
      cap|C2|2|0.01|

3  GND
      cap|C2|1|0.01|
      tim555|U1|GND|SE555T|
      cap|C1|1|0.1|

4  Output
      tim555|U1|OU|SE555T|

5  Vcc
      tim555|U1|Vcc|SE555T|
      tim555|U1|RE|SE555T|
      resis|R1|2|10K|

ERROR 2 Part has unconnected ports.
```

The header in the file contains the file name, logical level and a list of errors generated. Following the header is a numbered list of the connection nets that were found. For the last three nets the numbers are followed by the net names you placed in the drawing.

Each connection in a net is identified by one line in the list. For example, the first connection in the 'Vcc' net is written as 'tim555/U1/Vcc/SE555T/'. This notation indicates a connection in the tim555 part, with a reference designator of U1. The actual port has a logical name of Vcc and a part type (or value in some cases) of SE555T. The above list is an example of the file format, so you can see that a program could be written to reformat the file any way you want.

At the end of the connection list errors may be listed. This connection list indicates that two errors were generated, but it is not obvious where these errors occurred in the schematic. The connection list error file addresses this issue. As shown below, this file contains ADD NOTE commands in which the note string contains the error number (for example, 2) that corresponds to the list file, as well as the error value which indicates what is wrong (CONERR22). Refer to *Understanding HP EGS* for definitions of the error values. You can use the INPUT command to automatically add the error notes to your drawing so that you can see where the error occurred.

```
ADD N255 '1 CONERR16' 5.0000000000,1.5000000000;
ADD N255 '2 CONERR22' 4.0000000000,3.5000000000;
```

Return to the Schematic Drawing Editor to add the error notes to your drawing.

1. Press the space bar to return to the Connection Lister menu.
2. Move the -> to `QUIT` and press **Return** to return to the Electrical Engineering menu which now looks like this:

```
Use arrow keys to move cursor to desired function.
Select function with <RETURN> or <ENTER>.
    RETURN To Main Menu
    Printed Circuit Board Layout Editor
    Electrical Schematic Drawing Editor

    EE Parts File Editor
    EE Material Lister
    Convert Old EE Parts Files

    Printed Circuit Board Connection Lister
  ->Electrical Schematic Connection Lister
    Compare Connection Lists
    Merge Connection Lists
    Generate Rat's Nest

    PHOTOPLOT/DRILL Post-processor
```

3. Return to the Electrical Schematic Drawing Editor by selecting:

```
Electrical Schematic Drawing Editor
```

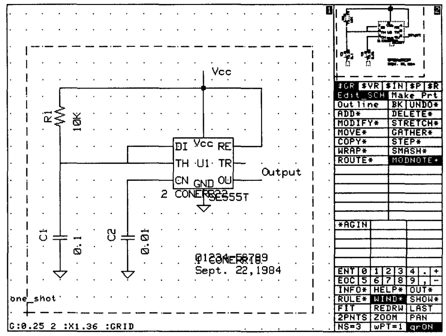
4. Prefix to your storage volume. To edit the schematic, enter:

```
EDIT one_shot;
```

5. To determine the location of the problems in your schematic, input the error file by entering:

```
INPUT one_shot_e;
```

Your screen should look like this:



The note '1 CONERR16' appears in the lower right corner of your schematic, on top of the drawing number. The note '2CONERR22' appears in the center of the drawing, at the bottom of the tim555 part.

'CONERR16' corresponds to the 'Part has no ports' error. You can ignore this error, because the part identification intentionally has no ports.

'CONERR22' corresponds to the 'Part has unconnected ports' error. You can also ignore this error, because the 'TR' (trigger) port was intentionally left disconnected.

6. To turn off the error file, enter:

```
SHOW* -E255;
```

To prepare for the next lesson, you need to produce another connection list that has physical port names instead of logical port names.

7. To create the physical port connection list, you must first exit the Schematic Drawing personality. Enter:

```
Quit;
```

to return to the Electrical Engineering menu in the manager, which now looks like this:

```
Use arrow keys to move cursor to desired function.
Select function with <RETURN> or <ENTER>.
    RETURN To Main Menu
    Printed Circuit Board Layout Editor
  ->Electrical Schematic Drawing Editor

    EE Parts File Editor
    EE Material Lister
    Convert Old EE Parts Files

    Printed Circuit Board Connection Lister
    Electrical Schematic Connection Lister
    Compare Connection Lists
    Merge Connection Lists
    Generate Rat's Nest

    PHOTOPLOT/DRILL Post-processor
```

Making a Physical Port Connection List

1. To enter the Electrical Schematic Connection Lister, select:

```
Electrical Schematic Connection Lister
```

2. The following text appears briefly on the screen:

```
Loading 'EWRC:verify.code'
Enter MESSAGE FILE name? EWRC:con
Initialization in progress
```

The first prompt asks for the name of the message file that tells the system how to create a connection list. As the previous example illustrates, the default message file, EWRC:con, is provided by the system.

Next the Connection Lister prompts for the name of the file containing default values to be used. HP EGS provides the file name, EWRC:schphy.

```
Enter DEFAULT FILE name? EWRC:schphy
```

3. The Connection Lister prompts for the name of the archived drawing file that you want to process:

```
Enter ARCHIVE FILE name?
```

When you enter the name of the archive file, it is not necessary to include the suffix `_a`. If you are using an SRM, type the full pathname of the file. If you are using local mass storage, type:

```
DRAW:one_shot
```

4. The Connections Lister prompts for the destination of the connection list.

```
Enter OUTPUT DESTINATION?
```

If you press (Return) the Connection Lister output is placed in the same volume as the input file (in this case DRAW:). If you want the output in another volume, you can enter the volume name now. You want the output to go to DRAW:, so press (Return).

5. The system presents a menu listing the available Connection Lister options:

```
Use the Arrow Keys to position cursor.
Press RETURN or ENTER to edit a field.

  ->QUIT
    Re-set to Default File          = EWRC:schphy.ASC
    Physical or Logical Rules       = Logical
    Flat or Hierarchical List       = Hierarchical,32767
    Reference Designator Paths      = External only
    Smash to Logical Level          = 0
    Components to Include           = #E -E7 -E51
    Net Properties Tag List         =
    Port Properties Tag List        =
    Instance Properties Tag List    = 1004
    Net or Instance List Order      = Net
    Output Destination              = DRAW:
    Archive File to Process         = DRAW:one_shot_a
    RUN Connection Lister
```

On the right side of the menu the Connection Lister displays the values found in the default file (EWRC:schphy) for each option and the names of the output volume and the archive file you entered in steps 3 and 4.

Because this example uses the values displayed in the menu for generating a connection list of physical ports, you are ready to run the Connection Lister.

6. Move the -> to `RUN Connection Lister` and press (Return).

The Connection Lister begins by displaying the option menu to verify any changes you may have made. In this case, you did not make any changes so the menu is the same. You could still change the menu by pressing (Stop) to return to the menu, but let the Connection Lister continue.

The Connection Lister will output dots on the screen to indicate it is processing the archive file.

When the connections list is complete the following message is displayed (your values for memory available and used may be different):

```
Connection list generated.  2 errors

Starting memory available                 = 750798
Total memory used                         = 99318
Overhead memory used                      = 58968
Drawing memory used                       = 20994
Processing memory used                    = 19356

All processing complete.

Press <space> to continue.
```

When HP EGS prompts you to overwrite files, respond by pressing Y. The connection list should look like this:

```
one_shot 1
ERROR 1 Part has no ports.

1
    tim555|U1|7|SE555T|
    tim555|U1|6|SE555T|
    resis|R1|?1|10K|
    cap|C1|?2|0.1|

2
    tim555|U1||5|SE555T|
    cap|C2|?2|0.01|

3  GND
    cap|C2|?1|0.01|
    tim555|U1|1|SE555T|
    cap|C1|1|0.1|

4  Output
    tim555|U1|3|SE555T|

5  Vcc
    tim555|U1|8|SE555T|
    tim555|U1|4|SE555T|
    resis|R1|?2|10K|

ERROR 2 Part has unconnected ports.
```

Note that the only difference between this connection list and the last connection list is that the logical port names have been replaced by physical port names.

To create a material list, return to the Schematic Drawing personality:

1. Press the space bar to return to the Connection Lister menu.
2. Move the -> to `QUIT` and press **Return** to return to the Electrical Engineering menu which now looks like this:

```
Use arrow keys to move cursor to desired function.
Select function with <RETURN> or <ENTER>.
    RETURN To Main Menu
    Printed Circuit Board Layout Editor
    Electrical Schematic Drawing Editor

    EE Parts File Editor
    EE Material Lister
    Convert Old EE Parts Files

    Printed Circuit Board Connection Lister
  ->Electrical Schematic Connection Lister
    Compare Connection Lists
    Merge Connection Lists
    Generate Rat's Nest

    PHOTOPLOT/DRILL Post-processor
```

3. Return to the Electrical Schematic Drawing Editor by selecting:

```
Electrical Schematic Drawing Editor
```

4. Prefix to your storage volume. To edit the schematic, enter:

```
EDIT one_shot;
```

Creating a Material List

1. To produce a material list, enter:

```
LIST_MAT
```

The display shows:

```
ENTer volume and file name for the source drawing file.
```

2. Enter:

```
one_shot
```

The display shows:

```
ENTer destination volume, PRINTER: or CONSOLE:, then EOC.
```

Destination volume names for material lists are treated the same as for connection lists. The only difference is that the file which is produced will have a file name of one_shot_m (note the suffix of _m).

3. For this exercise put the file in the prefixed volume by selecting:

```
EOC
```

The material list file appears as follows. (You can look at your file later using the text Editor.)

```
id      |Sept. 22,1984|01234-56789|
gndx    ||SMASH/IGNORE|
tim555  |SE55T|U1|
gndx    ||SMASH/IGNORE|
gndx    ||SMASH/IGNORE|
resis   |10K|R2|
cap     |0.01|C2|
cap     |0.1|C1|
```

This file contains a list of individual parts used in the schematic you created. Each line represents a separate part where the first entry is the library part name, the next entry is the part type, if one exists, and the last entry is the reference designator. The above list is in the exact format of the file, so you can see that a post-processor can be generated to manipulate this file any way you want.

4. To process this preliminary material list file to create a true bill of materials, you must first exit the Schematic Drawing personality. Enter:

```
QUIT;
```

to return to the Electrical Engineering menu in the manager, which now looks like this:

```
Use arrow keys to move cursor to desired function.
Select function with <RETURN> or <ENTER>.
    RETURN To Main Menu
    Printed Circuit Board Layout Editor
  ->Electrical Schematic Drawing Editor

    EE Parts File Editor
    EE Material Lister
    Convert Old EE Parts Files

    Printed Circuit Board Connection Lister
    Electrical Schematic Connection Lister
    Compare Connection Lists
    Merge Connection Lists
    Generate Rat's Nest

    PHOTOPLOT/DRILL Post-processor
```

5. Add the tim555 part to the parts file so that it will be adequately documented in the final bill of materials. Select:

```
EE Parts File Editor
```

Soon the following prompt appears:

```
Should the hpegs volumes be searched for parts files? (y/n)
```

6. Answer this prompt by pressing (Y). The part file editor then searches all of the HP EGS volumes for parts files. Wait for the following prompt to appear:

```
Enter the name of another volume to be searched for parts files.
(<RETURN> if no more)
```

7. You don't need to search any other volumes, so press (Return).

Next you will select the part file to change, and then add the new part to it. The following menu should now be displayed:

```
Use arrow keys to move marker.
Select with <RETURN> or <ENTER>.

  ->QUIT
    CREATE A New Parts File
    EWEE:eepart_pf
    EWME:mepart_pf
```

8. Select:

```
EWEE:eepart_pf
```

This enables you to add the 555 timer to this file. The following menu appears:

```
Use arrow keys to move marker.
Select with <RETURN> or <ENTER>.

->QUIT
  ADD A New Part
  CHANGE An Existing Part
  DELETE An Existing Part
  LIST The Existing Parts
  SELECT A New Parts File To Edit
```

9. Select:

```
ADD A New Part
```

The system prompts you for all of the data fields that need to be added for the new part. These data fields are defined as follows:

Data Field	**Description**
Part Name	The part name must be spelled exactly as it was when the part was saved. Do not include the _d portion of the part name.
Part Type	The part type must be spelled exactly as it was when the part was created.
List Flag	The list flag must be either a 'y' (yes) or an 'n' (no). It tells the material list post-processor whether to list the part in the final bill of materials. (The ground part has an 'n' here.)
Stock Number	This is your company's stock number for the part.
Description	This is any description you want included in the bill of materials for this part.

10. Answer each of the prompts as follows:

```
Enter the Part Name
 Max. length is 8: tim555
Enter the Part Type
 Max. length is 16: SE555T
Enter the list flag
 Max. length is 1: y
Enter the Stock Number
 Max. length is 11: 1820-1437
Enter the Description
 Max. length is 40: SE555T Monolithic Timer
```

11. Quit the parts file editor by selecting:

    ```
    QUIT
    ```

    The parts file which you changed, EWEE:eeparts_pf, is now automatically saved for you. The Electrical Engineering Menu is redisplayed.

Creating a Bill of Materials

Now you are ready to create the final bill of materials. The Electrical Engineering menu should look like this:

```
Use arrow keys to move cursor to desired function.
Select function with <RETURN> or <ENTER>.
    RETURN To Main Menu
    Printed Circuit Board Layout Editor
    Electrical Schematic Drawing Editor

  ->EE Parts File Editor
    EE Material Lister
    Convert Old EE Parts Files

    Printed Circuit Board Connection Lister
    Electrical Schematic Connection Lister
    Compare Connection Lists
    Merge Connection Lists
    Generate Rat's Nest

    PHOTOPLOT/DRILL Post-processor
```

1. Select:

   ```
   EE Material Lister
   ```

   Soon the following prompt appears:

   ```
   Enter the name of another volume to be searched for parts files.
   (<RETURN> if no more)
   ```

   You need to search a system volume, so press Return. Wait for the following prompt to appear:

   ```
   Do you want to search the system volumes (y/n)?
   ```

2. Answer yes by pressing Y. The material list post-processor then searches all of the HP EGS volumes for parts files. If some volumes are not online the system prompts for a space to continue.

3. Now you can produce two bills of materials for your schematic. The first will be sorted by reference designator and the second will be sorted by stock number. The following menu should now be displayed:

```
Use arrow keys and <RETURN> to select parts files.

  -> RETURN To The Material List Menu
    USE ALL Parts files
    EWEE:eepart_pf
    EWME:mepart_pf
    RESTART - Selecting Parts files
    DONE - Selecting Parts files
```

4. Select:

```
EWEE:eepart_pf
```

   so that your material list will be based on this parts file. An asterisk (*) appears next to this file name to indicate that it will be used.

5. Select:

```
DONE - Selecting Parts files
```

   The following prompt appears:

```
Enter the name of the FIRST LIST MAT file (<enter> if none)
```

6. Enter the name of your file. Remember to include the volume you were prefixed to:

```
EWYOU:one_shot
```

   (The _m is automatically appended.) The following prompt appears:

```
Enter the name of the NEXT LIST MAT file (<enter> if no more)
```

7. Press **Return**.

   The following prompt appears:

```
Do you want to sort by Reference or Stock number (<enter> to exit)?
```

8. Press **R** for reference. The following prompt appears:

```
Where do you want the output written (Crt, Printer or File)?
```

9. Decide where you want to list the file and press C, P, or F as appropriate. The system then prompts you for a one-line header. Enter a header. Your material list will look like this:

```
                              Material List
Sept. 22,1984
01234-56789
Your Header                                              25-Sep-84 Page 1

REFERENCE                                   STOCK NO  DESCRIPTION
-----------------------------------------------------------------------------
C1                                          1860-1110 0.1 microfarad Capacitor
C2                                          1860-1101 0.01 microfarad Capacitor
R1                                          1850-1210 10K ohm Resistor
U1                                          1820-1437 SE555T Monolithic Timer
```

When the list has been generated the following prompt appears:

```
Do you want to sort by Reference or Stock number (<enter> to exit)?
```

10. Press S for stock number, and continue as you did earlier to produce a bill of materials sorted by stock number. Your completed bill of materials should look similar to this:

```
                                 Material List
Sept. 22,1984
01234-56789
Your Header                                              25-Sep-84 Page 1

STOCK NO   QTY DESCRIPTION                REFERENCE
-----------------------------------------------------------------------------
1820-1437    1 SE555T Monolithic Timer    U1
1850-1210    1 10K ohm Resistor           R1
1860-1101    1 0.01 microfarad Capacitor  C2
1860-1110    1 0.1 microfarad Capacitor   C1
```

Notice that the date and drawing number are included in each of the material lists. The following prompt appears:

```
Do you want to sort by Reference or Stock number (<RETURN> to exit)?
```

11. To terminate the material list generation process, press Return. The following prompt appears:

```
Enter the name of the FIRST LIST MAT file (<RETURN> if none)
```

12. Press (Return). The following menu appears:

```
Use arrow keys and <RETURN> to select parts files.

-> RETURN To The Material List Menu
   USE ALL Parts files
   EWEE:eepart_pf
   EWME:mepart_pf
   RESTART - Selecting Parts files
   DONE - Selecting Parts files
```

13. Select:

```
RETURN To The Material List Menu
```

BREAK TIME

Relax, stretch, and take a break. You are now done with the schematic drawing example. In this section you:

- Prepared a connection list.
- Created a connection list.
- Created a material list.
- Created a bill of materials.

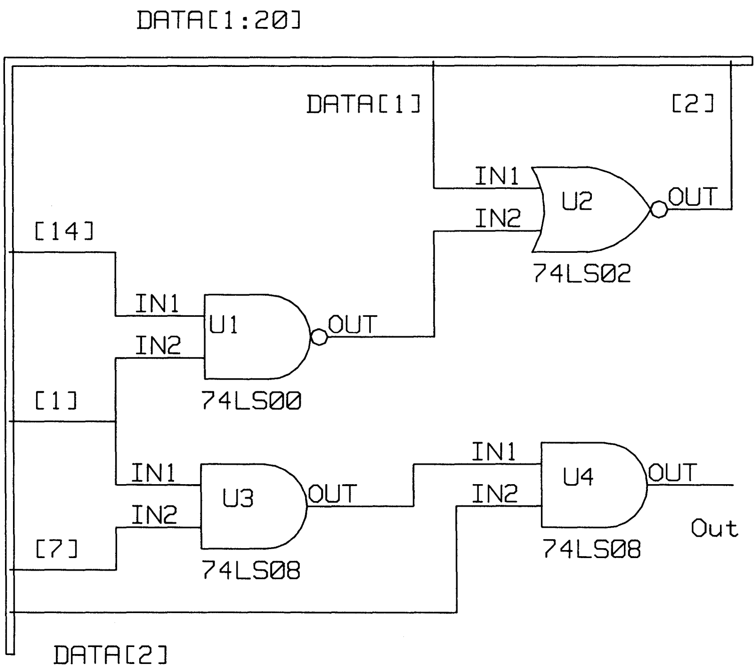
Using Buslines for Connections

To add a busline to a schematic you must change the layering mode to Busline. You can then add buslines the same way you enter other connections. The primitives you add can (and should) have width.

What makes a busline significantly different from a normal connection is its ability to carry multiple connections. This ability is specified through the use of conventions for naming buslines and the nets that connect to them. These conventions are as follows:

- For The Busline: All buslines must have a net name. The name of a bus is in the form NAME[1:n], where NAME is your name for the bus and n is the maximum number of discrete nets carried in the bus.
- For Connected Nets: All nets that are connected to a bus must have a net name. The name must be in the form NAME[k] or [k], where NAME is the name of the bus that the net is connected to and k is the value of the discrete bus net that is a part of the identified net. (k must be between 1 and n.)

The following figure is a section of a schematic that contains a bus. The bus is named DATA[1:20], which means that it can only contain 20 nets.



Example Schematic Using a Bus

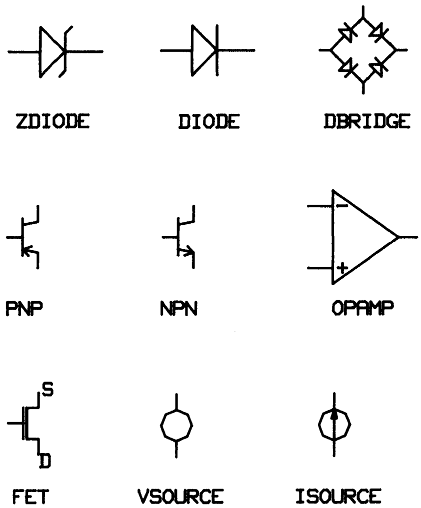
Listed below is the output from a connection list of the logical ports which was generated from the schematic in the previous figure. Only the nets which were used in the busline are listed.

```
|bus_ex 1

1
    nor2|U2|IN2|74LS02|
    nand2|U1|OUT|74LS00|

2
    and2|U4|IN1|74LS08|
    and2|U3|OUT|74LS08|

3 DATA[1]
    nor2|U2|IN1|74LS02|
    nand2|U1|IN2|74LS00|
    and2|U3|IN1|74LS08|

4 DATA[2]
    nor2|U2|OUT|74LS02|
    and2|U4|IN2|74LS08|

5 DATA[7]
    and2|U3|IN2|74LS08|

6 DATA[14]
    nand2|U1|IN1|74LS00|

7 Out
    and2|U4|OUT|74LS08|
```

Schematic Library Parts

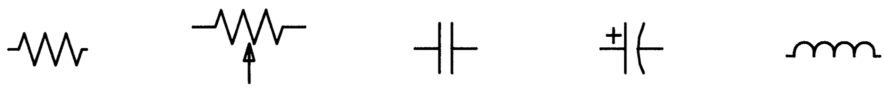
To help you start drawing schematics, a set of library parts is provided with this personality. These library parts are stored in the default volume, EWEE. The conventions used to construct these library parts are the same ones you used to create the tim555 library part.

The library parts supplied with this personality can be grouped into six general categories. The members of each group are listed below each figure.



Active Devices

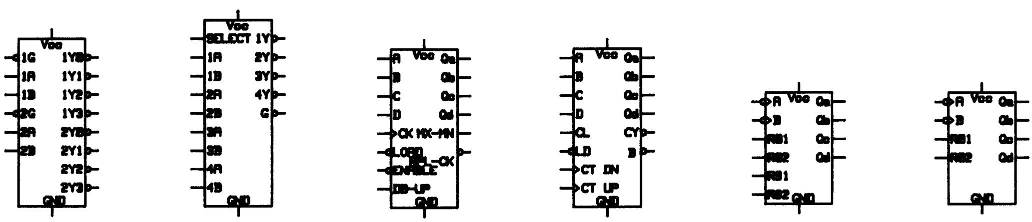
Library Part Name	Description
pnp	PNP transistor
npn	NPN transistor
fet	FET transistor
opamp	Operational amplifier
diode	Diode
zdiode	Zener diode
vsource	Voltage source
isource	Current source
dbridge	Diode bridge



RESIS POT CAP PCAP IND

Passive Devices

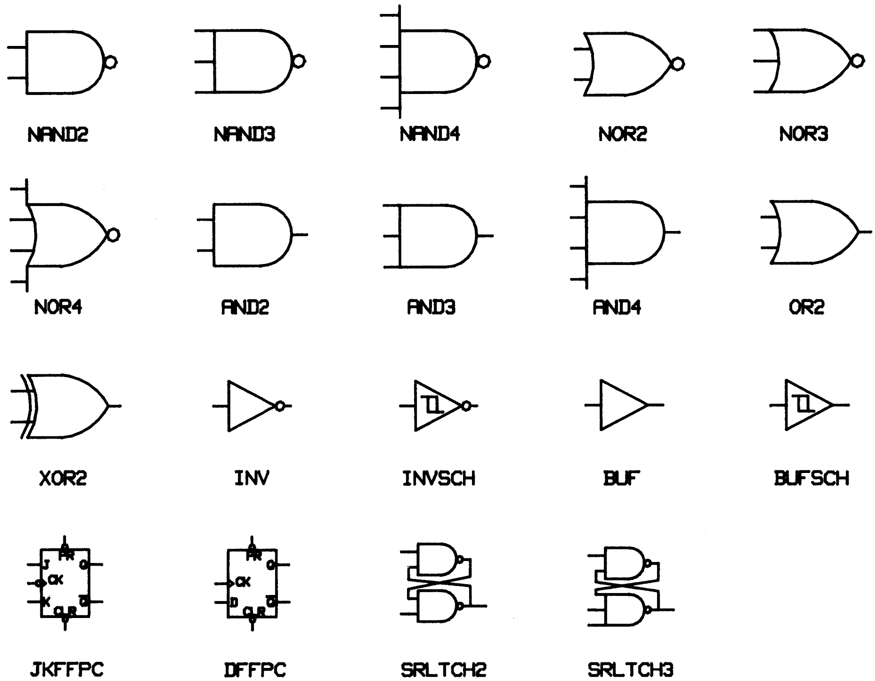
Library Part Name	Description
resis	Resistor
pot	Potentiometer
cap	Capacitor
pcap	Polarized capacitor
ind	Inductor



DCOD139 DSEL258 CNT190 CNT193 CNT290 CNT293

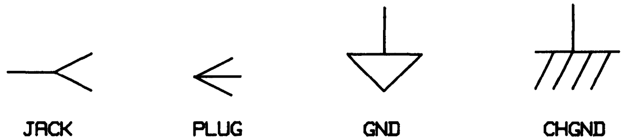
Complete Chips

Library Part Name	Description
cnt290	4-bit binary and decade counter (type-SN74LS290)
cnt293	4-bit binary and decade counter (type-SN74LS293)
cnt190	4-bit binary and decade up/down synchronous counter (type-SN74LS190)
cnt193	4-bit binary and decade up/down synchronous counter (type-SN74LS193)
dcod139	Decoder/Multiplexer (type-SN74LS139)
dsel258	Quadruple 2-line-to-1-line Data Selector/Multiplexer (type-SN74LS258)



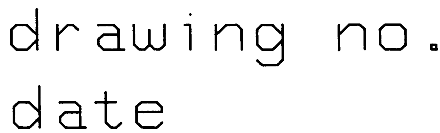
Digital Devices

Library Part Name	Description
and2	2 input AND gate
and3	3 input AND gate
and4	4 input AND gate
nand2	2 input NAND gate
nand3	3 input NAND gate
nand4	4 input NAND gate
or2	2 input OR gate
xor2	2 input Exclusive OR gate
nor2	2 input NOR gate
nor3	3 input NOR gate
nor4	4 input NOR gate
inv	Inverter
invsch	Schmitt Trigger Inverter
buf	Buffer
bufsch	Schmitt Trigger Buffer
jkffpc	JK Flip-Flop with Preset and Clear
dffpc	D Flip-Flop with Preset and Clear
srltch2	SR Latch (2 input latch type-SN74LS279)
srltch3	SR Latch (3 input latch type-SN74LS279)



Drawing Aids

Library Part Name	Description
gnd	Ground (system ground)
chgnd	Chassis ground
jack	Edge connector, in
plug	Edge connector, out



System's Library Part

Library Part Name	Description
id	Identification part

The library parts were created using the units of this personality and a display grid of 0.05,2 (as you did). All of the ports were added on 0.25 centimeter centers, so a display grid of 0.25,2 is recommended when using these parts in a drawing. The library parts supplied with this personality contain default pieces of text for material and connection listing. Like the part you created, these include:

- Logical Ports
- Physical Ports
- Part Type
- Reference Designator

All the furnished library parts have been saved at logical level 0, so that the LIST_ MAT command can distinguish base level parts from groups and wraps.

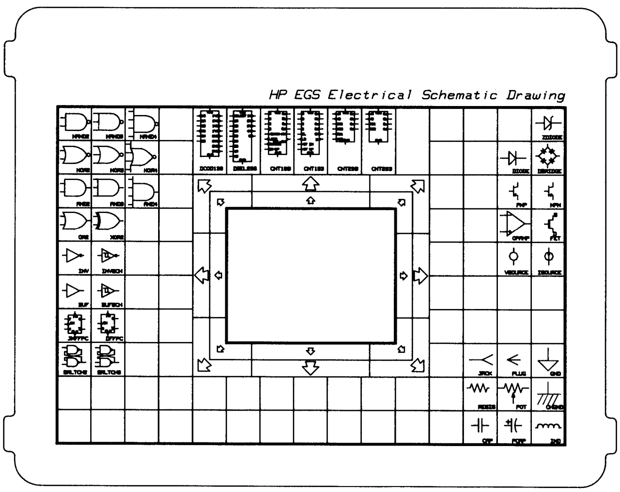
The gnd and chgnd library parts contain net names of 'GND' and 'CHGND' respectively. As a result, any net that is connected to one of these parts is automatically named.

The system library part, id, is a component whose sole use is to relay the drawing number and the date of creation to the material list. You may modify the associated notes in this component to change their font size and the characters that form the notes.

If the system library part is not used in your schematic, no space is allocated for it in the material list header.

The Schematic Tablet Menu

The tablet menu provided for use with the Electrical Schematic Drawing personality is shown in the following figure:



Electrical Schematic Drawing Tablet Menu

This menu is set up for the tablet rather than the screen so that you can select library parts without having to use the *SCRN option in the ADD* PART command. This tablet menu is convenient because it gives you a picture of the parts as well as the name.

Notice that the tablet menu has some port manipulation commands around the screen tracking block. These commands enable you to pan the large port in the direction of the arrows with a single stroke. The large arrows represent full window pans and the small arrows represent half window pans.

See an appendix in this manual to plot this tablet menu, or another appendix to add a new part to this tablet menu.

Printed Circuit Board Layout

Chapter 9

Introduction

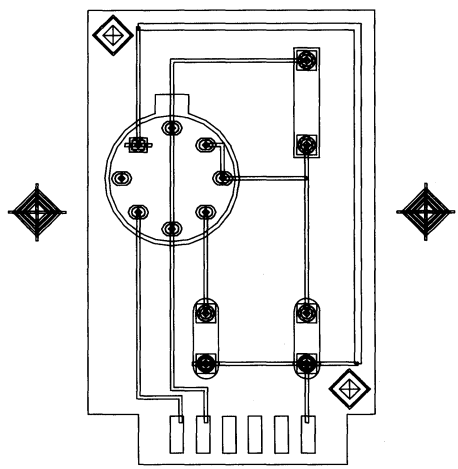
In this chapter you will learn the differences between the Printed Circuit Board Layout personality and the General Drawing personality. Then you will use the Printed Circuit Board Layout personality to create the printed circuit board representing the one-shot schematic you drew in Chapter 8.

Note

Before you begin the lessons in this chapter, be sure you have read Part II and Chapter 8 of this manual. They contain basic necessary information that is not repeated in this chapter.

You will also create a new library part representing an eight lead to5 package for the SE555T timer. When the printed circuit board is finished you will generate a connection list and a material list from the drawing. Finally, you will use the Compare Connect List post-processor to check that the printed circuit board is an exact representation of the one-shot schematic.

The following figure shows the finished drawing of the Printed Circuit board representing the one-shot schematic.



Completed Printed Circuit Board

Understanding Printed Circuit Board Layout

To use the Printed Circuit Board Layout personality you need to understand the basic differences between this personality and the General Drawing personality, which you should already understand. We will approach these differences by first examining how the process file has been changed and then how the macro file has been changed.

The Process File

The HP EGS process file defines the system units, the default parameters, and the layer definitions for the data in your drawing. For this personality, the basic unit of measurement is the mil (0.001 inch), with 2 system points per mil. This means that the location of every point in your printed circuit board can be specified to 0.5 mil. By changing the user grid setting and snapping modes, you may force points to fall on some multiple of the basic resolution.

Several defaults are different due to this change in units. For example, the default font size is 100 mils, where it was 10 millimetres in General Drawing.

The changes to two defaults values, LOCK and RESOLUTION, pose more interesting differences. LOCK, which is the default lock angle, is set to 45 degrees in this personality. (LOCK is 0 degrees in General Drawing.) This means that when you are adding straight segments of lines and polygons, HP EGS informs you if they are not at intervals of 45 degrees.

RESOLUTION, which is the default resolution angle for arcs and circles, is set to 15 degrees. (RESOLUTION is 0 degrees in General Drawing.) Circles and arcs appear rougher on your display. For example, a 360-degree circle appears to have 24 sides. For more information on the effects of these commands, see the *HP EGS Syntax Reference*.

As you learned by using the General Drawing personality, all of the components in your drawing are placed on discrete layers. Recall that each layer is like a transparent sheet which exactly overlays the other layers. Each layer specifies the color and line-type of the component as well as the pen it will be plotted with. The Printed Circuit Board Layout personality takes advantage of this layering capability by separating various aspects of the drawing on different layers. The definition of the layers is different than in General Drawing as follows:

Layer	Use	Color	Line Type
0 INSTBOUN	Instances	White	Short Dotted
1 PDRILL	Circles for plated holes	Blue	Solid
2 BDBLANK	PC board outline/tooling hole pads for the router	Yellow	Solid
3 PENPADM	Copper on component/circuit sides of the board for pen plots with fill	Magenta	Solid
4 KEEPOUT	Keepout master for all keepout areas common to multiple layers	Cyan	Solid
5 COMPONET	Traces, pads, and other metalization on the component side of the board	Red	Solid
6 CIRCUIT	Traces, pads, and other metalization on the circuit side of the board	Green	Solid
7 FIVEVOLT	Pads and traces for inner layer power plane	Magenta	Solid
8 GROUND	Pads and traces for inner layer ground plane	Magenta	Solid
9 FIVEKO	Special keepout areas for an inner layer power plane	Cyan	Solid
10 GNDKO	Special keepout areas for an inner layer ground plane	Cyan	Solid
11 INLAYRA	Traces, pads, and other metalization on the inner layer, A	Magenta	Solid
12 INLAYRB	Traces, pads, and other metalization on the inner layer, B	Magenta	Solid
13 INLAYRC	Traces, pads, and other metalization on the inner layer, C	Magenta	Solid
14 INLAYRD	Traces, pads, and other metalization on the inner layer, D	Magenta	Solid
15 INKOA	Special keepout areas for the inner layer, A	Cyan	Solid
16 INKOB	Special keepout areas for the inner layer, B	Cyan	Solid
17 INKOC	Special keepout areas for the inner layer, C	Cyan	Solid
18 INKOD	Special keepout areas for the inner layer, D	Cyan	Solid
19 REFRENCE	Registration graphics and text for the PC fabrication shop	Yellow	Solid
20 PENSLDM	Areas on both outer layers of the board void of solder resist for pen plots with fill	Cyan	Solid

Layer	Use	Color	Line Type
21 SLDRCOMP	Areas unique to the component side of the board void of solder resist	Cyan	Solid
22 SLDRCIRC	Areas unique to the circuit side of the board void of solder resist	Cyan	Solid
23 UPDRILL	Circles for unplated holes	Blue	Solid
24 CIRCKO	Special keepout areas for the circuit side of the board	Cyan	Solid
25 COMPKO	Special keepout areas for the component side of the board	Cyan	Solid
26 ASMBDRAW	Outlines of parts for assembly drawings	Yellow	Solid
29 DISPPADM	Copper areas on both component and circuit sides of the board for the display	Magenta	Solid
30 DISPSLDM	Areas on both outer layers of the board void of solder resist for the display	Cyan	Solid
50 LOGPORTS	Logical ports	Yellow	Solid
51 PHYPORTS	Physical ports	Yellow	Solid
52 PARTTYPE	Part type	Yellow	Solid
53 REFDESIG	Reference designator	Yellow	Solid
54 NETNAME	Net name	Yellow	Solid
55 FLASHTAG	Flash aperture identifier	Green	Solid
56 DRILLTAG	Drill size	Green	Solid
255 ERRORS	Errors	White	Solid

Note the use of colors in the process file definition. Layers performing similar functions are assigned the same display color. For example, all keepout process layers are displayed with the color cyan. In this way you can quickly identify and separate the information in your board design as it is displayed.

This process file is sufficient for laying out up to 8-layer PC boards, if you follow the layout concepts described in the following sections. You can change the parameters in the process file by using such commands as UNITS and EQUATE. For more information on changing the process file, see the Customizing chapter of *Understanding HP EGS*.

Process File Layers and Physical Board Layers

Physical board layers, also called board sides, are the actual physical layers of copper on a PC board. Two or more layers separated by insulating material make up a multi-layered board. Each physical layer referred to in this manual represents a layer of copper or of solder resist. Information on one process layer may be used to construct artwork for several physical board layers.

Similarly, the artwork for one board layer may be so complex that several process layers are needed to draw the artwork. All of the graphical information on the process layers represents features of the board as viewed from the top or component side of the board.

The Basic Layers

To create a simple printed circuit board (PC board), you might need only to add drill holes and geometry-defining pads for your parts and connecting traces. This PC board would have two physical layers, the component and circuit sides of the board. You would use the following layers:

Layer	Use
0 INSTBOUN	Instances of library parts
1 PDRILL	Circles representing plated drilled holes
23 UPDRILL	Circles representing unplated drilled holes
5 COMPONET	Traces, pads, and planes for metalization on the component side of the board
6 CIRCUIT	Traces, pads, and planes for metalization on the circuit side of the board

Adding Solder Resist

If your PC board requires a solder resist mask for each outer layer of the board (component, and circuit sides), you would place components on the following layers:

Layer	Use
21 SLDRCOMP	Areas of no solder resist on the component side of the board
22 SLDRCIRC	Areas of no solder resist on the circuit side of the board

These two process layers, SLDRCOMP and SLDRCIRC, contain components that represent areas in which NO solder resist is desired. For this reason, these process layers are referred to as negative areas, or keepout areas. This contrasts with the process layers that represent the physical sides of the board, where components in the process layers represent areas in which copper is desired. These other process layers are referred to as positive areas.

Although using this subset of the process file would provide you with the artwork necessary to create a two-layer board, it causes you to lay out each individual component on each layer just as it is on the physical board layer. This works well for pads, traces, and planes that are unique to an individual layer. However, for those layers that require pads, traces, or other metalization in identical locations such as pads on the circuit side and the component side for plated-through holes, extra time is required to make the identical entry on each layer.

To avoid this duplication of work, the process file can be defined to have individual layers "bound" to a master layer. In this way the contents of the master process layer are automatically added to each of the process layers that are bound to it when you plot your drawing.

For example, you have a master process layer containing all pads that are to appear on both the component and circuit sides of the board. You would enter those pads that appear on both sides of the board in the master process layer once. Then you would enter those pads, traces, and planes that are unique to an individual board side on the individual process layer representing that board side (this process layer is bound to the master layer). You would get the final plot for the component side of the board by creating a plot which contains both the pad master process layer and the component process layer.

Similarly, you have a master process layer, the keepout master layer, that contains components representing areas that should not be covered with solder resist.

These two master layers are bound to all layers that their components are required to appear on. For example, layers 5 and 6 are bound to layer 29 via the :B option in the EQUATE command. As a result, all components found in layer 29 will be processed in layers 5 and 6 as well as layer 29.

Manufacture and Assembly Layers
Because most PC shops require that a pen plot of the board outline, the board blank, and tooling holes be provided for the router, a separate process layer, BDBLANK, for the board blank is available.

Each physical board layer requires registration marks. Another process layer, REFRENCE, is defined to contain nothing but registration marks and other reference information needed by the fabrication shop such as text and labels. The information contained in this process layer may be added to the information contained on the other individual process layers when plotting.

For PC shops that require a separate assembly drawing for your PC board, a separate process layer, ASMBDRAW, has been provided to contain outlines of parts.

Layer	Use
2 BDBLANK	PC board outline and the tooling hole pads for a router
19 REFRENCE	Registration marks and text for the PC fabrication shop
26 ASMBDRAW	Outlines of parts for assembly drawings

Using Inner Layer Planes

When creating multi-layer PC boards, you may want large areas of copper to provide a ground or power plane. You could create these planes by adding components to the process layer representing the particular board side. However, you may not be able to obtain the shape of the plane that you want. An alternate method of creating solid planes on a PC board requires the definition of additional physical layers and the use of photographic reversal and combination techniques.

Consider an inner board side, inner physical layer A. This layer could consist of copper pads and traces, keepout areas, areas in which no copper is desired around plated-through holes and traces, and a large ground plane that is formed by the large copper area occupying the remainder of the board layer. Traces and pads for plated-through holes may be created by adding the appropriate components to the process layer, INLAYRA, that represents the inner physical board layer, layer A.

To add the ground plane, you must use the keepout, or negative, process layer, INKOA. Components added to this process layer represent areas in which no copper is desired. Keepout areas for plated-through holes and keepout areas around traces should be placed in this process layer. Pads for plated-through holes and traces should be represented by the appropriate components placed in the positive process layer, where components represent metalization, INLAYRA.

In addition to the two process layers, INLAYRA and INKOA, provided for the inner physical layer A, six more process layers are provided for three additional inner physical layers, B, C, and D.

Layer	Use
11 INLAYRA	Traces, pads, and other metalization on the inner physical board layer, A
12 INLAYRB	Traces, pads, and other metalization on the inner physical board layer, B
13 INLAYRC	Traces, pads, and other metalization on the inner physical board layer, C
14 INLAYRD	Traces, pads, and other metalization on the inner physical board layer, D
15 INKOA	Keepout areas for inner physical board layer A
16 INKOB	Keepout areas for inner physical board layer B
17 INKOC	Keepout areas for inner physical board layer C
18 INKOD	Keepout areas for inner physical board layer D

Defining a Master Keepout Layer

The individual keepout process layers, defined above, provide sufficient information to produce multi-layer PC boards with irregularly shaped plane areas. However, it requires that components for the keepout areas be added to each individual keepout process layer. This works well for keepout areas that are unique to each layer. However, for keepout areas that are common to multiple layers, for example, plated-through holes require identical keepout areas on each physical board layer. Again, an easier method is available. This method involves defining another process layer, KEEPOUT, to act as a master keepout process layer.

Layer	Use
4 KEEPOUT	Keepout master for keepout areas common to multiple layers

Components representing keepout areas common to multiple physical board layers are added to the keepout master while keepout areas unique to a physical board side are entered on the individual keepout process layer for that board side or inner physical layer. Plots of a keepout layer for a board side are then obtained by combining the keepout master and the keepout layer for the individual board side. (Note that in this case the layers are not bound, because only you will know which layers to combine when plotting a physical layer.)

Separate Power and Ground Layers

If you want to keep power and ground plane information separate from the information in the other process layers, you may use the separate process layers defined for them. These process layers can then be combined with the process layers representing the traces, pads, and other components on a board side.

Layer	Use
7 FIVEVOLT	Pads and traces for an inner layer power plane
8 GROUND	Pads and traces for an inner layer ground plane
9 FIVEKO	Special keepout areas for an inner layer power plane
10 GNDKO	Special keepout areas for an inner layer ground plane

Display Optimization Layers

The PC board personality is constructed to allow for graphic output to two types of devices: the graphic CRT display and pen plotters. In many cases, you can build library parts to more efficiently use either of these devices. For example, you may want your plotted pads to be smooth circles (a resolution of 6 degrees). However, the displayed version of this pad could be very coarse (a resolution of 15 degrees), which would enable HP EGS to draw it much more quickly. So by creating two versions of a pad you could increase display performance.

For this reason the supplied process file has two pad master layers and two solder resist master layers. You have already learned about the plotting version of these, PENPAD and PENSLDM. Here is the display version:

Layer	Use
29 DISPPADM	Pad master layer when using the graphic CRT during editing
30 DISPSLDM	Solder resist keepout master layer when using the graphics CRT, for example, during editing

You can use the DISPDISP macro command to turn on the display layers and to turn off the pen plotting layers, thus enhancing the display performance while you are creating a PC board. If you want to do artwork with fill on a pen plotter, you can use the DISPPEN command to turn on the pen plot layers and turn off the display layers.

The Macro File

The macro file contains the definitions of all the items that appear in the screen menu. You should be familiar with the operation of the Printed Circuit Board Layout personality because its macro file is based on the macro file for the General Drawing personality and has many features that are similar to the Electrical Schematic Drawing personality. Where appropriate the macro file has been changed. Some of the important changes follow.

Changes to the Modes

A major difference to the General Drawing personality is the presence of two overall drawing modes, Edit_PCB (edit a printed circuit board) and Make_Prt (make a part). These modes are defined as follows:

Mode	Description
Edit_PCB	Used when actually drawing a printed circuit board.
Make_Prt	Used to interrupt the drawing of a printed circuit board to create a new part. Some functions in this mode are changed from Edit_PCB to make the creation of parts easier. Upon completing the part, the system automatically returns to the printed circuit board that was interrupted.

The General Drawing personality has color and line-type modes (such as Solid and Red) that you set to determine the layer on which to place components. The Printed Circuit Board Layout personality has one mode, Layr, which determines the current layer for placing components.

Notice that the number of the current layer is displayed next to Layr. When you select Layr, a major portion of the menu will be overwritten with the name of all of the layers. At this time you can either select one of these names or enter the actual layer number.

Changes to the Primary Commands

Additional primary commands have been constructed, using macros, to make the creation of schematics easier. These commands are:

ROUTE*	Provides a convenient way to re-route an existing connection. This is similar to STRETCH* *REPL, except that the ends of a component remain fixed and the layer and width may also be changed as the component is re-routed.
LEVEL*	Provides a convenient way to set the logical level of a part (logical levels are explained later).
MOD_NOTE*	Provides a convenient way to modify notes. It is most often used to change the part type and reference designator defaults defined with library parts to the actual values.

LIST_MAT	Compiles a list of the library parts used in a printed circuit board. This list can then be post-processed to produce a formal bill of materials.
DISPDISP	Executes the proper SHOW* command to turn on specific display layers while turning off the pen plot layers.
DISPPEN	Executes the proper SHOW* command to turn on specific pen plot layers while turning off the display layers.

LIST_MAT, DISPDISP, and DISPPEN never appear in the screen menu. Therefore, you must enter these commands from the keyboard when you want to use them.

Changes to the Components

Some of the components presented in General Drawing (such as DIMEN and OVAL) are not useful for creating PC boards and therefore are not included in the Printed Circuit Board Layout personality. But other "special" components are useful. These components were created using macros and are included in the Printed Circuit Board Layout personality. These are:

- NET_NAME: This component specifies the name of a net. It is a NOTE component that you associate to a specific connection. This NOTE is automatically placed on layer 54 and given a tag value of 1002.
- PART_TYP: This component specifies the identifying type of a library part. It is a NOTE component that you associate to a part when it is created. This NOTE is automatically placed on layer 52 and given a tag value of 1004.
- REF_DESI: This component specifies the reference designator of a part. A reference designator is a unique name used to distinguish between multiple occurrences of the same library part. It is a NOTE component that you associate to a part when it is created, and later modify when the part is added to a schematic. This NOTE is automatically placed on layer 53 and given a tag value of 1001.
- PHY_PORT: This component specifies the name and location of a physical port. Physical port names are used to reflect the name of the port in the corresponding printed circuit board for the schematic. When you add this component you must also specify the location of the port in the part. This NOTE is automatically placed on layer 51 and given a tag value of 1003.

The tag values mentioned above are used by the material and connection listing processors to differentiate between the different notes in your drawing.

Changes to the Secondary Commands

The only change made to the secondary commands is in the operation of SHOW*. In the Printed Circuit Board Layout personality you may turn on, off, or see (turn on but not modify) any combination of layers you want. Methods include ON, OFF, SEE, LINES, PARTS, NOTES, and ALL. While the option sPT=1/sPT=2 still exists, a new option sLAYR is present so that you can easily select layers.

You will find that the operation of SHOW* is very flexible, but with this flexibility comes more complexity. You must always select the SHOW* command before selecting a set of methods (that is, it is never a "current" command). After selecting SHOW* you must select ON, OFF, or SEE. Then you must select one of the remaining methods, LINES, PARTS, NOTES, or ALL. You may then enter a layer number to restrict the effects of the SHOW* command to one layer. At this time you may either select EOC to terminate the command, or loop back to ON, OFF, or SEE to affect another layer.

If you don't know the number of the layer you want to affect, select sLAYR. The command menu will be replaced with layer names you can select instead of entering values. To get the command menu back you must select sLAYR again. Note that when you select sLAYR you will terminate and re-initiate the SHOW* command just as if you had selected EOC SHOW*.

Loading Printed Circuit Board Layout

1. If your computer is not displaying the main menu in the HP EGS manager, follow the instructions in Chapter 2, in the *Loading General Drawing* section, to get to the main menu. Your computer display should look like this:

```
Use arrow keys to move cursor to desired function.
Select function with <RETURN> or <ENTER>.
  ->System Utilities
    General Drawing Editor
    Electrical Engineering
    Mechanical Engineering
```

2. Move the cursor to the Electrical Engineering function and press (Return). The display should look like this:

```
Use arrow keys to move cursor to desired function.
Select function with <RETURN> or <ENTER>.
  ->RETURN To Main Menu
    Printed Circuit Board Layout Editor
    Electrical Schematic Drawing Editor

    EE Parts File Editor
    EE Material Lister
    Convert Old EE Parts Files

    Printed Circuit Board Connection Lister
    Electrical Schematic Connection Lister
    Compare Connection Lists
    Merge Connection Lists
    Generate Rat's Nest

    PHOTOPLOT/DRILL Post-processor
```

3. Select:

```
Printed Circuit Board Layout Editor
```

The Printed Circuit Board Layout personality will now load automatically.

In the next section of this chapter you begin the drawing example in which you learn to create a PC board using this personality. To preview this lesson, enter:

```
INPUT EWLRN:chap9;
```

Note

Before you begin to draw the printed circuit board, use the PREFIX command to prefix HP EGS to your personal mass storage volume. This keeps you from editing or destroying someone else's data as you edit and save drawings.

Starting the PC Board

In this section you will begin to create a PC board, which is the physical representation of the one-shot schematic, by adding the necessary library parts. This includes the creation of a new library part which represents an eight lead to5 package for mounting the SE555 monolithic timer.

1. Prefix to your storage volume, enter:

   ```
   PREFIX DRAW:;
   ```

2. To set the drawing mode to editing PC boards, select:

   ```
   Edit_PCB
   ```

   The display prompts you for the name of the PC board to edit.

3. Enter:

   ```
   shot_bd
   ```

   Because the shot_bd drawing does not exist yet, you will be editing an empty drawing. It is important to set the drawing mode, however, so that the Printed Circuit Board personality can store the name of the current PC board for later use.

4. Prepare the drawing area for the addition of components by turning the grid on and zooming in on a smaller portion of the large port. Select:

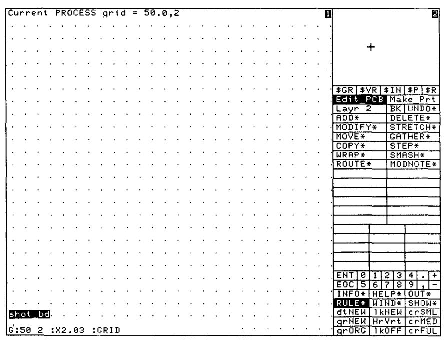
   ```
   WIND* grOFF 2PNTS
   ```

5. Enter the two points (−50,−50) and (2500,2500).

   Notice that the display grid is currently set to 25,4. For this example you will find a setting of 50,2 easier to use, so select:

   ```
   RULE* grNEW 50 ENT 2 ENT
   ```

   Your screen should look like this:



Adding Capacitors and Resistors

Now you are ready to add the PC board pad sets for the passive parts, 1 resistor and 2 capacitors.

1. Select:

   ```
   ADD* PART
   ```

   Enter the names of the parts you want to add.

2. Because you don't know the actual file names for resistors or capacitors, select the option:

   `*SCRN`

   Your screen menu is redrawn, and it displays a list of the parts provided with HP EGS.

3. From the upper left corner of the menu, select:

   `C300`

   (The C stands for capacitor, and 300 is the distance between the center of the pads in mils.)

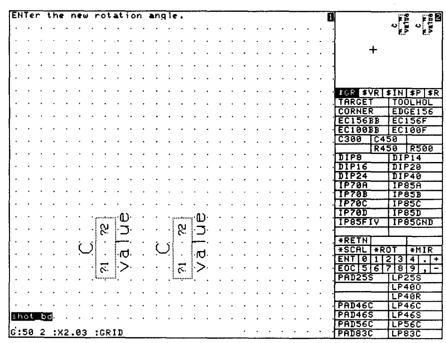
4. Move the cursor into the drawing area. Notice that you are tracking the boundary of the capacitor, which is horizontal. As shown in the first figure of this chapter, the capacitor needs to be rotated to a vertical orientation before it is placed. From the middle of the menu, select the option:

   `*ROT`

5. Enter

   `90`

   for a 90-degree rotation. Now place the capacitor at (700,500) and (1300,500). Your screen should look like this:



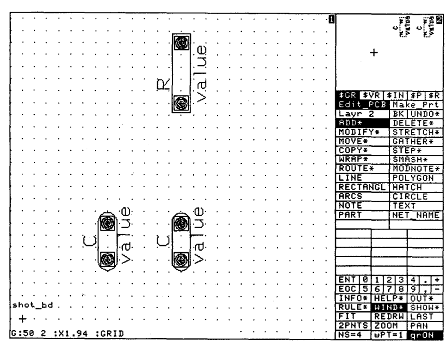
6. Add the resistor to the schematic by selecting:

   `R500`

   and placing it at (1300,1800). Notice that the resistor is rotated 90 degrees.

   Now that all of the passive parts have been added, return to the original menu by selecting the option:

   `*RETN`



7. To set the nesting level to 4 and redraw the screen so that you can see inside the instances you have just added, select:

   `WIND* NS=1 4 ENT FIT`

   Your screen should look like this:

Adding a Pad Set

Next, you will add the pad set for the 555 timer, an eight lead to5 package. This part is not provided in the library, so you need to create it.

1. Select the drawing mode:

   ```
   Make_Prt
   ```

2. When prompted for the part name, enter:

   ```
   to58
   ```

3. To prepare the part drawing area for the addition of components, set the display grid to 50,2. Then zoom in on a smaller portion of the large port by selecting:

   ```
   RULE* ⅁rNEW 50 ENT 2 ENT WIND* ⅁rOFF 2PNTS
   ```

   and select the two points (−250, −250) and (1000,1000). Note that this is smaller than before, because you are making a single part.

Defining a PC Board Library Part

1. The first step in defining a PC board library part is to place the pads in the desired locations. To start, get the screen menu of library parts by selecting:

   ```
   ADD* PART *SCRN
   ```

   Notice the three kinds of pads: primitive (LP), primary (PAD), and inner layer (IP). For information on the differences between these pads see the last section of this chapter.

2. Also notice that each category of pads has different finished hole sizes (such as 56 for 56 mil), and that there are different shapes: circular (C), rectangular (R), and oval (O). For this eight lead T package, select:

   ```
   LP400
   ```

   and place it at (300,0).

3. Select:

   ```
   *RETN
   ```

4. To set the nesting level to 3 and redraw the screen so that you can see inside the part you have just added, select:

   ```
   NS=1 3 ENT REDRW
   ```

   Notice that this part already has the appropriate drill, pad, and solder mask components appropriately added to match the definitions presented in the first section of this chapter.

5. To complete the pad set, you must add seven more copies of LP400 in a circular array. To do this you will use the *POLR (polar) option of the step command. Select:

   ```
   STEP* *POLR
   ```

   The display shows:

   ```
   Select component, polar center point, ENTer number, (*UNRT) angle between ea.
   ```

6. To start completing the polar step, select the pad and then select (300,300) as the center point about which to step the pad. To make seven additional copies, enter:

   ```
   7
   ```

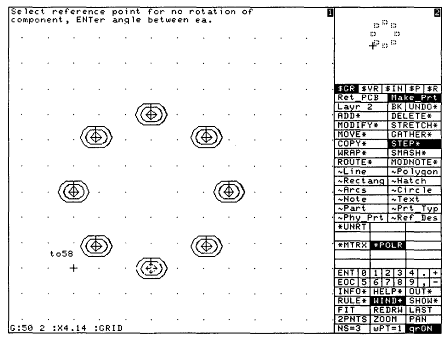
7. The default mode of polar step is to rotate the selected component as it is stepped so that its center axis always lies on the radial line through the polar center point. In this case you don't want the part to be rotated, because the oval pads must be kept orthogonal to the grid. To avoid rotation, select:

   ```
   *UNRT
   ```

8. Select (300,0) as the point about which the pad will be "un-rotated".
9. Enter:

   ```
   45
   ```

   as the angle between each pad in the final pattern. The values for this polar step insure that more than 100 mils are between pad centers, a design rule for this part which could be checked by measuring the distance with INFO* DIST?. Your screen should look like this:



Outlining a Part

Now you will draw the outline of the part to be used in assembly drawings.

1. To start, change the current layer to the assembly drawing layer named =AsmbDraw by selecting:

   ```
   Layr 2 =AsmbDra
   ```

   The outline of a T package is drawn as a circle with a small tab above lead number 8.

2. To draw the circle, select:

   ```
   ADD* CIRCLE
   ```

3. Select the center point (300,300), which is the same as the polar center point. Then select a point on the circumference outside the pads, such as (300, −100).

4. To add the small tab, select:

   ```
   LINE
   ```

   and add a line from (200,650) to (200,800) to (400,800), and finally to (400,650). Select EOC to complete the lines.
5. To complete the outline of the part, you must delete the segment of the circle between the vertical lines of the tab and the two short line segments inside the circle. To do so you must select the exact endpoints of these segments. First, change the snapping mode to intersection by selecting:

   ```
   $IN
   ```

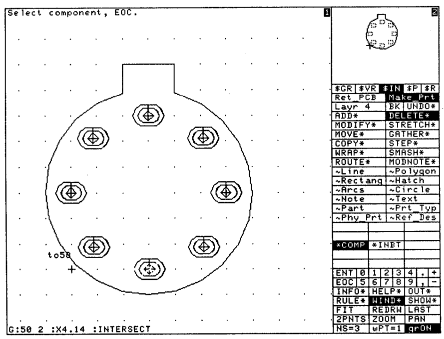
6. Select:

   ```
   DELETE* *INBT
   ```

   and select the left end of the circular segment (200,690), then the right end of the circular segment (400,690), and a point on the segment of the circle to delete (300,690). Repeat this selection process for the two line segments. Approximate selection points are:

endpoint	endpoint	segment point
200,650	200,685	200,660
400,685	400,650	400,660

Your screen should look like this:



Adding the Keepout Area

1. Remember that the keepout area is the area where metalization is *not* desired on a ground or power plane. For this part, the keepout area is a circle with a 375 mil radius placed at the parts center.
2. Change the current layer to the master keepout layer, =KOMs, by selecting:

   ```
   Layer 26 =KOMs
   ```

3. Select:

   ```
   ADD* CIRCLE
   ```

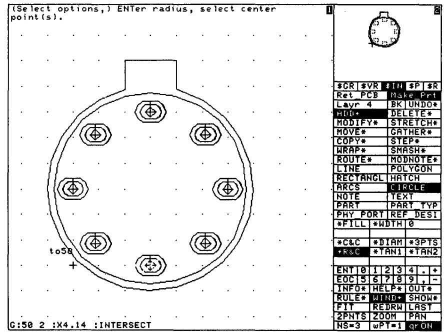
   ```
   *R&C
   ```

   *R&C is the radius and center point method of adding circles.

4. For the radius, enter:

```
375
```

and then place the circle at the center of the part, (300,300). Your screen should look like this:



Correcting Pad Locations

Next, consider how the part will be added to actual PC boards to make sure that the pad placement is correct. In most PC boards traces are always started and terminated on grid points. Notice that the four pads at angles of 45, 135, 225, and 315 degrees don't exactly lie on grid points. Because the leads of a device (such as an SE555T timer) can be shaped easily, the exact location of the pads is not critical. For this reason the center points of the above four pads should be moved to the nearest grid point.

1. To insure that the center point of the pads are selected as the reference point when they are moved, set the snapping mode to vertex. (Remember that the center of a circle is a valid vertex.) Select:

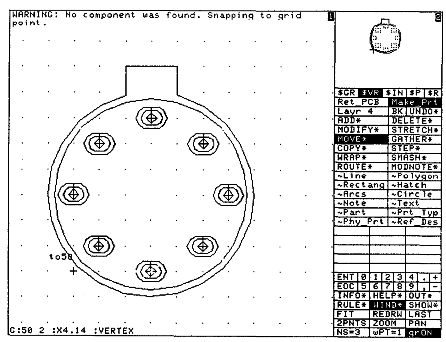
```
$VR

MOVE*
```

and then select a pad.

2. Select the center of the pad as a reference point, and then the nearest grid point as its new location. Make sure you are using the small cursor such that this works properly. These points for the four pads of interest are:

pad location	center point	new grid location
510,510	510,510	500,500
510,85	510,85	500,100
85,85	85,85	100,100
85,510	85,510	100,500

Your screen should look like this:



Redefining the Pad for Lead One

Now consider how the actual device will be inserted into the PC Board. There is no indication yet of how the device should be oriented to get the proper leads in the proper holes (unless the assembly layer is silkscreened on the board). In the library parts included with HP EGS, the rectangular pad corresponds with lead or pin one. So, you must now replace the pad corresponding to lead one with a rectangular pad.

1. Select:

   ```
   DELETE*
   ```

   and select the pad just to the left of the tab (100,500). Select EOC to complete the deletion.

2. Select:

   ```
   ADD* PART *SCRN
   ```

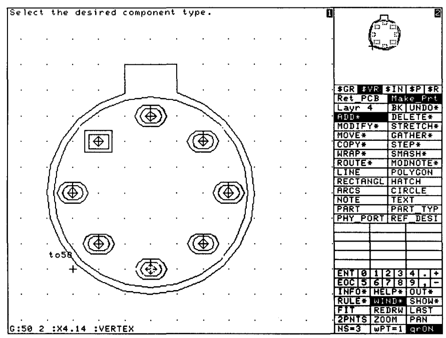
3. Select the rectangular pad corresponding to the oval pads already in the part:

   ```
   LP40R
   ```

4. Place this pad where you just deleted the original (100,500). Then return to the command menu by selecting:

   ```
   *RETN
   ```

   Your screen should look like this:



Aligning the Part

Now that you have created the basic pieces of the to58 library part, consider how it will be dynamically tracked and placed when it is added to a PC board. Remember that the cursor will track the (0,0) position of the part, so a good choice is to place the center of one of the pads at (0,0).

It is generally not important which pad is placed at (0,0), as long as it is always the same one. Because the lead or pin number one pad of the parts in the provided library are all located at (0,0), so will the pad in this example.

1. The device to be installed at the to58 location is an SE555T. Lead number one in this device corresponds to the rectangular pad. To align the part, select:

   ```
   WRAP*
   ```

2. Select the wrap origin, such as (300,300), and wrap all of the components on the screen by selecting (−200, −200) and (850,900). Select EOC to terminate the wrap.

3. Now move the instance you just created so that the pad just to the left of the tab is centered over (0,0). Select:

   ```
   MOVE*
   ```

   then select the instance (about 150,600), then select the center of the rectangular pad as the reference point (100,500), and then place the instance at (0,0).

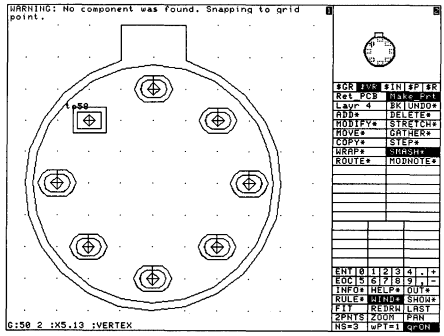
4. To fit the part to the screen, select:

   ```
   FIT
   ```

5. Select:

   ```
   SMASH*
   ```

6. Select the instance and EOC so it will be smashed back to the component level. Your screen should look like this:



Adding the Reference Designator and Part Type

You are now ready to annotate the part with the reference designator and part type. These notes must be in your library part so that you can create a valid rat's nest and connection list from the final PC board.

1. To begin, set the snapping mode back to grid by selecting:

   ```
   $GR
   ```

2. The reference designator is the name by which a single instance of a part is referred to, such as U37 or R54. When building a library part you need to add a default designator (i.e., U or R), which you will modify later. To add the default designator, select:

   ```
   ADD* REF_DESI
   ```

3. Enter:

   ```
   'U'
   ```

   (Don't forget the quotation marks.)

4. Select (100,-200) as the location of the note string.
5. The part type represents the value or type of your part. If your part has an exact part type, add it here.

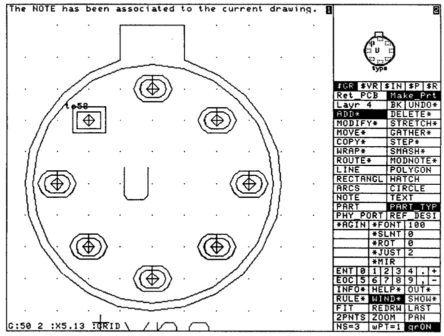
   For parts which do not have exact types, add the word 'value' or 'type' here. You would modify this word later when the part is added to a schematic (that is, change 'value' to '10K' for a resistor). To add the part type for this example, select:

   ```
   PART_TYP
   ```

6. Because the to58 can be used for any 8 lead part, enter:

   'type'

7. Select (0, −650) as the location of the note string. (Notice that part of the string is outside of the large port.) Your screen should look like this:



Adding Physical Ports

The final annotation task is to assign the physical port names of the pads in the part. Physical port names represent the pin numbers of the devices in a PC board part. These are the names that will be used when comparing connection lists between schematics and printed circuit boards. The sequence for adding a port name is to first select the logical location, then enter the port name, and then place the port name string.

1. To add the first physical port name, select:

   PHY_PORT

2. Select the center of the pad to the left of the tab (0,0), and enter:

   '1'

3. Place this string directly on top of the pad (-50,0).
4. To add the second port name, select:

   *AGIN

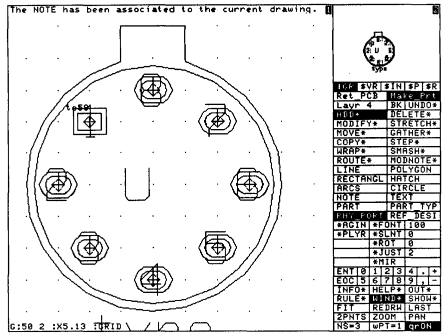
5. Select the center of the pad counter-clockwise from the first (−100, −200), then enter:

   '2'

6. Place this string in the center of this pad at (−150, −200).

7. Use *AGIN and repeat the process to to add the remaining physical port names for the following values:

Port Location	Name	Name Location
0, −400	‘3’	−50, −400
200, −500	‘4’	150, −500
400, −400	‘5’	350, −400
500, −200	‘6’	450, −200
400,0	‘7’	350,0
200,100	‘8’	150,100



Your screen should look like this:

Verifying the Part

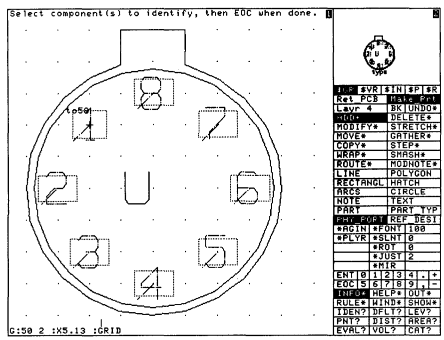
Next you will check to make sure that all of the logical locations of the ports are correct.

1. To make this process easier, set the nesting level to 1 and redraw the large port by selecting:

   `NS=3 1 REDRW`

2. To check the logical locations, select:

   `INFO* IDEN?`



3. Enter the tag value for port names (discussed at the beginning of this chapter):

   `:T1003`

   All port names should now be highlighted, and a line is drawn from the lower left corner of each port name string to its corresponding logical location. This point should be the center of the pad. If a port is incorrect, delete and re-add the port name properly. Your screen should look like this:

Understanding Logical Levels

The part is now complete. To return to the original PC board, select:

```
Ret_PCB
```

Notice that two commands, Continue and LEVEL*, appear at the bottom of the command menu. The display reads:

```
Remember to assign level, save the part, then select Continue.
```

This prompt reminds you to do two more tasks before the part is ready for use. Before you set the logical level, you need to know a little more about logical levels.

Logical levels are similar to nesting levels, except that they define logical rather than physical relationships. The logical hierarchy is created with a user assignment of a logical level number to each part or instance.

Unlike nesting levels, higher positive numbers mean a higher logical position in the hierarchy. When adding an instance to a drawing (or to another instance), the instance must have a logical level less than or equal to the logical level of the drawing (or instance) being drawn.

If you look in the process file for each personality you will find that the default logical level is set to 1 (LEVEL 1;). Therefore, any drawing you create will have a logical level of 1 unless you reset it. The logical level can be changed only by editing the instance as an individual drawing. If you create an instance with the WRAP or GROUP commands, you must edit the instance so you can execute the LEVEL command on it explicitly.

Because all library parts furnished with the system are at logical level of 0, the :D0 option can be specified in the LIST_MAT macro command. This command will then process only "base level" parts in a drawing, and ignore groups and wraps (which would be at a logical level of 1 or higher).

Through the use of logical levels, you can specify a hierarchy of instances within a drawing. Setting the logical level can also be useful for the ARCHIVE and GENERATE commands, since unwanted instances can be eliminated from the output file.

1. By convention the post-processors furnished with your system must have the schematic and printed circuit board parts set to a logical level of 0 or lower. To set the logical level of your part to 0, select:

   ```
   LEVEL*
   ```

2. Enter:

   ```
   0
   ```

3. You are now ready to save your part. Select:

   ```
   OUT* SAVE* OLD
   ```

   Your part will be saved on the default volume under the name to58, which you entered earlier.

4. Now that you have completed the final two tasks return to the original PC board by selecting:

   ```
   Continue
   ```

   Answer No to the prompt to input "shot_bd" from disc, since the version in memory is the one currently being edited.

BREAK TIME

Take a short break now. When you return you will complete the PC board. Review what you have done in this section:

- Learned about the process file and the macro file.
- Loaded the Printed Circuit Board Layout personality.
- Added capacitors and resistors.
- Defined a PC board library part.
- Added a pad set.
- Outlined a part.
- Added a keepout area.
- Moved and changed pads.
- Aligned a part.
- Added the reference designator and part type.
- Added physical ports.
- Verified a part.

Completing the PC Board

In this section you will finish placing the components on the PC Board and then modify the annotations to match the schematic. Then you will generate a rat's nest from the placed parts and then produce the connection list.

Re-Editing a Drawing

Your screen should be displaying the original PC board. The nesting level has been reset to 1 and the display grid has been turned off because the shot_bd drawing was automatically brought back using the EDIT command.

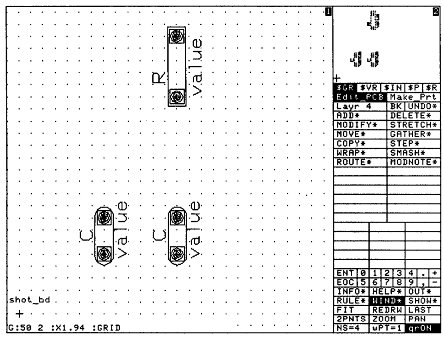
1. Set the nesting level to 4 and turn on the grid by selecting:

```
WIND* grOFF NS=1 4 ENT REDRW
```

2. As discussed in the previous chapter, the primary pad library parts contain geometry to produce both display and plotting versions of the pads. All of this data is now displayed on your graphics display. To decrease the time it takes to draw the pads on the display, use the DISPDISP macro command to turn off the plotting layers and insure that the display layers are turned on. Enter:

```
DISPDISP
```

   The large port is redrawn. The pads are much simpler and are drawn faster. Your screen should look like this:



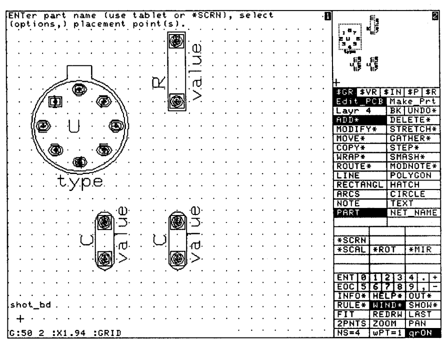
3. Add the to58 library part which you just created. Select:

```
ADD* PART
```

4. Enter:

```
to58
```

5. Place the part at (300,1800). Your screen should look like this:



Adding an Edge Connector

In this section you will create a six-finger edge connector and the board blank for the PC board. In this example you are provided with the exact locations for the components that determine the physical size of the board. You may often find that these components need to be placed using constructive geometry. To learn more about using the constructive geometry features of HP EGS, read Chapter 6 of this manual. You may find it easier to use the Mechanical Drafting personality to create complex board blank shapes.

You will make the edge connector by building half of it first, then copying the first half and mirroring the copy to build the other half.

1. To add half of the connector fingers to the drawing, access the parts menu by selecting:

   ```
   *SCRN
   ```

2. Select:

   ```
   EC156BB
   ```

3. Add the finger/board blank junction to the drawing at (450, −100).
4. Select:

   ```
   EC156F
   ```

   and add the finger junction at (550, −100).
5. Return to the command menu by selecting:

   ```
   *RETN
   ```

6. Move the finger so that it touches the board blank. To do this, first change the snapping mode to vertex snapping. Then select:

   ```
   MOVE*
   ```

   and then select the finger (550,0) and a reference point at the left end of the board blank line in the finger (475, −100). Now place this point at the right end of the board blank (450, −100).
7. Make two more copies of the finger and place them exactly adjacent to the existing one. Select:

   ```
   COPY*
   ```

   and then select the finger (500,0), and again select the reference point on the left end of the board blank line (450, −100). Place two copies at the right ends of the board blank line, (600, −100) and (750, −100).
8. To copy and mirror this half of the connector, first wrap these parts together. Select:

   ```
   WRAP*
   ```

9. Select (450,50) as the reference point, and then collect all of the connector parts by selecting (0, – 150) and (1000,250). When all of the parts are highlighted, terminate the wrap with EOC.
10. Select:

    COPY*

    and select the instance you just created at (450,50). Select (150, – 100) as the reference point and then place a copy at (1150, – 100).

    By following the next three steps you can mirror and move this copy with one command.
11. Select:

    MODIFY*

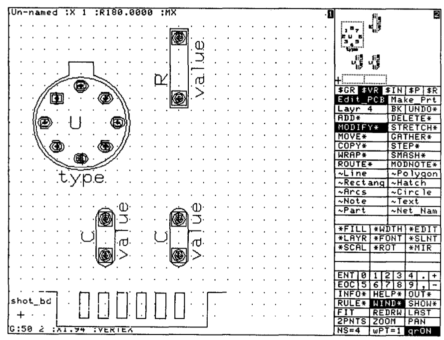
    and select the copy (1400, – 50). Select (1900, – 100) as the reference point.
12. To mirror the instance, select:

    *MIR

13. You are presented with the axis around which the instance can be mirrored. Select:

    YAXIS

    and then select (950, – 100) as the new placement point for the instance. Complete the modification with EOC. Your screen should look like this:



This procedure of selecting points in vertex snapping mode had to be used here, since the EC156F finger parts are 156 mils wide. Therefore, the endpoints of the contained board blank lines do not fall on grid points.

Adding a Board Blank Outline

1. To add the board blank outline, first change the current layer to =BdBlank by selecting:

    Layr 4 =BdBlank

2. Set the snapping mode to grid snapping. Then select:

   `ADD* LINE`

   Start the line at the left end of the connector (150,200). Place the second vertex at (0,200). The third vertex must be placed at (0,2600). If this point is beyond the top of the large port, select:

   `PAN`

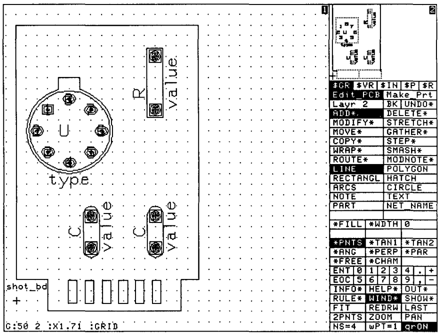
   and select (1000,2000) as the point to bring to the center of the large port.
3. Place the third vertex, and then place the fourth vertex at (1700,2600). To finish the board blank you must place two vertices near (1700,200). If you panned the large port, before you can obtain this point you must select:

   `LAST`

4. Place a vertex at (1700,200) and change the snapping mode to vertex snapping. Place the last vertex at the right end of the board blank line in the connector (1685,200) and complete the line with `EOC`.
5. Set the snapping mode back to grid snapping and select:

   `FIT`

   to fit the entire PC board to the large port. Your screen should look like this:



Modifying the Note Strings

Your next task is to modify the default reference designators and the passive component values to reflect the values used in the schematic of the last lesson.

1. Select:

   `MODNOTE*`

2. Select the reference designator for the to58, 'U', at (450,1600). The 'U' appears on your alpha screen. Use the editing softkeys you learned about in Chapter 2 to change the 'U' to 'U1'. Then press (**Return**).

3. To change the part type on the to58, select:

```
*AGIN
```

and then select the string 'type' at (500,1100). The 'U' you changed is un-highlighted, and 'type' appears on the alpha screen. Change this to 'SE555T' as you did before.

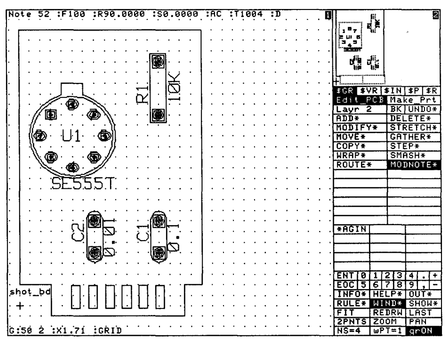
Repeat this process for the following strings as defined in the following table:

Default String	Approx. Location	New String
'R'	1100,1900	'R1'
'C'	1100,650	'C1'
'C'	500,600	'C2'
'value'	850,650	'0.01'

4. For the last two strings repeat the same process, except that after the string has been edited, select a new location for it.

Default String	Location	New String	New Location
'value'	1500,1950	'10K'	1500,2050
'value'	1500,550	'0.1'	1500,600

Complete the last `MODNOTE*` with EOC. Your screen should look like this:



Preparing the Post-Processor

You have now completed the necessary tasks to prepare the PC Board for the generation of the rat's nest. Before you continue with this example, save your drawing by selecting:

```
OUT* SAVE* OLD
```

The rat's nest post-processor requires two files to process. The first is the connection list from the previous lesson, and the second is the archive file representing the placed and properly annotated PC board for the schematic. Since the connection list file is already available (one_shot_c), you now need to produce the archive file. Enter:

```
ARCHIVE :D1 shot_bd;
```

This produces an archive file of your PC board and puts it on the default volume as shot_bd_a (note the _a suffix). The :D1 option specifies that the archive file is only to include instances with a logical level of one or greater. This saves memory because the rat's nest does not need the part definitions.

1. To process the connect list and archive files to create a rat's nest, you must first exit the Printed Circuit Board Layout personality. Enter:

   ```
   QUIT;
   ```

   The system returns you to the Electrical Engineering menu in the manager, which now looks like this:

   ```
   Use arrow keys to move cursor to desired function.
   Select function with <RETURN> or <ENTER>
       RETURN to Main Menu
     ->Printed Circuit Board Layout Editor
       Electrical Schematic Drawing Editor

       EE Parts File Editor
       EE Material Lister
       Convert Old EE Parts Files

       Printed Circuit Board Connection Lister
       Electrical Schematic Connection Lister
       Compare Connection Lists
       Merge Connection Lists
       Generate Rat's Nest

       PHOTOPLOT/DRILL Post-processor
   ```

2. Select:

   ```
   Generate Rat's Nest
   ```

   Wait to be prompted for the input and output file names. You must include the prefix to the volume where you placed the original files (EWYOU: is used here to indicate this). The following display appears:

   ```
   Enter the name of the schematic connect list file:
   ```

3. Enter:

   ```
   EWYOU:one_shot
   ```

   The _c is automatically appended. The following display soon appears:

   ```
   Enter the name of the PC Board archive file:
   ```

4. Enter:

```
EWYOU:shot_bd
```

The _a is automatically appended. Soon the following display appears:

```
Enter the name of the output (rat's nest) file:
```

5. Enter:

```
EWYOU:shot_rat
```

In this case an _r is appended to the file name when the rat's nest file is placed in your volume.

The post-processor asks you to specify the layers on which to put specific nets. Because the rat's nests are created with lines, the post-processor will also ask you for widths. For this example you will put the ground net (GND) on layer 8 (the ground layer) and make the trace widths 20 mils. The balance of the nets should be placed on layer 6 (the circuit side layer) and given a width of 20 mils. The display reads:

```
Enter a net number, net number range, or net name to be processed:
(ex: 5, 23-78, GND, * specifies 'the rest of the nets', Enter an
empty line when done)
```

6. Enter:

```
GND
```

The system displays:

```
Enter the layer number for the traces: (1 <= n <= 255)
```

7. Enter:

```
8
```

The system displays:

```
Enter the width for the traces: (n >= 0)
```

8. Enter:

```
20
```

The display reads:

```
Enter a net number, net number range, or net name to be processed:
(ex: 5, 23-78, GND, * specifies 'the rest of the nets', Enter an
empty line when done)
```

9. To set the layer 6 to receive the remaining nets, enter:

   ```
   *
   ```

   The display reads:

   ```
   Enter the layer number for the traces: (1 <= n <= 255)
   ```

10. Enter:

    ```
    6
    ```

    The system displays:

    ```
    Enter the width for the traces: (n >= 0)
    ```

11. Enter:

    ```
    20
    ```

    The display reads:

    ```
    Enter a net number, net number range, or net name to be processed:
    (ex: 5, 23-78, GND, * specifies 'the rest of the nets', Enter an
    empty line when done)
    ```

    Because you have no other nets to specify, press (Return) to terminate the specification of layers. The post-processor will now create the rat's nest. The system indicates its progress. When processing is complete, the display reads:

    ```
    Done.
    Press <space> to continue.
    ```

    Press the space bar to return to the Electrical Engineering menu.

BREAK TIME

Relax, take a break, and review how you generated the rat's nest before you continue. To find out more about the rat's nest post-processor, refer to *Understanding HP EGS*. In this section you:

- Added an edge connector.
- Added a board blank outline.
- Modified a note string.
- Prepared and made a rat's nest.

Completing the Printed Circuit Board

In this section you will complete the printed circuit board by inputting the rat's nest, rerouting selected traces, and adding other traces. Then you finish the artwork by adding tooling holes and registration marks.

The Electrical Engineering menu should be displayed as follows:

```
Use arrow keys to move cursor to desired function.
Select function with <RETURN> or <ENTER>.
    RETURN To Main Menu
    Printed Circuit Board Layout Editor
    Electrical Schematic Drawing Editor

    EE Parts File Editor
    EE Material Lister
    Convert Old EE Parts Files

    Printed Circuit Board Connection Lister
    Electrical Schematic Connection Lister
    Compare Connection Lists
    Merge Connection Lists
  ->Generate Rat's Nest

    PHOTOPLOT/DRILL Post-processor
```

Inputting Rat's Nest Traces

1. Return to the Printed Circuit Board Layout personality by selecting:

   ```
   Printed Circuit Board Layout Editor
   ```

2. When the personality is loaded again, prefix to your personal mass storage volume. Then edit the original PC board by selecting:

   ```
   Edit_PCB
   ```

3. When prompted for the PC board name, enter:

   ```
   shot_bd
   ```

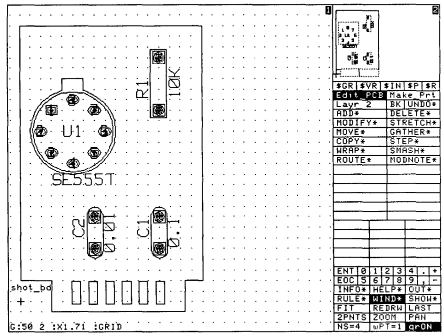
   The shot_bd drawing should appear on your screen. Turn the grid on and set the nesting level to 4 to prepare the drawing area for the addition of components. Select:

   ```
   WIND* grOFF NS=1 4 ENT REDRW
   ```

4. Turn off the plotting layers by entering:

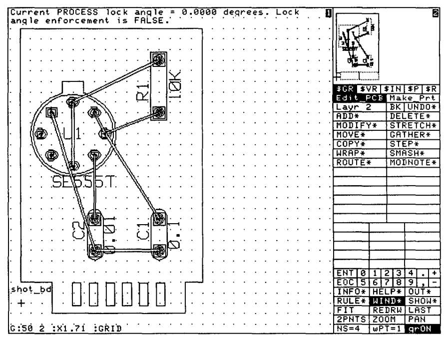
   ```
   DISPDISP
   ```

   Your screen should look like this:



5. You are now ready to input the traces produced by the rat's nest post-processor. Enter:

   ```
   INPUT shot_rat_r;
   ```



   Notice that straight line traces have been added where connections were specified by the one_shot schematic drawing. Also notice that the snapping mode indicator at the bottom left of the large port is now set to :RAWPOINT. Select:

   ```
   $GR
   ```

   to reset the snapping mode to grid snapping.

Connecting Ground Layer Traces to the Outer Layers

To insure that the traces on an inner physical layer are connected to the plated drill holes, an appropriate inner layer pad must be placed at the termination point of traces on inner physical layers. In this case this means that an inner layer ground pad (IP85GND) must be placed where each ground trace connects to a device lead.

1. To see only the traces on the ground layer to make the placement of the inner ground layer pads easier, turn off the circuit side layer with the show command by selecting:

   ```
   SHOW* sLAYR
   ```

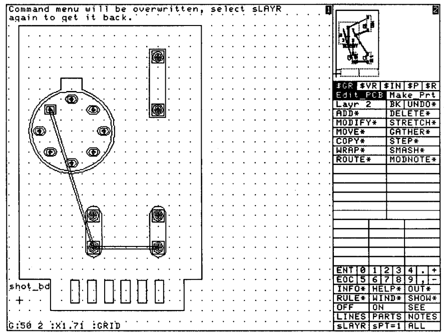
2. The SHOW* command is now ready and the command menu should be replaced with layer names. To turn off all notes and the lines on the circuit side physical layer, select:

   ```
   OFF NOTES OFF LINES =Circ EOC
   ```

3. To return to the command menu, select:

   ```
   sLAYR
   ```

   Your screen should look like this:



4. Add the inner ground layer pads at the termination points of the ground traces by selecting:

   ```
   ADD* PART *SCRN
   ```

5. Select the inner ground layer pad:

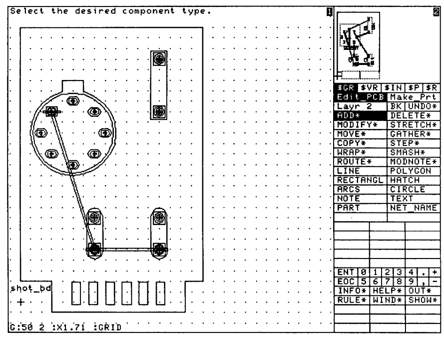
   ```
   IP85GND
   ```

   and place it at (1300,500), then (700,500) and (300,1800).

6. To return to the command menu, select:

   ```
   *RETN
   ```

   Your screen should look like this:



Re-Routing Traces using STRETCH*

The next several steps in this lesson show you how to re-route the traces on your PC board. As a design rule used here, all traces must be at angular increments of 45 degrees to vertical and horizontal. (This is why the default LOCK angle is 45 degrees.)

1. Turn on the traces on the circuit side of the board by turning on all lines with the SHOW* command. Select:

   ```
   SHOW* ON LINES EOC
   ```

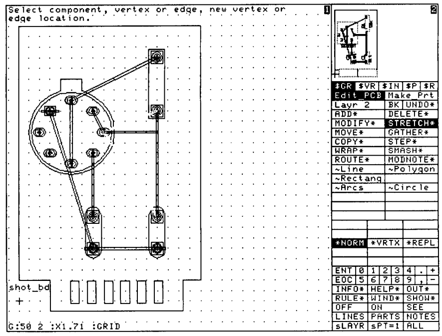
2. The first re-routing task is to make two of the three traces emanating from lead six of the to58 conform to the design rule above. Select:

   ```
   STRETCH*
   ```

   and then select the trace which connects to the top pad in capacitor C1 (1000,1350).

3. Select the end of this trace which connects to lead six (800,1600), and place this end on the lower pad of the resistor (1300,1800).

4. Select the the trace that starts at pin six and connects to the lower pad of the resistor (1100,1700). Then select the end of the trace that connects to the resistor (1300,1800) and place it at (1300,1600). This trace should now be horizontal and also should be connected to the trace you stretched previously. Your screen should look like this:



You will now re-route two more traces with the *VRTX (add a vertex) method of stretch.

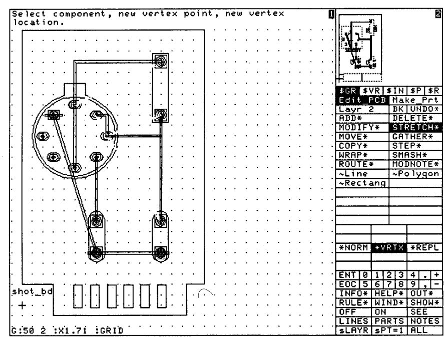
5. Select:

   `*VRTX`

   and select the final trace emanating from lead six (750,1700).

6. Specify a new vertex to be added in the middle of this trace by selecting the middle again for the addition of a vertex (750,1700). Now place the vertex so that there is a 90-degree bend in it producing one vertical and one horizontal segment (800,1800).

7. Put a 90-degree bend in the trace from lead 8 to the top of the resistor using the same process. Select the trace (850,2050), then select the segment to add the vertex to (850,2100). Place the vertex at (500,2300). Your screen should look like this:



Re-Routing Traces using ROUTE*

In the following steps you will use the ROUTE* command to re-route the ground layer trace from lead one to the lower pad of capacitor C2. The ROUTE* command maintains the endpoints of the trace as multiple vertices are added. This appears to work the same as adding a polygon, except that the trace is not closed. Several options to ROUTE* allow you to change the layer of the trace and change the width of the trace while it is being re-routed. The options are:

Option	Description
*LNUM	Changes the layer number of the following segments by entering the actual value of the layer.
*LMEN	Changes the layer number of the following segments by selecting the name of the layer from the layer menu.
*WDTH	Changes the width of the following segments.
*NOLK	Completes the route without lock angle enforcement.

The procedure for ROUTE* is to select the first endpoint of the trace to route, and then select the second endpoint of the trace to route, then identify the segment to route. You can then select as many new vertices for the trace as you want, from the first endpoint to the second endpoint.

You may also select options at any time during the routing process. To complete the route you must select EOC. If lock angle enforcement has been turned on, HP EGS may not allow the route to be completed. To override this without turning off lock angle enforcement, select *NOLK instead of EOC.

Note that when the layer or width of a component is changed during routing, the original component is broken into multiple components. This is because a single component can have only one width and one layer value.

1. Select:

   ```
   ROUTE*
   ```

   and then select the first endpoint of the trace at lead one (300,1800). Select the second endpoint at the bottom pad of capacitor C2 (700,500), and select the trace segment to route (550,950). Place the first new vertex at (300,2500).
2. Select:

   ```
   *WDTH
   ```

3. To change the width of subsequent traces to 40 mils, enter:

   ```
   40
   ```

4. Place the next vertex at (1600,2500) and another vertex at (1600,500). Select:

```
*WDTH
```

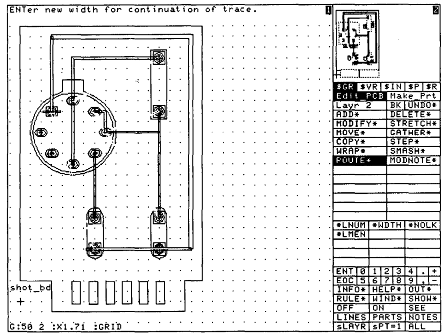
5. To set the trace width back to 20 mils, enter:

```
20
```

6. Complete the routing process by selecting:

```
EOC
```

   Look closely at the trace you just routed and notice that there are actually three lines. One has a width of 40 mils and two have a width of 20 mils. Your screen should look like this:



Stretching a Re-Routed Trace

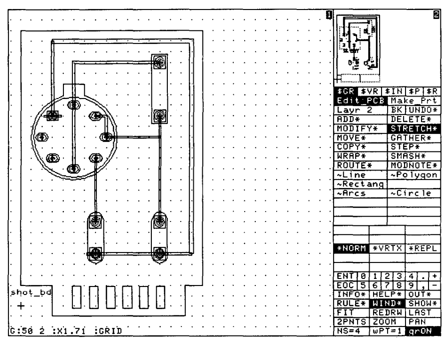
1. When you re-routed the trace, the final segment was completed on top of the trace which existed between the lower pads of the two capacitors. To remove this overlapping trace segment, select:

```
STRETCH*
```

2. Select the last segment of the trace you just routed (1450,500). Then select the end of this trace which connects to the lower pad on capacitor C2 (700,500) and connect it to the lower pad on capacitor C1 (1300,500). There now appears to be no connection between the two capacitors. To see that the original, short, trace still exists, redraw the screen by selecting:

```
WIND* REDRW
```

   Your screen should look like this:



Adding Traces

To complete the connections on the PC board, several traces must be added to connect the inputs and outputs to the edge connector.

1. First, change the current layer to the circuit side, =Circ, by selecting:

```
Layr 2 =Circ
```

2. Select:

```
ADD* LINE
```

3. Select:

   `*WDTH`

4. To set the trace width to 20 mils, enter:

   `20`

5. Add the ground trace by connecting the lower pad of capacitor C1 to the right finger of the edge connector using the following points:

   `(1300,500)  (1300,150)  EOC`

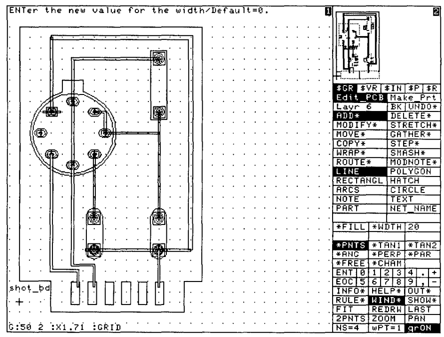
6. Next, add the input trace by connecting lead four to the second finger from the left of the edge connector using:

   `(500,1300)  (500,350)  (700,350)  (700,150)  EOC`

7. Add the output trace by connecting lead three to the left finger of the edge connector using:

   `(300,1400)  (300,300)`
   `(550,300)  (550,150)  EOC`

   Your screen should look like this:



Adding Tooling Marks

The final task in this section is to add the tooling marks required on the PC board. Before doing this you need to pan the large port so that the PC board is approximately centered in the large port.

1. Select:

   `PAN 800,1400`

2. Next you will add tooling holes to the lower right and upper left corners of the PC Board. To get the part menu, select:

   `PART *SCRN`

3. Select:

    TOOLHOL

    and place the tooling hole part at (1550,350) and (150,2450).

4. Next add registration targets to the right and left of the PC Board outline by selecting:

    TARGET

    and placing the target part at (2000,1400) and (–300,1400).

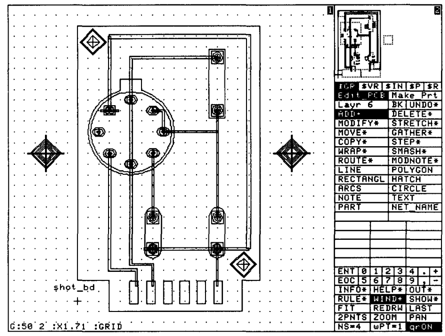
5. To return to the command menu, select:

    *RETN

6. Fit the drawing to the large port by selecting:

    FIT

    Your screen should look like this:



7. You have now completed the PC board. Before you continue with the next section, save your drawing by selecting:

    OUT* SAVE* OLD

BREAK TIME

Take a few minutes to stretch, relax, and review what you did in this section:

- Inputted rat's nest traces.
- Turned off the notes and circuit side.
- Added inner ground layer pads.
- Re-routed traces.
- Stretched a re-routed trace.
- Added traces and tooling marks.

Comparing Connection Lists and Creating Artwork

In this section you will prepare a connection list from the PC Board and then use the Compare Connect Lists post-processor to compare the shot_bd and the one_shot connection lists. This process insures that the two drawings are logically identical.

You will also use the SHOW* and PLOT* commands to prepare a few different pieces of artwork from the shot_bd drawing. The artwork produced is for a pen plotter. If you want to use the photoplot post-processor to prepare more exact output, see *Understanding HP EGS* for more information.

Before you create the connection list file, here are the rules for physical connection listing:

- Connections touching a port are connected.
- If any portion of a connection touches another, they are connected.
- Connection nets with the same associated net name are connected.

Notice that these rules are slightly different than those for generating a connection list for a schematic. Here, connections that cross are connected, whereas in a schematic a junction is required to make crossing connection lines connect.

Archiving the Drawing

Your drawing must be archived using the :P and :M options before the Connection Lister can process it.

1. Enter:

```
ARCHIVE :P :M shot_bd;
```

2. To create the connection list, you must first exit the Printed Circuit Board Layout personality. Enter:

```
Quit;
```

to return to the Electrical Engineering menu in the manager, which now looks like this:

```
Use arrow keys to move cursor to desired function.
Select function with <RETURN> or <ENTER>.
    RETURN To Main Menu
  ->Printed Circuit Board Layout Editor
    Electrical Schematic Drawing Editor

    EE Parts File Editor
    EE Material Lister
    Convert Old EE Parts Files
```

```
Printed Circuit Board Connection Lister
Electrical Schematic Connection Lister
Compare Connection Lists
Merge Connection Lists
Generate Rat's Nest

PHOTOPLOT/DRILL Post-processor
```

Making a Connection List

1. To enter the Printed Circuit Board Connection Lister, select:

```
Printed Circuit Board Connection Lister
```

2. The following text appears briefly on the screen:

```
Loading 'EWRC:verify.code'
Enter MESSAGE FILE name? EWRC:con
Initialization in progress
```

The first prompt asks for the name of the message file that tells the system how to create a connection list. As the previous example illustrates, the default message file, EWRC:con, is provided by the system.

Next the Connection Lister prompts for the name of the file containing default values to be used. HP EGS provides the file name, EWRC:pcbphy.

```
Enter DEFAULT FILE name? EWRC:pcbphy
```

3. The Connection Lister prompts for the name of the ARCHIVE FILE that you want to process:

```
Enter ARCHIVE FILE name?
```

When you enter the name of the archive file, it is not necessary to include the suffix `_a`. If you are using an SRM, type the full pathname of the file. If you are using local mass storage, type:

```
DRAW:shot_bd
```

4. The Connections Lister prompts for the destination of the connection list.

```
Enter OUTPUT DESTINATION?
```

If you press **Return** the Connection Lister output is placed in the same volume as the input file (in this case DRAW:). If you want the output in another volume, you can enter the volume name now. You want the output in DRAW:, so press **Return**.

5. The system presents a menu listing the available Connection Lister options:

```
Use the Arrow Keys to position cursor.
Press RETURN or ENTER to edit a field.

  ->QUIT
    Re-set to Default File         = EWRC:pcbphy.ASC
    Physical or Logical Rules      = Physical
    Flat or Hierarchical List      = Flat
    Reference Designator Paths     = External only
    Smash to Logical Level         = 0
    Components to Include          = -E #I #E51 #E52 #E53 #E54 #E5 #E6 #E7 #E8
                                      #E11 #E12 #E13 #E14 #E29
    Net Properties Tag List        =
    Port Properties Tag List       =
    Instance Properties Tag List   = 1004
    Net or Instance List Order     = Net
    Output Destination             = DRAW:
    Archive File to Process        = DRAW:shot_bd_a
    RUN Connection Lister
```

On the right side of the menu the Connection Lister displays values found in the default file (EWRC:pcbphy) for each option and the names of the output volume and the archive file you entered in steps 3 and 4.

Because this example uses values found in EWRC:pcbphy for the menu items, you are ready to run the connection lister.

Move the -> to `RUN Connection Lister` and press **Return**.

The Connection Lister begins by displaying the option menu to verify any changes you may have made. In this case, you did not make any changes so the menu is the same. You could still change the menu options by pressing **Stop** to return to the menu, but let the Connection Lister continue.

The Connection Lister will output dots on the screen to indicate it is processing the archive file.

When the connections list is complete the following message is displayed:

```
Connection list successfully generated.

Starting memory available              = 750798
Total memory used                      = 99318
Overhead memory used                   = 58968
Drawing memory used                    = 20994
Processing memory used                 = 19356

All processing complete.

Press space to continue.
```

The connection list is stored in the file shot_bd_c. (If you have made any errors, a file named shot_bd_e is also created. See Chapter 8, *Electrical Schematic Drawings*, to understand this error file.)

The connection list file appears as follows. (You can look at your file later with the Pascal editor if you want.)

```
|shot_bd 1

1
     to58|U1|6|SE555T|
     to58|U1|7|SE555T|
     r500|R1|?1|10K|
     c300|C1|?2|0.1|

2
     to58|U1|5|SE555T|
     c300|C2|?2|0.01|

3
     to58|U1|4|SE555T|
     to58|U1|8|SE555T|
     r500|R1|?2|10K|

4
     to58|U1|3|SE555T|

5
     c300|C1|?1|0.1|
     to58|U1|1|SE555T|
     c300|C2|?1|0.01|

6
     to58|U1|2|SE555T|
```

The header in the file contains the file name and a list of errors generated. After the header is a numbered list of the connection nets that were found. The net numbers may be followed by net names, as in the schematic example, but in this case they aren't.

Each connection in a net is identified by one line in the list. For example, the first connection in net 5 is written as 'c300/C1/?1/0.1/'. This indicates a connection in the c300 part, which has a reference designator of C1. The actual port has a physical name of ?1 and a value (or part type in some cases) of 0.1. The preceding list is the exact format of the file, so you can see that a post-processor can be generated to manipulate this file any way you want.

Comparing Connection Lists

1. To compare this connection list to the one generated from the schematic, you must first exit the Printed Circuit Board Layout personality by entering:

   ```
   QUIT;
   ```

   This returns you to the Electrical Engineering menu in the manager, which now looks as follows:

   ```
   Use arrow keys to move cursor to desired function.
   Select function with <RETURN> or <ENTER>.
       RETURN To Main Menu
       Printed Circuit Board Layout Editor
       Electrical Schematic Drawing Editor

       EE Parts File Editor
       EE Material Lister
       Convert Old EE Parts Files

     ->Printed Circuit Board Connection Lister
       Electrical Schematic Connection Lister
       Compare Connection Lists
       Merge Connection Lists
       Generate Rat's Nest

       PHOTOPLOT/DRILL Post-processor
   ```

2. Select:

   ```
   Compare Connection Lists
   ```

   and wait to be prompted for the input and output file names. It is important for you to include the prefix to the volume where you placed the original files (EWYOU: is used here to indicate this). The display reads:

   ```
   Enter the name of the first connect list file:
   ```

3. Enter:

   ```
   EWYOU:one_shot
   ```

   and the _c is automatically appended. The following display appears:

   ```
   Enter the name of the second connect list file:
   ```

4. Enter:

   ```
   EWYOU:shot_bd
   ```

   Again, the _c is automatically appended. The following display soon appears:

   ```
   Enter the name of the file for the results:
   ```

5. Enter:

```
EWYOU:compar
```

or `CONSOLE:` or `PRINTER:` as you desire. Answer yes to the following three prompts by pressing (Y):

```
Should nets with only a single port be ignored? (y/n)
Should net mis-matches (errors) be reported? (y/n)
Should net matches be reported? (y/n)
```

The post-processor indicates its progress. When it is done processing it displays:

```
Done.
Press <space> to continue.
```

6. Press the space bar to return to the Electrical Engineering menu.

The results obtained from the comparison are listed below. If you sent them to a file you can later edit the file with the Pascal editor.

```
                  HP EGS Connect List Comparison
Report Matching Nets
Report Nets That Do Not Match
Ignore Nets With Only One Port
         MANUAL:shot_bd_c       MANUAL:one_sh1_c
   1                      |  1
   2                      |  2
   3                      |  5
   5                      |  3
Connect Lists match exactly.
```

Notice that your specifications and the numbers of the matching nets are included in the file. The bottom line indicates that the two files did compare exactly.

The Electrical Engineering menu should be displayed as follows:

```
Use arrow keys to move cursor to desired function.
Select function with <RETURN> or <ENTER>.
    RETURN To Main Menu
    Printed Circuit Board Layout Editor
    Electrical Schematic Drawing Editor

    EE Parts File Editor
    EE Material Lister
    Convert Old EE Parts Files

    Printed Circuit Board Connection Lister
    Electrical Schematic Connection Lister
  ->Compare Connection Lists
    Merge Connection Lists
    Generate Rat's Nest

    PHOTOPLOT/DRILL Post-processor
```

7. Return to the Printed Circuit Board Layout personality by selecting:

    `Printed Circuit Board Layout Editor`

    When the personality is loaded again, prefix to your personal mass storage volume. Then edit the original PC board by selecting:

    `Edit_PCB`

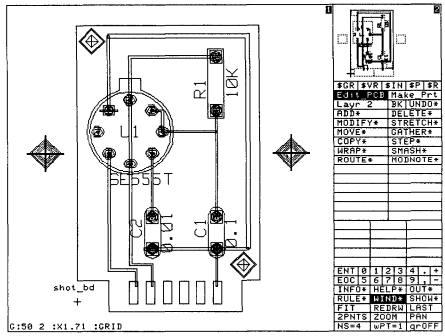
8. When prompted for the PC Board name, enter:

    `shot_bd`

    The shot_bd drawing should now appear on your screen. Next, set the nesting level to 4 and redraw the screen so that you can see inside the library parts. Select:

    `WIND* NS=1 4 ENT REDRW`

    Your screen should look like this:



Creating the Assembly Drawing

The first piece of artwork to create is the assembly drawing, which can also be used as the silk screen mask.

1. Start off by selecting:

    `SHOW*`

2. Turn on the layer menu by selecting:

    `sLAYR`

3. Turn off the display of all the layers by selecting:

    `OFF ALL EOC`

4. Turn on the library parts, the reference layer, the assembly layer, the board blank layer, and the reference designator layer by selecting:

`SHOW* ON PARTS ON ALL =Refrnce ON ALL =AsmbDra ON ALL =BdBlank ON ALL =RefDesi EOC`

5. To plot this piece of artwork to a local plotter, make sure your plotter has paper and pens in it and is ready to plot. Then select:

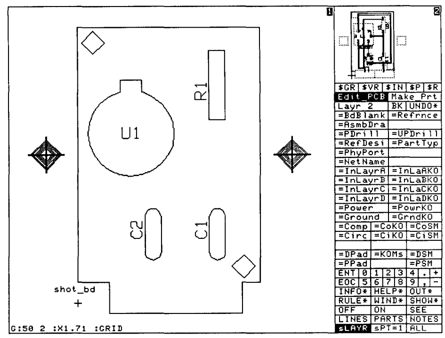
    `OUT* PLOT* FILL VLCTY 20 ENT SCALE 2 ENT CENTR EOC`

Recall that VLCTY 20 sets the plotter pen velocity to 20 cm/sec, SCALE 2 makes the plot two times larger than the original, and CENTR places the plot in the center of the paper. The FILL method specifies that all primitives with width are to be filled on the plot.

Note

The trace and pad widths on the plot you produce may not be the correct size because the width of the pen may not be specified correctly. To specify an exact pen width, see the PLOT command in the *HP EGS Syntax Reference*.

Your screen should look like this:



Creating the Circuit Side Physical Layer

The next piece of artwork to create is the circuit side, physical layer mask.

1. Select:

   ```
   SHOW*
   ```

2. Turn off the display of all the layers by selecting:

   ```
   OFF ALL EOC
   ```

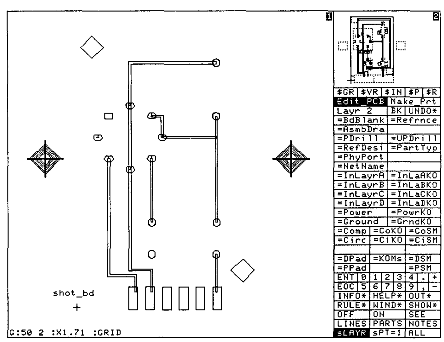
3. Turn on the library parts, the reference layer, the plotting pad master, and the circuit side layer by selecting:

   ```
   SHOW* ON PARTS ON ALL =Refrnce ON ALL
   =PPad ON ALL =Circ EOC
   ```

4. To plot this mask to a local plotter, make sure your plotter is ready. Then select:

   ```
   OUT* PLOT* FILL VLCTY 20 ENT SCALE 2 ENT
   CENTR EOC
   ```

   Your screen should look like this:



Creating the Ground Physical Layer

The final piece of artwork to create in this example is the ground physical layer mask.

1. Select:

   ```
   SHOW*
   ```

2. Turn off the display of all the layers by selecting:

   ```
   OFF ALL EOC
   ```

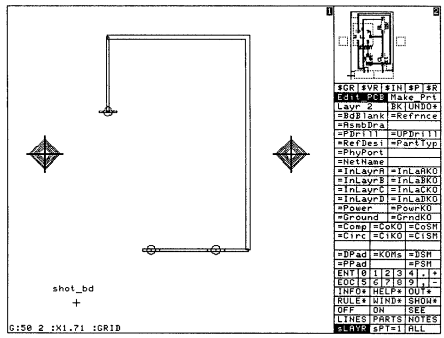
3. Turn on the library parts, the reference layer, and the ground layer by selecting:

   ```
   SHOW* ON PARTS ON ALL =Refrnce
   ON ALL =Ground EOC
   ```

4. To plot this mask to a local plotter, make sure your plotter is ready. Then select:

   ```
   OUT* PLOT* FILL VLCTY 20 ENT SCALE 2
   ENT CENTR EOC
   ```

   Your screen should look like this:



BREAK TIME

Relax, stretch, review, and take a break. You are now done with the Printed Circuit Board Layout drawing example. In this section you:

- Made a connection list.
- Compared connection lists.
- Created the artwork for the assembly drawing.
- Created the artwork for the circuit side physical layer.
- Created the artwork for the ground physical layer.

Connecting Traces Between Physical Board Layers

When you worked through the example in this chapter, you may have wondered how the connection listing process determines connections, especially between layers. This portion of the manual discusses how connection listing for PC boards works.

First is a brief discussion of the concepts involved. Then the concepts are illustrated with information on how to make workable inner layer pads, vias, and edge connectors.

Concepts

On any individual layer, connections are made where geometric components overlap or touch. But connections between layers must be generated in some other way.

The fundamental concept here pertains to logical locations of ports. Recall that when you added the physical port names you also specified a logical port location. However, you couldn't see that the PHY_PORT macro also specifies layers for this logical port location. This is because of the following rule:

Rule

Net elements on different layers can be connected by: 1) defining a port at the point of intersection; and then 2) defining this port on all of the layers to be connected.

For example, if you wanted to insure that traces on both the circuit and component sides of a PC board that touch the same pin on a dip are connected, you would follow these steps:

1. Draw a pad on the circuit side of the PC board (layer 5), and then draw another pad on the component side of the PC board (layer 6).
2. Next you would label this 'pad set' with a physical port by adding a note to the physical port layer (layer 51), specifying the logical location to be on layers 5 and 6 with a logical x,y location in the center of the 'pad set'.
3. You would then add traces to either side of the PC Board. Any traces that touch the pad on the circuit side of the board are then defined to be connected to the traces that touch the pad on the component side of the board.

This procedure works well for pad sets that are built containing pad geometry on all neccessary layers. But recall from the beginning of this lesson that the parts provided with this personality all obtain the pad geometry from the pad master layer (layer 29). The following connection listing rule should help explain how this works:

Rule

Components found on master layers are treated as if they existed as separate components, one on each of the layers to which the master layer is bound.

Recall that the pad master layer (layer 29) is bound to both the component and circuit side layers (layers 5 and 6). The above rule means that the connection listing operation internally copies the pad geometry from the pad master layer to both the circuit and component side layers.

So now you know how traces on the circuit side connect to their pads which are actually drawn on the master pad layer. But to find out how the two sides of the board get connected, let's look at another connection listing rule:

Rule

A port with a logical location on a master layer will be processed as if the logical location exists on all of the layers to which the master layer is bound.

This rule means that when you label a pad on the master pad layer with a physical port name, you need only to specify the pad master layer (layer 29) as the layer for the port's logical location.

Let's follow the above example using these new concepts. Again, you want to ensure that traces on both the circuit and component sides of a PC Board that touch the same pin on a dip are connected.

1. First you would draw a pad on pad master layer (layer 29).
2. Next you would label this pad with a physical port by adding a note to the physical port layer (layer 51) specifying the logical location to be on layer 29 with a logical x,y location in the center of the pad set.

When you built the to58 library part in this lesson, you did step 1 above when you added the pad parts, and step 2 when you used the PHY_PORT component macro.

Inner Layer Pads

So how does an inner layer pad get connected to the circuit and component sides of the board? Let's examine the one you used in this lesson, IP85GND. This library part consists of:

- An 85-mil circular pad with a small horizontal rectangle, on the ground layer (layer 8).
- A physical port name of 'P' (on layer 51) with a logical location on the ground layer and the pad master layer (layers 8 and 29), and a logical x,y location in the center of the pad.
- A reference designator of 'VIA'.

The geometry on the ground layer (layer 8) enables you to easily connect your traces to this pad. The logical location on the ground layer and the pad master layer (layers 8 and 29) actually specifies that the logical location is on the circuit side, component side, and the ground layer (layers 5, 6 and 8).

Therefore, any trace on the inner ground layer which touches the inner ground layer pad is connected to any trace, pad, or other component which is over the center point of the pad (logical x,y location of the port name) in the circuit or component side of the PC Board.

The 'VIA' reference designator indicates to the connect lister that the IP85GND part should not be included in the connection list output file, although the connectivity of the net is to be maintained.

Follow these steps to build an inner layer pad part:

1. Select the Make_Prt mode of the PC Board personality.
2. Add circles, rectangles, or other components to define the pad. You may also place pad geometry on multiple layers.
3. Add a reference designator of 'VIA' to this part, since you don't want to have the port name on this part included in connection list output files. Use the REF_DESI component to do this.
4. Add a physical port name of 'P' (any name will work) with a logical location on the pad master layer (layer 29) and all layers where you added part geometry. To do this select:

   ```
   PHY_PORT *PLYR
   ```

   The *PLYR option enables you to explicitly specify all of the logical location layers you want. Next, enter:

   ```
   [29,L1,L2,...,LN]
   ```

Where L1,L2,...,LN are all of the layers with your part geometry (you must include the square brackets). Then select the logical x,y location for the port, and enter:

```
'P'
```

and select the location for the port name.

5. In most cases you will never want to display the reference designator' or the port name. To turn off the display of the reference designator, select:

```
MODNOTE*
```

Select the reference designator, and then enter:

```
:D
```

Terminate the modification with EOC. Note that :D is an option that turns on or off the display of associated text. You can repeat this process to turn off the display of the port name.

6. Save the part.

Recall from the lesson that the command WIN :A; enables you to turn on and off un-displayed associated text.

VIAs

Here is how you could make a via which only connects the circuit and component sides of the PC Board.

1. Select the Make_Prt mode of the PC Board personality.
2. Add a primitive pad (LP series). Note that the pad geometry is defined on the pad master layer (layer 29). Also, solder mask and drill hole geometry is included in these parts.
3. Add a reference designator of 'VIA to this via, because you don't want to include the port name on this part in connection list output files. Use the REF_DESI component to do this.
4. Add a physical port name of 'P' (any name will work) with a logical location on the pad master layer (layer 29) and an x,y location in the center of the pad. You need only to use the PHY_PORT component, responding correctly to the prompts.
5. Again, you will likely never want to display the reference designator or the port name. To turn off the display of the reference designator, select:

```
MODNOTE*
```

6. Select the reference designator, and then enter:

```
:D
```

7. Terminate the modification with EOC. Then repeat this process to turn off the display of the port name.
8. Save the part.

It is important to understand that this via connects *only* the circuit and component sides of the PC Board. To connect inner layers you must add the appropriate inner layer pads to the drawing as you did in the lesson.

Edge Connectors

In the example of this lesson you constructed an edge connector that was not included in the connect list output because it did not include physical port names. If you want to build one that will be included in connection lists, use the following procedure:

1. Select the Make_Prt mode of the PC Board personality.
2. Add as many edge connector fingers as you want, then add the edge connector board blank parts, or otherwise complete the ends of the edge connector. Note that the edge connector fingers are made up of two rectangular pads, one on the circuit side and one on the component side, as well as a single line on the board blank layer.
3. Use the REF_DESI component to add any reference designator of your choice to this part.
4. Add physical port names to each of the fingers of this edge connector. Before doing so, you must decide if you want the circuit and component sides of the fingers to be connected or separate.

   If you want them connected, you need to add one physical port name for each finger in the edge connector. Use the PHY_PORT component, because it will place the logical location on the pad master layer (layer 29) and therefore set it to the circuit and component sides (layers 5 and 6). This causes a connection between the two rectangles of the finger. Be sure that the logical x,y location for each finger is inside the rectangle in that finger.

   If you want them separate, you must add two physical port names for each finger, one for each side.
5. To restrict the logical layer location to only one side of the PC Board, select:

   ```
   PHY_PORT *PLYR
   ```
6. For the circuit side of the board (layer 5), enter:

   ```
   [5]
   ```
7. Then select the logical x,y location for the port (inside the finger rectangle), and finally enter the port name and select its location.

8. To repeat this process for the remaining fingers, use:

```
*AGIN *PLYR [5] ENT
```

each time.

9. For the component side of the board (layer 6), add physical port names to all of the fingers of the edge connector again but each time use:

```
*AGIN *PLYR [6] ENT
```

10. Save the part.

Printed Circuit Board Library Parts

To help you start drawing PC boards, this personality provides a set of library parts. These library parts are stored in the default volume for this personality, EWEE. The conventions used to construct these library parts are the same ones you used to create the to58 library part. The library parts conform to the process file, pc_pr.TEXT, provided with this personality. Each library part consists of components: lines, rectangles, circles, and instances of other library parts added to specific layers.

Because each PC fabrication shop has its own set of specifications for PC board layout (such as tolerances, pad sizes, and drilled hole sizes), you will want to create your own set of library parts. Use the library parts supplied with this personality as a reference for constructing your own set of library parts.

Library parts are often created using other library parts as a foundation. For example, PAD25S actually consists of an instance of the library part LP25S and a 100x100 mil area on process layer 4, the keepout master. This means you must be careful when modifying a library part, because all parts using this part are modified as well as the specific part.

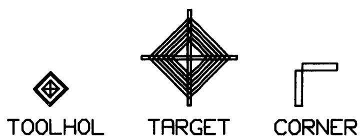
The following subsections explain the supplied library parts.

Reference Library Parts

Reference library parts are components added to a PC board layout to align the physical board layers. The reference parts supplied are a tooling hole, a shear mark (also called a corner), and a registration target.

Part	Description
TOOLHOL	Provides reference points to help locate and position the PC board for machining. The origin of this library part is at the center of the pads and drilled hole.
CORNER	The library part shear mark. It is required at each corner of the PC board to bracket the board outline for the routing process. Since only one orientation of the device CORNER is supplied, you will need to use the *MIR and *ROT options of the ADD* command to get the other three orientations. The origin of the library part CORNER is located at the intersection of the two line segments that form the part.
TARGET	The target registration mark used to align the different plots that comprise a physical board layer. It is also the mark used to align the physical layers to form a board.

The following figure illustrates the reference library parts.



Reference Library Parts

Primitive Pads

Primitive pads represent outer layer pads on the master layers for copper and solder resist keepout areas and a drilled hole in your drawing. Each primitive pad library part is named according to the finished hole size and the shape of the pad.

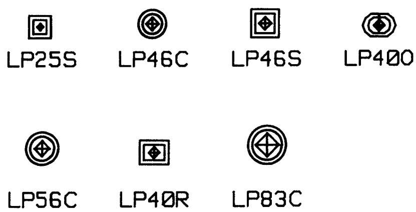
The two leading characters of the name (LP) in each primitive pad name identify the library part as a primitive pad. The next two characters in the library part name are digits that represent the finished hole size, in mils. The final character in the library part name is a letter specifying the shape of the pad: (C) for circular pads, (R) for rectangular, (S) for square, and (O) for oval.

Each primitive pad consists of an outer layer pad shape on process layers 3 and 29, the pad master process layers, a solder resist keepout shape on process layers 20 and 30, the solder resist keepout process layers, and a drill hole circle on process layer 1. The origin of each primitive pad library part is the center of the pad, which is also the center of the drill hole.

The solder resist keepout for the library part LP40O appears as a polygon in the shape of an oval on the display layer (30), and a line with width wrapped in on itself, to allow filling, in the shape of an oval on the pen plot layer (20).

The pad for the LP40R library part is the same as the LP40O part except that it consists of rectangles on the display layer (29) and pen plot layer (3).

The following figure illustrates primitive pads.



Primitive Pads

Primary Pads

Primary pads are groups of components that represent the components required for a plated-through hole on a PC board. Primary pads are named in the same fashion as primitive pads: two digits in the library part name specify the finished hole size of the library part, and a trailing character, (C) for circular, (O) for oval, (S) for square, or (R) for rectangular, that identifies the shape of the outer layer pad.

Each primary pad library part consists of a square in the keepout master layer (4) and an instance of a primitive pad library part. If you follow the techniques and concepts described with the process file, all of the components necessary for creating a plated-through hole are present in each primary pad library part.

The following figure illustrates primary pads.

Primary Pads

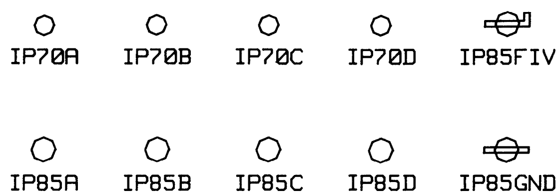
Inner Layer Pads

Inner layer pads are parts made of components that represent pads on inner layers which connect to plated-through holes. The names of the inner layer pad library parts begin with the two characters, IP. The next two characters of the library part name are digits representing the diameter of the inner layer pad. Pads with a diameter of 85 mils are designed for connecting to plated-through holes with a finished hole size of 40 or 46 mils. Pads with a diameter of 70 mils are designed for connecting to plated-through holes with a finished hole size of 25 mils.

The remaining characters in the library part name identify the process layer in which the inner layer pad is to be added. For example, the inner layer pad library part IP85C consists of an 85 mil diameter pad in process layer 13, the process layer reserved for positive components on inner layer C.

Two inner layer pad library parts, IP85FIVE and IP85GND, also provide a 20 mil wide line in the appropriate inner positive layer that bisects the inner layer pad and extends beyond the boundary of the pad. These lines connect the inner layer pad to traces and power and ground planes located in the same layer as the inner layer pad.

The following figure illustrates inner layer pads.



Inner Layer Pads

Analog Parts

The analog parts consist of two resistors and two capacitors. These parts consist of two primary pads and a part outline on the assembly layer (26).

The following figure illustrates analog parts.

Analog Parts

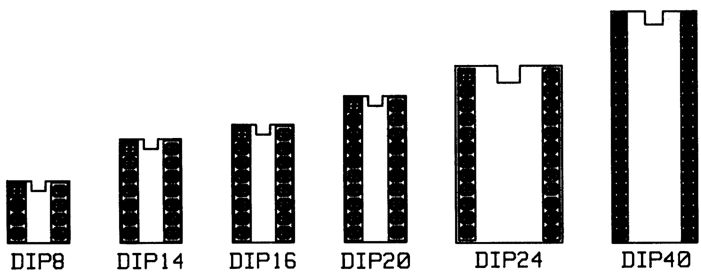
I.C. DIP Parts

I.C. DIP (Integrated Circuit Dual In-line Package) library parts are composed of components representing the pads, drilled holes, keepout areas, and an I.C. DIP. Each I.C. DIP library part consists of two 120 mil wide rectangles in the keepout master layer (4), an assembly outline in the assembly layer (26) and two columns of primitive pads.

The I.C. DIP library parts use instances of the primitive pad LP40O to provide oval pads for all of the DIP pins, except for pin number one. Pad LP40R is used to provide the pad for pin number one, since it provides a rectangular pad rather than an oval one.

The dimensions of each DIP library part are the same. For example, the physical spacing between adjacent pins of a dip are the same, and the physical distance between rows of pins are the same; only the number of pins changes.

The following figure illustrates I.C. Dip parts.



I.C. Dips

Edge Connectors

Edge connectors are made of components that represent connector fingers and the board blank outlines of an edge connector. Each edge connector library part represents a portion of a final edge connector. These portions of edge connectors can be combined in various combinations to provide the edge connector that you need in a specific application.

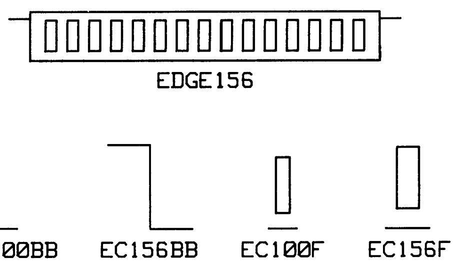
Edge connector fingers consist of a connector pad on the pad master process layers (3 and 29) and a line representing the edge of the board on the board blank layer (2). No keepout regions are provided for the solder resist process layer. The origin of the connector finger is at the intersection of the centerline of the pad and the board edge. The finger is oriented for a connector along the bottom edge of the board. However, you can get any orientation by using the *ROT and *MIR options of the ADD* command.

Two different connector fingers are provided in the library: EC100F and EC156F. These two fingers are provided to allow for different finger size and finger spacing requirements of an edge connector. EC100F is for connectors with a 100 mil spacing and consists of a 50 mil by 250 mil pad in the pad master process layers and a line of zero width in the board blank process layer.

The part EC156F is for connectors with a 156 mil spacing and consists of an 80 mil by 300 mil pad in the pad master process layers and a line of zero width in process layer two, the board blank process layer.

Note that the edge connector fingers provided with HP EGS have a reference designator of 'IGNORE'. This keeps them from being included in connection list output files.

Board blank outline parts, EC100BB and EC156BB, consist of lines on the board blank process layer (2) which form the outline of the edge connector. Two different sizes of board blank outlines are provided: one for edge connectors with 156 mil spacing and one for edge connectors with 100 mil spacing. Each part is oriented for an edge connector along the bottom edge of the board. Again, you can obtain different orientations by using the *MIR and *ROT functions of the ADD command.



Edge Connectors

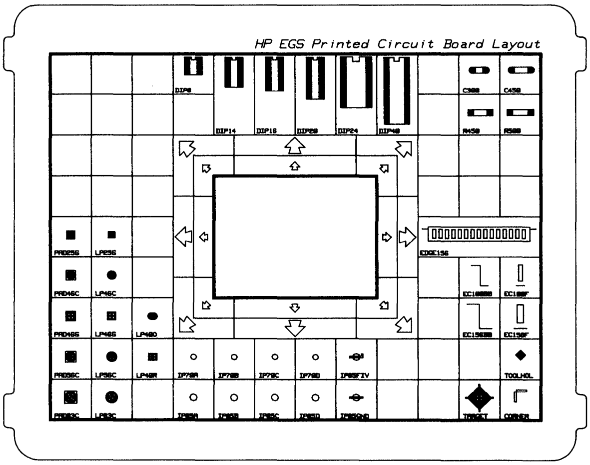
The library parts were created using the units of this personality and many different display grids. However, the display grids are always in multiples of 5 mils. Most of the pads were added at increments of 25 mils, so a display grid of (25,4) or (50,2) is recommended when using these parts in a drawing. The library parts supplied with this personality contain default pieces of text for material and connection listing. Like the part you created, these include:

- Physical Ports
- Part Type
- Reference Designator

All the furnished library parts have been saved at logical level 0, so that the ARCHIVE and LIST_MAT commands can distinguish base level parts from other parts.

The Printed Circuit Board Layout Tablet Menu

The tablet menu provided for use with the Printed Circuit Board Layout personality is shown in the following figure.



Printed Circuit Board Layout Tablet Menu

This menu is set up for the tablet rather than the screen so that you can select library parts without having to use the *SCRN option in the ADD* PART command. You may also find this tablet menu convenient because it gives you a picture of the parts as well as the name of the parts.

Also notice that the tablet menu has some port manipulation commands around the screen tracking block. These commands enable you to pan the large port in the direction of the arrows with a single stroke. The large arrows represent full window pans while the small arrows represent half window pans.

See Appendix A in this manual to plot this tablet menu, or Appendix B to add a new part to this tablet menu.

Appendices

Plotting and Activating Tablet Menus

Appendix A

This appendix describes how to plot and activate the tablet menus of the four HP EGS personalities. The Mechanical Drafting, Electrical Schematic, and Printed Circuit Board Layout tablet menus already have textual and graphical items that can be used for the particular personality. The General Drawing tablet menu is blank and can be customized for your application. Appendix C outlines how to add items to one of these four tablet menus. Chapter 4 of *Understanding HP EGS* describes how to create a tablet menu from scratch.

Although the tablet menus of the four personalities differ, the procedure for plotting and activating them is similar. The following directions note where the procedure is different for a particular personality.

1. You need to have on hand a plotter:
   - To produce an HP 9111A graphics tablet menu, you need a plotter that accommodates size C or larger paper.
   - To produce an HP-HIL 46087A graphics tablet menu, you need a plotter that accommodates size A or larger paper.
   - To produce an HP-HIL 46088A graphics tablet menu, you need a plotter that accommodates size B or larger paper.
2. Call up the appropriate personality for the tablet menu you plan to plot. See the introductory sections of *Learning HP EGS* if you do not know how to call up the personalities.
3. Enter the EDIT command for the appropriate personality:

Personality	Command
General Drawing	EDIT gdtmenu;
Mechanical Drafting	EDIT metmenu;
Schematic Drawing	EDIT schtmenu;
Printed Circuit Board Layout	EDIT pctmenu;

4. If you are making the tablet menu for an HP-HIL graphics tablet, delete the border containing the tabs which surrounds the tablet menu.

5. Select from the screen menu:

```
WIND* NS=1 10 ENT REDRW
```

or enter:

```
WIN :n10;
```

6. Enter the SHOW command for the appropriate personality:

Personality	Command
General Drawing	SHOW −e3;
Mechanical Drafting	SHOW −e8;
Schematic Drawing	SHOW −e50 −e51 −e52 −e53 −e54;
Printed Circuit Board	SHOW −e29 −e30 −e50 −e51 −e52 −e53 −e54;

7. Load the plotter paper:

Graphics Menu	Paper Size
HP-HIL 46087A	A-size or greater
HP-HIL 46088A	B-size or greater
HP 9111A	C-size or greater

8. Put black pens in the plotter pen stalls. A fine tip pen (such as P.3) is suggested.
9. To plot the tablet menu:

For an HP 9111A graphics tablet–

On a local plotter select:

```
OUT* PLOT* SCALE 1 ENT CENTR VLCTY 20 ENT EOC
```

To plot to a remote HP 7586B plotter, select:

```
OUT* PLOT* SCALE 1 ENT CENTR VLCTY 20 ENT SPOOL tabmen ENT C ENT
7586B ENT EOC
```

For an HP-HIL 46087A graphics tablet–

On a local plotter select:

```
OUT* PLOT* SCALE .80 ENT CENTR VLCTY 20 ENT EOC
```

To plot to a remote HP 7586B plotter, select:

```
OUT* PLOT* SCALE .80 ENT CENTR VLCTY 20 ENT SPOOL tabmen ENT A ENT
7586B ENT EOC
```

For an HP-HIL 46088A graphics tablet–

On a local plotter select:

```
OUT* PLOT* SCALE 1 ENT CENTR VLCTY 20 ENT EOC
```

To plot to a remote HP 7586B plotter, select:

```
OUT* PLOT* SCALE 1 ENT CENTR VLCTY 20 ENT SPOOL tabmen ENT B ENT
7586B ENT EOC
```

10. When the plotter finishes, cut out the tablet menu and place it on your graphics tablet.
11. Enter SAVE TMENU for the appropriate personality:

Personality	SAVE TMENU Command
General Drawing	SAVE TMENU gd;
Mechanical Drafting	SAVE TMENU me;
Schematic Drawing	SAVE TMENU sch;
Printed Circuit Board	SAVE TMENU pc;

The system asks you if the current tablet menu should be removed from mass storage. Enter:

```
Y
```

12. The system then prompts you to select the four corners of the tablet menu. These are the corners of the outside rectangle of the tablet menu. When you have selected the corners, the system should display:

```
The tablet menu is active.
```

You can now use this tablet menu while you remain in the current personality. To add any of the items displayed on the tablet menu, select ADD from the screen menu (or enter ADD from the keyboard) and then select the tablet menu item. If you quit the personality and re-enter later, the system will automatically load and activate the tablet menu.

If you want to turn off the tablet menu at any time, enter:

```
TMENU ;(Return)
```

TMENU is a toggle that turns the tablet menu on and off.

Modifying Parts Screen Menus

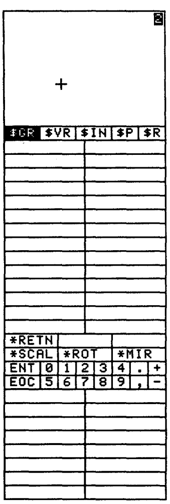
Appendix B

This appendix describes how to add command, macro, or part names to the parts screen menus of the four HP EGS personalities.

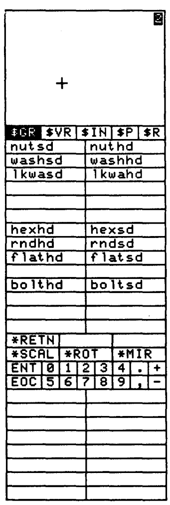
1. Call up the appropriate personality.
2. Select the commands to display the parts menu of the appropriate personality.

Personality	Commands
General Drawing	ADD* INSTANCE *SCRN
Mechanical Drafting, Schematic Drawing, and Printed Circuit Board	ADD* PART *SCRN
Mechanical Drafting Annotation Mode	ADD* SYMBOLS *SCRN

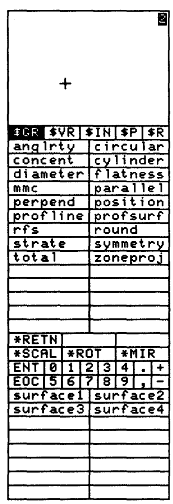
The parts menus are shown below:



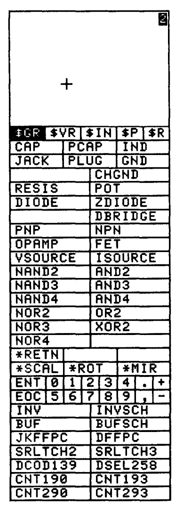
General Drawing



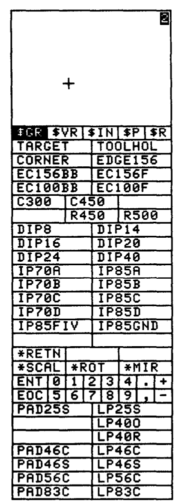
Mechanical Drafting



Mechanical Drafting Annotation Mode



Schematic Drawing



Printed Circuit Board Layout

You can add command, macro, or part names to any of the empty spaces of the parts menus without disturbing the personality function. If you decide you do not need a particular part on the menu, you can replace the part with the new menu item. You cannot, however, replace any of the menu items shown in the General Drawing parts menu. These same items cannot be replaced in any of the other personality parts menus.

3. To add a command, macro, or part name, enter:

   ```
   MENU
   ```

   and the text within quotes. Select the appropriate menu slot with the stylus.

4. After you have finished modifying the screen menu, enter:

   ```
   SAVE MENU;
   ```

   The system asks you if you want to overwrite the current screen menu. Enter:

   ```
   Y
   ```

If you quit the personality and re-enter later, the system will automatically load and activate the screen menu.

Modifying Tablet Menus

Appendix C

This appendix describes how to add textual or graphical parts to the tablet menus of the four HP EGS personalities.

1. Call up the appropriate personality.
2. Enter the EDIT command for the appropriate personality:

Personality	Command
General Drawing	EDI gdtmenu;
Mechanical Drafting	EDI metmenu;
Schematic Drawing	EDI schtmenu;
Printed Circuit Board Layout	EDI pctmenu;

3. Select from the screen menu:

```
WIND* NS=1 10 ENT REDRW
```

or enter:

```
WIN :n10;
```

4. Enter the SHOW command for the appropriate personality:

Personality	Command
General Drawing	SHO −e3;
Mechanical Drafting	SHO −e8;
Schematic Drawing	SHO −e50 −e51 −e52 −e53 −e54;
Printed Circuit Board	SHO −e29 −e30 −e50 −e51 −e52 −e53 −e54;

5. If you plan to add a part from a directory or volume not listed in the current SEARCH file, add this directory or volume name to it.
6. Enter ADD and the part name. If the part was created in another personality the system may ask you if you want to change the units. Enter `N`.
7. Place the part in an empty rectangle on the tablet menu drawing. If the part is too large or small for a particular area, enter `:X` and a scale factor that enlarges or shrinks the part to the correct size.

8. Enter:

```
ADD N :T1010 :AC
```

   and select the rectangle that surrounds the part you just added.
9. Enter the text string that associates the part to the tablet menu. For example, if your part is called nand2, enter:

```
"nand2 "
```

NOTE

A space must separate the associated text from the second quote.

---

10. Select the highlighted rectangle that surrounds the part you just added. If the boundary rectangle of the text is too large or small for the highlighted rectangle, enter `:F` and a font size that enlarges or shrinks the text to the correct size.
11. Repeat steps 5 through 10 for each part you wish to add to the tablet menu.
12. Enter the SAVE command for the appropriate personality:

Personality	SAVE Command
General Drawing	save gdtmenu;
Mechanical Drafting	save metmenu;
Schematic Drawing	save schtmenu;
Printed Circuit Board	save pctmenu;

   The system asks you if you wish to overwrite the current tablet menu. Enter `Y`.
13. Enter:

```
SAVE SEARCH;
```

   to save the search file. This is necessary if you modified the current search file before adding any parts to the tablet menu.
14. Plot out and save the tablet menu as described in Appendix A.

You can now use this tablet menu while you remain in the current personality. To add any of the items displayed on the tablet menu, select ADD from the screen menu (or enter ADD from the keyboard) and then select the tablet menu item. If you quit the personality and re-enter later, the system will automatically load and activate the tablet menu.

If you want to turn off the tablet menu, enter:

```
TMENU ;(Return)
```

TMENU is a toggle that turns the tablet menu on and off.

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