p-cad

From ACCEL Technologies, Inc.

Master Designer

Version 8.5

Tutorial
Volume 2

October 1995

October 1995

Contents

Conventions

This manual uses the following conventions:

! or →	Connects commands. Commands following the arrow appear on submenus. For example File!Load or File→Load
*	One or more characters can occupy the asterisk's position. Also known as a wildcard.
.	In text, introduces a procedure that explains how to do a task.
Ctrl-F	Press the keys simultaneously.
↵	The return key. Press this key after typing data in a data entry box or on a message line or to accept a default. You can click left in many P-CAD tools instead of pressing ↵.
Enter	Indicates you need to press ↵ after typing data.
Spacebar	The space bar. You can use this key to digitize a point within the drawing area.
italics	Indicates variable characters.
boldface	Indicates characters you enter from the keyboard; for example Enter **sheet7** in the data entry box.
boldface italics	Indicates variable characters you enter from the keyboard; for example Enter ***filename*** in the data entry box.
Click left	Press and release the left button (button 1) on the mouse.

Click middle	Press and release the middle button (button 2) on the mouse.
Click right	Press and release the right button (button 3) on the mouse. If you are using a two-button mouse, press **Ctrl** and button 2 simultaneously to clickright.
Cycle	Click left repeatedly on a cycle box until the item you want is selected. A cycle box is indicated by ◊.
Select	Move the cursor to an item or point and press ↵, **spacebar**, or click left.

Creating a Plot Instructions File 7

What You'll Do in This Chapter

In this chapter you create a plot instructions file of tutor1 in your schematic design. You can use this file to create a hard copy of tutor1. This exer cise is written for those of you who have an HP 7475 pen plotter. If you do not have this pen plotter, you can create a plot using the plotter you have available. See the “Hardcopy" chapter in the Schematic Tools or PCB Tools *manuals to find out how to create a plot using your plotter.* To create a plot you must first create a plot file that consists of the area you want to plot in the design. You specify this plot file in the Hardcopy *tool to create your plot. This chapter tells you how to*

- view the design you want to plot
- choose the layers to plot
- select the pen plot positions for the layers
- create the plot file
- create the plot configuration file
- create the plot instructions file

What to Do if You Need Help

Here are some suggestions for things you can do if you need help:

- Read the What to Do If You Get Stuck section in the “Introducing P-CAD" chapter.
- Read the “Hardcopy" chapter in the Schematic Tools or PCB Tools manuals to find out how to plot files.

Viewing the Design You Want to Plot

Before you create a plot, you need to view the design you want to plot and create a plot file from it. The plot file is what you plot using the *Hardcopy* tool.

View the design you want to plot

1. Enter P–CAD.

2. Verify that tutor is the current design in the Design directory list box.
3. Verify that tutor1 is the current sheet in the Schematic Sheet list box.
4. Select Schematic Tools.

In this chapter you use the Schematic Editor to create a plot file. You can also create a plot file by using one of the other three graphics editors:

- ◊ the PCB Editor in the PCB Tools module
- ◊ the Part Editor in the Library Manager module
- ◊ the Symbol Editor in the Library Manager module

5. Select *Schematic Editor*. The system automatically loads tutor1.sch. The entire sheet is shown on the screen.

You can create a plot file either for an entire design or for part of a design. The portion of the design you want to include in the plot file must be completely visible on the screen. In this exercise, you create a plot file for an entire sheet.

If you want to create a plot file for part of a sheet, you select *View Window* rather than *Fit View* and select the corners of the area you plan to plot.

Choosing the Layers to Plot

The layers you set to either *ON* or *ABL* are included in the plot file.

Choose the layers to plot

1. Select *View Layer*.
2. Set the WIRES, GATE, PINNUM, PINCON, REFDES, ATTR, SDOT, DEVICE, NOTES, NETNAM, and BORDER layers to either *ON* or *ABL*.
3. Set the remaining layers to *OFF*. The Layer Table should look like Figure 7-1.

WIRES	1	ABL	A
BUS	1	OFF	
GATE	2	ON	
IEEE	2	OFF	
PINFUN	3	OFF	
PINNUM	1	ABL	
PINNAM	6	OFF	
PINCON	4	ON	
REFDES	2	ABL	
ATTR	6	ABL	
SDOT	1	ON	
DEVICE	5	ON	
OUTLIN	5	OFF	
ATTR2	6	OFF	
NOTES	6	ABL	
NETNAM	4	ABL	
CMPNAM	5	OFF	
BORDER	5	ON	

Figure 7-1. Layer Table for Plotting

Selecting the Pen Positions

In this example, you assign all the layers to pen position 1 so the layers have the same width and color. You can also choose another pen position or choose different pen positions for different layers.

If you are using a laser printer, you can skip this procedure.

Select the pen position for the layers to plot

1. Select the number in the first column to the right of WIRES. The cursor moves to the colored boxes at the lower portion of the screen. The numbers, shown in Figure 7-1, are pen positions in the plotter turret.
2. Select the box that contains 1. The number to the right of WIRES is changed to 1.
3. Repeat steps 1 and 2 for all layers to be plotted (the layers set to *ON* or *ABL*).
4. Select *EXIT* to return to the Schematic Editor.

Creating the Plot File

To create a plot file, you define a rectangular area you want to plot by selecting two opposite corners of the rectangular area. You'll create a plot file of the entire sheet, so select the corners outside the sheet border, as shown in Figure 7-2.

INSERT FIGURE 7-2 HERE

After you define the area you want to plot, you name the plot file.

Create the plot file

1. Select *File*!*Create Plot File*. A prompt appears in the response area:

   Select Page Corner 1

2. Select a point in the upper left corner of the drawing area (outside the sheet border). A prompt appears in the response area:

   Select Page Corner 2

3. Select a point in the lower right corner of the drawing area (outside the sheet border). This completes the window.

   The response area prompts you to enter a plot filename and shows tutor1.plt as the default filename.

4. Select *OK* to accept the default name and create the plot file. The system displays a message that the plot file is complete.

   If you select a plot filename that already exists, a decision box appears. Select *Yes* to override the existing filename.

5. Select *File*!*Quit.*

Now you've returned to the P–CAD screen. Next you'll plot the file you've created.

Creating the Plot Configuration File

Plot files are used by the *Hardcopy* tool to generate printer plots, pen– plots, and photoplots.

Create the plot configuration file

1. Select *Hardcopy*. The Hardcopy screen, shown in Figure 7-3, appears.

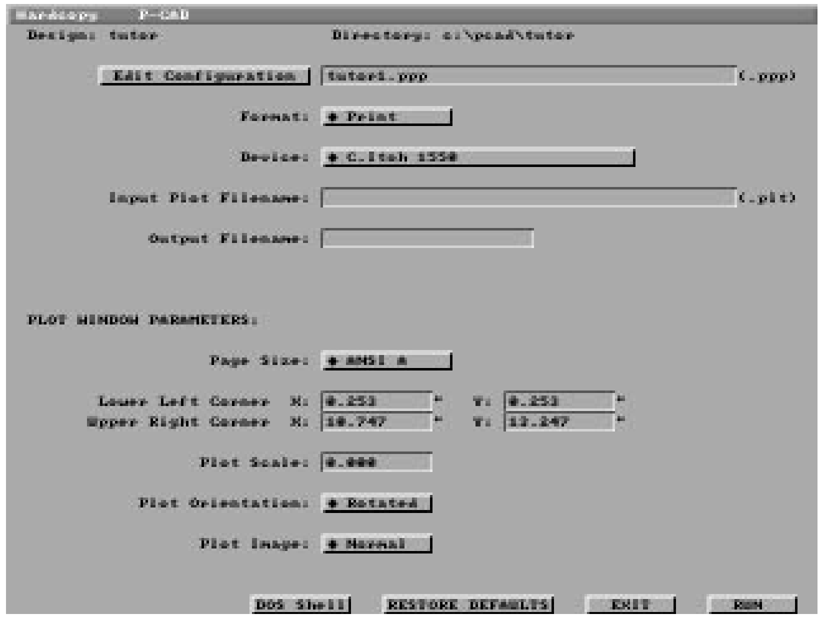

Figure 7-3. Hardcopy Screen

2. Cycle Format to *Plot*.
3. Cycle Device to *HP 7475*.

   In this exercise, you're instructed to choose an HP 7475 plotter. Cycle through the devices until you find a plotter that matches your hardware. You can check with your system administrator to see what plotters are available. For detailed information, see the chapter on printing and plotting files in the *Schematic Tools* manual or *PCB Tools* manual.
4. Select *Edit Configuration*. The Edit Configuration screen, shown in Figure 7-4, appears.

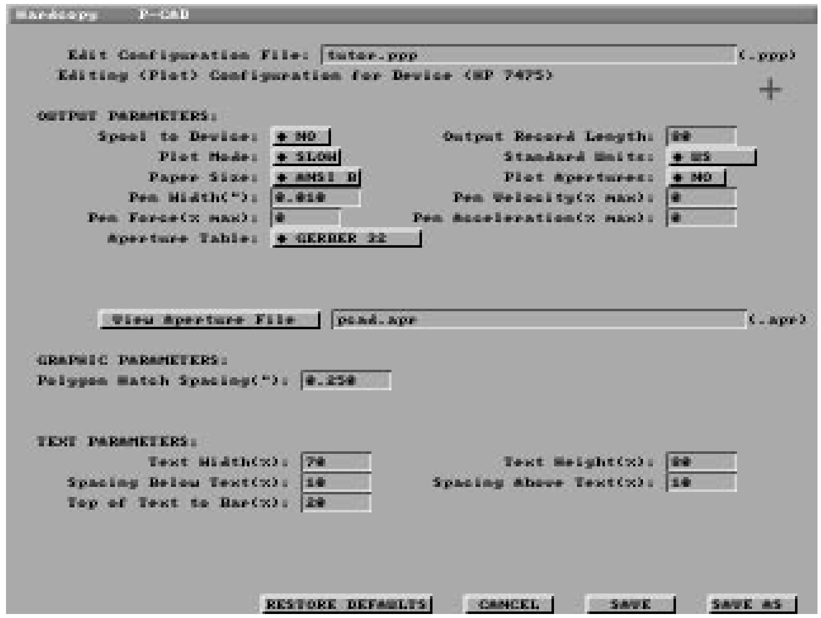

Figure 7-4. Edit Configuration Screen

Each time you run a plot, the system saves the plotting instructions in a disk file in the current design directory. In this exercise the plotting instructions are saved to a file named tutor.ppp. The Edit Configuration File data entry box shows tutor.ppp as the filename.

5. Cycle Spool to Device to *No*. When the Spool to Device option is set to *No*, the system stores the configuration information for later use.
6. Use the default values for all the other options.
7. Select *SAVE* to save the configuration information.

   You're now back in the Hardcopy screen.

Creating a Plot Instructions File

You created two files, the plot file called tutor1.plt and the plot configuration file called tutor.ppp. You'll use both these files to create a plot instructions file, called tutor1.hp. You use the plot instructions file to create a hardcopy plot.

Create a plot instructions file

1. Select the Input Plot Filename data entry box. A file selector box appears.

2. Enter **tutor1.plt** in the data entry box or select it from the list box. The system makes the output filename the same as the input filename. The system changes the extension of the output filename to correspond to the device you selected. In this case, the plot instructions file is named tutor1.hp. The .hp extension signifies a plot instructions file for the HP 7475 plotter.

   The default plot size is the largest that the paper selected can accommodate and still maintain a margin around the plot image. The scale factor can be reduced, but not increased, from the default.

   The system calculates the plot scale according to the relationship of the image size to the paper size. The higher the number that's shown, the larger the scale factor. Although you can change the plot scale to a lower value, use the default value for this exercise.

3. Select *RUN*.

   Status messages and error messages appear on the P–CAD screen. You can view these messages in the file PCAD.LOG, using the *Report Editor* tool in the Design Manager module or by typing out the file in DOS or UNIX.

   You created a plot instructions file called tutor1.hp. You can plot this file if you have an HP 7475 plotter. See the "Hardcopy" chapter in the *Schematic Tools* manual or the *PCB Tools* manual.

4. Select *Quit* to exit P–CAD and return to the DOS or UNIX prompt.

Take a break and relax. In the next chapter, you'll do some more work in a graphic editor.

What You've Learned So Far

Up to this point in the tutorial, you learned how

- the AutoLoader or installation program configures your system with the directories and files you need and some procedures you can follow if you get stuck
- to create a symbol
- to create a schematic
- to check for electrical rules violations
- to create the plot instructions file that you later use to create a hardcopy of a schematic sheet

What You'll Do Next

In the next chapter you'll create a part that corresponds to the 5404b.sym symbol you created earlier. You'll use this part in the pcb layout you'll create later on in this tutorial.

Creating a Part 8

What You'll Do in This Chapter

In this chapter you learn to create a part that corresponds to the 5404b.sym symbol you created in Chapter 4. The part you'll create is shown in Figure 8-1. This chapter tells you how to

- set up the editor environment
- enter the pins
- enter the origin
- draw the outline of the IC (integrated circuit) package
- draw a semicircle showing the orientation of pin 1
- enter a device name
- enter the packaging information
- enter the component ID type
- assign pin types
- save the file
- translate the packaging information
- check your work

What to Do if You Need Help

Here are some suggestions for things you can do if you need help:

- Read the What to Do If You Get Stuck section in Chapter 1.
- Use the *Library Manager* manual to get information on how to use the Part Editor.
- Use the 5404b.prx symbol file in the tutor directory.

Setting Up the Editor Environment

Before you create the part, you must set the layer table and the grid in the Part Editor.

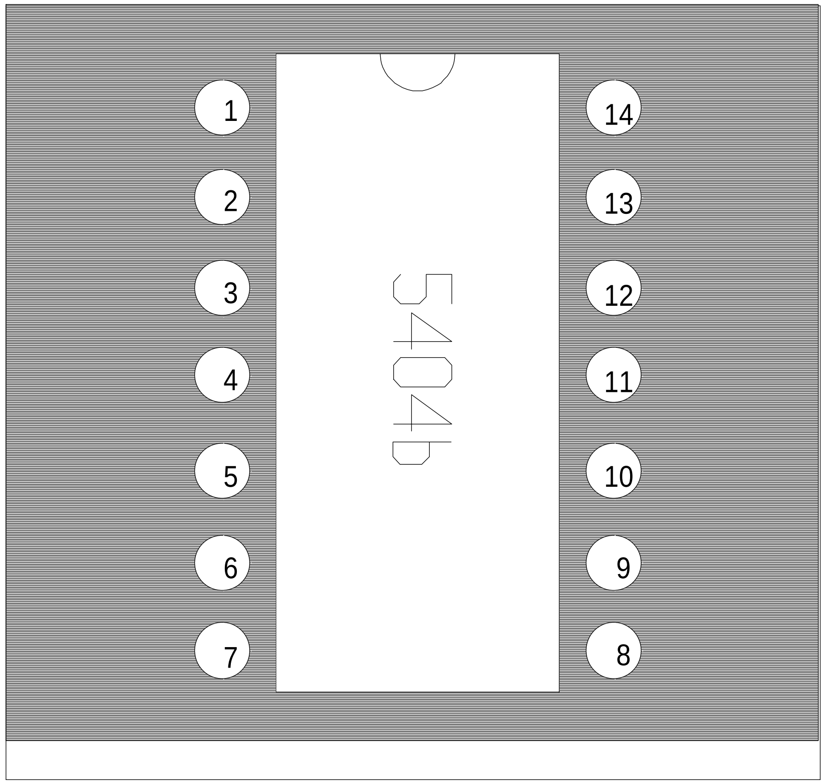

Figure 8-1. 5404b.prt Part

Set up the editor environment

1. Enter P–CAD.
2. Select Library Manager.
3. Verify that tutor is the current design directory highlighted in the list box.
4. Select *Part Editor*.
5. Select *View Layer* from the view command line. The Layer Table you create will look like Figure 8-2.
6. Enable the PIN, SLKSCR, and DEVICE layers by cycling them to *ABL*.
7. Make the PIN layer active by cycling to *A*.
8. Cycle the remaining layers to *OFF*.

9. Select *EXIT* or press **Esc** to return to the Part Editor.
10. Select Grd in the status area.
11. Enter **50 50**. In the Part and PCB Editors, one DBU represents 1 mil (0.001 inch). A grid spacing of 50:50 represents a 50 mil by 50 mil grid on a scale of 1:1.
12. Select *Zoom In* from the view command line.
13. Select the center of the drawing area (0,0) as the view center.
14. Press **Esc** to cancel the Zoom command.

PADCOM	7	OFF		INT 1	14	OFF
FLCOMP	7	OFF		INT 2	6	OFF
PADSLD	8	OFF		DRLGIN	5	OFF
FLSOLD	8	OFF		DRLFIN	6	OFF
PADINT	9	OFF		PINTOP	4	OFF
FLINT	9	OFF		PINBOT	3	OFF
GNDCON	10	OFF		MSKGTP	13	OFF
FLGCON	10	OFF		MSKGBT	14	OFF
CLEAR	12	OFF		MSKFTP	8	OFF
FLCLER	12	OFF		MSKFBT	9	OFF
PWRCON	13	OFF		PSTGTP	1	OFF
FLPCON	13	OFF		PSTGBT	2	OFF
SLDMSK	14	OFF		PSTFTP	12	OFF
FLSMSK	14	OFF		PSTFBT	13	OFF
DRILL	15	OFF		SLKTOP	6	OFF
FLDRLL	15	OFF		SLKBOT	5	OFF
PIN	4	ABL	A	DVCTOP	1	OFF
BRDOUT	4	OFF		DVCBOT	2	OFF
FLTARG	4	OFF		REFDTP	3	ON
SLKSCR	6	ABL		REFDBT	6	OFF
DEVICE	5	ABL		PINFTP	8	OFF
ATTR	6	OFF		PINFbP	9	OFF
REFDES	6	OFF				
COMP	1	OFF				
SOLDER	2	OFF				

Figure 8-2. Layer Table for the Part

Entering the Pins

The first graphics you'll create are the pins.

Enter the pins

1. Select *Enter*!*Pin.* A prompt appears in the response area:

   Select pin location:

2. Select the center of your drawing area (0,0). A prompt appears in the response area:

   Select pin name location. (Attrb OK?)

3. Press **Esc** to bypass giving the location of the pin name. Since the pin name or number won't be displayed, the location of the pin name isn't needed. A prompt appears in the response area:

Enter pin name [pin number]:

4. Enter **1**. A prompt appears in the response area:

   Select pin location:

5. Select a point two grid points down (0,–100) for the next pin. A prompt appears in the response area:

   Select pin name location. (Attrb OK?)

6. Press **Esc**. A prompt appears in the response area:

   Enter pin name [pin number]:

7. Enter **2**.

8. For the locations and names of pins 3 through 7, follow the same procedure you used for pin 2. See Figure 8-3.

   A prompt appears in the response area:

   Select pin location:

9. Select a point six grid points to the right (300,–600) to enter the location of pin 8. A prompt appears in the response area:

   Select pin name location. (Attrb OK?)

10. Press **Esc**. A prompt appears in the response area:

    Enter pin name [pin number]:

11. Enter **8**. A prompt appears in the response area:

    Select pin location:

12. Select a point up two grid points (300,–500) to enter the location of pin 9. A prompt appears in the response area:

    Select pin name location. (Attrb OK?)

13. Press **Esc**. A prompt appears in the response area:

    Enter pin name [pin number]:

14. Enter **9**.

15. For the locations and names of pins 10 through 14, follow the same procedure you used for pin 9.

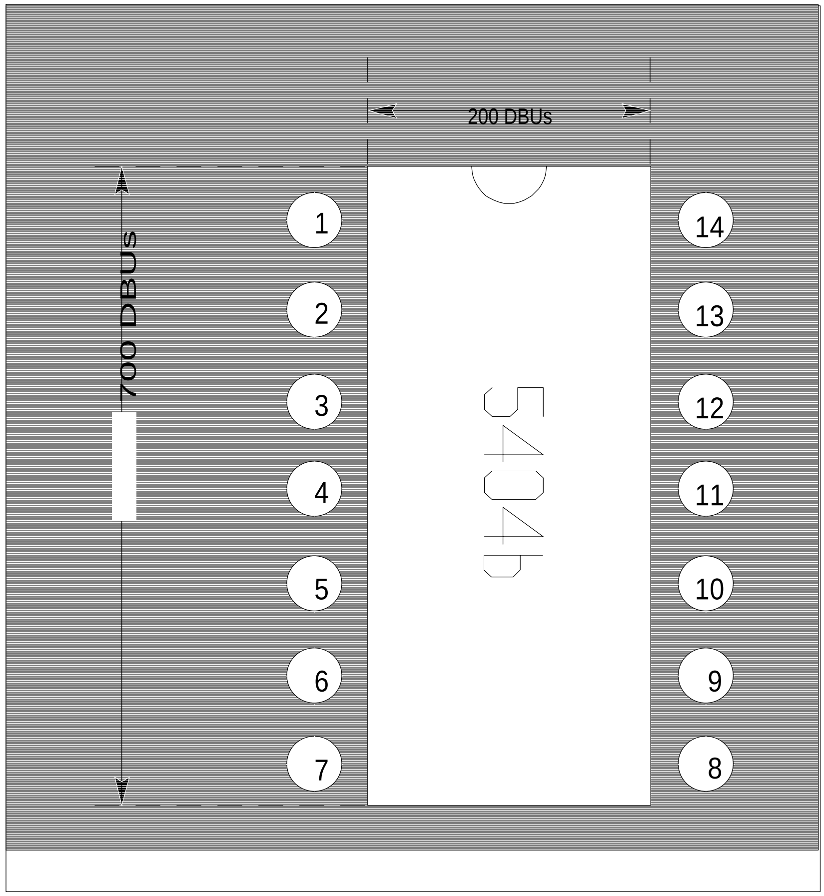

Figure 8-3. Pin Locations

Your drawing should look like Figure 8-4.

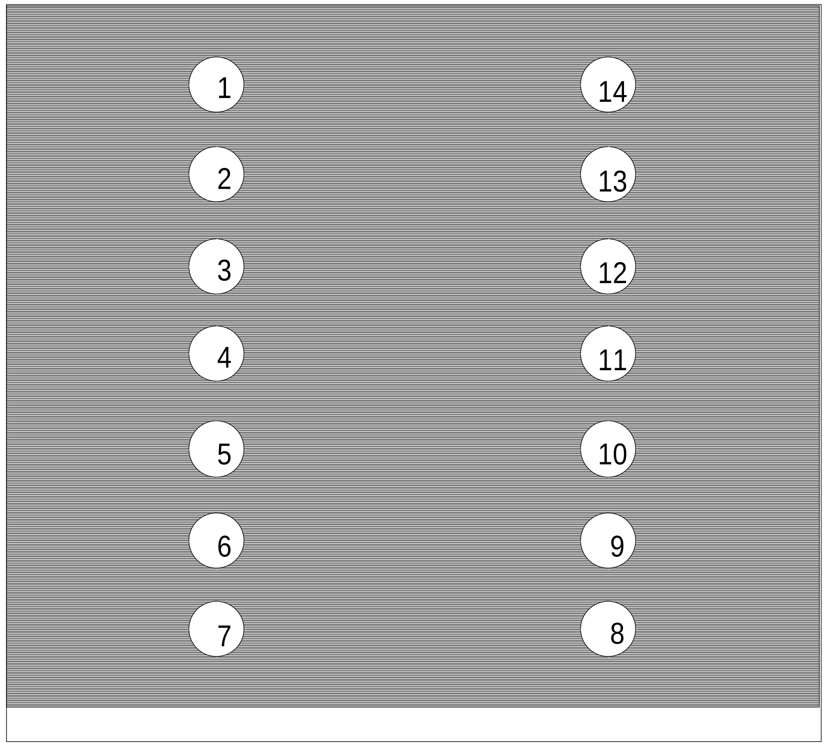

Figure 8-4. Pins

Entering the Origin

You now need to enter an origin, or drag point, for the cursor to attach to when you enter the part in a PCB layout. Pin 1 is usually the origin point of a part.

Enter the origin

1. Select *Enter*! *Origin.*
2. Select the center of pin 1 (0,0). The highlighted cross hairs and circle remain until you select another command.

Drawing the Outline of the IC Package

Now that you've created the pins, draw the outline of the IC (integrated circuit) package, shown in Figure 8-5.

Draw the outline of the IC (integrated circuit) package

1. Verify that the Grd setting in the status area is 50:50.

2. Select *Draw*!*Line.*

   You could also use *Draw*!*Rectangle* to draw the outline of the IC package.

3. Cycle the active layer in the status area to *SLKSCR*.
4. Move the cursor to Wid in the status area and enter **0**.
5. Select the point at the coordinates (50,50) as a start point for the line.
6. Draw a rectangle 200 DBUs across and 700 DBUs high.
7. Press **Esc** to finish using *Draw*!*Line.* Your drawing should look like Figure 8-5.

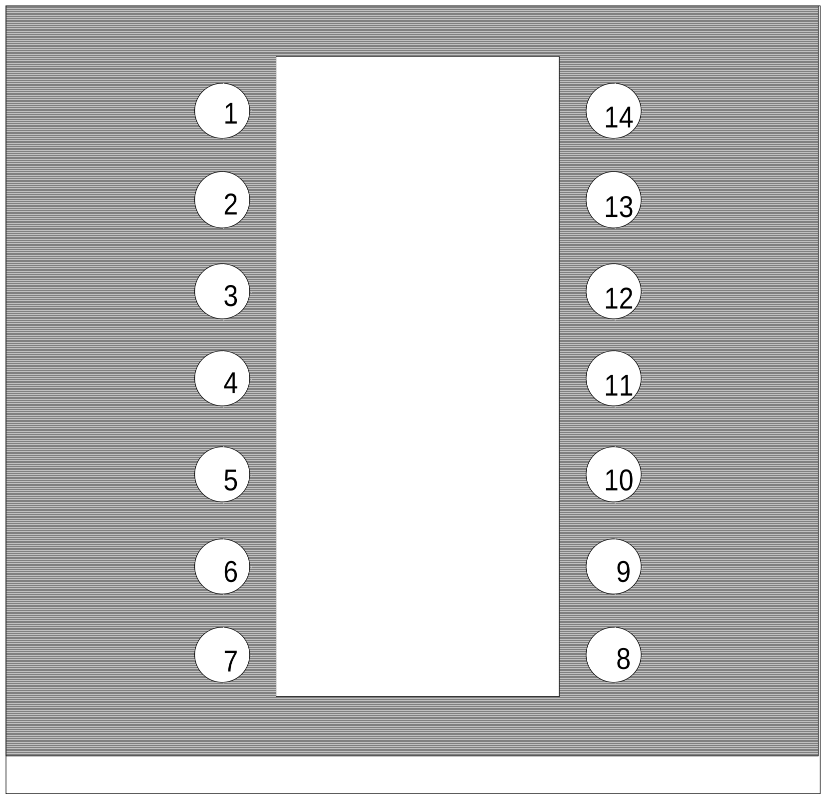

Figure 8-5. Pins and Outline of IC Package

Drawing a Semicircle Showing the Orientation of Pin 1

You need to show the orientation of pin 1. To show the orientation, you'll draw a semicircle.

Draw a semicircle showing the orientation of pin 1

1. Select *Draw*! *2–Point Arc.*
2. Select the midpoint of the top line of the rectangle (150,50) as the center point of the arc.
3. Move the cursor one grid point to the left (100,50) and select this location as the point defining the radius and starting angle.
4. Select (200,50) as the point defining the ending angle of the arc.
5. Select *Redraw* from the display command line. Your drawing should look like Figure 8-6.

   Redraw restores grid points and lines that disappear during editing or manipulating objects in the drawing area. *Redraw* is a nested command, and you can use it anytime without having to cancel the current command.

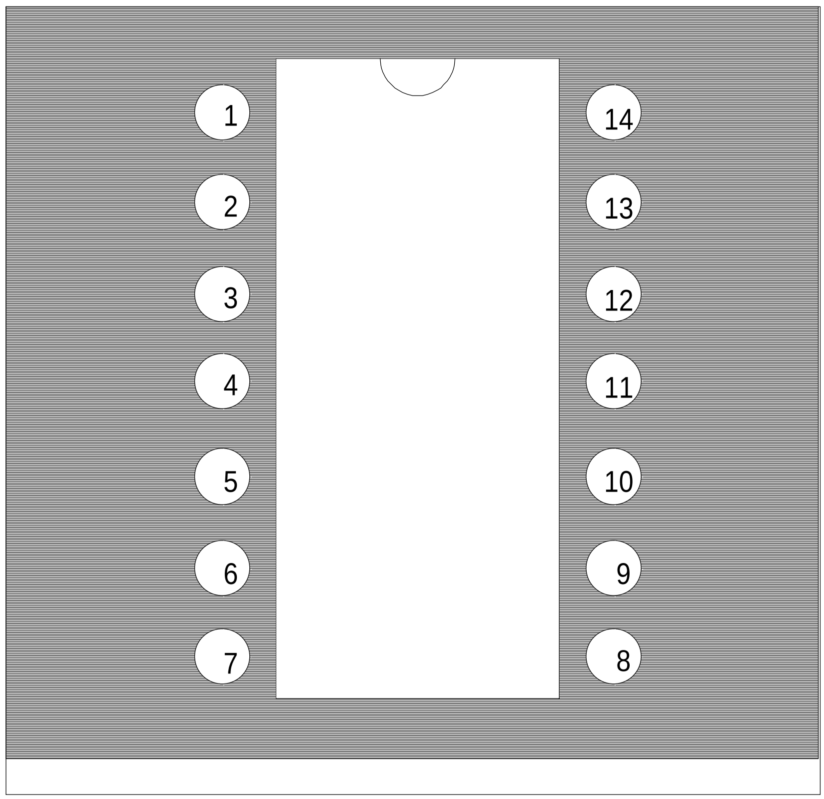

Figure 8-6. IC Package with Semicircle

Entering the Device Name

You'll assign the part the same device name as the 5404b.sym symbol you created in Chapter 4.

Enter the device name

1. Select *Draw*!*Text.*
2. Cycle the two text justification fields to *Center Center*.
3. Cycle the active layer in the status area to the *DEVICE* layer.
4. Select Size in the status area and enter?**125**.
5. Cycle the text orientation, shown in the Deg cycle box, to 270 degrees.
6. Select the center of the rectangle (150,–300).
7. Enter **5404b**.

Entering the Packaging Information

Packaging information maps the physical pins on the part to the symbol pins in each gate.

Enter the packaging information

1. Select *Enter*!*Packaging Data.* A prompt appears in the response area:

   Enter number of gates:
2. Enter **6**. A prompt appears in the response area:

   Enter number of pins per gate:
3. Enter **2**. A prompt appears in the response area:

   Enter name of gate pin 1:
4. Enter **ina**. A prompt appears in the response area:

   Enter name of gate pin 2:
5. Enter **out**. A prompt appears in the response area:

   Select gate 1 pin INA.
6. Select pin 1. A prompt appears in the response area:

   Select gate 1 pin OUT.
7. Select pin 2. A prompt appears in the response area:

   Select gate 2 pin INA.

8. Select pin 3. A prompt appears in the response area:

   Select gate 2 pin OUT.

9. Select pin 4.

10. Follow the same procedure as steps 6 through 9 above to enter the rest of the pins shown in Figure 8-7.

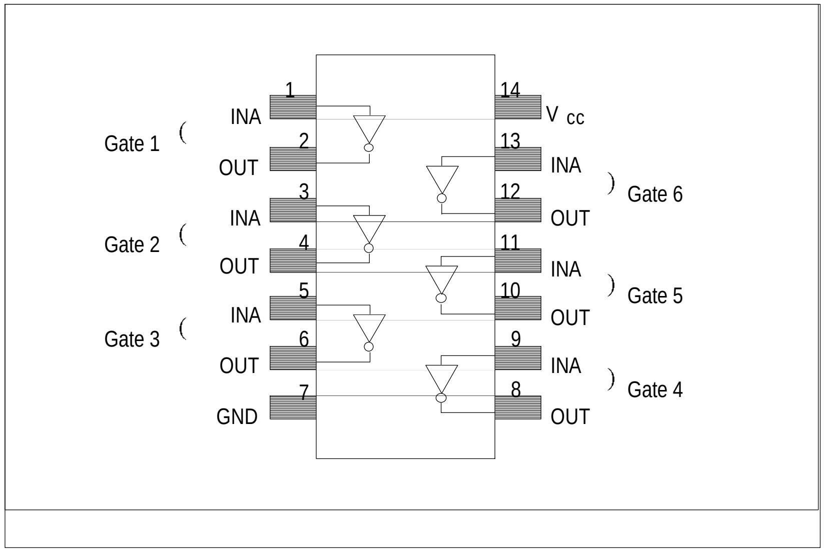

Figure 8-7. Input and Output Pins

Entering the Component ID Type

Component ID types identify the class or family to which a component belongs. You'll enter a component ID type that tells the system this part is a through–hole IC device. For more information on component ID types, see the "Using the Part Editor" chapter in the *Library Manager* manual.

Enter the component ID type

1. Select *Enter*!*Component Type.* A prompt appears in the response area:

   Part Old type = 10000. New type =

2. Press ↵ to accept the default value of 10000. This tells the system the part is a through–hole IC device. A decision box appears asking if the part is an SMD (surface–mount device).

3. Select *NO*.

Assigning Pin Types

Pin types assign physical and intelligent information to each pin.

Part pin types also determine what type of padstack will be attached to the pin when you use the part in a PCB layout. Padstacks contain the graphic pad description for pen–plotting, flash aperture information for photoplotting, and drill targets for creating fabrication drawings. You will learn to create a padstack in the next chapter.

The LEQ code for the part pins is the same as for symbols; it determines whether or not the pin can be swapped with any other pins having the same logical equivalence. In this example, use Table 8-1 to determine pin types and LEQ codes.

Table 8-1. Sample Pin Types and LEQ Codes

Pin	Pin Type	Meaning	LEQ Code
1	1	pin 1, DIP device	0
2	2	signal pin	0
3	2	signal pin	0
4	2	signal pin	0
5	2	signal pin	0
6	2	signal pin	0
7	3	ground pin	0
8	2	signal pin	0
9	2	signal pin	0
10	2	signal pin	0
11	2	signal pin	0
12	2	signal pin	0
13	2	signal pin	0
14	4	power pin	0

Each pin is highlighted as the prompt for the pin is displayed.

Assign pin types

1. Select *Enter*!*Pin Type.*

   A prompt appears in the response area:

   Enter new type:
   Type of pin 1 is 1.

2. Press ↵ to accept the default. A prompt appears in the response area:

   Enter new code:
   Pin LEQ code is 0.

3. Press ↵ to accept the default. A prompt appears in the response area:

   Enter new type:
   Type of pin 2 is 1.

4. Enter **2** to change the pin type from a 1 to a 2. A prompt appears in the response area:

   Enter new code:
   Pin LEQ code is 0.

5. Press ↵ to accept the default.

6. Use the data in Table 8-1 to finish entering the pin types and LEQ codes.

   Use *Query*!*Pin* and select each pin to verify that you correctly assigned the pin type and LEQ values to each pin.

7. Select *Fit View* to make the image fit the screen.

Saving the File

Don't forget to save all the work you've done.

Save the file

1. Select *File*!*Save.*
2. Enter **5404b.prt** in the data entry box.
3. Select File!Quit to return to the P–CAD screen.

Translating the Packaging Information

One method of transferring symbol pin information into part information is to use *Translate Packaging* in the *Component Editor* tool of the Library Manager module.

You use this option to translate the packaging information from one or more symbols into packaging information for one part. This is useful when you want to create nonhomogeneous parts. When you translate packaging information from symbols to parts, the pin LEQ values, pin names, and name of sections the pins are assigned to are transferred to the part. This information is listed in a tabular format. You can edit the pin name, pin type, or LEQ value.

It isn't necessary to translate the packaging at this point. However, if you made any errors when you were creating the part or symbol, it's easy to check your work using this tool.

Translate the packaging information

1. Select *Component Editor.*

2. Select *Translate Packaging*. The Translate Packaging screen appears, as shown in Figure 8-8.
3. Select the Destination Part Name data entry box.
4. Enter **5404b.prt** in the data entry box.
5. Select *ADD*.
6. Enter **5404b.sym** in the Source Symbol Name data entry box.

   If you can't find 5404b.sym, copy 5404b.syx to 5404b.sym and repeat this step.
7. Select *OK*.

The symbol packaging information is translated into part packaging information.

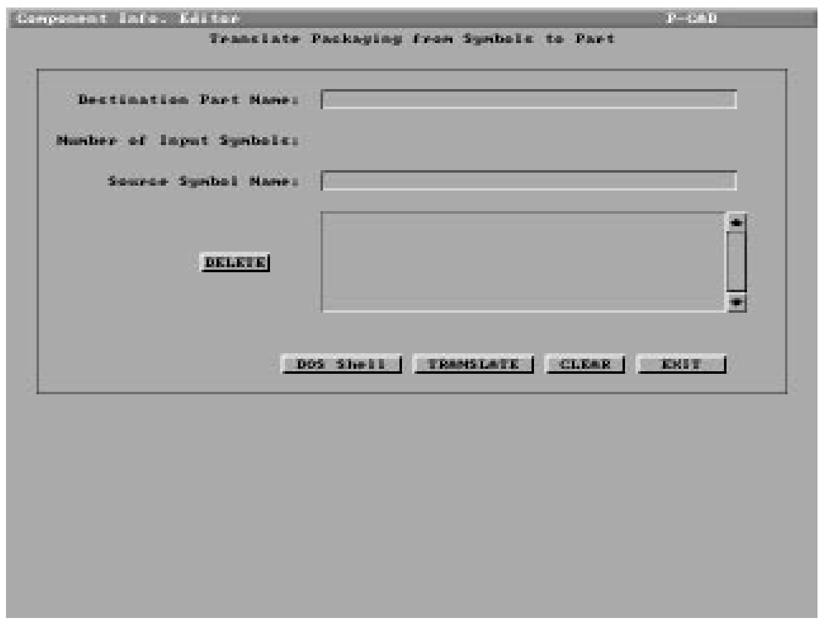

Figure 8-8. Translate Packaging Screen

Checking Your Work

The part pin table displays pin information for a part in table format. You'll use the part pin table to check your work and make sure it's correct.

Check your work

1. Select *Open Part* in the Component Editor screen.
2. Enter **5404b**.

   If you can‘t find 5404b.prt, copy 5404b.prx to 5404b.prt and repeat this procedure.

   It isn’t necessary to enter the extension of the filename.

   The part pin table appears, as shown in Figure 8-9.

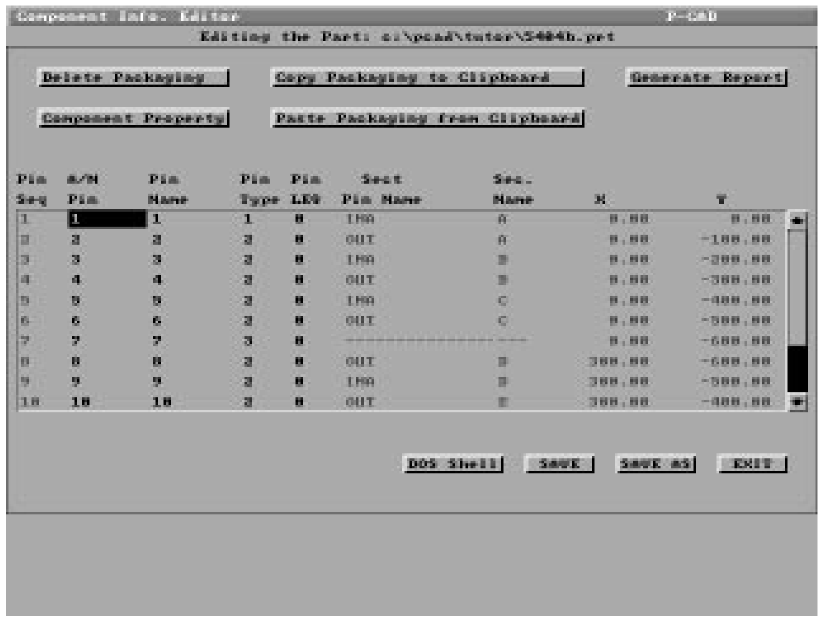

Figure 8-9. Part Pin Table

3. Move the cursor to the scroll bar in the list box. Part of the data may not be shown in the list box. To see all the data, drag the mouse up or down the scroll bar.

   The pin names, types, and LEQs are found in columns 3 through 5 in the list box. If you need to change the pin type or pin LEQ, select the value from the table and enter the new value in the data entry box that appears.
4. Select *Save*.
5. Select *EXIT*. You’re now in the Component Editor screen.
6. Select *EXIT*.

   You’ve finished creating a part and you’ve returned to the P–CAD screen.
7. Select *Quit* to return to the DOS or UNIX prompt.

What You've Learned So Far

Up to this point in the tutorial, you learned how

- the *AutoLoader* or installation program configures your system with the directories and files you need and some procedures you can follow if you get stuck
- to create a symbol
- to create a schematic
- to check for electrical rules violations
- to create the plot instructions file that you later use to create a hard copy of a schematic sheet
- to create a part

What You'll Do Next

In the next chapter you'll create an aperture table that defines aperture shapes and sizes used to photoplot your PCB database.

Creating an Aperture Table 9

What You'll Do in This Chapter

In this chapter you learn how to create and aperture table that defines aperture shapes and sizes that will be used to photoplot your PCB database. This chapter describes how you use the Aperture Table Editor to

- load an existing aperture table file into the Aperture Table Editor
- define an aperture shape description for a thermal flash
- add an aperture description to the end of an aperture table
- define an aperture shape description for a rectangular flash
- save your aperture table to an external file

What to Do if You Need Help

Here are some suggestions for things you can do if you need help:

- Read the What to Do if You Get Stuck section in Chapter 1.
- Use the *PCB Tools* manual to get information on using the Aperture Table Editor.
- Use the tutor1.apr aperture table in the tutor directory to see how your aperture table should look.

Understanding the Aperture Table Editor

What is the Aperture Table Editor and Why Do I Need It?

The Aperture Table Editor (ATE) tool lets you create and edit an aperture table file. The aperture table file describes the positions and shapes of the apertures on the photoplotter. The photoplot operator uses your aperture table file to set up the device correctly when processing your Gerber file into film.

P–CAD uses padstacks to define flash and graphic information at the location of each pin on a component in the PCB database. Each flash used in the padstack corresponds to a numbered aperture in the aperture file. You'll learn more about padstacks in the next chapter. Using the ATE you enter information that describes each aperture. The information includes its shape, size, rotation, and corresponding D–Code, and whether it's used to flash a shape or draw a line on the film.

Once the aperture file is created, you attach the file to your database in the PCB Editor and view the actual shape, size, and rotation of the apertures as they'll appear in your final photoplot. When the aperture file is attached to the database and you save the database, the aperture table information is saved internally with the database. You export the aperture data from your PCB database to an external file that you can print at any time.

The internal aperture table is used by the Design Rules Check (DRC) tool to check for spacing violations between entities in the PCB database.

Editing the P–CAD Aperture Table File

P–CAD supplies a sample aperture table file. This file is located in the directory where the program executables are found. The tutorial files also include the aperture table file, which you can use.

In this exercise, you use the P–CAD aperture table file supplied with the program and modify it for the tutorial database.

Open the external Aperture Table Editor from the P–CAD screen

1. Select PCB Tools.
2. Select *Aperture Table Editor*. The Aperture Table Editor screen,shown in Figure 9-1, appears.
3. Select the Load Aperture Table data entry box. A file selector appears.
4. Select pcad.apr from the listbox.
5. Select *OK* to return to the Aperture Table Editor screen. The aperture table data fills the list box, as shown in Figure 9-2.

Figure 9-1. External Aperture Table Editor screen

You'll use several apertures in your design. Use the default descriptions for most of the flash shapes as they are shown in the listbox. You'll change the description for aperture 24, and add aperture numbers 25 and 26 to the list and change their descriptions to identify a rectangular shape. Table 9-1 shows the aperture numbers and their corresponding shapes and sizes.

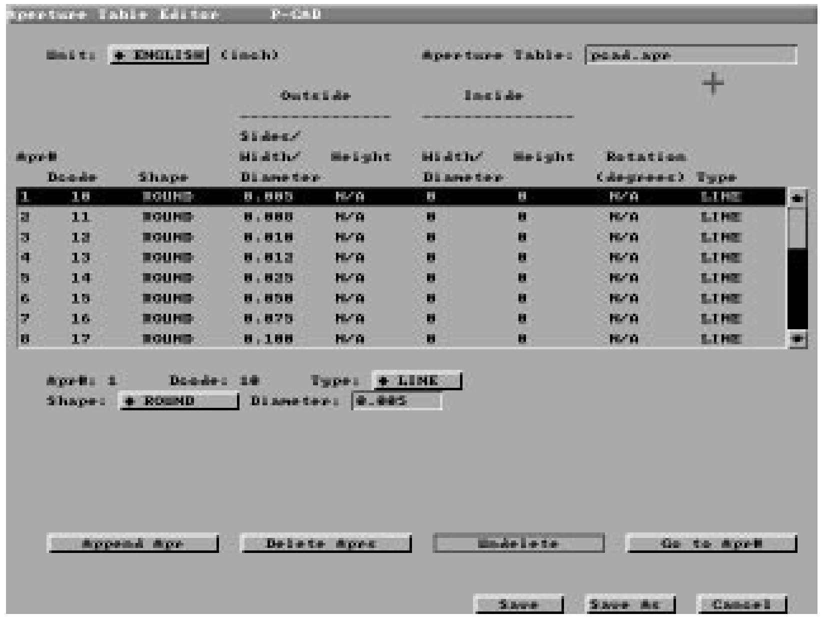

Figure 9-2. pcad.apr

Table 9-1. Aperture Shape Descriptions

Aperture Number	Shape	Size	Description
9	Round	.050	Flash for 50 mil via pad
11	Round	.060	Flash for 60 mil through –hole pad
12	Square	.060	Flash for 60 mil square pad
13	Round	.075	Flash for 75 mil through–hole pad
14	Square	.075	Flash for 75 mil square soldermask
15	Round	.080	Flash for 80 mil clearance
17	Round	.090	Flash for 90 mil round soldermask
24	Thermal	.060	Flash for 60 mil generic thermal
25	Rectangle	.070 x .026	Flash for 70 x 26 mil SMD land
26	Rectangle	.080 x .036	Flash for 80 x 36 mil SMD land mask

Change the shape description for aperture 24

1. Select the Go To Apr# button in the lower right of the Aperture Table Editor screen. The system prompts

   Enter Apr#:

2. Enter **24**. The description is highlighted.
3. Select the Shape option below the listbox and cycle to THERMAL.
4. Select the Diameter data entry box.
5. Enter **.100**. This is the outer clearance diameter.
6. Select the Inner Diameter data entry box.
7. Enter **.060**. This is the diameter of the interior of the thermal.
8. Select the Ties data entry box.
9. Enter **4**. This is the number of ties to an internal plane the thermal will have.
10. Select the Tie Width data entry box.
11. Enter **.025**. This is the width of the thermal tie.

Even though you identify aperture number 24 as a thermal shape, the image you'll see in the PCB Editor and on a penplot is a generic thermal shape. The generic shape shows you that you have a thermal at this location.

You can define the shape and size of the thermal in the aperture table so it can be penplotted generically and photoplotted correctly, but you must use a graphic image to show the actual shape and size in the PCB Editor.

Add descriptions for aperture numbers 25 and 26 to the bottom of the list

- Select Append Apr twice. A generic description for aperture numbers 25 and 26 are added to the bottom of the list.

Change the shape description for aperture 25

1. Select the description for aperture 25 in the listbox. The description is highlighted.
2. Select the Shape option and cycle to RECTANGLE.
3. Select the Width data entry box.
4. Enter **.070** for the width.
5. Select the Height data entry box.
6. Enter **.026** for the height.
7. Verify the flash Type is set to Flash, Rotation is set to 0 degrees, and Inner Hole is set to NONE.

Change the shape description for aperture 26

1. Select the description for aperture 26 in the listbox. The description is highlighted.
2. Select the Shape option and cycle to RECTANGLE.
3. Select the Width data entry box.
4. Enter **.080** for the width.
5. Select the Height data entry box.
6. Enter **.036** for the height.
7. Verify the flash Type is set to Flash, Rotation is set to 0 degrees, and Inner Hole is set to NONE.

You're finished creating your aperture table for the tutorial database. Save your file and print it. Later, you'll attach this file to your PCB database to see the flashes as they'll appear when they're photoplotted.

Save your aperture table to an external file

1. Select *Save As.*
2. Enter **tutor.apr** as the filename. The Aperture Table Editor screen closes and you return to the P–CAD screen.

Now You're ready to create a padstack.

Note on Flash Aperture Rotation

Unless you're planning to use an RS–274–X photoplot device that supports Extended Gerber photoplotting format, always keep the Rotation value in the Aperture Table Editor set to 0 degrees. If you specify a rotation for a flash aperture, whether it's in the aperture table or during entry of a drawn flash in the Part or PCB Editor, and you aren't using an RS–274–X photoplotter, flashes won't photoplot correctly, or they might not plot at all. Check with your photoplot vendors to see what devices they support.

See the *PCB Tools* manual for information regarding the Aperture Table Editor.

What You've Learned So Far

Up to this point in the tutorial, you learned how

- the *AutoLoader* or installation program configures your system with the directories and files you need and some procedures you can follow if you get stuck
- to create a symbol
- to create a schematic
- to check for electrical rules violations
- to create the plot instructions file that you later use to create a hard copy of a schematic sheet
- to create a part
- to create an aperture table file

What You'll Do Next

In the next chapter you'll create a padstack that describes the solder pads attached to a component pin.

Creating Padstacks 10

What You'll Do in This Chapter

In this chapter you learn to create a padstack file. This chapter describes a padstack and tells you how to

- create padstack graphics for pen plotting
- create flash graphics for photoplotting
- define the origin and save the file

What to Do if You Need Help

Here are some suggestions for things you can do if you need help:

- Read the What to Do if You Get Stuck section in Chapter 1.
- Use the *PCB Tools* manual to get information on using padstacks.
- View the c_60r32g.ps padstack file in the tutor directory to see how your padstack should look.

Understanding Padstacks

What is a Padstack?

A padstack defines how the pen–plotter or photoplotter draws (or flashes) pad graphics. A padstack is a stack of pad graphics and flash graphics located precisely one over another and all centered on the same X,Y coordinates. The graphics represent data associated with pins on through–hole and SMT (surface–mount technology) parts and vias.

Each padstack corresponds to a pin type on the (physical) part outline. In a PCB database, a pin type is a number that you assign to a component pin to link it to a padstack. This is the pin type we are referring to when we talk about padstacks. Don't confuse this with a symbol pin type in a schematic database, which describes the function of the pin (input, output, ground).

The padstack definition incorporates the number of board layers that are needed to create photoplots. The pin type does not. The pin type says: at this position, bring in this padstack. When you create a new pin type, you have to create a new padstack.

A padstack file consists of

- graphics data used for penplotting and graphics display.
- flash data used in photoplotting. Flash data represents the shape found at the corresponding position on an aperture wheel, or indicates a custom shape will be used.
- When you use the Integrated Aperture Table, you can view most of your flash data in the PCB Editor as it will appear in a photoplot. Some shapes can't be viewed or penplotted as they'll appear, so you create graphics data to show these shapes in the PCB Editor and on a penplot. The graphics data is paired with the flash data it represents. Each flash layer has a corresponding graphics layer.

Figure 10-1 shows an example of a ground pin padstack for pin type 3.

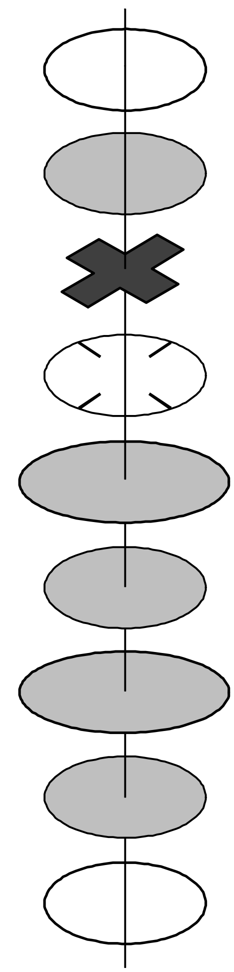

Figure 10-1. Ground Pin Padstack as Viewed in the PCB Editor

Pad graphics can contain drawn lines, rectangles, filled rectangles, polygons, circles, arcs, text, flashes, or clearance areas. The graphic representations in a padstack file are added to the graphics already at the pin. SMT parts can have padstacks, or you can create SMT part footprints without using a padstack by creating the drawing and flash graphics for each layer in the part file. See the *Library Manager* manual for information on creating padstacks for SMT parts.

An ASCII special symbol file (*filename*.ssf) pairs the pin types in the PCB database to the appropriate padstack file. By changing the special symbol file, different sets of padstacks can be incorporated into the same PCB database and you'll be able to create new databases for different applications. In the next chapter, you'll create the special symbol file.

P–CAD supplies some sample padstacks. The padstacks are in the prt directory. Each padstack is in a separate padstack file. Padstack files are saved with a .ps file extension. They're linked to or unlinked from a PCB database by using *Environment*!*Attach Padstacks* in the PCB Editor. Sample padstacks for use in the tutorial are supplied with the tutorial files.

You can place your own padstacks in any directory, as long as you specify a search path when using them in a design.

Why Use Padstacks?

Padstacks store repetitive graphic and flash data. By not incorporating the padstack information until the end of the design cycle, you save time and computer memory. If you want to change the padstack, you open the padstack file once and make the change to the padstack rather than having to make a change for each instance of the pad graphic in the PCB database. You can also use padstacks to create unique pad graphics, and provide a way to locate custom flash graphics in your design.

What You Need to Know Before You Create a Padstack File

The graphics and aperture data for each layer in the padstack must be located at a common origin point. All pad and flash graphic centers must be located at the same coordinate as the origin for the padstack unless your pad or graphic is asymmetric. Before you begin creating them, you must know

- the pin type the padstack will be associated with
- the precise size and shape of the pad
- how the pad size and shape might vary from layer to layer

This chapter presents a typical way to create a padstack file. Your company's requirements for padstack file contents may be different.

Creating Padstack Graphics and Flashes for Pen and Photo Plotting

Each layer of the padstack represents a unique connectivity for the type of pin associated with the padstack. The padstack graphics and flashes you'll pen and photoplotting. The flash data in your padstack is used to create film for PCB manufacture. Each flash aperture represents a size and shape that is flashed on the film during photoplotting.

You use flash layers for photoplotting, and graphics layers to identify a shape that is not easily represented by a flash, or for penplotting. You can make penplots of your database, and see the actual shape of the flashes in your database. If you don't have access to a printer that can accept Postscript files or a penplotter, you won't be able to see the actual size and shape of your flash aperture when you create a penplot. In this case, you need to create graphics in the padstack to represent flash shapes that can't be printed.

When you use the Integrated Aperture Table available in the PCB Editor, a flash is used in a padstack to represent a pad in a P–CAD PCB database. The shape of the flash is shown when the padstack is attached in the PCB Editor. The shape is the actual shape and size of the flash aperture assigned to that pin at that location.

The Integrated Aperture Table shows the actual shape and size for most of the flash types. Some shapes appear as a generic shape, or by a bounding box that shows the maximum generic outline of a special shape. If you want to see these shapes in a penplot as they'll photoplot, you must also create graphics for them. If you don't create graphics for them, the penplot will show a generic flash shape at the location so you know a special shape is located there.

In the Part Editor, a flash is shown as an outlined generic circle with the aperture number in the center. The aperture number assigned to this flash appears inside the circle. The flash graphic's size is set in the *Editor Configuration* screen of the PCB Tools module. When you are in the PCB Editor, you attach the padstacks to your database, and then attach the aperture table to your database to see the flash shown at it's actual size and shape. When you make a change to the aperture table data while in the PCB Editor, the change is reflected in the database as soon as you exit the Aperture Table Editor and return to the PCB Editor. Thermal target flashes and special shape flashes are still shown as a generic outline.

The Aperture Table Editor that you access in the Part Editor is not associated with your PCB database. You can view the aperture table file, make changes, and export the table to an external file from the Part Editor. You must attach the file to your PCB database by using the Aperture Table Editor in the PCB Editor.

When you attach padstacks to a database in the PCB Editor, the flash graphic shown is the actual size and shape of the aperture at the pin location.

In this exercise, you create a sample padstack that can be used for pen and photoplotting. Table 10-1 gives a summary of the graphics and flashes you'll create. This exercise is meant only to walk you through the procedure of creating a padstack. Layers, shapes, graphics, and apertures for your own application will differ.

Table 10-1. Pads You'll Create

Layer	Meaning
PADCOM	60–mil round pad on the component side, for penplots
FLCOMP	Flash for 60–mil round pad on the component side
PADSLD	60–mil round pad on the solder side, for penplots
FLSOLD	Flash for 60–mil round pad on the solder side
FLINT	Flash for 60–mil round internal layer pad
GNDCON	Thermal relief ground ties graphic, internal plane
FLGCON	Generic flash shape to identify location of thermal
FLCLER	Flash for clearance around internal isolated pads
FLPCON	Flash for internal power pad not tied to plane
FLSMSK	Flash for 75–mil soldermask clearance
DRILL	60–mil x 60–mil cross hairs to mark hold for drill legend

To create your padstack, you

- print your aperture table file
- set up the editor environment
- create a round pad graphic on the PADCOM layer
- create a flash on the FLCOMP layer
- create a round pad graphic on the PADSLD layer
- create a flash on the FLSOLD layer
- create a flash on the FLINT layer
- create a thermal relief graphic on the GNDCON layer
- create a generic thermal flash on the FLGCON layer
- create a clearance flash on the FLCLER layer
- create a flash on the FLPCON layer
- create a flash on the FLSMSK layer
- create cross hairs for the drill targets on the DRILL layer
- add the drill size
- add an origin point for the padstack
- save the file

Printing Your Aperture Table File

You'll print the aperture table file you created earlier. This give you a guide for creating padstacks using apertures listed in the table.

Print your aperture table file

1. Enter P–CAD.
2. Select Library Manager.
3. Verify that tutor is the current selection in the Designs listbox.
4. Select *Part Editor*.
5. Select *File*!*Print*.
6. Enter tutor.apr. The file is sent to your online printer.

If you don't have an online printer set up, you can use any DOS or UNIX command that prints an ASCII file.

Setting Up the Editor Environment

You create padstack graphics in the Part Editor. You need to set the layer
table and the grid first.

Set up the editor environment

1. Select *View Layer* from the view command line.
2. Enable the following layers by cycling them to *ABL*.

   PADCOM
   FLCOMP
   PADSLD
   FLSOLD
   FLINT
   GNDCON
   FLGCON
   FLCLER
   FLPCON
   FLSMSK
   DRILL

   The meaning of these layers is shown in Table 10-2.

Table 10-2. Padstack Layers

Layer	Meaning

PADCOM	Graphic, component side pads
FLCOMP	Flash, component side pads
PADSLD	Graphic, solder side pads (optional)
FLSOLD	Flash, solder side pads (optional)
FLINT	Flash internal layer signal pads
GNDCON	Graphic, ground ties to internal plane
FLGCON	Flash, thermal ground ties to internal plane
FLCLER	Flash, clearance for pads or ties to internal plane
FLPCON	Flash, power pad not tied to internal plane
FLSMSK	Flash, solder mask, through–hole pads
DRILL	Graphic, drill legend

3. Make the PADCOM layer active by selecting the space to the right of *ABL* in the PADCOM line.
4. Cycle through all the remaining layers until they all show *OFF*.
5. Select *EXIT* to return to the Part Editor.
6. Select Grd in the status area.
7. Enter **5 5**.
8. Select *Zoom In* from the view command line.
9. Move the cursor to the center of the drawing area (0,0).
10. Select this point four times. The grid points are visible in the drawing area.
11. Press **Esc** to stop zooming in.

Creating a Round Pad Graphic on the PADCOM Layer

You'll draw an outline of a 60–mil pad on the PADCOM layer, as shown in Figure 10-2. This is the graphic representation of a 60–mil pad on the component side of the PC board.

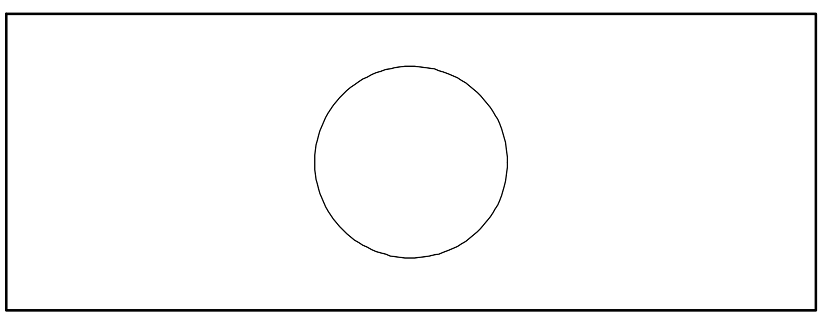

Figure 10-2. Round Pad Graphic on the PADCOM Layer

Create a round pad on the PADCOM layer

1. Select *Draw!Circle.*
2. Select Wid in the status area.
3. Enter **0**. This is the line width.
4. Select (0,0) as the center of the pad graphic.
5. Select a point six grid points to the right (30,0). This is a point on the circumference of the circle.

Creating a Flash on the FLCOMP Layer

You'll place a flash for a 60–mil round pad on the FLCOMP layer.

Create a flash on the FLCOMP layer

1. Make the FLCOMP layer active by using one of the following methods:
   - Press **F1** to cycle to *FLCOMP*. Each time you press **F1** the active layer name appears in the status area.
   - Select *Layer* in the status area and cycle to *FLCOMP*.
   - Select *View Layer* from the display command line. Select the space to the right of ABL in the FLCOMP line of the Layers screen.
2. Select *Draw*!*Flash*.
3. Select Aper in the status area.
4. Enter **11** as the aperture number for a 60–mil round flash.
5. Place the flash at (0,0).

The outline of a circle that represents the flash appears, with the number 11 in the center. This corresponds to aperture number 11 in your aperture table file.

Creating a Round Pad Graphic on the PADSLD Layer

You'll draw an outline of a 60–mil pad on the PADSLD layer. The graphic is directly on top of the pad of the PADCOM layer.

Create a round pad on the PADSLD layer

1. Select *Draw*!*Circle*.
2. Make the PADSLD layer active.

3. Select (0,0) as the center point.
4. Select (30,0) as the point on the circumference. The graphic is directly on top of the pad and flash you just created.

Creating a Flash on the FLSOLD layer

You'll place a flash for a 60–mil round pad on the FLSOLD layer.

Create a flash on the FLSOLD layer

1. Select *Draw*!*Flash*.
2. Make the FLSOLD layer active.

   You already set the aperture number to 11, which represents a 60–mil round flash.
3. Place the flash at (0,0).

Creating a Flash on the FLINT layer

You'll place a flash for a 60–mil round pad on the FLINT layer.

Create a flash on the FLINT layer

1. Make the FLINT layer active. You'll use aperture number 11 again.
2. Place the flash graphic at (0,0).

   Remember to periodically save your work using the *File*!*Save* command.

Creating a Thermal Relief Graphic

You'll create a thermal relief graphic, shown in Figure 10-3. There are comments in this figure to help you create the graphic.

This procedure gives you an example of creating a thermal relief graphic that could be used to create a photographically reversed internal ground plane.

If you're using the Aperture Table Editor and have padstacks and an internal aperture table attached to your PCB, you'll need a drawn graphic on a graphic layer to see the exact thermal shape when you print a plot file on a non–PostScript printer.

Create a thermal relief graphic on the GNDCON layer

1. Make the GNDCON layer active.

2. Select *Draw!2–Point Arc.*
3. Select Wid in the status area and enter **20** as the line width.
4. Select (0,0) as the center point for the arc.
5. Select (40,10) as the start point of the first arc.
6. Select (10,40) as the point defining the end point of the first arc.
7. Complete the thermal relief graphic by using the coordinates for each arc in the quadrants shown in Table 10-3.

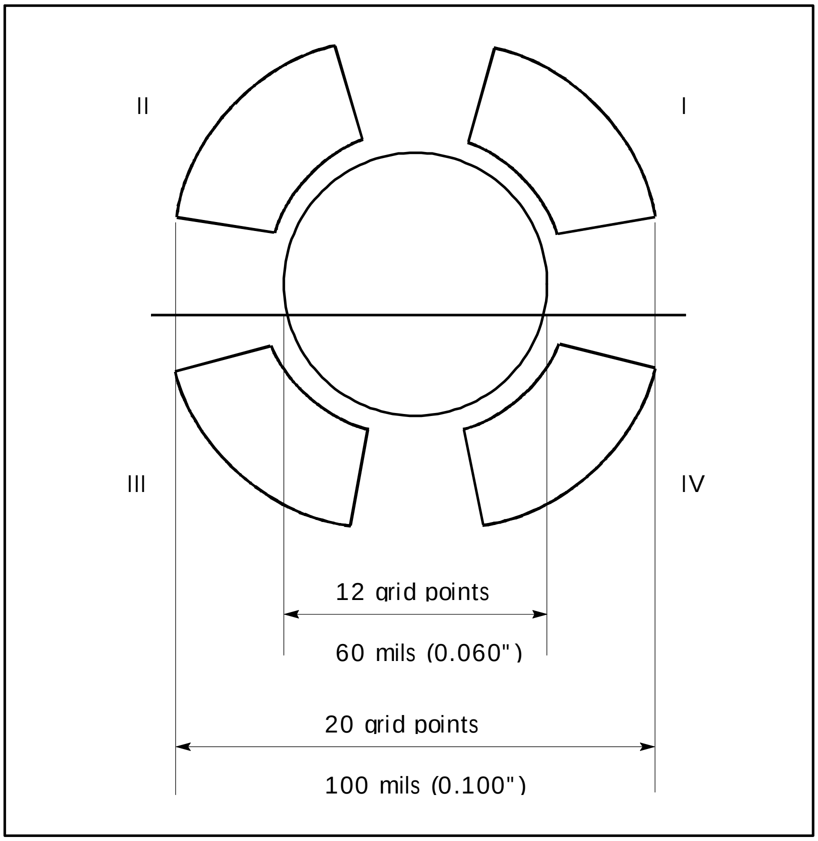

Figure 10-3. Thermal Relief Graphic

Table 10-3. Thermal Relief Coordinates

Location	*Quadrant I*	*Quadrant II*	*Quadrant III*	*Quadrant IV*
Center	0,0	0,0	0,0	0,0?
Start of Arc	40,10	-10,40	-40,-10	10,-40
End of Arc	10,40	-40,10	-10,-40	40,-10

Creating a Generic Thermal Flash on the FLGCON Layer

You'll place a generic flash for a 60–mil thermal flash on the FLGCON layer.

Create a flash for the graphic on the FLGCON layer

1. Select *Draw*!*Flash*.
2. Make the FLGCON layer active.
3. Select Aper in the status area.
4. Enter **24** as the aperture number. This corresponds to aperture number 24 in your aperture table file.
5. Place the flash at (0,0). The outline for the flash is placed on the stack. The shape you see when you're in the PCB Editor and have both your padstacks and the aperture table attached to your database will be a generic outline for a thermal.

Creating a Clearance Flash on the FLCLER Layer

You'll place a clearance flash for internal polygon clearance on the FLCLER layer. The flash represents an 80–mil diameter clearance area.

This clearance, combined with a flash for an internal pad, would be used to create film composites, meaning you use two layers of film to create a photoplot for an internal plane.

Even though this method of creating an internal plane differs from the previous method, it's only to give an example of another way you could build your padstacks images for creating internal planes.

Note: **I**f you use polygons to create internal planes, flashes will cbe cleared on specified layers when you enter the polygon over any flashes. Refer to the Command Reference manual for information on using *Environment*!*Clear Flashe*s with polygons.

Create a clearance flash on the FLCLER layer

1. Make the FLCLER layer active.
2. Select Aper in the status area.
3. Enter **15** as the aperture number. This corresponds to aperture number 15 in your aperture table.
4. Place the flash at (0,0).

Creating a Flash on the FLPCON Layer

You'll place a flash for a 60–mil round pad on the FLPCON layer.

Create a flash on the FLPCON layer

1. Make the FLPCON layer active.
2. Select Aper in the status area.
3. Enter **11** as the aperture number.
4. Place the flash at (0,0).

Creating a Flash on the FLSMSK Layer

You'll create a 75–mil round flash on the FLSMSK layer.

Create a graphic for the FLSMSK layer

1. Make the FLSMSK layer active.
2. Select Aper in the status area.
3. Enter **13** as the aperture number. Aperture number 13 represents a 75–mil round flash.

Creating Cross Hairs of the Drill Targets

You'll create the cross hairs for the drill target, as shown in Figure 10-4.

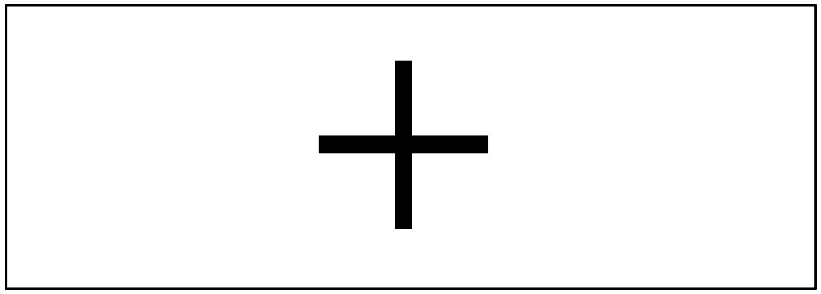

Figure 10-4. Graphic with Cross Hairs

Create cross hairs for the drill targets on the DRILL layer

1. Make the DRILL layer active.
2. Select *Draw!Line.*
3. Select Wid in the status area.
4. Enter **5**.
5. Select (–30,0) as the starting point of the horizontal line.
6. Select (30,0) as the ending point of the horizontal line.
7. Press **Esc**.
8. Select (0,30) as the starting point of the vertical line.
9. Select (0,–30) as the ending point of the vertical line.
10. Press **Esc**.

Adding the Drill Size

You'll specify the drill size.

Add the drill size on the DRILL layer

1. Select *Draw!Text.*
2. Select *Size* in the status area.
3. Enter **50** to change the text size.
4. Cycle the text justification to *Center Bottom*.
5. Select (–5,55) as the location for the text.
6. Enter **32** for the drill size.

Defining the Origin and Saving the File

You must define an origin for your padstacks before using them in a database.

Define the origin and save the file

1. Select *Enter*!*Origin*.
2. Place the origin at (0,0).
3. Select *File*!*Save.*
4. Enter **60r32g.ps**.

   You've created a padstack file for pen–plotting or photoplotting. Many companies use unique names to identify padstack files. The filename in this example uses these abbreviations:

   60 = 60–mil pad
   r = round pad
   32 = hole size
   g = ground ties to plane
   ps = padstack
5. Select *File*!*Quit.*

Remember, the padstack files supplied in the P–CAD tutorial may not be correct for your individual application outside the tutorial. You may want to create your own custom library of padstack files.

Now you've finished creating the padstack file and you've returned to the P–CAD screen. Go ahead and take a break if you need one. You'll create the special symbol file next.

What You've Learned So Far

Up to this point in the tutorial, you learned how

- the *AutoLoader* or installation program configures your system with the directories and files you need and some procedures you can follow if you get stuck
- to create a symbol
- to create a schematic
- to check for electrical rules violations
- to create the plot instructions file that you later use to create a hard copy of a schematic sheet
- to create a part

- to create an aperture table file
- to create a padstack

What You'll Do Next

In the next chapter you'll create a special symbol file that links the pin types to the padstacks in the PCB database.

Creating a Special Symbol File 11

What You'll Do in This Chapter

In this chapter you'll create a special symbol file that maps the padstacks you created in the previous chapter and other padstacks supplied with the system to the pin types you'll use in the PCB Layout.

What to Do if You Need Help

Here are some suggestions for things you can do if you need help:

- Read the What to Do if You Get Stuck section in Chapter 1.
- Use the *Design Manager* manual to get information on how to use the *Report Editor*.
- Use the *PCB Tools* manual to get information on linking padstacks to a database.

Understanding a Special Symbol File

A special symbol file is an ASCII input file that defines which padstack graphics are assigned to which pin type in a PCB design. When you execute *Environment*!*Attach Padstacks* in the *PCB Editor* tool, it links the padstack you created in the previous chapter, as well as any other padstacks specified in the file, to the specified pin type in the database.

If you assign pin types in a database and don't attach padstacks, you can't produce correct plots or film during photoplotting.

Figure 11-1 shows an example of a special symbol file.

```
% tutor.ssf
% special symbol file
%
%            pin type      connectivity  filename      remarks
%             number          status
%
             0             *             50r.ps        % vias only, .050
%
% pin types 1 through 4 have .060 pads intended to be
% with a .032 plated through*hole
             1             *             60s.ps        % dip pin
             2             *             60r.ps        % signal
             3             *             60rg.ps       % dip gnd pins, except pin 1
             4             *             60rp.ps       % dip pwr pins, except pin 1
             5             *             75rg.ps       % gnd pin, .046
             8             *             75r.ps        % other pins, .046
             98            *             50rp.ps       % via, SMD to internal power plane
             99            *             50rg.ps       % via, SMD to internal ground plane
```

Figure 11-1. Special Symbol File

The first few lines are comment lines. Also notice that the eighth and ninth lines are comment lines and the right sides of the last eight lines contain comments. Comments are preceded by a percent sign (%). Any text to the right of the percent sign is ignored.

Each entry in the special symbol file consists of three items: the pin type number, the connectivity, and the padstack filename. A description of each of these items follows:

- *pin type number* indicates the pin type assigned to the component part. It's entered by using *Enter*!*Pin Type* in the *Symbol Editor* tool.
- *connectivity* indicates whether the pin is committed or not committed. The connectivity is also known as the status. An asterisk indicates that the same padstack can be used both for committed and for uncommitted pins when you have an aperture table file attached to your PCB database.
- *padstack filename* indicates the name of the padstack file to which the pin type is to be linked. The padstack must be in the current design directory or the correct library search path must be specified in the *Set Libs. & Search Paths* tool of the Library Manager module.

Creating a Special Symbol File

To create a special symbol file, you

- open the P–CAD text editor
- enter and save the data for the special symbol file
- print the file

Opening the P–CAD Text Editor

You create ASCII files in the *Report Editor* tool of the Design Manager module, or you can use any text editor. To specify a default text editor for your system when running P–CAD, see the chapter on system configuration in the *Design Manager* manual. The default text editor for DOS systems is Edlin. The default text editor for UNIX systems is vi.

Open the P–CAD text editor

1. Enter P–CAD
2. Select Design Manager.
3. Select *Report Editor*. The Report Editor screen appears, as shown in Figure 11-2.

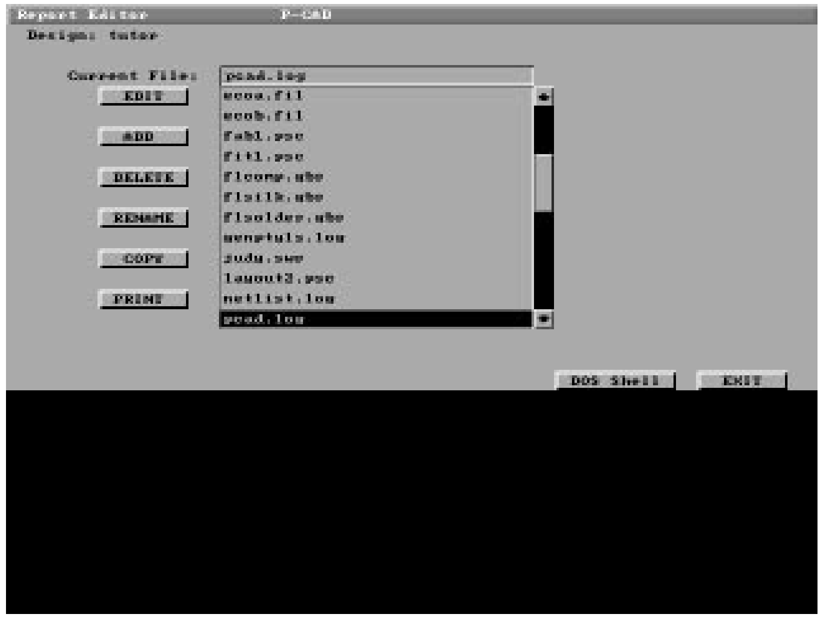

Figure 11-2. Report Editor Screen

4. Select *ADD* from the Report Editor screen.
5. Enter **tutor.ssf**.
6. Select *EDIT*. The screen shows the text editor. The text editor may be different according to the way your system is configured.
7. If you're using a UNIX machine, move your cursor to the editor window. The editor name appears in the title bar of this window.

Entering Data for the Special Symbol File

Figure 11-3 shows the special symbol file you'll create.

```
% tutor.ssf
% special symbol file
%
%            pin type     connectivity  filename     remarks
%             number         status
%
             0            *             50r.ps       % vias only, .050
%________________________________________________________________________
% pin types 1 through 4 have .060 pads intended to be
% with a .032 plated through*hole
             1            *             60s.ps       % dip pin
             2            *             60r.ps       % signal
             3            *             60rg.ps      % dip gnd pins, except pin 1
             4            *             60rp.ps      % dip pwr pins, except pin 1
             5            *             75rg.ps      % gnd pin, .046
             8            *             75r.ps       % other pins, .046
             98           *             50rp.ps      % via, SMD to internal power plane
             99           *             50rg.ps      % via, SMD to internal ground plane
```

Figure 11-3. Special Symbol File You'll Create

When you create the special symbol file, enter information in a free–form, tabular format and separate adjacent items by at least one space or one comma.

Enter data for the special symbol file

1. Enter **%**.
2. Enter **% special symbol file**.
3. Complete the file by entering all the data shown in Figure 11-3.
4. Save and quit your file using commands specific to the text editor you are using.

You've returned to the Report Editor screen.

Printing Your File

If you have an online printer, perform the following steps to print your file.

Print your file

1. Select *tutor.ssf* from the Current File list box in the Report Editor screen.
2. Select *PRINT*.
3. Select *EXIT* to return to the P–CAD screen.

You've finished creating a special symbol file and returned to the P–CAD screen. Take a break. In the next chapter, you'll create a board outline.

What You've Learned So Far

Up to this point in the tutorial, you learned how

- the *AutoLoader* or installation program configures your system with the directories and files you need and some procedures you can follow if you get stuck
- to create a symbol
- to create a schematic
- to check for electrical rules violations
- to create the plot instructions file that you later use to create a hard copy of a schematic sheet
- to create a part
- to create an aperture table file
- to create a padstack
- to create a special symbol file that maps the padstack to the pin types you'll use in the PCB layout

What You'll Do Next

In the next chapter you'll create a board outline that you'll use for packaging the schematic.

Creating a Board Outline 12

What You'll Do in This Chapter

Creating a board outline is part of preparing the PCB database for layout. In this chapter you load a board outline from the P–CAD library. Then you create your own board outline. This chapter tells you how to

- load a board outline from a P–CAD library
- save the board outline with a new filename
- create your own board outline

After finishing the tutorial, you can create your own library of board outlines according to your installation's specifications, mechanical interface requirements, critical part placement, and manufacturing constraints.

What to Do if You Need Help

Here are some suggestions for things you can do if you need help:

- Read the What to Do if You Get Stuck section in Chapter 1.
- Use the *PCB Tools* manual to get information on how to use the PCB Editor.

Loading a Board Outline from a Library

P–CAD supplies blank board outlines that you can edit to create your own. In this chapter, you'll create your own, but, in order for you to understand how to access the board outline supplied by P–CAD, you'll load the std.pcb board outline, then clear the database and create your own.

The *Designers Component Library* manual contains pen plots of the blank board outlines supplied by P–CAD. These board outlines are standalone files in \pcad\prt on a DOS system. If you're on a UNIX system, the board outlines are in (*pcaddir*)/libs/prt.

Load a board outline from a library

1. Enter P–CAD.
2. Verify that tutor is the current design in the Design directory list box.
3. Select PCB Tools.
4. Select *PCB Editor*.
5. Select *File*!*Load*.
6. Enter **std.pcb** as the filename.

   The std.pcb file is located in \pcad\prt as a standalone file. If you do not find it there, look for std.pcb in the electro.plb library.

Saving the Board Outline with a New Filename

If you were going to use this board outline to create your own, you'd save it with a different filename.

Save the board outline with a new filename

1. Select *File*!*Save*.
2. Enter **tutor1.pcb** as the filename.

Creating Your Own Board Outline

You can create your own board outline if the board outlines supplied with P–CAD don't meet your requirements.

If you intend to use the *Autorouter* tool of the PCB Tools module or the *Placement* commands of the PCB Editor, the board outline must be one continuous line shaped like a polygon. The line can have a complex shape but can't have overlapping segments or curved lines. You can ensure that your outline meets these requirements in one of two ways:

- Use Draw!Rectangle.
- Use the absolute coordinate entry method with *Draw*!*Line*.

When you use the absolute coordinate entry method, you draw lines by selecting XY in the status area and entering the coordinates instead of moving the cursor in the drawing area. You use this method to avoid overlapping segments.

If your board requirements are too complex to use either of the methods, you can create two board outlines on separate layers. (Make a new layer named FABDAT, for example.) Use the BRDOUT layer to create a simple outline for autorouting or placement. Create a complex outline on the FABDAT layer by using any of the *Draw* commands. This complex outline can be implemented into the fabrication and assembly drawings when you prepare for manufacturing.

To create your own board outline, you

- set up the editor environment
- draw the outline
- draw the tooling holes and cross hairs
- assign a board origin

Setting Up the Editor Environment

Before you create the board outline, you set the grid and the active layer.

Set up the editor environment

1. Select *File*!*Clear Database*. The system blanks out the drawing area.
2. Select Grd.
3. Enter **100 100**.
4. Verify that the SnpGrd check box is filled.
5. Select *View Layer*.
6. Enable the BRDOUT layer and make it active.
7. Select *Exit* to close the layer table.

Drawing the Outline

Figure 12-1 shows the board outline you'll create. This figure contains coordinates to help you create the board outline.

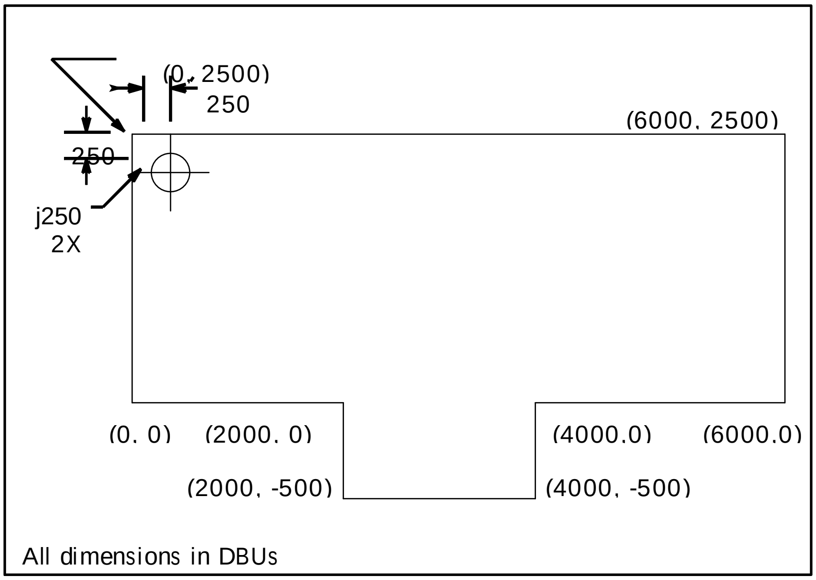

Figure 12-1. Board Outline

Draw the board outline

1. Select *Draw*!*Line*.
2. Select Wid.
3. Enter **0** as the line width.
4. Select XY.
5. Enter **0 0**.
6. Select XY.
7. Enter **0 2500**.
8. Continue to select XY and use the absolute coordinate entry method to complete the board. Use the coordinates in Figure 12-1 as a guide.
9. Select *Fit View*. The board outline is centered in the drawing area.

Drawing the Tooling Holes and Cross Hairs

You'll create the tooling hole shown in Figure 12-2.

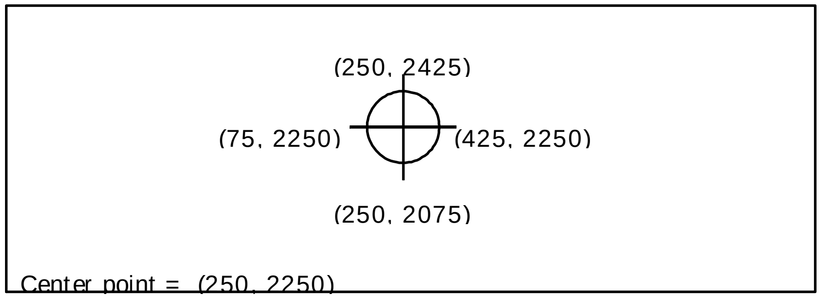

Figure 12-2. Tooling Hole with Cross Hairs

Draw the tooling holes and cross hairs

1. Enable the SLKSCR layer and make it active.
2. Select Grd.
3. Enter **25 25**.
4. Select *Zoom In*.
5. Select a point in the upper left corner of the board outline.
6. Select the point again. Grid points appear on the screen.
7. Select *Draw*!*Circle*.
8. Select XY.
9. Enter **250 2250**. These coordinates are the center point of the tooling hole.
10. Select a point five grid points to the right (375,2250) as a point on the circumference of the tooling hole. You've created a circle 250 DBUs in diameter with a center point at (250,2250).
11. Select *Draw*!*Line* and draw cross hairs for the circle using the coordinates in Figure 12-2 as a guide.
12. Select *Fit View* to see the entire outline again.
13. Use *Copy*!*Objects* or *Copy*!*Window* to copy the tooling hole and cross hairs to the other side of the board. The center of the circle at the new location is (5750,2250).

Assigning a Board Origin

Autorouting and placement both require a pre–assigned board origin.

Assign a board origin

1. Select *Enter*!*Board Origin*.

2. Select (0,0) as the origin point.
3. Select *File*!*Save*.
4. Enter **tutor2** as the name of the file. The system adds the .pcb extension. You'll use this board outline in Chapter 13, when you package a schematic database into a PCB database.
5. Select *File*!*Quit*.

You've loaded a board outline from the P–CAD library. You've also created a board outline. Now you've returned to the P–CAD screen.

What You've Learned So Far

Up to this point in the tutorial, you learned how

- the *AutoLoader* or installation program configures your system with the directories and files you need and some procedures you can follow if you get stuck
- to create a symbol
- to create a schematic
- to check for electrical rules violations
- to create the plot instructions file that you later use to create a hard copy of a schematic sheet
- to create a part
- to create an aperture table file
- to create a padstack
- to create a special symbol file that maps the padstack to the pin types you'll use in the PCB layout
- to create a board outline

What You'll Do Next

In the next chapter you'll create a cross–reference file that maps symbols to physical parts. You'll use the board outline and the cross–reference file to package the schematic in Chapter 13.

Creating a Cross–Reference File 13

What You'll Do in This Chapter

In this chapter you'll create a cross–reference file that maps symbols to parts. You'll map the 5404b.sym symbol you created in Chapter 4 to the 5404b.prt part you created in Chapter 8. In the next chapter, you'll use the cross–reference file when you package the schematic database.

What to Do if You Need Help

Here are some suggestions for things you can do if you need help:

- Read the What to Do if You Get Stuck section in Chapter 1.
- Use the *Design Manager* manual to get information on how to use the *Report Editor*.
- Use the *Schematic Tools* manual to get information on using cross–reference files when you package a schematic.

Understanding a Cross–Reference File

The *Package Schematic* tool in the Schematic Tools module uses the cross– reference file to define which symbols are packaged into which parts on the PCB. Figure 13-1 shows a cross–reference file.

```
% tutor
% cross*reference file
%
%         #gates    device    part name              pin/net    symbol name
% -----------------------------------------------------------------------
          1         cap       ck06.prt               caph.sym
          1         cap       ck06.prt               capv.sym
          1         diode     do15.prt               diode.sym
          1         zener     do15.prt               znr.sym
          1         npnebc    t92ebc2.prt                       npnebc.sym
          1         polcap    case*aa.prt                       polcapv.sym
          1         polcap    case*bb.prt                       polcapv.sym
          1         res       rc07.prt               resh.sym
          1         res       rc07.prt               resv.sym
          4         c4011b    c4011d.prt             (7=gnd,14=+5V)  c4011b.sym
          6         5404b     5404b.prt (7=gnd,14=+5V)          5404b.sym
```

Figure 13-1. Cross-Reference File

Each entry in the cross–reference file consists of either four or five items. These items are the number of gates, the device, the part name, the pin/net description (optional), and the symbol name. Following is a description of each of these items:

- *number of gates* is the number of schematic symbols or gates contained in the physical PCB part.
- *device* is the manufacturer's part number or description that identifies the PCB package.
- *part name* is the filename of the PCB part to which you want your schematic symbols mapped. If the file doesn't exist in your current design directory (as *filename*.prt), the file must exist in another directory or library and the search path must be specified in the *Set Libs. & Search Paths* tool of Library Manager.
- *pin/net* is an optional item used for automatic power and ground pin and net assignment. This item allows *Package Schematic* to assign part pin numbers to power and ground nets when the part pin numbers don't appear on a symbol.
- *symbol name* is the filename of the schematic symbol to be packaged into the PCB part. If the file doesn't exist in your current design directory (as *filename*.sym), the file must exist in

another directory or library and the search path must be specified in the *Set Libs. & Search Paths* tool of Library Manager.

The last 11 lines in the table provide information in a free–form, tabular format. When you set up your file, separate adjacent items by at least one space or one comma.

Special restrictions apply to pin/net descriptions, as shown in the last two lines of the table.

- Enclose the pin/net descriptions in parentheses.
- Separate the pin number and net name by an equals sign.
- Separate all pin/net descriptions from each other by at least one space or one comma.

In this example, the pin/net items are separated by a comma.

Creating a Cross–Reference File

To create a cross–reference file, you

- open the P–CAD text editor
- enter and save the data for the special symbol file
- print the file

Opening the P–CAD Text Editor

The Report Editor uses edlin, or any other text editor available on your system. You specify which text editor you want to use in the *System Configuration* tool of the Design Manager module. For information about using the editor, refer to the documentation for your text editor.

Open the P–CAD text editor

1. Enter P–CAD.
2. Select Design Manager.
3. Select *Report Editor*.
4. Select *ADD*.
5. Enter **tutor.fil** in the Current File data entry box.

6. Select *EDIT*. The Report Editor screen closes and the text editor screen is visible.

Entering Data for the Cross–Reference File

Figure 13-2 shows the cross–reference file you'll create.

```
% tutor
% cross*reference file
%
%       #gates   device    part name              pin/net      symbol name
% -------------------------------------------------------------------------
        1        cap       ck06.prt               caph.sym
        1        cap       ck06.prt               capv.sym
        1        diode     do15.prt               diode.sym
        1        zener     do15.prt               znr.sym
        1        npnebc    t92ebc2.prt                         npnebc.sym
        1        polcap    case*aa.prt                         polcapv.sym
        1        polcap    case*bb.prt                         polcapv.sym
        1        res       rc07.prt               resh.sym
        1        res       rc07.prt               resv.sym
        4        c4011b    c4011d.prt             (7=gnd,14=+5V)  c4011b.sym
        6        5404b     5404b.prt (7=gnd,14=+5V)    5404b.sym
```

Figure 13-2. Cross-Reference File You'll Create

The first five lines in Figure 13-2 are comment lines. Text from the percent sign to the end of a line is a comment and is ignored by the system.

Figure 13-2 contains 11 entries, each of which is on a separate line. Each entry has either four or five items.

Although each entry in this exercise is on a separate line, an entry can also extend over more than one line. However, each item within an entry must be contained entirely on one line.

A pin/net description enclosed in parentheses is considered one item.

Use **Backspace** or **Delete** to correct typographical errors as you make these entries.

Enter data for the cross–reference file

1. Enter **% tutor**.
2. Enter **% cross–reference file**.

3. Enter **%**.
4. Complete the file by entering all the data shown in Figure 13-2.
5. Save and quit your file using commands specific to the text editor you are using. When you quit the file, you return to the Report Editor screen.
6. Select *EXIT* to return to the P–CAD screen.

You created a cross–reference file that you'll use in the next chapter when you package the schematic.

What You've Learned So Far

Up to this point in the tutorial, you learned how

- the *AutoLoader* or installation program configures your system with the directories and files you need and some procedures you can follow if you get stuck
- to create a symbol
- to create a schematic
- to check for electrical rules violations
- to create the plot instructions file that you later use to create a hard copy of a schematic sheet
- to create a part
- to create an aperture table file
- to create a padstack
- to create a special symbol file that maps the padstack to the pin types you'll use in the PCB layout
- to create a board outline
- to create a cross–reference file that maps symbols to parts

What You'll Do Next

In the next chapter you'll package the schematic, using the schematic you created in Chapter 5, the board outline you created in Chapter 11, and the cross–reference file you created in this chapter.

Index