

Quick Start Guide

ProCAD PowerStation 32[®] **FOR WINDOWS**

INTERACTIVE CAD SYSTEMS P.O. Box 4182 SANTA CLARA, CA. 95056

Sixth publication June, 1998 Revision 5.0

Contact Information:

Sales & Technical Support (408) 970-0852
Bulletin board system (BBS) (408) 986-0524
Fax (408) 986-0524

General e-mail sales@icadsys.com
Technical support e-mail support@icadsys.com

Web site http://www.icadsys.com

Sales Office (USA) Interactive CAD Systems
Attn: Sales
P.O. Box 4182
Santa Clara, CA 95056-4182

****DISCLAIMER****

THIS PROGRAM AND DOCUMENTATION ARE PROVIDED "AS IS", WITHOUT WARRANTY OF ANY KIND, EITHER EXPRESSED OR IMPLIED, INCLUDING, BUT NOT LIMITED TO WARRANTIES OF PERFORMANCE, OR FITNESS FOR ANY PARTICULAR PURPOSE. THE ENTIRE RESULTS AND PERFORMANCE OF THIS PROGRAM ARE ASSUMED BY YOU.

© 1986-1998 INTERACTIVE CAD SYSTEMS. ALL RIGHTS RESERVED.

No part of this publication may be reproduced or transmitted in any form or by any means, electronic, mechanical, photocopying or otherwise without the prior written consent of Interactive CAD Systems and, except for personal backup purposes, may not be copied or distributed in any form or by any means without the prior written consent of Interactive CAD System.

This publication could contain technical inaccuracies or typographical errors. Changes are periodically made to the information herein; these changes will be incorporated in new editions of this publication.

PROCAD, **PROCAP** and **PROPLOT** are trademarks of Interactive CAD Systems.

IBM is a trademark of International Business Machines Corp.

MSDOS and **WINDOWS** are trademarks of Microsoft Corp.

GSS and **VDI** are trademarks of Graphics Software Systems, Inc.

GX and **PCX** Graphics are trademarks of Genus Microprogramming, Inc.

Contents

TUTORIAL OVERVIEW	5
CONVENTIONS	5
ENTERING PROCAD	6
Mouse	7
Context Sensitive Commands	8
Keyboard Shortcuts	8
Quick Edit Feature	9
PROCAD Database Screen	9
The Command Line	10
The Status Line	10
X-Y Coordinates	11
The Grid System	11
Ribbon Bar	12
Tool Bar	12
Screen Menu	12
 CHAPTER 1 (SCHEMATIC DESIGN)	 17
OPENING A SCHEMATIC DATABASE	17
Open	17
Schematic Macro	19
Layering Conventions	20
Function Key Definitions	22
PLACING DEVICES	23
INPUT CELL	24
Cell References	24
Reference Designators	24
Change Value	27
PACK and SAVE	28
Schematic Interconnects	28
SET LAYER	28
INPUT WIRE	29
Net Names	33
Adding A Connection	36
Show Signal	37
CONNECTIVITY	38
Bus Structures	38
Step & Repeat Method	40
Using the Copy Method	42
Using Schematic Design Rule Checks	45
Clear DRC	48
Exiting PROCAD	48

CHAPTER 2 (NETLIST EXTRACTION)	51
INTRODUCTION	51
Using PROCAP	51
Menu	53
Netlist Extraction	54
Generic Netlist	54
Simulation Netlist	55
Bill-of-Materials	55
Input Multiple Databases	55
RATS NEST	56
Printing the Output	57
Comparing Output with Input	58
Generic Netlist:	58
PARTS	58
PINS	60
Bill-of-Materials	60
Rats Nest	61
Multiple Sheet Schematics	62
Sheet Prefix	62
Set Mode	63
Pack/R	64
 CHAPTER 3 (PRINTED CIRCUIT BOARD DESIGN)	 69
INTRODUCTION	69
OPENING A PCB DATABASE	69
PCB Macro	71
Layering Conventions	72
Show Layers	75
Function Key Definitions	75
CREATING THE BOARD OUTLINE	76
Board Limits	76
Relative Coordinate Input	77
Keepout Areas	79
Toolholes	80
Placing Components	81
Packaging File	82
Connection File	85
Optimizing the Component Placement	86
Drag and Rubberband	86
Manual Routing Techniques	87
Connectivity	88
Preparing for Manual Routing	89
Board Stitching	91
MANUAL ROUTING OF COMPLEX BOARDS	94

Vertical and Horizontal Routing	94
Defining Stitch Parameters	98
Modifying Existing Routes	98
REROUTE	98
MOVE VERTEX	98
ADD VERTEX	99
DELETE VERTEX	99
CHANGE LAYER	99
CHANGE WIDTH	99
DELETE SEGMENT	99
Autorouting with PROROUTE	99
Routing Passes	99
POWER/GROUND BUS	100
MEMORY	100
LOOKOUT STRATEGY	100
Pre-Routes	100
USING PROROUTE	100
Router Control File	102
Graphic Display	105
Processing RATS NEST	105
Status Line	106
ABORTING AND TERMINATING PROROUTE	106
Set True Width	107
Using Design Rule Checks	107
Clear DRC	110
Clean-up and Exit	111
CHAPTER 4 (CREATING SYMBOLS)	115
INTRODUCTION	115
Data Attributes	115
Regular Cells	116
Change Library	117
Cell Formation	118
Template Cells	124
Hierarchical Cells	131
Schematic Hierarchy	131
Cell Editor	132
Show Hierarchy	138
PCB Hierarchy	139
General Hierarchy	143
Library Management	144

CHAPTER 5 (HARDCOPY OUTPUTS) ----- 151
INTRODUCTION-----151
Plot -----151
PROPLOT-----154
FINAL ARTWORK -----157

APPENDIX A (QUICK REFERENCE GUIDE) -----159
 INTRODUCTION-----159

INDEX ----- 167

Tutorial Overview

INTRODUCTION

This Tutorial is a self-paced instructional guide to using Interactive CAD Systems' PROCAD, for both Schematic and Printed Circuit Board design.

The Tutorial is separated into five major sections: Schematic design, Netlist Extraction, Printed Circuit design, Cell Formation, and Generating Hardcopy Outputs.

Each section of the Tutorial builds upon the previous section to demonstrate the methods by which PROCAD may serve as a vertical design workstation. It is recommended that you work through each section in order, paying particular attention to the sections of specific interest.

This tutorial assumes you have a basic knowledge of MS DOS, MS Windows, and an understanding of PROCAD overview. For an understanding of MS DOS & MS Windows, you should consult your DOS & Windows User's manuals respectively.

This tutorial was written with the assumption that ProCAD is configured for QUICKSET OFF (Use menus). Refer to the CONFIGURATION utility section of the User's Guide for details. Also the left and right mouse buttons are configured for entity select and ESCAPE respectively. See [View]Preference... command for details.

QUICKSET is the command interpreter which allows PROCAD to assume the command execution will take place at the present cursor location. If QUICKSET is toggled OFF, PROCAD will prompt you for the point of command execution. For instance, with QUICKSET turned OFF, PROCAD will prompt you for the first point of a line when using INPUT/LINE. With QUICKSET turned ON, PROCAD places the first point of the line at the cursor location when using INPUT/LINE.

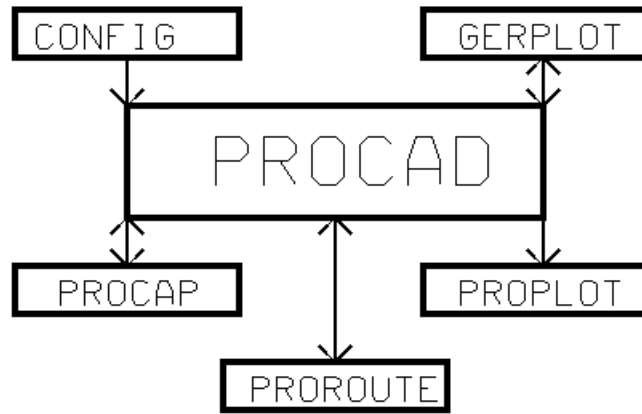
CONVENTIONS:

<i>italics</i>	Operator responses (keyboard entry)
<ENTER>	Press the ENTER key on the keyboard or the left mouse button.
<ESCAPE>	Press the ESCAPE key on the keyboard or the right mouse button.
Cntrl-X	Pressing the control key and the next character simultaneously (in this example, an X)

[MenuX]Cmd Select command **Cmd** from menu **MenuX**

Note: This Tutorial is meant to be used in conjunction with the Quick Reference Guide.

THE OVERVIEW
INTERACTIVE CAD SYSTEM'S
PROFESSIONAL ENGINEERING DESIGN SYSTEM



ENTERING PROCAD

PROCAD must be completely installed onto the hard disk of your computer as described in the User's Manual.

CAUTION: Make sure your computer has the proper date and time. Once you start ProCAD, you may NOT reset your system date backwards. The date is important for PROCAD to keep Design Creation, Revision Dates and maintenance information accurate. ProCAD is Y2K compliant.

ProCAD PowerStation 16 (Windows 3.1x version):

From the DOS prompt **C:>** type **CD\WPROCAD** to change to the sub directory where PROCAD is stored. Type **WIN** and press the **<ENTER>** key once. This will start up the Windows program. From the Program Manager, double click on the PROCAD icon on the PROCAD PowerStation 16 for Windows group. (Please refer to your windows manual if you are not familiar with starting a program from the windows environment).

ProCAD PowerStation 32 (Windows 95/98/NT version):

Start Windows, and select the **[Start][Programs]ProCAD PowerStation 32 group**, then click on ProCAD 32 icon. (Please refer to your windows manual if you are not familiar with starting a program from the windows environment).

You must start ProCAD from its icon, NOT from file manager (explorer in Win 95/98/NT).

The PROCAD Title Screen will appear on your graphic display with your serial number and copyright information. After about 2 seconds, the title screen will disappear, and the default database (TEMPFILE.DBF) will be opened.

NOTE: If this is the first invocation of a CD-ROM version of ProCAD, the registration dialog box will appear, and you will be required to provide your authorization code located on the inside cover of your CD-ROM jewel jacket.

If you do not have TEMPFILE.DBF, an empty one will be initialized for you. You may start your design on the TEMPFILE.DBF and later save it with your desired file name (**[File]SaveAs** menu item), or you may either open an existing or create a new database using the **[File]Open** or **[File]New** commands respectively.

MOUSE:

Your mouse will be active while the PROCAD Title Screen is displayed. Try to control the movement of the on-screen cursor by utilizing the mouse; if the cursor is unresponsive your mouse is not functioning correctly with PROCAD. Abort PROCAD at this time (**[File]Exit** menu item) to correct any problem with the mouse or PROCAD System Configuration.

If your mouse is functioning correctly with PROCAD, or you prefer to work exclusively with the arrow keys, you may proceed with this learning guide.

On a three button mouse: the left button selects an entity at the current cursor position for further editing, the middle button is equivalent to the ENTER key on your keyboard, and will repeat the last command entered, and the right button is ESCAPE.

On a two button mouse: the left button selects an entity at the current cursor position for further editing, the right button is ESCAPE.

*Note: You may re-assign the function of your mouse buttons by choosing the desired assignment using the **[View]Preference..** menu item.*

Note: On a three button mouse, the middle button will operate if and only if you installed the manufacturers supplied drivers for Windows. Make sure you are not emulating Microsoft mouse, since Microsoft drivers only support two mouse buttons.

CONTEXT SENSITIVE COMMANDS

When the left mouse button is setup for entity selection mode (default), click drag on a vertex of a selected wire or line will automatically enter Move Vertex mode, while click drag on a segment of a wire or line enters entity Stretch mode. If the mouse is however, click dragged any where outside the corners or segments of the selected wire or line, entity Move operation is performed.

Note: Context sensitive command works best when the left mouse button is setup for Entity Selection mode.

KEYBOARD SHORTCUTS

The "**PgUp**", "**PgDn**", "**Home**" and "**End**" keys performs **Zoom in (x2)**, **Zoom out (x0.5)**, **Pan** and **Refresh** screen functions respectively. Also, the "**Del**" key may be used for deleting an entity. If the Del key is pressed when the mouse cursor is over a horizontal or Vertical line/wire segment in select environment, that segment only is deleted. If the cursor was on a vertex of a selected line/wire, that vertex only is deleted. If the cursor is outside the vertex or segment of a selected wire/line, the whole entity will be deleted. *Note: When the Del key is pressed while nothing is selected, ProCAD prompts for an entity to delete. Once the entity is identified, it is deleted regardless of where the cursor was placed to select the entity.*

"PgUp"	Zoom In (x2)
"PgDn"	Zoom Out (x0.5)
"Home"	Pan
"End"	Refresh
"Del"	Delete Entity

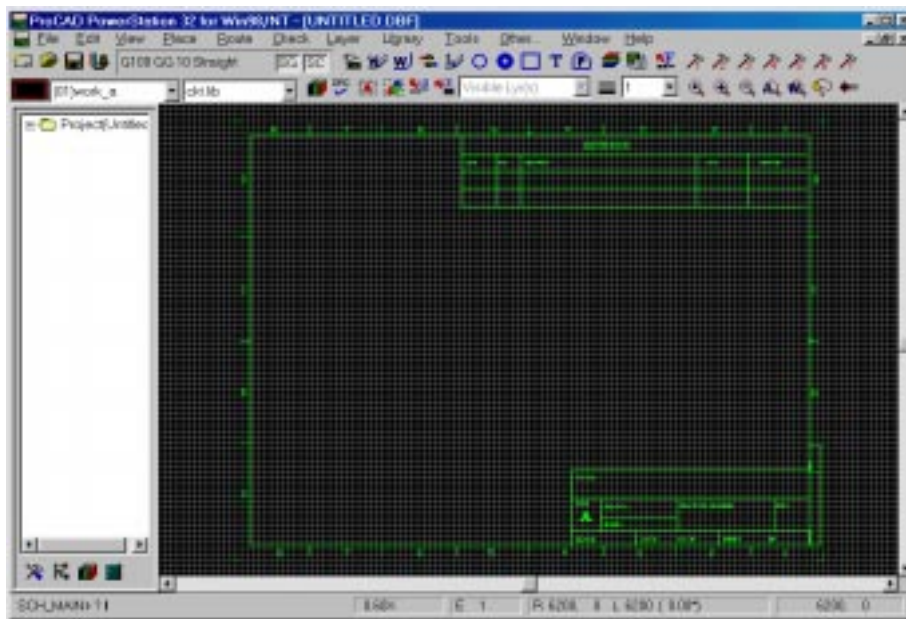
Tips: You may use [View]Preference... menu item to re-program the above keys (except "Del") to perform diagonal cursor movements.

QUICK EDIT FEATURE

This feature is invoked when you double click the left mouse button on an entity. The Quick Edit feature allows for editing or modifying limited entity placement parameters, such as placement layer, trace width, text string editing, instance name and value placements, signal name and placement etc. Most of the functions performed in the **[Other..][Modify]** menu can be accomplished from the Quick Edit dialog box.

PROCAD DATABASE SCREEN:

Take a moment to become familiar with the PROCAD Database Screen.



Default Screen

Looking at the graphic display you will see the PROCAD Database Screen, which includes: the Command Line, the message center, Tool and Ribbon bars, the Status Line, and the Grid.

The Command Line:

The Command Line is located at the lower left hand corner of the display:

SCH_MAIN>

This is where commands typed from the keyboard will be displayed. This area is also used for displaying messages while PROCAD is processing a command.

The Status Line:

The Status Line is the same as the command line, with status information displayed to the right of the command/message section.. Status information is displayed in 3 or 4 fields depending on your graphics resolution. Information which may be displayed can include:



The information displayed across the Status Line is continuously updated as you work on your design in PROCAD.

Magnification: is the current magnification factor. (Status Bar)

E: is the number of entities currently placed in the database. (Status Bar)

G: is the current displayed grid setting (x-value) (Tool Bar).

GG: is the current ghost grid setting (x-value). (Tool Bar)

TR: is the current default trace/track width setting. (Tool Ribbon)

SG: displays the current status of the "Snap on Grid" toggle. (Tool Bar)

SC: displays the current status of the "Snap on Connection" toggle. (Tool Bar)

XX,YY: displays the current cursor position (absolute) in database units. (Status Bar)

R XX,YY: displays the current cursor position (relative to user defined origin) in database units. (Status Bar)

L dd (aa): displays the current distance and angle of the cursor position relative to user defined origin. (Status Bar)

At the upper left hand corner of the graphic screen, on the ribbon bar, is the Active Layer Selection drop down combo box, which displays the current layers with name, number, and a box representing layer color and fill attributes. You may click on this layer box to change its attribute of color, line type and raster mode. (See the SET COLOR command for details).

X-Y Coordinates:

At the far right of the status line, the X-Y Coordinates of the cursor are displayed. You may toggle between relative and absolute display by entering *Cntrl-X* from your keyboard. When the coordinate display is in relative mode, it is prefixed by the letter "R". Use *Cntrl-Y* to set the relative display origin.

The Grid System:

The Grid System is displayed in the actual working area, which takes up the largest area at the center of the graphic display. The format of the Grid System, which is selected during the configuration process, consists of a series of lines or dots.

NOTE: If you are using CGA, or Monochrome, the use of Grid dots greatly increases visibility within the working area.

The Grid System controls the increments of movement which the cursor is allowed during the design process. There are actually two grids within the working area, the Grid (visible on the display) and the Ghost Grid (not visible).

The default Grid established, when a database is opened, is in 100 Mil increments (0.100"). This Grid is generally used for component placement.

The default Ghost Grid is established in 10 Mil increments (0.010") and is used for finer movements during the design. Applications of the Ghost Grid include cell creation, and PCB routing.

Note: The line style grid displays and refreshes significantly faster than the dot style. You should select the dot style (from the configuration utility) only if you have a strong preference for it.

Ribbon Bar:

Across the top of the Grid area is the Ribbon bar. The Ribbon Bar contains buttons for selecting active layer, default library and trace width. It also includes buttons for controlling the screen display, such as zooming in and out, refreshing the screen, pan, etc. The ribbon bar may be disabled from the **[View]Preference** menu item.

Tool Bar:

Directly above the Ribbon Bar is the Tool Bar. The Tool Bar contains buttons for activating other utility programs such as ProCAP (Netlist extractor), ProRoute (Auto Router), etc. Common input primitive buttons, such as Wires, Lines, Circles, Cells, etc. are also included in the Tool Bar. The Tool bar may be disabled from the **[View]Preference..** menu item.

*Note: If you plan to disable the tool and or ribbon bars, you may enable the floating Tool and or Ribbon Boxes from the **[Window]** menu. These boxes are versions of the tool and ribbon bars respectively, except that they occupy less space, and could be re-located any where on the screen, and rolled up and down as needed.*

Screen Menu:

Across the top of the Tool and Ribbon bar area, is the Screen Menu which allows you to access buried menus in a direct fashion. The Screen Menu should appear as follows:

[File] [Edit] [View] [Place] [Route] [Check] [Layer] [Library] [Other] [Window] [Help]

By positioning the cursor on one of these command blocks, and pressing the left mouse button, you may access the menu. The following is a brief description of the function of these menu items:

File: This is the menu for basic I/O operations such as opening database file, saving or closing the design, macro command file execution, hardcopy control file generation, GERBER import and import & export of other CAD ASCII files, etc.

Edit: This is the menu for performing editing operation on placed objects. Commands such as copy, move, rotate, delete, mirror etc. are included in this menu. This menu is also invoked automatically when select mode is entered.

View: This is the menu for basic screen operations such as zooming, panning, display grid spacing selections, screen preference selections, etc.

Place: This is the menu for placing components, nets, rats nests, text, via, etc. in the design

Route: This is the basic menu for placing and routing tracks, stitching tracks, auto routing, etc.

Check: This is the menu for performing design integrity checks such as Design rule checking, Node hi-lighting, DRC\$PADn token verifications, connectivity resolution and floating pins checking, etc.

Layer: This is the menu for controlling basic layer attributes of color, fill style & pattern, mapping for SMT designs, etc.

Library: This is the menu for controlling basic cell library creation, cell editing, automatic padstack cell generation, Library manager, etc.

Other... This is the miscellaneous menu with a number of submenus for performing quick edits, Toggles & Constants dialog boxes controls, Instance attribute assignment and modification, current database status information and mode selection (PCB or Schematic), etc.

Window: This is the menu for performing window manipulation operations such as Tile, Cascade, Arrange Icons, Enable or Disable floating Utility & Drawing tool boxes, database window selection, etc. This menu is especially useful if you are working with multiple databases.

Help: This is the menu for selecting various help topics, and viewing the ProCAD About information of version number, serial number, etc.

NOTE: Again, if you plan to use menus, it is strongly suggested that the QUICKSET feature stay toggled OFF (default setting).

This page is intentionally left blank.

Schematic Design

C*hapter*

1

Schematic Design

Topics Covered:

OPENING A SCHEMATIC DATABASE
PLACING DEVICES
PACK AND SAVE
SCHEMATIC INTERCONNECTS
CONNECTIVITY
USING SCHEMATIC DESIGN RULE CHECKS
EXITING PROCAD

OPENING A SCHEMATIC DATABASE

OPEN:

To open a Schematic Database type:

OPEN/N TESTCKT <ENTER>.

or use the **[File]New** menu item.

PROCAD will display the message "Initializing Database File TESTCKT.DBF" and then opens a dialog box with the following requested information:

Database file:

Here you enter the name you want assigned to the database (schematic) file. For this exercise, enter the name TESTCKT.DBF as shown below.



Database Units (Mils,IN,CM,MM)[MILS]:

Here you should determine the units of measure which will be used during the design process. It is highly recommended to utilize the default selection [MILS] to maintain compatibility with all PROCAD Libraries.

Screen Display Factor (1..99)[10]:

This controls the grid/pixel ratio based upon a complex equation (See SCREEN DISPLAY FACTOR in the Reference Manual for a complete explanation). Use the default selection [10].

Database Type [P]CB or [S]chematic [S]:

This selection controls the method of interconnect, logical or physical, which PROCAD will utilize during the design process.

If you select [S]chematic the following information will also be requested:

Schematic Sheet [1]:

This selection allows you to link up to thirty schematic sheets together for one PCB design. Simply specify which sheet in the sequence this database will represent.

Schematic Sheet Size [A-E]:

This may be a value of A,B,C,D, or E, and allows PROCAD to place a drawing sheet border cell ([CKT]ADWG, [CKT]BDWG, [CKT]CDWG, [CKT]DDWG or [CKT]EDWG respectively) into the design database for your convenience. This template cell may be used for documenting various aspects of the design such as part number, date, and revision numbers. It is only there for your convenience, and may be removed from the design database using the **[Edit]Delete** command.

Other variables may be set if you click on the "More.." button. For now, you should use all default settings.

Refer to the appropriate commands in the reference manual for details on other database variables.

If you are opening an existing database file, PROCAD will open it, and will not issue any of the above prompts, since these values were already saved as part of the database.

SCHEMATIC MACRO:

Before beginning a design, the database should be configured for use in schematic design, or PCB design.

Database parameters may be set for schematic designs using the macro command CKT.CMD.

A macro command, or macro, is a series of predefined commands which execute in sequence when accessed through the macro.

For instance, the CKT.CMD macro performs a variety of commands to configure database parameters for schematic design.

To use the CKT.CMD macro simply enter the following command onto the command line:

`MACRO/P CKT <ENTER>`

or select the **[File]Run command file...** menu item, selecting the "CKT.CMD" file when prompted to select a macro command file.

PROCAD echoes the command as they are being executed.

CKT.CMD then defines layer information such as color, fill, and name, as described in the PROCAD Quick Reference Section on page 163.

CKT.CMD makes the active library CKT.LIB as is displayed in the Ribbon bar across the top of the screen.

CKT.CMD also defines all 25 function key positions as described in the Quick Reference Section.

Note: ProCAD will automatically execute the CKT.CMD macro command file after initializing a schematic type database file.

LAYERING CONVENTIONS:

Layering concept applied to Schematic design is only used for color and line type effects on the schematic drawing. If you don't care for these effects or you plan to use a monochrome display, you may ignore all references to placement layering conventions, and skip this section.

One important task performed by CKT.CMD is the assignment of layer parameters.

The importance of layering conventions is most apparent when trying to generate graphic output containing only specific types of information.

However, maintaining proper layering conventions is also important when trying to display specific information on the graphic screen, while designing in PROCAD.

NOTE: For most effective use of PROCAD, and the libraries provided with PROCAD, you should strictly adhere to the prescribed layering convention.

The Schematic Layering Convention recommended for use with PROCAD, and actually used in all PROCAD standard libraries is as follows:

Layer 1 - The Work Layer/Placement Layer is used exclusively for the placement of Schematic symbols, and their reference designators. A symbol is any device stored in a library for repeated use within PROCAD. A Schematic symbol may include resistors, TTL or CMOS devices, linear devices, or a drawing border.

Layers 2 thru 4 are used for drawing Schematic Symbols during the cell creation process.

Layer 6 is used to place interconnect wires into the Schematic design. This layer should be used for connection wires only.

Layer 9 is used for placing non-electrical text into the design. Non-electrical text is the type which would be used in the corner of a drawing border to specify design name and revision number.

Electrical text is text used in reference designators, and is placed on layer 1 by PROCAD because it is part of the symbol in the design database.

All other layers should be used as specified below:

SCHEMATIC LIBRARY LAYER ASSIGNMENTS

<u>LYR</u>	<u>TYPE/FILL</u>	<u>USE</u>	<u>COLR</u>	<u>CODE</u>
1	SLD/OFF	WORK LYR & TEXT	RED	[2]
2	SLD/OFF	SCHEMATIC SYMBOL	GRN	[3]
3	SLD/ON	SCHEMATIC SYMBOL	GRN	[3]
4	DOT/OFF	SCHEMATIC SYMBOL	GRN	[51]
5	SLD/OFF	CONNECTION DOTS	BLU	[4]
6	SLD/OFF	WIRE CONNECTIONS	YLW	[5]
7	SLD/ON	LINES(non-elect.)	RED	[2]
8	DAS/OFF	LINES(non-elect.)	RED	[34]
9	SLD/OFF	TEXT (non-elect.)	YLW	[5]
10-18		USER DEFINED		
19	SLD/OFF	LOGO/DWG BDRS	GRN	[3]
20-29		USER DEFINED		
30-39		RESERVED		
40-48	SLD/OFF	NESTED INFORMATION		
49	SLD/OFF	RESOLVE ERRORS	RED	[2]
50-98		USER DEFINED		
99	SLD/OFF	HIGHLIGHTING	WHT	[129]

Layers 31 through 49 are reserved for special functions and should not be used for schematic design.

The CKT.CMD macro sets up the following default entity layer placement configuration:

<u>Entity type</u>	<u>Placement Layer</u>
Cells (Components)	1
Wires(traces)	6
Text(Non-electrical)	9
All other (lines,circles, rectangles etc)	Current active layer

Spend a few moments familiarizing yourself with PROCAD's Schematic layering convention.

FUNCTION KEY DEFINITIONS:

The function keys are divided into three groups of nine: F1-F9, Shift F1-F9, and Cntrl F1-F9 (except Cntrl F4 & F6). All of these 25 function keys are user definable, and are provided to allow PROCAD more flexibility for the user.

CKT.CMD predefines all 25 function keys for use in schematic design as described in the Quick Reference Section.

F1-F9 are utilized primarily for screen manipulation, and placing special symbols such as in-sheet & off-sheet connectors, connector dots, power & ground symbols etc.

Shift F1-F9 are data manipulation commands utilized for moving, copying, or otherwise manipulating data in the database.

Cntrl F1-F9 are schematic attribute commands used for assigning, and changing, reference designators and node names.

It is recommended that you spend a couple of minutes familiarizing yourself with the general structure of the function key definitions as described above. Here it is again:

F1-F9	Screen Controls & special symbol input
Shift F1-F9	General Manipulation
Cntrl F1-F9	Schematic Manipulation

As you familiarize yourself with PROCAD, and the Schematic Function Key definitions, you may find that different commands will suit your purposes more effectively. Feel free to redefine the function keys as you see fit. See the SET FUNCTION command in the Reference manual for details.

PLACING DEVICES

With a fundamental understanding of the function key definitions, menu structure, and the layering conventions, you are now ready to begin building a schematic design. In this exercise, we will be accessing commands using primarily menus. You may choose to type in the commands or use function keys if you so desire. As a simple exercise, build the schematic shown in Figure 1 by following the step-by-step instructions provided.

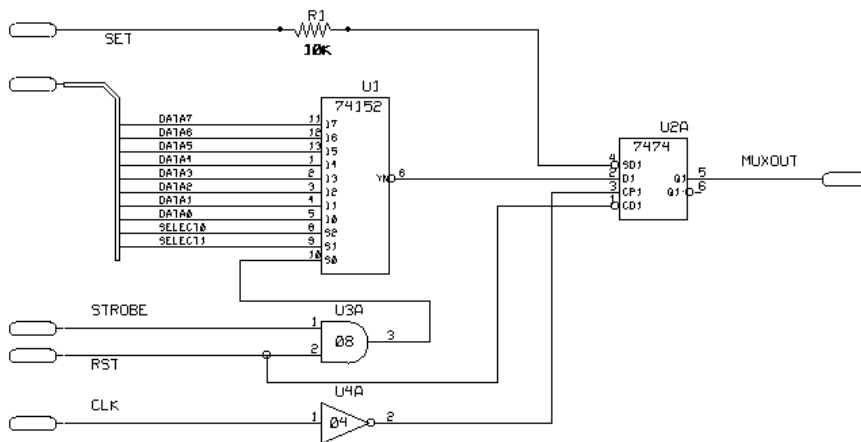


Figure 1

CHANGE LIBRARY:

To begin with, make the TTL library active. You will recall that CKT.CMD macro has established CKT.LIB as the active library.

Access the TTL library from the drop down combo box on the Ribbon bar. This becomes the default active library.

INPUT CELL:

Access the **Component** command from the Tool bar or **[Place]Get Cell...** menu item. PROCAD opens a window of available cells. Select the cell 74152 (You may have to scroll through the library to find this cell). Now move your cursor to the upper left area of the grid and press the left mouse button to place the first device. Click the Right mouse button to exit cell placement command.

CELL REFERENCES:

Notice the device is drawn around your cursor at one point (normally the upper left hand corner of the device is drawn at your cursor).

The point of the cell, which is located on the cursor upon placement into a database is known as the Cell Reference or Origin Point.

The Cell Reference Point is important for placing and manipulating schematic symbols stored in the libraries.

REFERENCE DESIGNATORS:

Notice that the 74152 device has been assigned a reference designator of U1.

PROCAD automatically assigns numerically sequenced reference designators as devices are called from the libraries and placed into the design database.

Access the **[Place]Get Cell...** command again, selecting the device 7474 for placement. Move your cursor to the right of the 74152 device and click the left mouse button to place the 7474 device, and the right mouse button to exit the command. Notice the position of the cell reference point and the reference designator, U2.

MOVE:

At this time you may want to move a component to place it in better alignment with another device, or simply to reposition the device within the design.

Position the cursor on the newly placed 7474 cell, and press the left button to select it (you may also use the **[Edit][Select]Entity** menu item or access the **Select Entity or Cell** command from the ribbon bar). The system will hi-lite the cell signifying its selection, and the **[Edit]** menu will then become active. Access the **Move** command from this menu to Move the

selected entity (in this case the 7474 cell). PROCAD will then prompt you for the reference location and the location to move it to.

*Note: After an entity is selected, click dragging your left mouse button outside the selected entity automatically places ProCAD in entity move mode (See the **Move** command in the Reference manual for details).*

Move the device to position it relevant to the 74152 as illustrated in Figure 2. Click the left mouse button to place the device, and the right mouse button (<ESCAPE>) to exit the select environment.

Tips: This method of selecting an entity and then applying any of the editing operations is very useful if you plan to perform repeated editing operations on the selected entity or a group of entities. If you only wanted to perform a single edit operation on a single entity, you may simply access the appropriate command from the [Edit] menu, and the system will automatically execute the implied selection, and exit sequence after the editing operation has been performed. This is by far a faster method than that illustrated above, but either method has its merits.

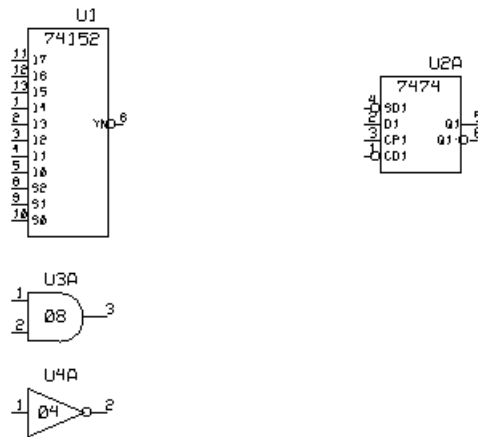


Figure 2

PAN:

Access the **Pan** command from the ribbon bar or **[View]** menu. Move the cursor to the bottom of the 74152 symbol, and move it down from the device, and click the left mouse button to **PAN** the cursor location to the center of the screen. Notice the screen display is refreshed with the old cursor coordinates now displayed at the center of the screen.

The **PAN** command is one of the Screen Control commands assessable from the ribbon bar and **[View]** menu. You may also use the "**Home**" key (Keyboard shortcut)

You will find these commands are useful for moving around in your design.

Now that you have panned the display to a new location in your design database, place the 7408 and 7404 devices as is also illustrated in Figure 2.

A rough placement is fine when initially placing the components into your design. This is how you will actually begin placing components in your own schematics when you are ready to start your own designs.

Move the cursor up, away from the top of the 74152 device to perform the next step. You may **PAN** the screen if it is required.

Input a resistor into the drawing by typing the following command:

GET [CKT]RES <ENTER>

Notice that in order to place the resistor device into your design you must specify the library [CKT]. The reason for this is that the resistor symbol is not stored in the active library, TTL.LIB, so you must specify the library which it is stored in.

This command structure allows you to retrieve symbols from all libraries without changing the active library. *Note: If you prefer to use the **Component** command from the Tool bar or **[Place]Get Cell...** menu item, you must first change your active library (**[Library]Change Lib...** menu item), since this command will automatically open a window of cells in the current active library when issued from the Tool bar or **[Place]** menu.*

Place the resistor as shown in Figure 1.

CHANGE VALUE:

After placing the resistor, you may assign it a value by accessing the **[Other..][Modify]Instance Value** menu item, then position the cursor onto the device and click the left mouse button. This command allows you to specify the value reference (justification) coordinate, text height, width & angle, and the value assigned to the instance.

Enter Value Ref. Coord:

Position your cursor about 100 mils below the resistor symbol as shown in Figure 1 and click the left mouse button.

Value Height [75]:**Value Width [1]:****Value Angle [0]:**

Accept the default height of 75 mils, default width of 1 mil and default angle of 0 degrees by clicking your left mouse button in response to the three prompts above.

Enter Component Value [NONE]:

Enter a value of *10K* <RETURN>.

*Note: You may also use the **[Other..][Quick Edit...** command or double click on the device to accomplish the same operation above. If you use the **Quick Edit** method to modify device value placement position, you will need to enter the absolute coordinate of its display reference (justification) point.*

PROCAD allows you to assign individual values of up to eight alpha-numeric characters to symbols as they are placed into the design.

Some cells do not need values, but it is a useful capability for defining the values of discrete components.

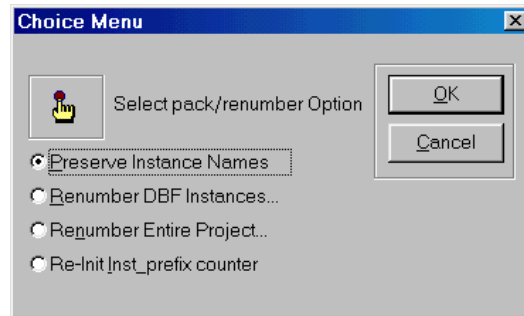
The value of each device may be displayed in the schematic design, and may also be extracted into the Bill-of-Materials. See Chapter Three of this Tutorial for more information on Bill-of-Materials.

You have now placed all the schematic symbols from the library, and are ready to begin interconnecting them to form your schematic.

PACK AND SAVE

NOTE: At this time you should SAVE your work in progress. PROCAD may perform automatic database save if this feature is enabled (SET AUTOSAVE CONSTANT from [Other]Constants... dialog box). You must form the habit of saving all work periodically.

Before saving the database execute the **[File]Pack/Renumber...** command. This command empties an area in memory in which PROCAD stores all deleted entities.



When you delete an entity (cell, wire, text...) PROCAD maintains the deleted information in a buffer for error recovery.

When you are ready to save the work in progress, it is a safe assumption that you have not recently made any errors. Select the **"Preserve instance Names"** option from the choice menu, then click the Okay button to execute the command.

After packing the database access the **[File]Save** command.

PROCAD saves your work to disk, and returns to the command mode to continue working.

SCHEMATIC INTERCONNECTS

SET LAYER:

Occasionally, you may want to change the current entity placement layer. To do this, select the desired layer from the layer selection combo box in the Ribbon bar. This will make the selected layer the active placement layer. You may also accomplish the same operation by accessing the **[Layer]User Select** menu item, or entering the following keyboard command:

SET LAYER <ENTER>

PROCAD will then respond with the following message:

Enter Working Layer [xx] ?

Type the desired layer number and press <ENTER>. The new layer selection will be displayed in the Ribbon bar. Layer 6 will be used in the next section.

INPUT WIRE:

You are now ready to enter your first wire interconnect.

Recall that the CKT.CMD macro sets up layer 6 for placement of wire interconnections into the design. This is where all wires entered will be placed.

The first wire you will enter into your design goes from the Output of the 7408 to the Input S0 of the 74152 as illustrated in Figure 3.

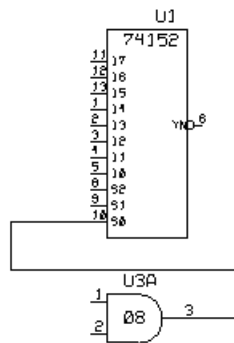


Figure 3

Begin by accessing the **Wires** command from the Tool bar or **[Route]Wire (Digitized)** menu item. Placing the cursor on the Output of the 7408, and click the left mouse button to start placing the wire .

Move the cursor horizontally away from the Output pin about 200 Mils. Click the left mouse button again.

PROCAD places a wire segment between the starting point (Output pin) and the location of the cursor when you hit return or click the left mouse button.

Now, move the cursor vertically up about 300 Mils and click the left mouse button again. PROCAD adds another wire segment.

Continue to move the cursor, and click the left mouse button until the cursor is positioned at the Input S0 of the 74152 device. Your line should look like the one illustrated in Figure 3.

Click the right mouse to close the wire and input it in the design. Click the right mouse button the second time (<ESCAPE>) to leave the INPUT WIRE mode.

If you make a mistake while inputting a wire, you may press *U* to undo the last segment entered. You may also press "*Home*" to pan, "*PgUp*" or "*PgDn*" to zoom IN or OUT the screen while you are still inputting the wire. Pay attention to the status line prompts while inputting a wire.

Enter the next wire from the Output of the 7404 to the Input CP1 of the 7474 as shown in Figure 4. Also connect pin 6 of U1 to pin 2 of the 7474.

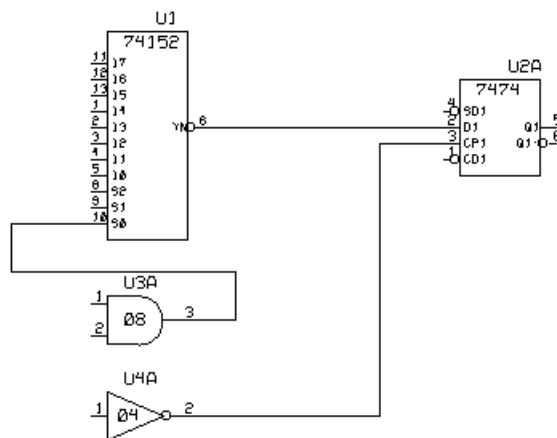


Figure 4

Pan the display so that the 7408 device is centered on the screen.

The **SELECT FILTER(KINDS)** command provides a filter mechanism to manipulate a specific kind of information such as **WIRE, TEXT, or CELL**.

By using the **SELECT KIND WIRE** command, all subsequent manipulations will be directed at wires only.

COPY:

The **COPY** command allows you to easily duplicate portions of a design.

Access the **[Edit]Copy** command. Position your cursor on Input 1 of the 7408 device and click the left mouse button to start the copy process.

PROCAD highlights the wire you have selected and prompts for a copy reference point. Simply click the left mouse button to use the current cursor position as the copy reference point. PROCAD then returns the following message:

Copy To:

Position the cursor on the Input 2 of the 7408 and click the left mouse button. Notice how PROCAD copies the wire from Input 1 to Input 2.

Position the cursor on the Input of the 7404 device and click the left mouse button again. Notice how PROCAD copies the wire again to the new location.

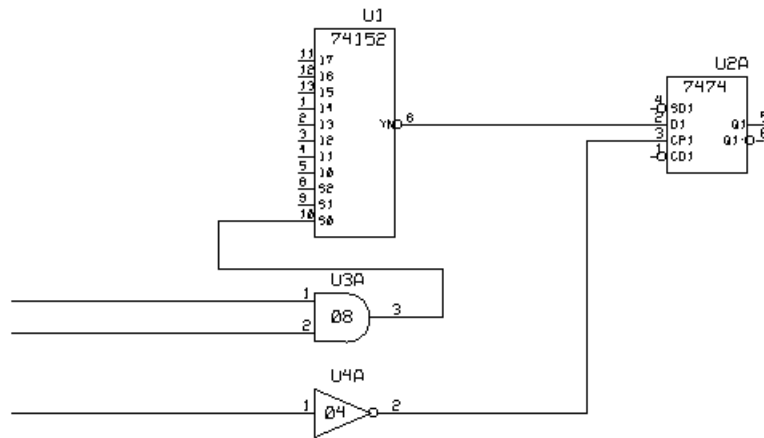
Click the right mouse button (<ESC>) to stop copying the wire.

You now have three wires as illustrated in Figure 6.

Type the following command:

SEL KIND ALL <RETURN>

This will reset the select mask to access all entity types (You may use the menu as in the previous step to accomplish this task if you so desire).

**Figure 6****NET NAMES:**

The wires which have been INPUT into the schematic design, have so far been undefined signals. The wires do not have any user-defined signal names.

PROCAD automatically assigns a signal name to all wires as they are input into the design database.

The automatically assigned signal names are sequentially numbered as N0000QQ, N0001QQ, N0002QQ...

By automatically assigning signal names as wires are added to the design, PROCAD saves you the effort of having to name every node manually.

CHANGE SIGNAL NAME:

There are times, however, when you will want to assign signal names which will reflect a function, or are part of a bus structure.

Access the **[Other..][Modify]Net Name** command. Position your cursor on the wire which connects to Input 1 of the 7408 device. Click the left mouse button to start the Change Signal Name command. PROCAD returns the message:

Is Signal Name Display Required? [Y]

Select **YES**.

Signal Name Justification Coordinate:

The Justification Coordinate is the bottom left coordinate of the text as it is to be positioned. See Figure 7.

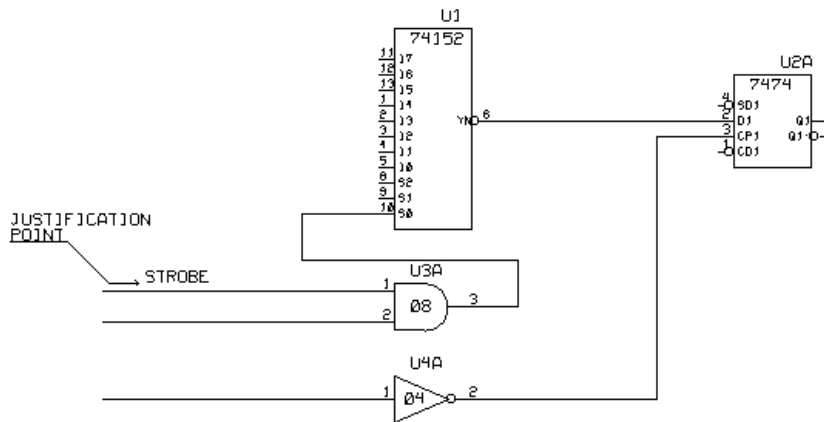


Figure 7

Move the cursor off the line and position it where you would like the signal name to be displayed. Click the left mouse button.

Signal Name Placement Height: [75]

This is the height of the text in database units. In this case the database units are Mils, and 75 Mils is fine for the signal name height. Press <ENTER> or click the left mouse button to accept the default height of 75 Mils..

Signal Name Placement Width: [1]

This is the width of the text in database units. In this case the database units are Mils, and 1 Mil is fine for the signal name width. Press <ENTER> or click the left mouse button to accept the default.

Signal Name Placement Angle: [0]

This is the angle at which the signal name will be displayed. The possible choices are 0 thru 359 degrees rotated in a counter-clockwise direction. Press <ENTER> or click the left mouse button to accept the default.

Signal Name [N0000QQ]:

Here you may assign a new signal name to the wire. Type:

STROBE <ENTER>.

PROCAD now changes the name of the signal and displays the new name according to the display values which you have defined.

*Note: You may also use the **Quick Edit** (double click on the wire) to accomplish the same operation above in a dialog box. If you use the Quick Edit method to modify signal name placement position, you will need to enter the absolute coordinate of its display justification point.*

If you have a three button mouse, simply press the middle mouse button to use the PROCAD repeat feature to repeat the last command (in the case the Change Signal Name Command).

Repeat these steps for the wire on Input 2 of the 7408 device naming it RST.

Repeat this exercise again on the wire connected to the Input of the 7404 device, renaming this wire CLK.

You have now named all three of these wires as illustrated in Figure 8.

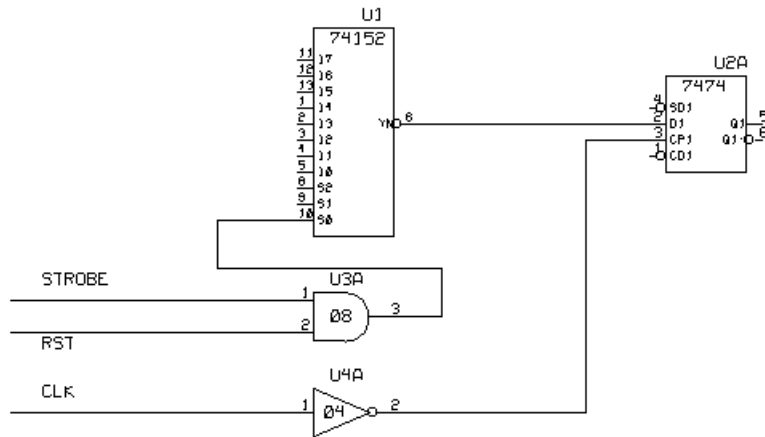


Figure 8

ADDING A CONNECTION:

PROCAD utilizes two techniques for recognizing logical connections of signal wires: **Common Signal Names** , and **Common Vertexes** .

You will now add a connecting wire from the RST signal to connect to the INPUT CD1 of the 7474 device using Common Vertexes.

Access the **Wires** command from the Tool bar or **[Place]Wires (Digitized)** menu item. Position the cursor on the RST wire, about 200 Mils off of the Input 2 of the 7408 and click on the left mouse button to start the **Input Wires** process. Now move the cursor to the Input CD1 of the 7474 device as illustrated in Figure 9. and click the left mouse button.. Click the right mouse button to place the wire and <ESC> to exit the Input Wires command.

Notice that ProCAD automatically places a connection dot at the "T" junction together with the necessary vertexes on the horizontal or vertical wires to establish connectivity.

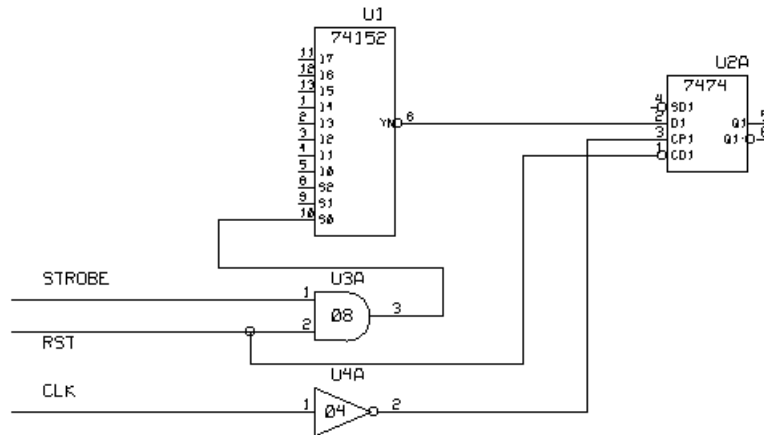


Figure 9

SHOW SIGNAL:

To verify that the new wire is connected to the original RST wire you can use the **SHOW SIGNAL** command.

Type the command from the keyboard:

SHO SIG RST <ENTER>

PROCAD highlights the RST wire with all other wires which are connected to, or are part of the RST signal net.

Note: You may also access the Show Signal command from the **[Check]** menu, then type in the name RST from the keyboard when prompted. Since this command requires an input from the keyboard, the first method of invocation is usually preferred.

The new wire should be highlighted as part of the RST net. Press <ENTER> to remove the highlighting.

CONNECTIVITY

This is a very important demonstration of schematic connections in PROCAD.

PROCAD makes logical connections of wires by means of Vertex Recognition.

In order for two wires to be logically connected they must share a common vertex at some point.

You have added a wire to the RST signal by placing a common vertex at the connection point. Two concepts must be firmly understood from this demonstration:

***Tips:* PROCAD does not consider wires to be logically connected unless they share a common vertex or connection dot. Two wires which crossover, and share a common vertex (or have a connection dot at the cross over point), will be logically shorted.**

By understanding these two concepts, you can greatly reduce the number of shorts and opens which your design may contain.

BUS STRUCTURES:

The second method which PROCAD utilizes to recognize logically connected signals is the Common Signal Name such as might be used in a Data Bus.

Access the **[View]Zoom All** command, then Zoom In around the 74152 device.

Access the **Pan** command from the Ribbon bar or **[View]Pan** menu item. Position the cursor in the center of the 74152 device and click the left mouse button to center the 74152 in the screen.

You are now ready to construct the bus illustrated in Figure 10.

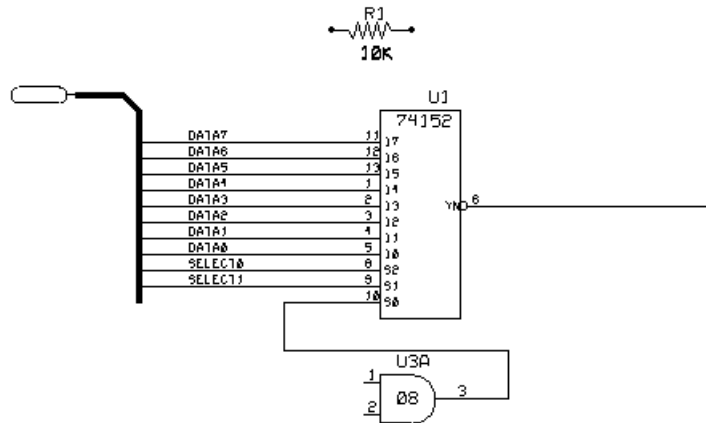


Figure 10

Access the **Wires** command from the Tool bar or **[Route]Wire (Digitize)** menu item. Place the cursor on the Input I0 of the 74152 and click the left mouse button to start inputting a wire from U1 pin 5. Move the cursor horizontally to the left, away from the device about 800 Mils and click left mouse button., and then right mouse button twice to complete the wire input.

Change the name of this new signal by using the **CHANGE SIGNAL** command from the keyboard or **[Other..][Modify]Net Name** menu item.

CHANGE SIG <ENTER>

PROCAD then responds with the following messages and prompts:

Is Signal name display required? YES <ENTER>

Signal Name Justification Coordinate?

Place justification on the wire, 200 Mils away from the unconnected end.

Signal Name Placement Height? 75 <ENTER>

Signal Name Placement Width? 1 <ENTER>

Signal Name Placement Angle? 0 <ENTER>

Signal Name? DATA0 <ENTER>

The name of this signal should now be displayed at the specified justification point as illustrated in Figure 11.

Note: You may use the **Quick Edit** feature (double click on the wire) to accomplish the above task in a dialog box.

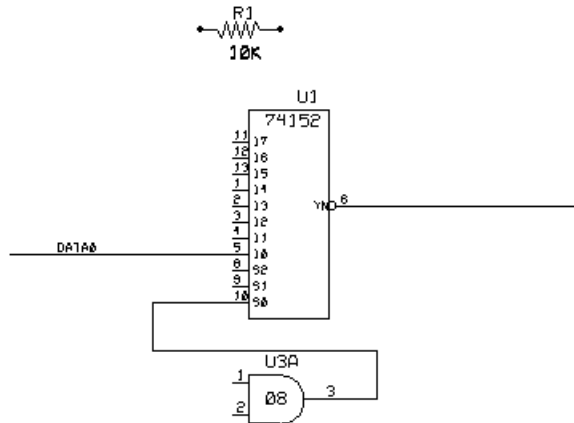


Figure 11

STEP & REPEAT METHOD

Access the **Step & Repeat** command from the **[More..]** submenu of the **[Edit]** menu. Move the cursor to the end point of the newly entered wire (DATA0) and click the left mouse button. PROCAD will highlight the wire and the following series of prompts will be displayed:

X-Step Offset [0]:

This is the spacing desired between the stepped entity in the X direction. Since the bus wire will be placed vertically, accept the default of 0 by pressing <ENTER> or the left mouse button.

Y-Step Offset [0]:

This is the spacing desired between the stepped entity in the Y direction. Type 100 <ENTER> to space the wire 100 database units (mils in the case) apart.

X-Repeat Count [0]:

This is the number-1 times the entity will be replicated in the X direction. Accept the default of 0 by just pressing <ENTER> or the left mouse button.

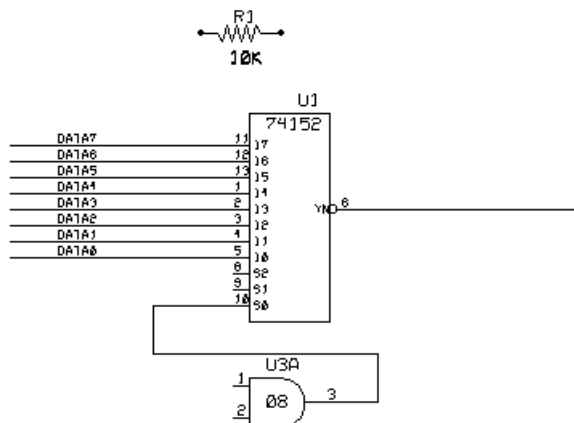
Y-Repeat Count [0]:

This is the number-1 times the entity will be replicated in the Y direction. Type 8 <ENTER>.

X-Offset:0 Y-Offset:100 X-Repeat:0 Y-Repeat:8 Bus_Start_Name:DATA0

Please Confirm (Y/N) [Y]:

Accept the values entered by typing Y <ENTER>. Notice how PROCAD creates the bus structure, sequentially naming the signal DATA0DATA7. The Step & Repeat command can also be used to duplicate the placement of any selected item(s) in both the X and Y directions. See the STEP & REPEAT command in the PROCAD Reference manual for an explanation of this powerful command. Your display should now be similar to Figure 12.

**Figure 12**

USING THE COPY METHOD

You can also use the copy method to enter a bus wire. This method is most useful if you want to place the same signal in different section of your design without actually placing the thick bundle wire.

Access the **Wires** command from the Tool bar or **[Route]Wire (Digitized)** menu item. Place a wire from U1 pin 8 and line it up horizontally to DATA0 wire previously placed. Now use the **[Other...][Modify]Net Name** command (or double click on the wire) to assign the name SELECT0 to it.

Access the **[Edit]Copy** command. Copy this signal to U1 pin 9. Press <ESC> to exit the copy process. Notice that both signals now have the same name "**SELECT0**". At this point, they are logically connected together.

To illustrate this connectivity of signals, type the following command from the keyboard:

SHO SIG SELECT0 <ENTER>

PROCAD will highlight all signals which are part of a common signal net. In this case, PROCAD highlights all two **SELECT0** signals. Now access the **[Other...][Modify]Net Name** command. Position the cursor on the Input S1 pin of device 74152 press <ENTER>. Accept the default signal name justification point by pressing <ESC>, and the default height, width and angle also by pressing <ESC>. Change the name to *SELECT1* <ENTER>.

Note: Since you are not changing the justification point of the signal name, you may be better off using the faster Quick Edit command (double clicking on the wire) and then changing the signal name.

Signal already assigned name SELECT0. Modify? [N].

This message simply reminds you that you are changing the name of a previously named signal. Notice that the default answer is **NO**, however you should select **YES** in order to complete the name change.

Your screen should now look like the illustration in Figure 13.

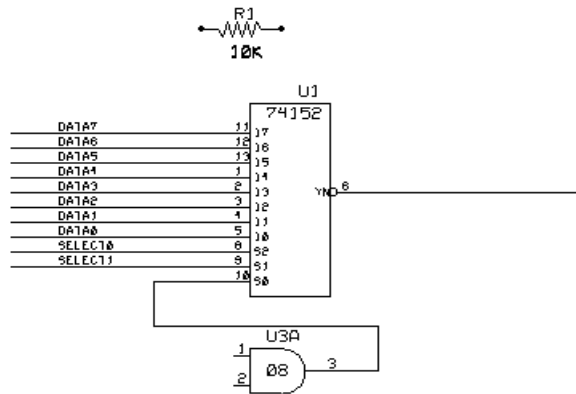


Figure 13

Now repeat the *SHO SIG SELECT0 <ENTER>* command, and notice that PROCAD only highlights the single remaining **SELECT0** wire.

Now to add the non-electric bundle; Set layer 7 as the active working layer (use the layer selection combo box from the Ribbon bar). Access the **Lines** command from the Tool bar. Position the cursor at the unconnected end of the wire labeled **SELECT1** and click the left mouse button to start inputting the line/bundle.

Enter the bundle as illustrated in Figure 14. Remember to press <ESC> when you have finished entering the bundle to exit the Lines input mode.

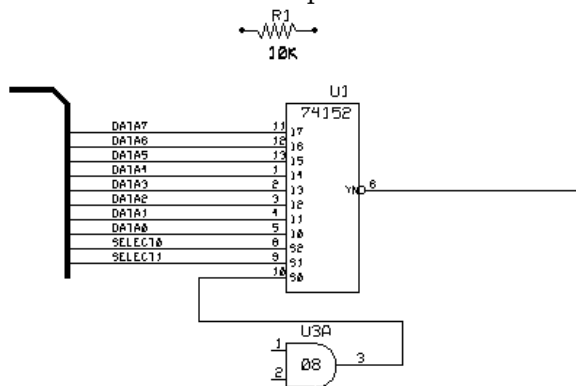


Figure 14

CHANGE WIDTH:

To change the thickness of the bundle line, access the **[Edit][More..]Change Entity Width** command. Position your cursor on any of the vertices (corners) of the bundle line and click the left mouse button.

PROCAD will highlight the line and issue the following prompt:

Enter New Width [1]:

Type **25 <ENTER>** to change the width of the selected line to 25 Mils. The bundle should now be displayed as a line with a visible width. If the layer attribute for layer 7 is solid fill, the bundle will be solid filled. You may also use the faster Quick Edit method to modify the line width.

Complete this sample circuit as seen in Figure 15, by adding the in-sheet and off-sheet connector symbols from the [CKT] library, or use function keys F7 and F6 respectively.

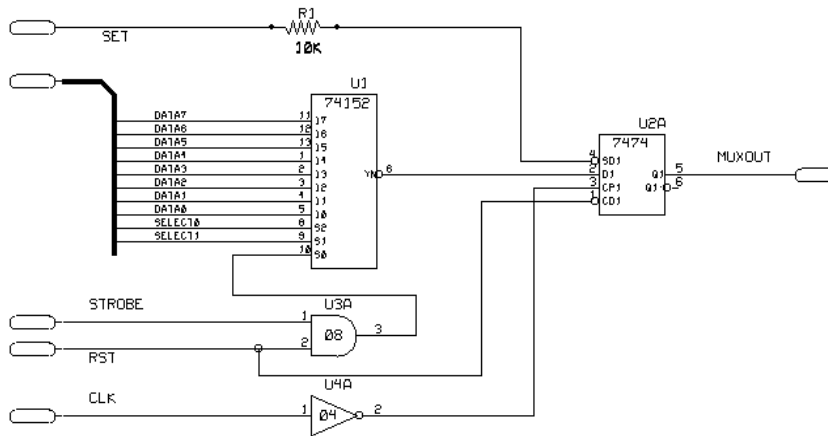


Figure 15

USING SCHEMATIC DESIGN RULE CHECKS

PROCAD has an on-line Design Rule Check utility which allows you to quickly check certain aspects of your design without leaving the PROCAD editor.

To use the Design Rule Check feature simply type the DRC <ENTER> command from the keyboard or access the **[Check]Design Rules Check** menu item.

The Design Rule Check dialog box will appear in the upper right hand corner of the graphic display and should contain the following information/prompts:



These are the default values assigned by PROCAD when the DRC utility is accessed.

Only five fields of the DRC values are of interest when checking a Schematic database. They are:

DRC Layer Combinations(s) 6

This represents the layers which will be tested by PROCAD's DRC.

Layer 6 must be included in this list. If you have followed the Schematic Layering Convention described in the PROCAD Quick Reference Guide, all wire interconnects should be placed on Layer 6.

Domain_check OFF

This is used to establish a window area within a large database, which you wish to check.

With Domain_Check toggled OFF, PROCAD will check the entire database.

Checking a small area of the entire design will greatly speed up the DRC utility, and is sometimes all that is required.

Floating input check ON

This flag causes ProCAD to place markers on layer 49 at all inputs that does not have a wire connected to it. Toggle it OFF.

Shorts_check ON

Checking for shorts in a Schematic database can be useful for locating graphical shorts (wires which cross on the same layer without sharing a common vertex).

However, in a Schematic design, it is often correct for two wires to cross without being shorted. This check simply provides you with a quick means of locating potential shorts and opens within the design. Toggle it ON.

Save DRC Markers ON

This statement allows you to view all DRC flags as temporary or saved within your database.

With Save DRC markers toggled OFF, all DRC flags will be erased when the screen is REFRESHed or REDRAWN. With Save DRC markers toggled ON, all DRC flags will remain in the design database until they are cleared.

Click the Begin button to start execution of the DRC function. PROCAD will display status information regarding the progress of the DRC, on the Message Line.

If you have designed your CKT circuit as described in the Tutorial, PROCAD will return a message of:

1 DRC Error(s) Found

This error will be marked with a DRC flag on layer 34. DRC flags are assigned layers as follows:

Error	Layer
Tr-to-Tr	31
Pd-to-Tr	32
Pd-to-Pd	33
Shorts	34

So, all flags for shorts will appear on layer 34. In the example schematic which you have designed, TESTCKT.DBF, the single possible short should be located at the crossover of the wires on Input pins CD1 and CP1 of the 7474 device, as illustrated in Figure 16.

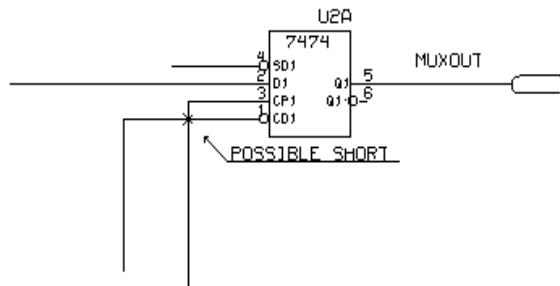


Figure 16

ZOOM:

You may zoom in, by accessing the **Zoom In** command from the Ribbon bar or **[View]Zoom In** menu item or using "**PgUp**" keyboard shortcut, on the intersection of these wires where the DRC flag should be located. There is a small X at the intersection flagging this point. The flag, in this case, merely points out where two distinct signals cross each other without sharing a common vertex.

NOTE: PROCAD is a "vertex recognition" system. This means that PROCAD will not recognize the connectivity of two signals which intersect, unless there is a common vertex or connection dot at the point of intersection.

CLEAR DRC:

To clear the DRC flags:

CLEAR DRC <ENTER>

or **[Check]Clear DRC Errors** menu item. This command deletes all entities from layers 31 thru 34, and layer 49.

Do not use layers 31- 34, and 49 for any reason. The Clear DRC command will delete all entities from these layers.

EXITING PROCAD

When you have completed the design, or are at least temporarily finished working within a design database, you may execute one of the following commands:

[File]Save and [File]Exit to save any changes made in the database onto the hard disk, leave PROCAD and return to Windows.

[File]Close to close the current database and remain in PROCAD without saving any changes to the hard disk. You may also use the **[File]Exit** command and don't save your changes if you want to leave PROCAD and return to Windows.

[Window]Close All same as **[File]Close** above, but will close all windows (databases) currently opened. PROCAD will prompt for save request for any database that had been modified since the last save operation.

[File]Open to open an existing design database without saving any changes to the current design. The current database is not closed in the Multiple Document Interface (MDI) version of PROCAD for Windows.

[File]New to open a new design database without saving any changes to the current design. The current database is not closed in the Multiple Document Interface (MDI) version of PROCAD for Windows.

Tip: Remember to SAVE any changes before leaving the database, if you wish to keep these changes.

You have now learned most of the commands available for designing a Schematic in PROCAD. Take the time to review if you feel the need.

C*hapter*

2

Netlist Extraction

Topics Covered:

- INTRODUCTION
- USING PROCAP
- NETLIST EXTRACTION
- BILL-OF-MATERIALS
- INPUT MULTIPLE DATABASES
- RATS NEST
- PRINTING THE OUTPUT
- COMPARING OUTPUT WITH INPUT
- MULTIPLE SHEET SCHEMATICS

INTRODUCTION

Now that you have completed your first simple circuit, you are ready to begin the process of Netlist Extraction.

PROCAD utilizes an automatic Netlist Extraction utility called PROCAP.

PROCAP accepts a PROCAD design database as input, and generates a variety of ASCII format output files for several applications.

For the purpose of acquainting yourself with PROCAP, as well as understanding the principal of multiple sheet schematics, this section of the Tutorial utilizes two predefined schematic databases.

CKT-3.DBF and CKT-4.DBF are the two schematic designs which will be utilized.

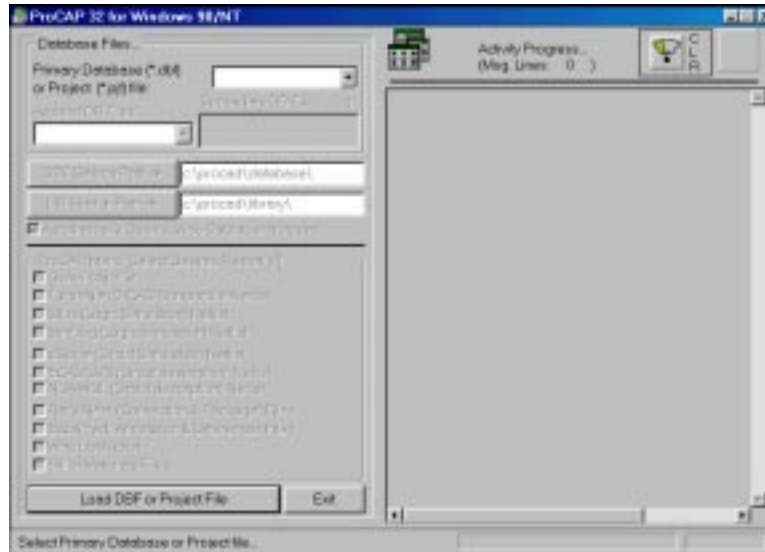
You will examine these files after completing the study of PROCAP.

USING PROCAP

You may invoke the PROCAP utility by clicking on the Netlist icon from the Tool bar in PROCAD, or invoke it from the icon in the ProCAD Advanced for Windows group in the Windows Program Manager. Please refer to your Windows manual if the above concept is not familiar to you.

Note: If you don't have enough memory to invoke PROCAP from within PROCAD (typically you get MSLANGLOAD... error exit), you should invoke it from its icon in the Program Advanced for Windows group of the Program Manager.

When PROCAP is invoked by either method, it displays the signon message with your serial number. After a few moments, the PROCAP main window will appear as follows:



PROCAP Window Control-menu Box:

The Upper left corner of the PROCAP window is a Control-menu box [-]. This Control-menu box when invoked drops down a menu for controlling the various aspects of the PROCAP window, and also submenus for displaying the current PROCAP version (About box) and setting its execution priority. Go ahead, click on the [-] box and experiment. Note: All PROCAD utility programs contains this common Control-menu box, and they all perform essentially the same functions.

Primary Database or Project File:

At this time you are ready to input a database file, which contains the Schematic design, or input the first in a series of up to 30 schematic design databases which are to be linked into one Printed Circuit Board layout. If you have organized your design into a project file (See PROJECT command in PROCAD Reference manual), you will supply the name of the file here.

ProCAD Learning & Tutorial Guide...

select **CKT-3.DBF** as your primary database file,
then click on the “Load DBF...” button.

PROCAP returns the following messages in the activity progress window:

Simulatable Macro (xxx\$macro) Cells:	0
Gate/Device (xxx\$gate/xxx\$device) Cells:	0

This information tells you that PROCAP has loaded the design database, processed it into a format which is utilized by PROCAP, and separated out the various devices into tables.

MENU:

The PROCAP contains the following Netlist formats that may be selected:

[Append Database Files..]	
[DBF Search path]	
[Library Search path]	
[G]eneric	Generate GENERIC Netlist
[F]uturenet	Generate FUTURENET Netlist
[S]ilos	Generate SILOS Netlist
[SI]mulog	Generate SIMULOG Netlist
[SP]ice	Generate SPICE Netlist
[M]dl	Generate NCA MDL Netlist
[L]vs	Generate ECAD'S LVS Netlist
[R]at	Generate Rats Nest Data files
[BA]ck	Generate Back Annotation/Diff
[W]ire	Generate Wire List
[BI]ll	Generate Bill-of-Materials
[Generate...]	Start Execution
[Exit]	Exit to PROCAP utility

Notice that the menu is divided into two halves. The first half of the menu is concerned with various netlist formats; while the second half of the menu is concerned with a variety of output data formats.

You have entered CKT-3.DBF into PROCAP, and may now either generate documentation from this database, or input another database to work with, or append databases to the primary database file..

PROCAP allows you to link multiple schematic databases, up to 30, to generate RATS NEST, BACK ANNOTATION, etc. files.

Note: Some of the above output formats are only supported for a single database. PROCAP will issue a warning message if you attempt to process multiple databases for these formats.

At this time, you should generate documentation which is specific to the CKT-3 schematic.

NETLIST EXTRACTION

GENERIC NETLIST:

Select **Generic Netlist**,
then click on the “Generate Report” button.

PROCAP returns the messages:

Generating GENERIC Netlist
Output File Name [CKT-3.NET]:

The default output file name has the same name as the input file, with the .NET extension. Use the default selection this time, but you may change this in the future.

Press <ENTER>

The Generic Netlist is an ASCII file format which is unique to PROCAP. It includes a list of all parts, data attributes, and pin-node assignments.

The data attributes listed in the Generic Netlist are straight reflections of the data attribute files of each schematic symbol as they are stored in the library.

Data Attribute files are ASCII characters stored with a library cell, which provide information regarding packaging, reference designators, and pin-numbers.

For more information on data attributes, see Appendix-A .

SIMULATION NETLIST:

The other netlist files which PROCAP will output are in the form of many popular simulation packages. These simulation netlists allow you to design a circuit in PROCAD, and pass an output file from PROCAP to any of the simulators listed in the menu.

Using these output formats requires special data tokens as part of the cell's data attribute file. For a complete understanding of the requirements of each Netlist format, see Appendix A.

BILL-OF-MATERIALS:

When PROCAP is finished generating the Generic Netlist, you are prompted for another command.

Select the **[Bill of Materials]** option and click the RUN button to generate a Bill-of-Materials. PROCAP returns the message:

Generating Bill-of-Materials
Output File Name [CKT-3.BOM]

The default file name has the extension .BOM for Bill-of-Materials. Press <ENTER> to accept it.

To generate the Bill-of-Materials, PROCAP searches the schematic database for special BOM tokens which are stored in the cell's data attribute file.

When PROCAP is finished generating the Bill-of-Materials, you are prompted for another command.

INPUT MULTIPLE DATABASES

You are now ready to generate the RATS NEST data files which are used by PROCAD during Printed Circuit Board layout.

However, before beginning the Rats Nest generation, you must add the second sheet of this design, the CKT-4.DBF.

Append the CKT-4.DBF database from the Append DBF Files... combo box.

If there were more files to append, up to 30, you would simply keep adding them to the list at this time.

Note: If you organize your files in a project (See the Project command in the Reference manual), you may specify the project file as the primary file, and ProCAP will automatically load all the files needed for the design project.

RATS NEST

Now that you have appended CKT-4.DBF to CKT-3.DBF in PROCAP, you are ready to generate the Rats Nest data files.

Select the **[Rats Nest]** option from the menu and click on the Run button. Then select ProCAD Standard ".CON" & ".PKG" file formats. PROCAP returns the message:

**Generating Rats Nest data file
Output File Name [CKT-3]**

PROCAP has utilized the file name from the first schematic database input, or in this case CKT-3, as the default.

Press <ENTER> to accept the default.

Generating Connection Data File

The connection data file will be assigned the .CON extension. The output file will be named CKT-3.CON.

The connection file includes a pin-to-pin map of all signals used in the design, and is used by PROCAD to automatically connect devices on a PCB.

This file will also automatically connect all cross sheet connectors, or signals which exist on multiple pages of the schematic such as VCC and GND.

It can take PROCAP several minutes to combine multiple databases and extract the data files.

**Rats Nest CON data file generated
Generating Packaging Data File**

PROCAP has finished generating the connection file CKT-3.CON, and is now generating the packaging file CKT-3.PKG.

The packaging data file includes information regarding the schematic symbols physical packaging for PCB design, and is used by PROCAD to automatically place components onto a PCB.

The .PKG file specifies a library location and a cell name for each physical package equivalent of a schematic symbol. This file also groups gate level devices together into a single physical package. For instance, four 74LS00 devices on a schematic will be packaged into a single 14 pin DIP package.

PROCAP generates the .PKG file by searching the schematic symbols for RAT data tokens stored in the cell's data attribute file.

When the packaging file CKT-3.PKG has been generated, you may exit PROCAP by clicking on the Exit button.

You have completely generated a Generic Netlist, and a Bill-of-Materials from the CKT-3.DBF; and you have generated Rats Nest connection and packaging files from the linked CKT-3 and CKT-4 database files.

PRINTING THE OUTPUT

From the DOS operating system or Windows Notepad, print the files:

Note: If you plan to use Windows Notepad, you may temporarily minimize the PROCAD's window, to allow for efficient selection of this utility.

CKT-3.NET	Generic Netlist
CKT-3.BOM	Bill-of-Materials
CKT-3.CON	Rats Nest Connections
CKT-3.PKG	Rats Nest Packaging

by using the **PRINT** command.

NOTE: If you do not have a printer, or the resources to print these ASCII files onto hardcopy, the next section will be more difficult to work through.

Try to obtain hardcopies of these files.

COMPARING OUTPUT WITH INPUT

At this point, you may compare the hardcopy output which was generated by PROCAP, with the schematic database files in PROCAD.

Use PROCAD to open and examine the CKT-3.DBF schematic database file.

You should be able to see schematic devices U4, and U2.

Now, review the four output files while viewing the database.

Look at the printed copy of the Generic Netlist, CKT-3.NET.

GENERIC NETLIST

The Generic Netlist is divided into three portions: **PARTS, ATTRIBUTES, and PINS**, which describe the CKT-3.DBF schematic.

PARTS is divided into **Instance, Cell_name, Device Value** etc.

INSTANCE is the reference designator of the schematic symbol.

CELL_NAME is the [Library] which the device is stored in, and the name.

VALUE is an eight character user-defined attribute to assign electrical values to discreet components.

Look at device U2A, on the graphic display, and locate it on the Generic Netlist.

U2A [TTL]7474

U2A is the reference designator, [TTL] is the library which the symbol is stored in, and 7474 is the device name in that library.

By looking at the Generic Netlist, you can quickly determine which reference designators have been used in this sheet of the schematic design, and which cells and libraries have been used.

ATTRIBUTES: displays a complete reflection of the data attribute files associated with each of the cells used in this design database.

For instance, locate the 7474 device in the Attribute section of the Generic Netlist. It should be the second devices listed, and should read as follows:

```
CELL:[TTL]7474
  REFERENCES: U2A  U2B
  CELL DATA:
    BOM$CO_PN      7474
    BOM$MF_PN      SN74LS74
    BOM$DESCN      DUAL D FLIP FLOP

    RAT$PKG        [PKG]IC14A

    RAT$VCC1       14
    RAT$GND1       7

    RAT$TCNT       2
    RAT$DEVA       6 5 4 3 2 1
    RAT$DEVB       8 9 10 12 11 13
```

SELECT INSTANCE:

Now, locate the U2A device on the graphic display. You may need to use the **Select Device** command from the **[Edit][Find]** menu for easy location.

If the U2A device is not displayed on the graphic screen, type *SEL INS U2A* <ENTER> on the PROCAD command line.

This command causes PROCAD to search the database to locate the specified device, pan to the location of the device, and select that device for manipulation.

NOTE: If you are using SEL INS just for the purpose of finding and displaying a specific device, be sure to press <ESCAPE> to deselect the device before issuing any other commands.

SHOW ATTRIBUTES:

Now Access the **[Library]Cell Attribute...** command. Position the cursor on the U2A device, and click the left mouse button to display the data attributes of the schematic symbol.

Compare the data attribute file which is displayed on the left half of the graphic screen, with the ATTRIBUTES printed on the Generic Netlist. They should be the same.

PINS described at the end of the Generic Netlist is a simple device pinout. This section lists the devices by their reference designator, shows pin numbers and types, and finally shows what signal is attached to a specific pin. The two most important pieces of information listed in the PIN section is the PINTYPE and SIGNAME.

PINTYPE illustrates whether the pin is defined as an INPUT, OUTPUT, or INOUT (bi-directional). This information is useful when describing the circuit to a simulator.

The technique for defining pins is described in Chapter 4 of this Tutorial, and takes place during cell formation.

SIGNAME illustrates which signal is attached to the specified pin, and is useful when generating a connection list.

BILL-OF-MATERIALS:

Now compare the Bill-of-Materials with the data attribute file of the 7474 device.

You will notice the BOM\$ data tokens have been extracted from the device, and formatted into the Bill-of-Materials file by PROCAP.

Notice the Bill-of-Materials shows the quantity, QTY, of 7474 devices as one. However, you will notice in the Generic Netlist, there are actually two schematic 7474 devices placed, U2A and U2B.

The Bill-of-Materials is actually related to the PCB design. PROCAP groups the schematic symbols into their physical counterparts before counting quantity. Therefore, PROCAP counts the two 7474 schematic devices as a single device in the Bill-of-Materials.

RATS NEST:

Finally, compare the Rats Nest .CON and .PKG files with the schematic design, by looking at the data attribute file of the 7474 device.

The RAT\$ data tokens are used in generating an accurate connection and packaging file for use in PROCAD for PCB design:

RAT\$PKG [PKG]IC14A

This data token tells PROCAP what the physical packaging of the 7474 schematic symbol is. [PKG] represents the library where the cell is stored, and IC14A is the cell name.

Locate the instance U2 on the hardcopy of the CKT-3.PKG, and you will see this token listed.

By using this token, you can effectively pre-package your entire design for quick entry into PCB design.

RAT\$VCC1 14
RAT\$GND1 7

These tokens allow PROCAP to automatically connect this device to a power and ground net.

Search the hardcopy of the CKT-3.CON file for the \VCC1 node, and the pin U2,14 should be listed.

By using this token, you can greatly simplify the necessity to manually connect every schematic symbol to the power and ground bus.

RAT\$TCNT 2
RAT\$DEVA 6 5 4 3 2 1
RAT\$DEVB 8 9 10 12 11 13

These data tokens are utilized by both PROCAD and PROCAP to perform device grouping.

RAT\$TCNT defines the number of schematic devices in a physical package, in this case two. PROCAP uses this token to group devices into a single package.

Notice that the device U2 shows up only once in the CKT-3.PKG file, although there are two devices in the schematic, U2A and U2B.

RAT\$DEVA provides the pin numbers assigned to the first device.

RAT\$DEVB defines the pin numbers assigned to the second device.

By utilizing these RAT\$ data tokens PROCAP generates an accurate description of the schematic design, for quick entry into the PCB design.

The actual use of the Rats Nest .CON and .PKG files will be discussed in Chapter Four of this Tutorial.

MULTIPLE SHEET SCHEMATICS

Because the CKT-3 and CKT-4 database files are sequential sheets of the same schematic design, some special attention must be paid to the creation of these databases.

Look at the hardcopy of the CKT-3.CON file, which is the Rats Nest of connections generated by PROCAP. The first half of this file contains computer assigned nodes labeled as:

```
\P01_N0000QQ  
\P01_N0001QQ  
\P01_N0002QQ
```

while the second half of the file contains computer assigned nodes labeled as:

```
\P02_N0000QQ  
\P02_N0001QQ  
\P02_N0002QQ
```

These computer assigned nodes are from the two different design databases, CKT-3 and CKT-4, and are therefore labeled with different computer assigned sheet prefixes.

SHEET PREFIX:

This sheet prefix, which is assigned by PROCAD to all computer named signals as they are entered, is the only method by which PROCAP can distinguish one node from another.

Without the sheet prefix the computer assigned nodes from CKT-3 and CKT-4 would be indistinguishable. You will recall from the first section of this Tutorial, PROCAD prompts you for a sheet number when you open a schematic database.

NOTE: PROCAD utilizes the user assigned sheet number when assigning sheet prefixes to computer assigned nodes.

SET MODE:

You may change the sheet number assigned to a schematic database by utilizing the **SET MODE** command ([**Other...**]**Switch mode** menu item) or [**Other**]**Constants...** menu item dialog box.

SET MODE allows you to define the type of database you are working in, either schematic or PCB. If you select schematic mode, you may also assign a sheet number.

This command allows you to manually renumber existing schematic databases, if you find it necessary.

When placing symbols into the schematic design, the first TTL symbol placed into the CKT-3 database was assigned the reference designator of U1A. The first resistor placed was assigned a reference designator of R1, and the first capacitor was named C1.

All subsequent devices placed into the design were sequentially assigned reference designators of U2, R2, and C2, respectively.

The assigned reference designators are sequentially ascending.

Type the command *OPEN CKT-4 <ENTER>* or Open the file using the **Open** command from the [**File**] menu. to open the second sheet of the schematic design.

SELECT INSTANCE:

Use the command *SEL INS U6 <ENTER>* to locate the schematic symbol labeled as U6.

Press *<ESCAPE>* to deselect the device.

In this design database, CKT-4, which is the second sheet of the schematic design, the first device placed was automatically assigned a reference designator of U1.

However, because this is the second sheet, U1 had already been placed in the CKT-3 design database.

When creating multiple sheet schematics, it is very important to maintain the sequential numbering of devices as they are placed into the schematic design.

In order to do this, the 74378 device was placed into the CKT-4 database. The **CHANGE INSTANCE** command was used to assign a reference designator of U6 to this device.

The next TTL device which was placed into CKT-4, then automatically received a reference designator of U7.

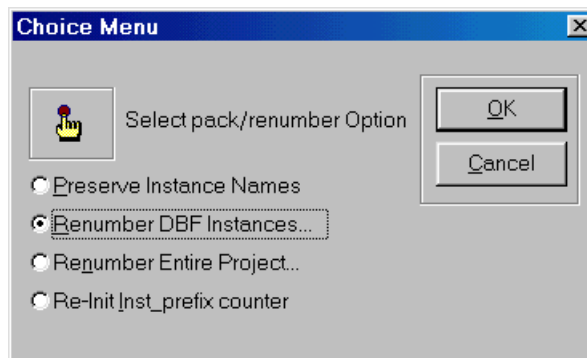
NOTE: When you change a reference designator of a device in the design database, PROCAD begins numbering each subsequent device sequentially from the new value.

This entails manually tracking reference designators from one schematic sheet to the next.

One effective way to track instance names is to generate a Generic Netlist in PROCAP after each sheet of the schematic is complete.

PACK/RENUMBER:

WARNING: If you are manually renumbering the reference designators in a schematic, as was done in CKT-4, you must never use the PACK/R option.



PACK/R empties the garbage from the delete buffer in the database just as the standard PACK command does. However, the Renumber instance names options tells PROCAD to renumber the reference designators of the schematic.

One method of using the PACK/Renumber command to automatically update multiple sheet schematics is by specifying a base value for renumbering.

PACK/R=nn

Where **R=nn** is the base value used for renumbering. For instance, on sheet one of a schematic design use PACK/R=100, and on sheet two use PACK/R=200. This command will cause PROCAD to renumber all devices placed into the schematic in a sequential order from the base of 100, 200, 300...

You may also issue the *PACK/R=CKT-3* <ENTER>. This will cause the renumbering to start from where it ended in schematic database CKT-3.

You have now completed the study of PROCAP, as well as some advanced features of PROCAD.

This page is intentionally left blank.

Printed Circuit Board Design

C*hapter*

3

Printed Circuit Board Design

Topics Covered:

- INTRODUCTION
- OPENING A PCB DATABASE
- CREATING THE BOARD OUTLINE
- PLACING COMPONENTS
- OPTIMIZING THE COMPONENT PLACEMENT
- MANUAL ROUTING TECHNIQUES
- MODIFYING EXISTING ROUTES
- AUTOROUTING WITH PROROUTE
- USING DESIGN RULE CHECKS
- CLEAN-UP AND EXIT

INTRODUCTION

PROCAD allows you to complete the design process begun with schematic design, by providing a full function PCB layout editor.

From the DOS prompt **C:>** type *CD\WPCAD* to change to the subdirectory where PROCAD is stored. Type *WIN* and press the *<ENTER>* key once. This will start up the Windows program. From the Program Manager, double click on the PROCAD icon on the PROCAD Advanced for Windows group. (Please refer to your windows manual if you are not familiar with starting a program from the windows environment).

OPENING A PCB DATABASE

To open a new PCB database from the PROCAD editor, access the **[File]** menu. From this menu you will see a variety of commands having to do with file manipulations for loading, saving, and leaving a design.

Access the **[File]New** command. PROCAD will display the message "Initializing Database File" in the message area. PROCAD also returns the file open dialog box. Enter the Database file name *TESTPCB.DBF* for this exercise. Select PCB type database, then accept all other default settings.



The following defaults should be established:

Database Units (Mils,IN,CM,MM)[MILS]:

Here you should determine the units of measure which will be used during the design process. It is highly recommended to utilize the default selection [MILS] to maintain compatibility with all PROCAD Libraries.

Screen Display Factor (1..99)[10]:

This controls the grid/pixel ratio based upon a complex equation. (See SCREEN DISPLAY FACTOR in the Reference Manual for a complete explanation). Use the default selection [10].

Database Type [P]CB or [S]chematic [S]:

This selection controls the method of interconnect, logical or physical, which PROCAD will utilize during the design process.

A logical connection system is based upon the vertices of wires crossing at common points.

The physical connection system, which is used by PROCAD in a PCB type database, is dependent upon layers as well as common vertexes. If two traces which are on the same layer intersect at any point (whether they actually share a common vertex or not) they are considered to be connected. The reasoning here is that all artwork for actual PCB manufacturing is generated from these graphic layers, and intersecting traces on artwork translate into shorted traces on a fabricated board.

In a PCB design database traces which are not on the same layer are only connected through a VIA or PAD of a physical package. This is based on the understanding that connectivity of traces on a PCB travel is consistent from one layer of the board to another when the trace crosses through a via of some sort.

Select the **[PCB]** option to open this database for PCB design.

Rats Nest Usage Enforced [YES]:

This flag controls the amount of checking ProCAD performs when you manually enter or stitch traces. With this flag set, ProCAD assures that you have a rat's nest connected to a device before you can route a trace to or from that device. This flag may also be enabled or disabled from the **[Other..]Toggles..** dialog box. Select the ON position for this database.

Click the OKay button to complete the command.

If you are opening an existing database file, PROCAD will open it, and will not issue any of the above prompts, since these values were already saved as part of the database.

PCB MACRO:

Before beginning a design, the database should be configured for use in schematic design, or PCB design. Certain parameters like layering definitions, color values, default libraries, and function key definition may all be established through the use of macro commands.

A macro command, or macro, is a series of predefined commands which execute in sequence when accessed through the macro. See the MACRO command in the Reference Manual for a complete explanation of writing macro commands.

Database parameters for PCB designs may be set using the macro command PKG.CMD.

After initializing the new database, ProCAD automatically executes the PKG.CMD macro command file, echoing the commands as they are executed. You may also invoke the **[File]Macro** command, and then select the PKG.CMD file when prompted.

PKG.CMD defines layer information such as color, layer fill, and name, as described in the following section.

PKG.CMD makes the active library the PKG.LIB as is displayed in the ribbon bar across the top of the screen.

PKG.CMD also defines 25 function key positions as described in the **Quick Reference section (Page 160)**. Function keys may be used instead of or in conjunction with menus as a user interface method. PROCAD allows you to define up to 25 different function keys for immediate command access. Refer to the SET FUNCTION KEY command in the Reference Section for a more complete explanation of the function keys.

LAYERING CONVENTIONS:

Before beginning any actual layout, you should understand the database environment which you have established by utilizing the PKG.CMD macro. The importance of layering conventions is most apparent when trying to generate graphic output containing only specific types of information. However, maintaining proper layering conventions is also important when trying to display specific information on the graphic screen, while designing in PROCAD.

NOTE: For most effective use of PROCAD, and the libraries provided with PROCAD, you should strictly adhere to the prescribed layering convention.

The PCB Layering Convention recommended for use with PROCAD, and actually used in all PROCAD standard libraries is as follows:

Layer 1 is the **WORK/PLACEMENT LAYER**, and is used exclusively for the placement of components from the libraries. You should not use Layer 1 for anything but placing components.

Layer 5 is the **PADMASTER**, and contains the copper defined for all pads and vias for either the component or solder side of the PCB. Do not use Layer 5 for anything unless you are defining a new pad or via to be stored in a library. See Chapter 4 of this Tutorial for more information.

Layer 11 is the **CARD_OUTLINE**, and should only be used for defining the physical outline of the circuit board. In general, you will know the dimensions and outline of your board, before component placement.

Layer 20 is the **COMPONENT SIDE**, and is used exclusively for routing component side traces.

Layer 21 and 22 are **VCC and GND** layers, and are provided for internal power and ground planes.

Layer 23 is the **SOLDER SIDE**, and should be used exclusively for routing the solder side traces.

Layers 24-29 are **INTERNAL** layers, and may be optionally used for routing multilayer boards if your design requires them.

Layer 30 is the **RATS NEST** layer, and will be used for placing Rats Nest connection lines.

All other layers should be used as specified below:

PCB LAYER ASSIGNMENTS

<u>LYR</u>	<u>TYPE/FILL</u>	<u>USE</u>	<u>COLR</u>	<u>CODE</u>
1	SLD/OFF	WORK Lyr #1 DBASE	YLW	[5]
2	SLD/OFF	WORK Lyr #2 CELLS	RED	[2]
3	SLD/OFF	ASSEMBLY DWG TOP	GRN	[3]
4	SLD/OFF	ASMB_DWG BOTTOM	GRN	[3]
5	SLD/ON	PADMASTER	CYN	[6]
6	SLD/ON	SLDR_MSK(TOP)	GRN	[3]
7	SLD/OFF	DRILL_TARGETS	CYN	[6]
8	SLD/OFF	DRILL_CODES	RED	[2]
9	SLD/ON	SLKSCRN_LGND(TOP)	WHT	[1]
10	SLD/OFF	SLKSCRN_TEXT(TOP)	WHT	[1]
11	SLD/OFF	CARD_OUTLINE	GRN	[3]
12	SLD/ON	CARD_DIMENSIONS	GRN	[3]
13	SLD/ON	TARGETS_ & MARKS	BLU	[4]
14	SLD/OFF	NOTES_FABRICATION	CYN	[6]
15	SLD/OFF	NOTES_ASSEMBLY	GRN	[3]
16	SLD/ON	CONNTOR_FINGERS	YLW	[5]
17	DOT/OFF	SPCL_1 (User_defined)	RED	[50]
18	SLD/ON	SPCL_2 (User defined)	RED	[2]
19	SLD/OFF	LOGO/DWG BDRS	RED	[2]
20	SLD/ON	TOP SIDE TRACES[L1]		
		SMD TOP PADS	YLW	[5]
21	SLD/ON	LYR 2 VCC {L2}	[RVSE]	[5]
22	SLD/ON	LYR 3 GND {L3}	[RVSE]	[5]
23	SLD/ON	SLDR_SIDE TRACES{L4,n}		
		SMD BOTTOM PADS	YLW	[5]
24-29	SLD/ON	LAYERS #6-9 {L6-L9}	YLW	[5]
30-34		RESERVED FOR DRC		
35	SLD/ON	SMD SLKS_TXT (BOT)	WHT	[1]
36	SLD/ON	SMD SLDR_MSK (BOT)		
37	SLD/ON	SMD PAST MSK (TOP)	WHT	[1]
38	SLD/ON	SMD PAST MSK (BOT)	WHT	[1]
39	SLD/ON	SMD SLKS LGND(BOT)	WHT	[1]
40-48	SLD/OFF	NESTED INFORMATION		
49	SLD/OFF	RESOLVE ERRORS		
50-98	SLD/OFF	USER DEFINED		
99	SLD/OFF	HIGHLIGHTING	WHT	[129]

The PKG.CMD macro sets up the following default entity layer placement configuration:

<u>Entity type</u>	<u>Placement Layer</u>
Cells (Components)	1
All other (traces,circles, rectangles etc)	Current active layer

LAYER COMBINATIONS FOR PCB ARTWORK GENERATION

COMPONENT SIDE	- (Layers 5,13,16,20),
ASSEMBLY DWG	- (Layers 3,4,10,13,15,19)
CIRCUIT/SLDR SIDE	- (Layers 5,13,16,23)
FABRICATION DWG	- (Layers 7,8,11,12,13,14,19)
VOLTAGE PLANE	- (Layers 13,21)
GROUND PLANE	- (Layers 13,22)
SOLDERMASK	- (Layers 6,13)
DRILL MASTER	- (Layers 7,13)
SILKSCREEN	- (Layers 9,10,13)
INNERLAYER n?	- (Layers 5,13,n(24..29))

SHOW LAYERS:

To view the layers assignments of the database simply execute the command **SHOW LAYER** **<ENTER>** ([**Layer**]Layer Attribute... menu item)

This command opens a window in the database with layer names, numbers, and color samples.

The color code may be used to redefine the existing layer parameters by using the **SET COLOR** **<ENTER>** command ([**Layer**]Set Layer Color... menu item). You may also scroll through the layer selection combo box in the ribbon bar to see the layer names and their corresponding fill and color attributes.

FUNCTION KEY DEFINITIONS:

The PKG.CMD macro predefines all 25 available function keys as illustrated in the Quick Reference section.

The function keys are divided into three groups of nine: F1-F9, Shift F1-F9, and Cntrl F1-F9 (except Cntrl F4 & F6). All of these 25 function keys are user definable, and are provided to allow PROCAD more flexibility for the user.

F1-F9 are utilized primarily for screen manipulation and viewing various artwork layer combinations.

Shift F1-F9 are data manipulation commands utilized for moving, copying, or otherwise manipulating data in the database.

Cntrl F1-F9 are PCB attribute commands used for assigning, and changing, reference designators and node names. There are also commands for changing working layers, and setting up routing grids.

It is recommended that you spend a couple of minutes familiarizing yourself with the general structure of the function keys as described above. Here it is again:

F1-F9 Screen Controls, and artwork views
Shift F1-F9 General Manipulation
Cntrl F1-F10 PCB Manipulation

As you familiarize yourself with PROCAD, and the Function Key definitions, you may find that different commands will suit your purposes more effectively. Feel free to redefine the function keys as you see fit. To redefine a function key execute the SET FUNCTION command (See the Reference Manual for details).

CREATING THE BOARD OUTLINE

BOARD LIMITS:

Now that you have familiarized yourself with the database parameters which PROCAD establishes for a PCB design database, you are ready to begin your first design.

The first step in creating a PCB is to define the actual Board outline, and limitations of the circuit board.

Board Limits represent the actual physical limitation of the board. This should be outlined on Layer 11. Using the layer combo box, Select layer 11.

Input the following commands to enter the board outline illustrated in Figure 17.

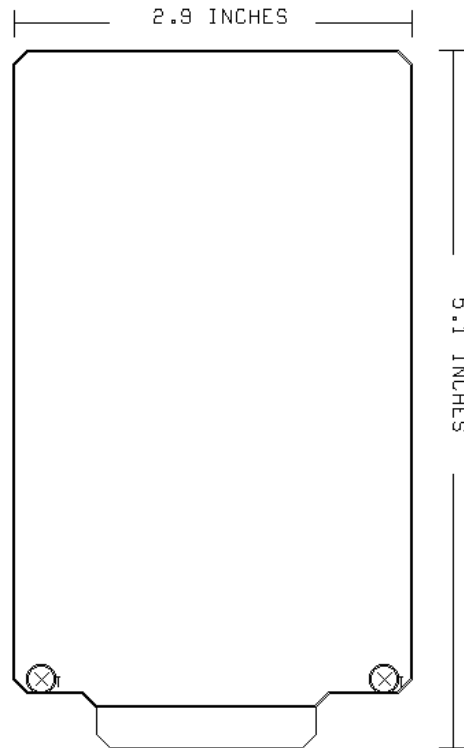


Figure 17

RELATIVE COORDINATE INPUT:

To toggle the coordinate display at the bottom of the screen between relative and absolute modes, you should press *Cntrl-X*. When in relative mode, the coordinate is prefixed by the letter "R". If you are entering coordinate from the keyboard (when rubberband mode is off), you may prefix the coordinate with the letter "R" to instruct PROCAD that these are values from the current cursor position.

For instance, *R 1500 0* defines a relative grid movement from the first point entered (which should be the current cursor position) of 1500 database units (Mils) in the X-direction, and 0 database units in the Y-direction.

Toggle the coordinate to relative display mode by press *Cntrl-X*.

Establish a **trace width** of say 5 mils from the **[Other...]Constants** dialog box, then access the **Lines** command from the Tool bar.

Position your cursor at the top left corner of the screen and press the left mouse button to start the line drawing process.

Note: You will be inputting a line instead of a wire, because the board outline is non-electrical.

Move the mouse horizontally to the right until the coordinate display at the bottom shows "**R 1500 0**", and click the left mouse button to accept it. Now move down until "**R 100 -100**" is displayed and click the left button to accept it. Continue this process for the following relative coordinates to complete the board outline:

```
R 0 -4500 <ENTER>
R -100 -100 <ENTER>
R -500 0 <ENTER>
R -100 -100 <ENTER>
R 0 -200 <ENTER>
R -100 -100 <ENTER>
R -1400 0 <ENTER>
R -100 100 <ENTER>
R 0 200 <ENTER>
R -100 100 <ENTER>
R -400 0 <ENTER>
R -100 100 <ENTER>
R 0 4500 <ENTER>
R 100 100 <ENTER>
R 1200 0 <ENTER>
```

Click the right mouse button twice to close the line being input and leave the input mode. Now press *Cntrl-X* to return the display to absolute display mode.

NOTE: PROCAD allows you three methods of graphic entry: Cursor position, coordinate entry, relative coordinate entry.

You have just utilized relative coordinate entry to create a board outline. This is how you would most likely create your own board outlines, working from a mechanical drawing and dimensions.

KEEPOUT AREAS:

A keep out is an area of the board that is not used for routing purposes. The keepout is utilized by PROROUTE (PCB Autorouter) to understand where traces may and may not be routed. A keepout may be a physical hole in the actual circuit board. See Figure 18.

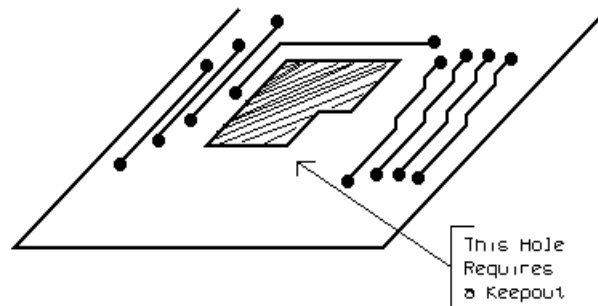


Figure 18

A keepout may be an area used for an imbedded device.

In order to define a keepout area of a PCB, you must outline the keepout by entering a line onto the actual routing layer on which the keepout exists.

For instance, if there is a physical hole in the board, you must place the keepout around this hole on all routing layers.

If there is an imbedded device on the component side of the board, you must only draw the keepout on the component layer of the database.

One important keepout is also the board outline. For instance, the board you just created is limited in routing area by the shape of the board.

In order for PROROUTE to understand where it may or may not route, you must duplicate the outline of the PCB on all routing layers.

To duplicate this board outline, access the **Select Entity** command from the Ribbon bar or **[Edit]** menu. Position your cursor on any of the vertices of the newly entered outline and click the left mouse button to select it.

Notice the outline is highlighted in white signifying its selection, and the automatic change of the Main menu to the selection Edit menu.

Now access the **Copy** command from the now active **Edit** menu. Press <ENTER> to accept the current mouse position as the copy reference point.

Without moving your mouse, press <ENTER> again to copy the outline onto its self.

Now press <ESCAPE> to stop the copy process. Access the **[Change]Entity(s) Layer** command and type 20 <ENTER> to move this newly copied outline to layer 20.

Now repeat the **Copy** process as outlined above, this time moving the newly copied outline to layer 23.

Click the right mouse button or Press <ESCAPE> to leave the select environment.

You have now bound the interior of the board, on Layers 20 and 23, as the allowable routing area for PROROUTE.

Note: Although you have 3 outlines drawn at the same physical coordinates, but on different layers (11,20&23). When you refresh the screen, the outline will be drawn with the color attribute of the last layer which it was copied to, in this case layer 23.

Note: If you plan to use CCT shaped based Autorouter or Massteck Router, you should not duplicate the board outline on routing layers 20 & 23. These routers will use the outline polygon placed on layer 11 as the allowable keep-in area.

TOOLHOLES:

Place two toolholes onto the board outline in the positions illustrated in Figure 19. The tool hole cell should be placed on layer 1. Recall that the PKG.CMD macro sets this up for you. Layer 1 is the only layer which you should place components on.

Access the **Component** command from the Tool bar or **[Place]Get Cell...** menu item. Select the cell *TOOLHOLE* from the list of cells (you may have to scroll through the list to find it). Position the cursor where you would like the toolhole and click the left mouse button to place it.

Tips: It might be quicker to type the command `GET [PKG]TOOLHOLE <ENTER>`, to avoid having to scroll through a list of cells. You should always do this if you know the name of the desired cell and the library the cell is in.

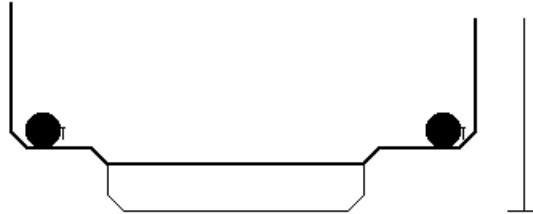


Figure 19

You have finished the board outline, and placed the toolholes. At this time you may complete the board design by adding dimensions to Layer 12, or placing a silkscreen pattern on Layer 10.

Once you have completed the board outline to your satisfaction you might want to save it as a library cell. By saving the completed outline, you may retrieve it from the library any time you are using it on future projects.

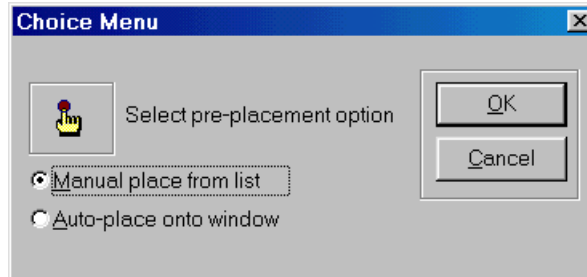
Do not save this outline at this time.

PLACING COMPONENTS

To begin placing components into your database you must first return to Layer 1. Select Layer 1 from the layer selection bar. Now you will use the Rats Nest packaging file CKT-3.PKG, which you generated from the two sheet schematic in Chapter Three of this Tutorial.

PACKAGING FILE:

Access the **[Place]Rats Nest** command. Select Load package (.PKG) file from the dialog box, press OK. Then enter or select the file name *CKT-3.PKG* when prompted for the package file name by PROCAD.



Select the **"Manual place from list"** option, then press Okay.

Do you want package file sorted ?

You may sort the data in the "package" file by reference designators if you wish, however, for this exercise, do not sort it (select NO).

Processing Rats Nest Package Placement

PROCAD then opens a window with a list of devices to be placed. Select a device, then click on the Okay button, for example, select U1 device. PROCAD will then issue the following prompt:

Enter Placement for Instance U1 [PKG]IC14A

Here PROCAD provides you with information concerning the device about to be placed, and requires a cursor position for placement.

The device reference designator is displayed, as well as the cell name and library.

The displayed cell name, and library, allows you to know what the component which is being placed may appear as. This information will help you in placing components.

Note: If you choose the "Auto place onto window" option, ProCAD allows you to define a window, preferably outside the board outline area where it will pre-place all required cells by component outline size. You may then move components from this window to the desired locations in the board as desired.

Place U1 in the upper-left corner of the board which you have just completed outlining, as illustrated in Figure 20.

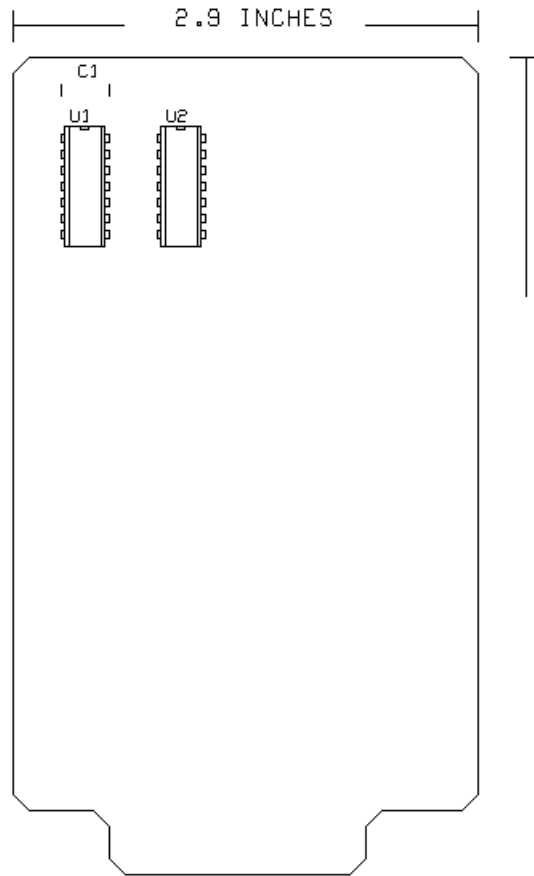


Figure 20

Now select U2 device. PROCAD will then return the message:

Enter Placement for Instance U2 [PKG]IC14A

Place U2 directly to the right of U1, about 300 Mils away.

Now select R1 device. PROCAD will then return the message:

Enter Placement for Instance R1 [PKG]RC07

Place R1 off to the side of the circuit board.

Now select CR1 device. PROCAD will then return the following message:

Enter Placement for Instance CR1 [PKG]DIODE

Place CR1 off to either side of the circuit board as illustrated in Figure 21.

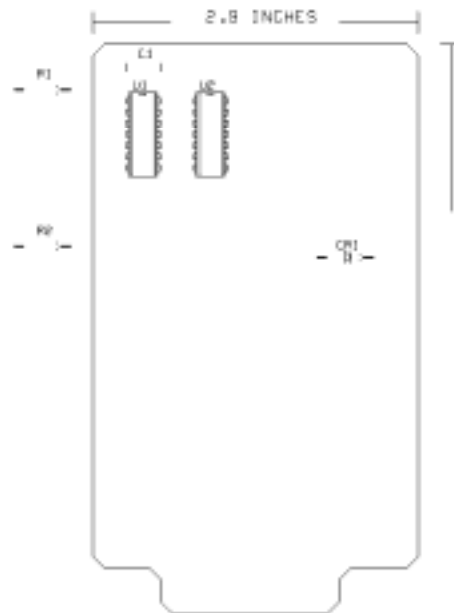


Figure 21

By placing the discrete components to the side of the PCB, and all DIPs, and larger packages, on the board, you are leaving yourself room for flexibility.

PROCAD allows you to press <ESCAPE> to skip the current component, "**R**" to Rotate, "**PgUp**" to Zoom in, "**PgDn**" to Zoom out or "**Home**" to PAN while placing, or left mouse button click (<ENTER>) to place the component.

Feel free to place some components outside the outline of the board during initial placement.

After all components have been placed, you will most likely need to adjust the position of these devices, to achieve the optimum placement of components.

Continue placing components, until all components have been placed. You may temporarily stop placing cells by clicking on the cancel button. PROCAD will then return you to the command mode. You may then re-arrange already placed components. Re-Issue the rat's nest command to complete the placement of the remaining components. PROCAD will only present the list of components that have not yet been placed.

When placing the connector, P1, take special care to place it as illustrated in Figure 22.

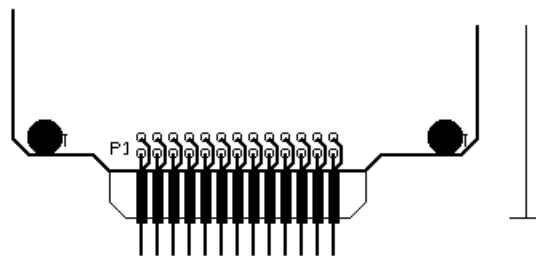


Figure 22

CONNECTION FILE:

Now that the components have been placed on or around the PCB, you may enter the **Rats Nest** connection file CKT-3.CON.

Access the **[Place]Rats Nest** command, Select Load connection (.CON) file option and then select the *CKT-3.CON* file when prompted by PROCAD.

PROCAD enters all connections from the CKT-3.CON file which was generated by PROCAP onto the default rat's nest layer (normally layer 30)..

When the Rats Nest is fully input, PROCAD displays the following prompt:

Do you want to optimize the traces ?

*Note: ProCAD will automatically optimize rats nest traces without issuing the above prompt if the Rats Enforced flag was set prior to invoking the **[Place]Rats Nest** command .*

You should accept the default (YES). This allows PROCAD to optimize the traces for shortest manhattan trace connections and ensures that all rats nests are single segments (point-to-point).

Note: You must optimize your traces if you plan to have ProCAD automatically remove rats nests after they have been manually routed.

PROCAD will then return to the COMMAND mode.

Your PCB design is completely interconnected with the logical interconnects specified from the schematic design.

OPTIMIZING THE COMPONENT PLACEMENT:

Before beginning the routing process, either manual or autorouting, you should optimize the component placement.

To do this, simply select a device and move it to the best location based on the connections to other devices.

DRAG AND RUBBERBAND:

Two features of PROCAD allow you to perform this function efficiently.

RUBBERBAND allows you to move a component in real time graphics. This means that you can select a device, and move it to new locations, watching the movement for the best position.

DRAG allows you to move devices while maintaining the connections made to that device. This command is very important when rearranging the component placement of a fully Rats Nested PCB.

Access the **[Other..]Toggles** command. Now make sure the RUBBERBAND & TRACE DRAG flags are ON. Use your mouse to toggle them ON if necessary.

Click on a device to select it, then click the left mouse button the second time to enter the Move mode. Move the component around (you may rotate, mirror, zoom, pan etc while you are moving the device. Pay attention to the prompts at the status line). and find its best placement.

*Note: If you are having a hard time selecting a device (selecting a rats nest instead), you may want to use the **SelCell** command from the tool ribbon or use the select filter command **[Other...]/Select Filter...** to enable cell only selection .*

The task of finding the best placement, based on the logical connections of all components, may take some time. However, the time invested now, in finding the best placement, will pay off in the routing stage.

To see a possible optimum placement of this same board, enter the command:

OPEN PCB-RAT <ENTER>

Do not save the TESTPCB.DBF database. It is alright to abandon the current changes if you want.

Look at the placement of this board, and how it relates to the Rats Nest.

You are now ready to begin routing.

MANUAL ROUTING TECHNIQUES

To understand manual routing within PROCAD, open the database file PCB-MAN.DBF.

PCB-MAN is a prepared database which represents a portion of the CKT-3 and CKT-4 circuit design. This database is used for simplifying the explanation of manual routing.

OPEN PCB-MAN <ENTER>

CONNECTIVITY:

Before beginning any actual routing there are a couple of concepts which you should understand.

Connections are the logical connections made between components by the Rats Nest .CON file.

Traces or **Routes** are the physical connections made between the components, which translate to copper on the PCB.

Connectivity for a Printed Circuit Board design, is not too much different than for a Schematic design. You will recall, PROCAD is a **VERTEX RECOGNITION SYSTEM**, which means that all connections must be made by a common vertex.

It is important to maintain logical connections, not just graphical ones, in order to use PROCAP's Back Annotation/ECO feature. Back Annotation/ECO can help you locate any differences between the completed PCB and the original schematics.

In PCB design databases, connectivity is also based on layers. This is because PCB connectivity is a graphical connection, which is dependent on layers, unlike schematics.

In a PCB database a signal is considered connected by PROCAD only if it shares a common vertex on a common layer with another signal.

A means of connecting signals through different layers is provided by use of the DRC\$VIA token which is found in all pads and vias. The DRC\$VIA token specifies layers which the via electrically connects to.

So, a signal which begins on Layer 20, the component side of the board, and passes through a via cell, will be connected to a trace on Layer 23, the solder side, through that via.

The **SHOW NODE** command will only display the node, or portion of node, which is on the current layer. The DRC\$VIA token is actually used by PROCAP for the purpose of Back Annotation.

PREPARING FOR MANUAL ROUTING:

Before routing, there are some system parameters which should be configured for manual routing.

To establish a manual routing environment, access the **[Other..]Toggles...** command.

Establish the following parameters:

Trace Drag:	ON
Rubber Banding	ON
Trace Display	ON
Snap-on-Grid	OFF
Rats Enforced	ON
Ghost Grid Check	ON

Now access the **[Other..]Constants...** command, and establish the following parameters:

Ghost Grid:	50,50
Display Grid:	100,100
Trace Width:	12
Nesting Level:	2

The following parameters have the following meaning to PROCAD:

Ghost Grid: 50,50; This establishes a ghost_grid, or a smaller routing grid, of 50 Mils in the X and Y axes.

Snap-on-Grid: OFF; This toggles the SNAP command OFF so that you may move the cursor on the finer established routing grid. At this point, you should be able to move the cursor between pins of the IC packages.

Trace Width: 12; This predefines the width of all newly entered traces at 12 Mils.

Nesting Level: 2; This tells PROCAD to display up to the second level of cell hierarchy.

NOTE: This is a very important concept.

Notice that at this time you cannot see the padstack information for the components. After setting nesting level to 2, you will be able to see the padstacks. The padstack is an integral part of all PCB packaging symbols which are stored in the PROCAD libraries. To display them Set Nesting level to 2; to hide them (for faster screen refresh) Set Nesting level to 1.

For a better understanding of cell hierarchy in both schematic and PCB design, see chapter 4 of this Tutorial.

Access the **Refresh** command from the Ribbon bar or **[View]Referesh** menu item. This refreshes the screen to allow you to see the padstacks which are stored hierarchically within the component packages.

Access the **[Layer]Layer Attribute...** command. Toggle all layers OFF (by clicking on the OFF entry button in the Visibility Attr. box). Now turn on the following layers: 1 5 11 20 23 30, by clicking on the appropriate layer entry under the Visible column.

Tips: You may type the following command for quicker operation: **SET VIS 1 5 11 20 23 30 <ENTER>**

The above operation sets visible layers so that you can see the cell references, the Layer 5 padmasters, the board outline, the component and solder side traces, and the Rats Nest of connections.

Now make layer 30 active (by selecting it from the layer selection combo box in the ribbon bar) so that you can manipulate the Rats Nest wires if you need to. You may also choose a color for layer 30 that will be used for rats nest by clicking on the layer color box on the far left of the ribbon bar, and selecting say dark purple. **Refresh** your screen and notice the change in rats nest color.

OPTIMIZE TRACES BEFORE ROUTING

In order for PROCAD to automatically delete rats nest traces as they are routed, you must make sure all your rats nests have been optimized for point-to-point. You may accomplish this task by executing the **[Place]Rats Nest** command, and selecting the "**Optimize Rats nest option**".

BOARD STITCHING:

Enter the command: *SEL K W <RETURN>* or access **[Other..]Select Filter** menu item, and enable only **Wires** to specify the selection filter for wires only. This enables you to quickly select wires only.

Locate U1, pin 2 (U1,2). The signal on this pin goes to U2,13. This is the first signal which you should route.

You must have Layer 30 defined as the active layer, in order to manipulate the RATS NEST of connections. Make sure that the active layer is 30 (you may select it from the layer selection combo box if necessary).

Position the cursor on U1,2 and click the left mouse button to select the rats nest. PROCAD highlights the node on U1,2. You may issue the SHOW SUMMARY Command (**Info...** menu item) to see some vital information about the net such as the name, length, etc. This net should be named P01_N0012QQ.

Route the node U1,2 to U2,13 by entering the following command, and following the pattern illustrated in Figure 23:

STI B <ENTER>

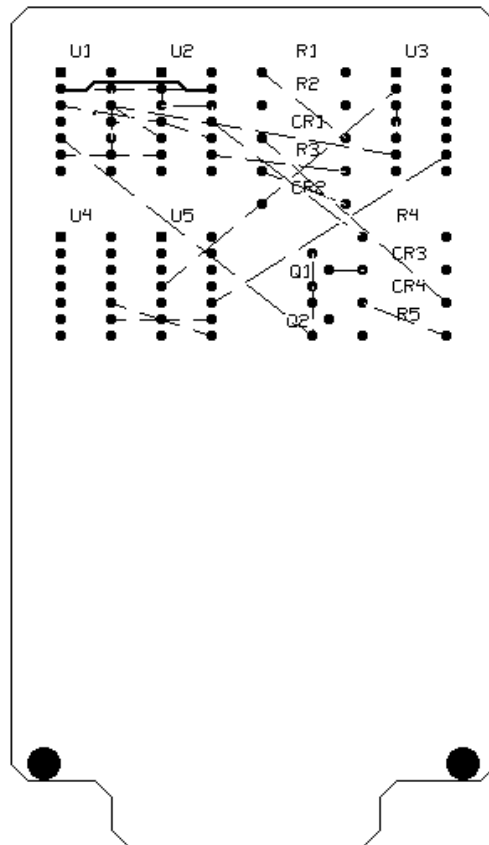


Figure 23

Click the right mouse button twice when you finish routing the trace to U2,13, to close the trace and exit the STITCH command.

If you routed the trace correctly, ProCAD will automatically delete the corresponding rats nest signal. If you did not route the trace to the proper pin, ProCAD will issue a warning message, and gives you the opportunity to discard the newly routed trace or accept it as a partial or an erroneous route.

You have now completed the first trace.

So, the route order would be: first select the desired rats nest signal, and stitch the trace into place as needed using the STITCH command.

The **STI B** command is used to stitch a trace beginning from the bottom side of the PCB. This command is the strength of PROCAD's interactive routing mode.

STITCH allows you to specify which layer to use as the top-side of the PCB (default: Layer 20), which layer to use as the bottom side (default: Layer 23), and which cell to use as the via or feedthrough (default: [PKG]VIA). Stitch also allows you to assign signal names to the traces as you enter them into the database.

Now select the rats nest on U4 pin 6 (by positioning your cursor on U4 pin 6 and clicking the left mouse button). Enter the command:

STI T signal_name <ENTER>

Position the cursor on U4,6 to begin routing a new trace up to U1,2. Route this new trace as illustrated in Figure 24.

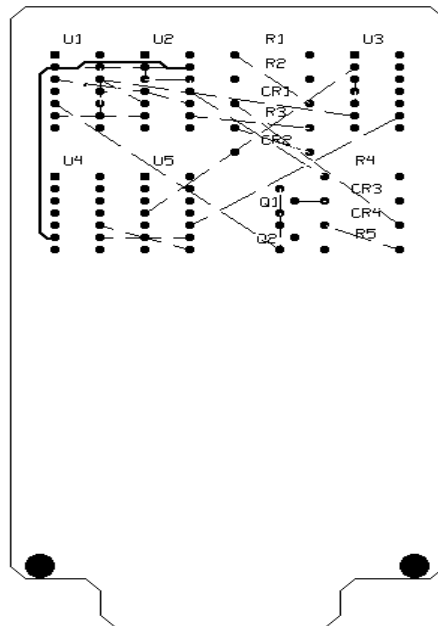


Figure 24

The **STI T** command begins routing on the top side of the PCB, or Layer 20. The signal_name specified in the **STITCH** command will be assigned to the trace as it is entered. If no signal_name is specified, PROCAD will automatically use the name assigned to the rats nest.

Note: If you assign a signal name which is different from the rats nest name, ProCAD will issue a possible "short" warning. You may ignore the warning and proceed if you wish.

Hint: The stitch command may be accessed from the [Route] menu. When using menus, you may not be able to specify the signal name as part of the stitch command. It is better to issue this command from the command line in order to gain complete control of its optional parameters.

MANUAL ROUTING OF COMPLEX BOARDS:

On complex boards, you may have a lot of rats nests clouding your design. This could cause difficulties in manual routing. You may lessen the impact of cloudiness by selecting a dim color (say dark gray) for your rats nest layer. When you select the rats nest to be routed, its color becomes bright white (hi-lighted), while other rats nests remain in a dim dark gray color. Also optimized rats nest lines are always point-to-point which assures you that they only make 2 connections (the end points only), even though they may cross other pins without making connections to them. You should only concentrate on the end points.

VERTICAL AND HORIZONTAL ROUTING:

Notice that the horizontal route which you have completed is placed on Layer 23, while the vertical route is on Layer 20. This is a standard convention which is also utilized by PROROUTE during autorouting.

Vertical traces on Layer	20
Horizontal traces on Layer	23

This standard is utilized to make routing easier between the interactive routing and autorouting.

Position the cursor on U1,3 and press the left mouse button to select and highlight the rats nest. This net goes from U1,3 to U2,4.

Begin routing on the horizontal layer by typing **STI B <ENTER>**. Route this signal from U1,3 through U2,4. Also select and route the rats nest on U2,10 as illustrated in Figure 25.

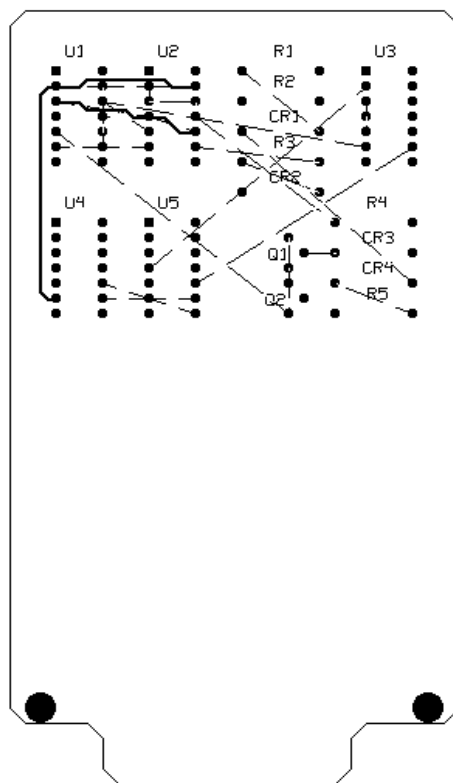


Figure 25

Now select the rats nest on U4,12. Route from U4,12 to a vertex of the just routed trace as illustrated in Figure 26, using **STI T**.

When you reach the vertex of the existing route, **DO NOT** press <ESCAPE>, instead press **V** when prompted for the next point. Pressing **V** any time while in the **STITCH** mode causes PROCAD to complete the trace up to that point, place a via at the cursor position, and move to the other side of the board to continue routing the trace. After pressing **V** to place a via, press <ESCAPE> to exit the STITCH mode.

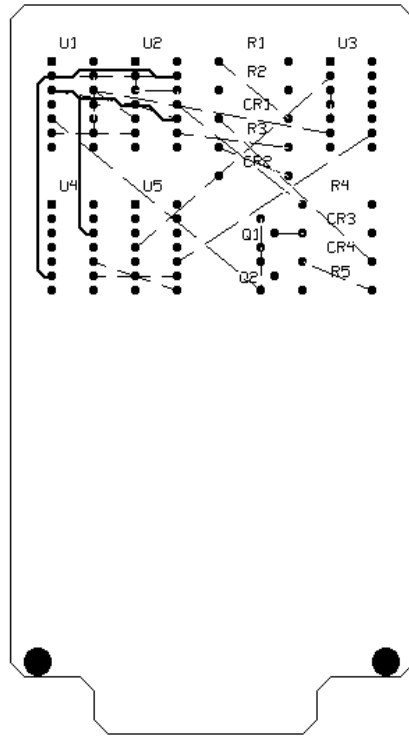


Figure 26

Notice that ProCAD did not delete the rats nest. This is because you routed to an existing route instead of routing to U2,10 where the selected rats nest was connected to. In real life, this is not an error condition, you have only chosen to make a better (and possibly shorter) route, while maintaining correct connectivity.

When you make these types of routes, you should manually delete the selected rats nest by issuing the select command from the ribbon bar (ProCAD returns to the held-in select item). Next delete the rats nest. ProCAD will issue a warning that you are trying to delete a rats nest. You should ignore the warning, and proceed to delete it.

Note: The ECO (back annotation) command in PROCAP will recognize these types of routes as valid, and will not flag them as inconsistency between Schematic and PCB databases.

Position the cursor on U1,5 and click the left mouse button to select and highlight the signal on this pin. This signal goes from U1,5 to Q1,3

Start routing this signal with **STI T**, to begin routing on the top layer of the board. Route this signal as illustrated in Figure 27. Press **V** to place a via and begin routing on the other side of the board if necessary.

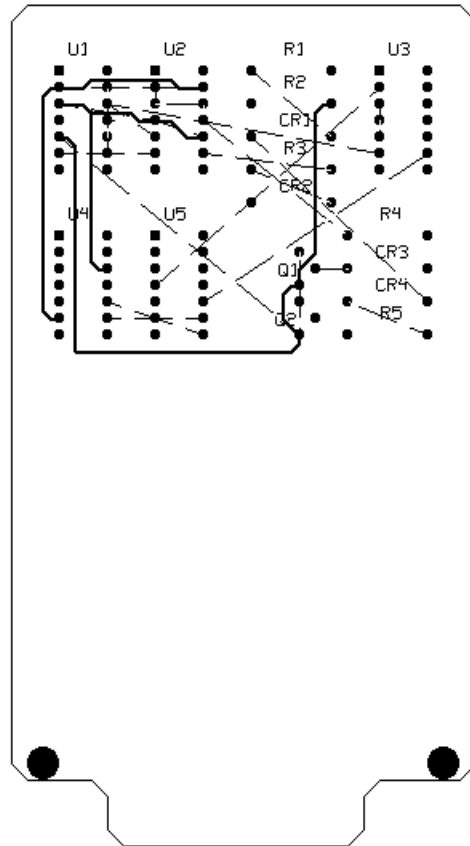


Figure 27

Continue routing this database to get a solid understanding of interactive routing within PROCAD.

DEFINING STITCH PARAMETERS:

When you are working with the STITCH command, you may need to change the default settings of top and bottom layers, or of the via cell.

STI/P allows you to view the STITCH parameters, and change them if you desire.

Typing **STI/P** without any arguments is exclusively for viewing the existing parameters.

To change the default values type (or use the **[Other]Constants...** dialog box):

STI/P nT nB [LIB]cell

nT represents the layer number of the new top layer. In this case, top does not mean component side, it just specifies one of two routing layers to work between.

nB represents the layer number of the second routing layer to work between.

[LIB]cell represent the library and cell name which the via cell is stored in. When you change this, be sure to specify a library. The new STITCH via cell specified will be placed whenever you press **V** in the STITCH mode.

MODIFYING EXISTING ROUTES

Some of the tracks which you have routed may need to be modified as your PCB design progresses.

PROCAD provides many commands for modifying routed traces. However, whenever modifying an existing trace be sure to consider the connectivity of the whole signal.

NOTE: When changing a routed signal, pay close attention to the connectivity.

Commands which are useful for modifying existing routes are:

REROUTE to add a new vertex to an existing line,
and move that vertex in real-time.

MOVE VERTEX to grab a vertex of a route and move it to a
new location.

ADD VERTEX	to add a vertex to an existing route for modification.
DELETE VERTEX	to remove a vertex from an existing route.
CHANGE LAYER	to move a route or route segment from one layer of the board to another.
CHANGE WIDTH	to change the width of an existing trace or trace segment.
DELETE SEGMENT	to remove a segment of a trace from the board. Use this command to delete a segment, and immediately add in a new segment.

These commands allow PROCAD to be a powerful interactive routing tool for PCB design. You may access these commands and some other editing commands from the **[Edit]** menu.

Hint: You should select a trace before performing any of the editing operations above. This allows you to remain in the select environment while you apply different editing commands as need to the selected trace to complete its routing path.

AUTOROUTING WITH PROROUTE

PROROUTE is a fully re-entrant autorouter utilizing state-of-the-art algorithms. The system is an unlimited iteration router which allows you to stop the route at any time, make engineering changes to the design, and continue routing. You specify the route passes and strategies, and establish the design rules which PROROUTE will utilize. The re-entrant capability allows you to retain control over the way PROROUTE will complete your design.

ROUTING PASSES:

PROROUTE provides you with the capability of performing a variety of routing passes to insure the highest completion rate of any personal computer based autorouter available. These routing passes include:

POWER/GROUND BUS - routes only nodes which are labeled with a signal name of VCC (VCC1, VCC2, VCC3, VCC4), or GND (GND1, GND2). This allows you to make a Power/Ground Bus route iteration, view the results in PROCAD to make any modifications you might require, and then return to the router.

MEMORY - to automatically route bussed arrays of traces for fast completion of Memory circuitry. The memory router is a two pass (internal passes A and B) 100% route algorithm which places no vias, and utilizes only one side of the PCB for completion of the routing. While the memory router contains two internal passes (one horizontal, the other vertical) it only requires one of the six available passes per route iteration.

LOOKOUT STRATEGY - performs a series of three strategic Lookout routes on each specified trace as required. The Lookout Strategy router insures the least amount of feedthroughs on the PCB, while completing the routing of traces on the board.

PRE-ROUTES:

So far, you have designed the board outline, placed the components, and connected the devices in a RATS NEST fashion. At this point the board is ready to be passed to PROROUTE by simply packing and saving the database, and exiting the PROCAD editor.

Before routing, you may want to place some pre-routes in the design which will insure routing of critical traces in a manner you consider most effective. Generally this step is not required, however for certain technologies you will want to define the route, for instance in the case of ECL technology.

Place prerouted traces into the board by utilizing the manual routing techniques previously discussed. PROROUTE will recognize any existing traces in a PCB database, and route around them utilizing the defined design rules.

USING PROROUTE

To invoke PROROUTE, click on the Router button on the Tool bar, or From the Program Manager, double click on the ProRoute icon.

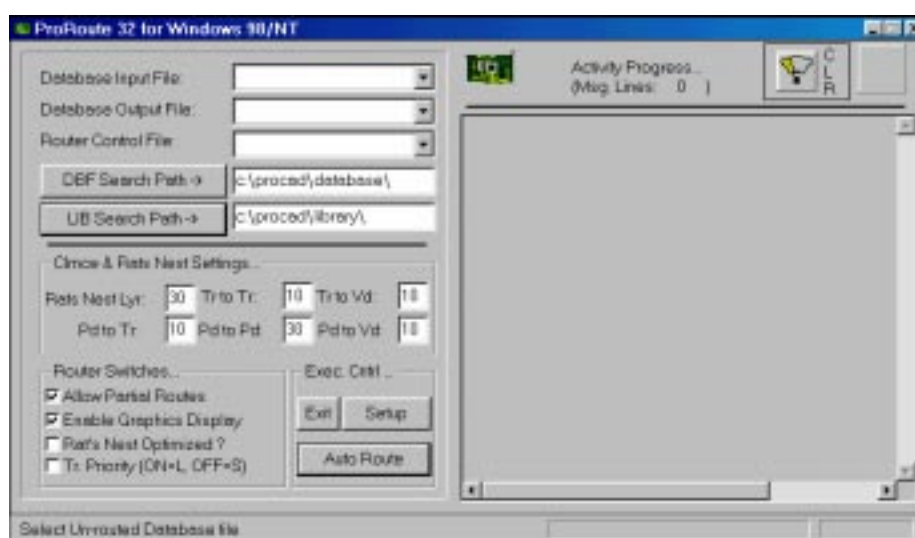
Hint: You may also use the integrated version of PROROUTE while in ProCAD by using the [Route]Auto Route menu item. This version of PROROUTE allows for Interactive Auto Routing within ProCAD, and offers better control of the routing process.

Note: If you plan to use the external PROROUTE utility, you must save your database before invoking it. When PROROUTE is invoked from within ProCAD, the current database is automatically saved.

All files associated with the database being routed (.DAT and .BIN) must be in the same working directory as PROROUTE. The .DBF and .LIB files must be placed in the sub-directories defined by the WCONFIG utility (CONFIG.BIN file).

For this tutorial, you should use the external PROROUTE utility. When invoked, PROROUTE starts execution by displaying a signon message with your serial number. After a few moments the PROROUTE main window will appear as shown below:

PROROUTE Window Control-menu Box:



The Upper left corner of the ProROUTE window is a Control-menu box [-]. This Control-menu box when invoked drops down a menu for controlling the various aspects of the ProROUTE window, and also sub-menus for displaying the current ProROUTE version (About box) and setting its execution priority. Go ahead, click on the [-] box and experiment. Note: All PROCAD utility programs contains this common Control-menu box, and they all perform essentially the same functions.

Database input file [.DBF]:

Select the PROCAD database containing the placed components and the rats nest connections. For this exercise select *PCB-RAT*.

Database Output file:

Enter the name of the output file to be generated by PROROUTE. It is important here to chose a different name from the input file, so that the original rats nest file can be used for backup. For this tutorial example, simply enter:

PCB-3 (file extension .DBF) is assumed.

Router Control file:

Enter the name of the file containing the parameters to be used by PROROUTE. For this tutorial session, use select the *TUTOR2.DAT* file.

ROUTER CONTROL FILE:

PROROUTE requires a predefined data file (.DAT) which describes certain parameters to be used during the route process. Information which PROROUTE requires include such things as source layer of RATS NEST information, destination layers for horizontal and vertical routes, and the allowable number of vias per route.

The control file is created by the user utilizing a standard line editor, or word processor in non-document mode. It must be a straight ASCII file, with no word processor control characters, with a single route parameter per line.

The Control file may include comment lines for ease of understanding by preceding each comment line with the \$ character. A series of predefined Route Control files may be created to define route strategies for up to six passes per route session, and used consecutively to create an unlimited pass router.

There are two types of parameters defined in the Control file PROROUTE PASS, which are variables required for each of the six allowable passes within one route session, and which are applied over all six passes during a single route session. Each parameter is established in the .DAT file by the use of a keyword which is four characters in length (except Design Rule Check keywords which are only two characters).

The parameters described in the keyword data file (TUTOR2.DAT) are used to define both PASS and GLOBAL variables. An explanation of the parameters established for the first pass, and the Global parameters, are explained below.

\$ Begin Pass Parameters

The \$ represents the beginning of a comment line. Any lines containing a comment must begin with the \$ character.

begin pass 1 establishes the beginning of Pass 1 variables.

pass 1 on turn Pass 1 on for use by PROROUTE

mode=p set the route mode for PWR/GND routing. This route mode will only select nodes which are named either VCCx or GNDx (where x is a blank or a single digit number). Other modes include m and a, for memory and algorithm.

maxv=0 allow no vias to be placed during this route pass.

maxa=90 these variables are ignored during power and

mina=0 ground bus routing (mode=p).

maxl=-1 set length of lines to be routed to unlimited

maxd=250 allow a maximum deviation in trace placement of 250 mils. This maintains mostly horizontal/vertical routing structure. If you are doing a single-sided board you will need to extend this variable significantly.

grid=50 to establish a minimum route grid movement for placement of traces and vias. A grid of 25 mils is standard, a grid of 20 mils will allow the placement of two traces between IC pads.

trac=25 set a routing trace width of 25 mils for this pass.

bend=on set bend reduction on, which allows for a higher completion rate, but takes longer to complete.

45db=off this variable is set to OFF during power routing (mode=p).

hlay=23 define the horizontal trace layer

vlay=20 define the vertical trace layer.

vgrid=100 define the via placement grid.

viacel=[pkg]pad030 define the via cell, and its library, which will be used every time PROROUTE places a feedthrough onto the PCB.

The PWR/GND Bus router does not guarantee 100% completion of all VCCx, and GNDx nodes; however with several successive route passes in this mode (mode=p) it is possible to achieve a high completion rate.

Take a look at the other pass parameters which the *TUTOR2.DAT* defines during execution of PROROUTE.

Pass 2 begins the Look-Out algorithm pass. This board does not utilize the memory route pass. This pass is also dedicated to routing the vertical traces first because of the definition of both **maxa**, and **mina**.

Pass 3 is the first pass to allow via placement. The number is kept low (below 4) to allow PROROUTE to attempt routes without feedthroughs first. This pass also opens up routing to all traces because of the new definition of both **maxa**, and **mina**.

Pass 4 allows wide use of vias in order to obtain a higher completion rate.

Pass 5 and 6 add two new layers (**hlay=24**, **vlay=25**) to this route iteration, to demonstrate PROROUTE's multilayer routing capability.

\$ Begin Global Parameters

Another comment line establishes the beginning of the Global parameters definitions. The Global Parameters will be used by PROROUTE during all six route passes.

mono=OFF define the display being utilized during execution of PROROUTE as monochrome. With this value toggled OFF, the monitor is not monochrome. Toggle this value ON if you are using a monochrome screen.

slay=30 define the source layer of RATS NEST lines. This is the PROCAD database layer which PROROUTE will search for nodes that must be routed.

tt=10 define the Trace-to-Trace clearance to be maintained during routing. PROROUTE performs continuous Design Rule Checks (DRC) while routing the board. All the DRC values are user-definable.

pt=10 define the Pad-to-Trace clearance.

pp=25 define the Pad-to-Pad clearance.

tk=10 define the Trace-to-Keepout clearance.

pk=10 define the Pad-to-Keepout clearance.

After PROROUTE has processed the Router Control File, the parameters are displayed on the graphic screen. At this point you may change any of the displayed parameters simply by moving the on screen cursor to the parameter which you wish to change. In this way you may use one Control file, and slightly change the parameters if desired.

GRAPHIC DISPLAY:

Enable Graphic Display?

If you enable Graphics display (Box checked), PROROUTE will display a representation of the PCB database so that you may watch the routes being placed into the design. By watching the route action as it takes place, you may be able to see potential problems which could affect the overall completion rate. However, by utilizing graphic display of routes, you will increase the route time which is required by PROROUTE.

Disabling graphic display (Box un-checked) causes PROROUTE to place routes more quickly, but does not allow you to participate with PROROUTE in a truly interactive fashion.

PROCESSING RATS NEST:

PROROUTE begins defining the route requirements by first breaking down the entire into single pin-to-pin connections. This process effectively sorts each net into a series of distinct nodes which may then be routed.

An important consideration for completion of this task is the DRC\$PAD Data token which PROROUTE looks for at each connection of the net. Every connection of the net must include the DRC\$PADx token, or PROROUTE defines that connection as invalid and deletes it from the route database.

You must insure that every cell in the CAD libraries of packages, pads, and vias, must include the DRC\$PADx token.

PROROUTE then sorts these distinct, point-to-point, nodes of every connection net to insure the shortest possible routing for each net.

Finally PROROUTE sorts each segment of the entire board connection structure by length, routing the shortest portions of each net first, then moving on to the next shortest, and so on. In this way PROROUTE effectively breaks down the entire routing task into a series of simpler trace segments.

STATUS LINE:

When PROROUTE begins routing the PCB it displays a constant progress report in the form of a status line at the bottom of the display. The status line illustrates the pass in use and it's percentage of completion, the node being processed, the number of unrouted nodes, the number of routed nodes, the number of feedthroughs placed, and the percentage of completion of the entire board.

By tracking this data you will be able to determine the effectiveness of each router pass, and each router iteration, and make changes in the variable definition as desired.

When PROROUTE has completed routing of the PCB-RAT design database, it should have achieved approximately a 94.9% completion rate. This will leave about eight nodes unrouted on the entire PCB design.

ABORTING AND TERMINATING PROROUTE

You may abort a route session at any time by using the "ESCAPE" key. When you abort a route session, PROROUTE will save the results of the route up to that point, so that you can view the results with PROCAD and continue PROROUTE at a later time.

When PROROUTE finishes a route iteration (up to six passes), you will be prompted for another database to be routed, and PROROUTE is ready to go again. If you want to route another board simply provide the database name and go from there, or click on the "**Exit**" button to exit PROROUTE.

SET TRUE WIDTH:

When you have left PROROUTE, return to PROCAD to view the newly routed PCB design database, PCB-3.DBF. This is the routed design, with approximately eight unrouted nodes.

When you open the database you should notice that all the traces appear to be lines without width. This is because PROROUTE turns off the true width of all traces for quick redraw when returning to PROCAD.

To view the lines with their actual width, you must execute the **SET TRUE_WIDTH <ENTER>** command. This command restores the defined widths to the database. This command may also be accessed from the **Toggles** dialog box in the **[Other..]** menu.

You are now ready to complete the routing of the PCB, by utilizing the manual routing techniques previously discussed.

USING DESIGN RULE CHECKS

PROCAD has an on-line Design Rule Check utility which allows you to quickly check certain aspects of your design without leaving the PROCAD editor.

To use the Design Rule Check feature simply type the **DRC** command from the keyboard:

DRC TUTOR2 <ENTER>

By specifying the TUTOR2.DAT file in the command syntax, you cause PROCAD to extract the DRC values assigned in TUTOR2.DAT and used during the routing of the PCB-RAT database. If you do not specify a .DAT file, PROCAD provides default values for the DRC.

The Design Rule Check window will appear in the upper left hand corner of the graphic display and should contain the following prompts as shown below:



These are the values assigned by TUTOR2.DAT when the DRC utility is accessed. These values are defined as follows:

DRC Layer combinations(s) 5,23

This represents the layers which will be tested by ProCAD's DRC.

Layers 5 and 23 represent the copper padmasters and traces which make up the solder side of the PC board.

NOTE: You must check all layers which combine to graphically represent a single layer of the PCB.

In this way you can check each layer of the board and make corrections as necessary. To check the routed version of PCB-RAT, you must check Layer combinations: 5 and 20, 5 and 23, 5 and 24, 5 and 25.

Domain only check OFF

This is used to establish a window area within a large database.

With Domain only check toggled OFF, PROCAD will check the entire database.

With Domain only check toggled ON, PROCAD allows you to define an area of the design to be checked.

Checking a small area of the entire design will greatly speed up the DRC utility, and is sometimes all that is required.

Shorts check ON

This is the only DRC device actually used during checking of a Schematic design database. The default is toggled ON in PCB databases.

The DRC utility checks a PCB database for traces and pads on the same layers specified (i.e. Layers 5 and 23 for solder side traces), which intersect but do not share a common vertex or connection point. PROCAD flags these instances as shorts for your inspection.

Save DRC markers ON

This statement allows you to view all DRC flags as temporary or saved within your database.

With Save DRC markers toggled OFF, all DRC flags will be erased when the screen is REFRESHed or REDRAWN.

With Save Errors toggled ON, all DRC flags will remain in the design database until they are cleared.

DRC flags are assigned layers as follows:

Error	Layer
Tr-to-Tr	31
Pd-to-Tr	32
Pd-to-Pd	33
Shorts	34

All Trace-to-Trace spacing violations will be flagged on Layer 31.

All Pad-to-Trace spacing violations will be placed on Layer 32.

All Pad-to-Pad spacing violations will appear on Layer 33.

All flags for shorts will appear on layer 34.

In order for PROCAD to check the spacing from a pad (via, or device pad) to another pad or trace, you must include the data token **DRC\$PADx XX lyr** in the data attribute file of the pad or device.

XX is the diameter (sides if rectangular) of the pad. For instance, PAD030 will have a value of 30.

lyr is the Layer number, on which the pad exists. In most cases this will be Layer 5, which is the copper padmaster.

Click on the "Run DRC..." button to begin execution of the DRC function. PROCAD will display status information regarding the progress of the DRC, on the Message Line.

PROCAD is a "VERTEX RECOGNITION" system. This means that PROCAD will not recognize the connectivity of two signals which intersect, unless there is a common vertex at the point of intersection.

CLEAR DRC:

To clear the DRC flags, issue the menu command **[Check]Clear DRC errors** or type the command:

CLEAR DRC <ENTER>

This command deletes all entities from layers 31 thru 34.

NOTE: Do not use layers 31 thru 34 for any reason. The CLEAR DRC command will delete all entities from these layers.

CLEAN-UP AND EXIT

Once the final design is complete there are a few items which will need to be taken care of. The design must be cleaned-up before final artwork can be generated.

First of all you will want to look through the entire design to see if there are any last minute improvements which can be made to the board layout which may reduce the length of some routes, or reduce the number of feedthroughs on the PCB. This process is not required on a board which has been autorouted 100%, but usually a designer will be able to make minor improvements to an autorouted layout.

It is a good idea at this point to utilize the on-line DRC facility provided with PROCAD to insure that PROROUTE did not violate any of the user-defined design rules established prior to routing. While this process might take some time for a large board design, it is still faster than finding errors the hard way... in manufacture. The PROCAD DRC may also assist you in discovering regions of the layout where some improvements might be made.

Finally, when the entire layout has been checked, and modified where necessary, you must remove the limit areas established in the board prior to routing. The area limits, as well as any keepout areas, are actual lines and polygons which will appear on any artwork masters. Therefore, prior to generating any artwork which is to be used in production, you must remove all area limits and keepouts from the board. After removing the unnecessary vias, keepouts, and board limits, you should begin placing the information which completes the design. You should place alignment targets, fabrication notes, and assembly notes.

To see the demonstration board in a finished state, open the database file PCB2-END.

This board has been run through the DRC utility, had all keepouts removed from the routing layers, and had assembly and fabrication notes added to the specific layers. This board is ready to be output to hardcopy.

To view the various layers use the **SET VIS** command on these layer combinations:

Component Side	5,13,16,20
Solder Side	5,13,16,23
VCC Plane	13,21
GND Plane	13,22
Inner Layer 1	5,13,24
Inner Layer 2	5,13,25
Soldermask	6,13

Drill Master	7,8,13
Silkscreen	9,10,13
Fabrication	7,8,11,12,13,14
Assembly	3,4,10,15

You may also view appropriate artwork layer combinations by accessing the **[View][Artwork]** menu and selecting the appropriate artwork entry. These layer combinations are the same layers which must be output to generate production artwork of the PCB design.

For more information on generating output see Chapter 5 of this Tutorial.

You have now completed the vertical design, from schematic through PCB design. Utilizing PROCAD, and the many support programs available from Interactive CAD Systems, you may find new avenues to more creative designing.

Creating Library Symbols

C***chapter***

4

Creating Symbols

Topics Covered:

- INTRODUCTION
- DATA ATTRIBUTES
- REGULAR CELLS
- TEMPLATE CELLS
- HIERARCHICAL CELLS
 - SCHEMATIC HIERARCHY
 - PCB HIERARCHY
 - GENERAL HIERARCHY
- LIBRARY MANAGEMENT

INTRODUCTION

At times you will require a symbol for a design which is not currently available in the PROCAD libraries.

PROCAD allows quick entry of new components to existing libraries, as well as easy creation of completely new libraries.

This chapter is concerned with all aspect of cell formation and library management.

DATA ATTRIBUTES

Data Attribute files are ASCII characters stored with a library cell, which provide information regarding packaging, reference designators, and pin-numbers.

Data attribute files contain data tokens which provide information for specific functions such as Design Rule Check, Rats Nest, and Bill-of-Materials.

For instance, within the 7474 device the data attribute file reads as follows:

```

CELL:[TTL]7474
  REFERENCES: U2A  U2B
  CELL DATA:
    BOM$CO_PN  7474
    BOM$MF_PN  SN74LS74
    BOM$DESCN  DUAL D FLIP FLOP

    RAT$PKG    [PKG]IC14A

    RAT$VCC1   14
    RAT$GND1   7

    RAT$TCNT   2
    RAT$DEVA   6 5 4 3 2 1
    RAT$DEVB   8 9 10 12 11 13

```

Each of these data tokens provide specific information to PROCAD, PROCAP, or one of the interfaces to other programs.

For instance, the BOM\$ data tokens are utilized by PROCAP to generate a Bill-of-Materials from a schematic design.

The RAT\$ data tokens are used by both PROCAD and PROCAP to perform functions like gate grouping, and automatic connection of power and ground.

NOTE: The importance of the data token utilized in PROCAD is the openness which it provides to the user for interfacing to other applications.

The Data Attribute file provides you with an ASCII interface from PROCAD to one of your own in-house applications if it is required. This is a powerful feature of the data token system established within PROCAD. If you have any needs for special interfaces, there is a strong possibility it may be readily available through the Data Attribute file.

REGULAR CELLS

A regular cell is the simplest form of device stored in a library. The regular cell requires very little information, and has no depth. A good example of a regular cell is a resistor. The resistor has two bi-directional pins, and two attributes.

Another example of a regular cell is a microprocessor, which has more complicated pin definitions and has four attributes.

A regular cell is a single device in a physical package, such as a resistor or microprocessor.

To demonstrate cell formation of regular cells, you will create a D-type Flip Flop, and store it in a new library called TUTOR.LIB.

CHANGE LIBRARY:

To change to a different library, or create a new library, execute the command:

CHANGE LIB [TUTOR] <ENTER>

This causes PROCAD to make the TUTOR.LIB the active library. If PROCAD can not find the TUTOR.LIB file, it prompts you to create a new library. Use this command to create the new TUTOR.LIB, and use this new file for storing all cells created in this Tutorial.

NOTE: All new cells will be stored in the active library as they are created.

Make layer 2 active by selecting it from the layer selection combo box. This is one of the Symbol Layers, used for generating schematic symbols.

Access the **Rect** command from the Tool bar. This command allows you to enter a rectangle. PROCAD prompts you for the first, and second corners of the rectangle you are entering. You should specify the two opposing corners of the rectangle you want to enter.

Make a rectangle about 600 Mils square.

Click the right mouse button (<ESCAPE>) to stop entering rectangles.

Access the **Lines** command from the Tool bar to begin entering lines. Enter 200 Mil lines around the rectangle as illustrated in Figure 28.

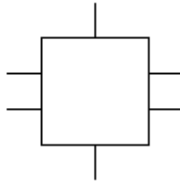


Figure 28

To begin entering text for pin numbers and part name, first

SET SNAP OFF <ENTER>

to allow movement of the cursor off the defined grid.

Now access the **Text** command from the Tool bar to **Input Text**, and position text into the cell as illustrated in Figure 29.

Click the right mouse button or Press <ESCAPE> when you have finished entering text. At this time you may want to adjust the position of some of the text, if it is required. Simply Access the **[Edit]Move** command (or click and drag the text) for this operation.

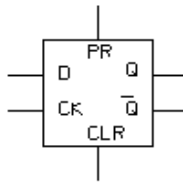


Figure 29

Notice that one of the **Q** pins has a bar over it. Simply access the **Lines** command to **Input Line**, and enter a small line over the letter. You may want to **ZOOM** into the symbol by accessing the **Zoom In (PgUp)** key) command from the Ribbon bar in order to enlarge the view while entering this line.

CELL FORMATION:

You have now completed the graphical representation of the cell, and are ready to store this in a library with logical information. Make sure the Snap-on-Grid is on by issuing the following command:

SET SNAP ON <ENTER>

Tips: Make sure Snap-on-Grid is on at this point before you continue. Cells with references points off grid may be very difficult to connect to !!!

Access the **SelBlk** command from the Ribbon bar or **[Edit][Select]Block** menu item to draw a selection window around this entire symbol (you may also click drag the left mouse button to accomplish the same task). PROCAD is now in the SELECT mode, and the symbol should be highlighted, and menu changed.

Access the **[Other..]Cell Formation** command to begin CELL FORMATION process. PROCAD returns the following prompts:

Cell Name:

Name this cell:

FLIPFLOP <ENTER>.

At this point, PROCAD opens a dialog box as shown below:



You should respond to the prompts in the Cell Properties dialog box as follows:

Cell Database Units:

This is the interpretation given to the units the cell is defined in. For example, if you selected millimeter units, the cell will be automatically scaled (converted) to the proper size when placed in a drawing defined in other units, say mils. If you select Database units (default), no scaling (conversion) will be performed when the cell is placed in a drawing.

For this exercise, leave the default Database units.

Number of Attribute Lines:

This represents the number of lines opened in the data attribute file. You want to be sure to assign enough lines for this file to be useful. Enter 15 just to be safe.

Attribute Window Left Margin:

This defines a left-margin to begin editing. This is only useful if you have a predefined data attribute file which you do not want to edit. In this case enter 0, so you may use the whole file.

Number of I/O Connections:

There are 6 I/O, or pin numbers, on this device.

Assign I/O types & Nets (Use cell for simulation) ?

If you do not plan to use this cell for simulation purposes or designate pins as inputs, outputs or bi-directional for the DRC floating pin checker, you should simply say *NO* (Leave box unchecked), even if it is a schematic symbol. If you answer *YES* (Box checked), the Pin Editor will then be invoked later after defining I/O points below.

For this exercise, answer *Y* (Box checked)

Attribute Data File: [NONE]

If you have a prepared attribute data file you may enter it at this time. The use of a prepared attribute data file insures that all cells created will have the same attribute data structure, and can also speed the cell formation process if you are creating a long series of cells. For this exercise, leave the default of None, since you have not created one.

Press **Okay** to continue the cell properties definition process. PROCAD will then prompt for the following information to complete the Cell formation process:

Cell Reference Coordinate:

This is the reference point which the cell is placed around when called from the library. Position the cursor at the upper-left corner of the rectangle and click the left mouse button.

Instance_name Placement Coordinate:

This is the point at which PROCAD will assign a reference designator to this symbol as it is placed into a design database. Position the cursor 100 Mils diagonally away from the upper-left corner, to allow room for the reference designator. Click the left mouse button at this point.

Component Value Reference Coordinate:

This is the position at which the cell value is placed when defined after placement. This device does not have a value, so this point may be placed in the center of the device. Click the left mouse button at this point. Cells which do have values are resistors, capacitors and other discreet devices. The value of these devices is assigned after placement into a design database by using the **CHANGE VALUE** command ([**Other..**][**Modify**]**Instance Value** menu item), or using the Quick Edit feature (double clicking on the component).

A window will open in the upper-right of the display. This window is the attribute data editor, and allows you to enter text in a line fashion.

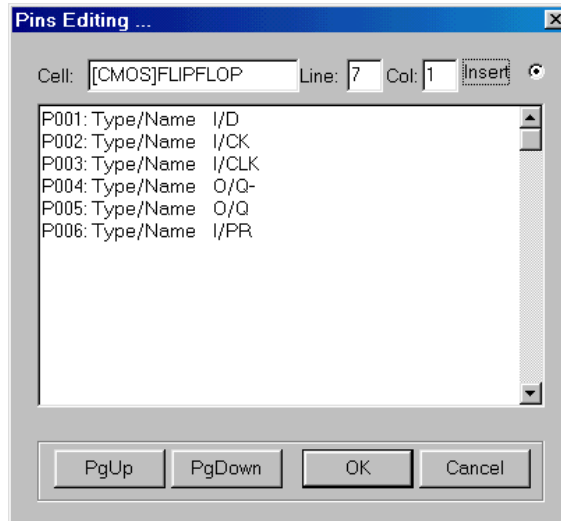
Enter the following data tokens into the data attribute file:

```
D-Type Flip Flop
Created By:.....
Date:.....
INS$PRFX U
```

The only data token in this file is **INS\$PRFX U**, which is the instance prefix which will be used by PROCAD when automatically assigning reference designators to this device as it is placed into a design database. The rest of the information is just reference material for your own use; it is optional. Press **Okay** to exit the data attribute file editor.

Since you checked the "**Use Cell for Simulation**" box previously, the Pin Editor window will be opened with all 6 pins listed. This window allows you to define each pin as **I**, **O**, or **B**, (input, output, or bi-directional). Pay close attention to naming the pins, as well as defining their nature, because you will be using this device later in this section.

Enter the following pin information:



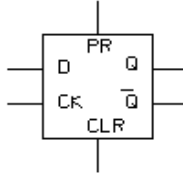
Move the text cursor (caret) to column 17 of the attribute window and enter the signal types (**I, O or B**) and signal names as shown above.

Note: Items in columns 1 through 16 are write protected; they are the necessary prompts and required entry format guides.

Press Okay to exit the pin editor.

I/O Connection Reference Coordinate:

This queries you for the actual reference point of each pin number, beginning with pin 1 through the last. Position the cursor at the end of the line which is labeled **D**, when prompted for Pin #1, and click the left mouse button, as illustrated in Figure 30.

**Figure 30**

Position the cursor at the end of line **CK** for Pin #2.

CLR is Pin #3. **Q-** (or Q-Bar) is Pin #4. **Q** is Pin #5. **PR** is Pin #6.

PROCAD returns the message:

Saving Cell FLIPFLOP in [TUTOR]

This ends the CELL FORMATION command, and PROCAD returns you to the design database in the SELECT mode.

At this time it may be a good idea to place the new cell into the database from the library. While still in the SELECT mode, access the **[Other...]****Hold Selection** command or press *Cntrl-Z*. This command suspends the SELECT mode, and returns to the COMMAND mode, while retaining all selected entities in suspension.

With the SELECT mode suspended, you may place the new cell into the database by accessing the **Component** command from the Tool bar or **[Place]****Get Cell...** menu item.

Place the FLIPFLOP cell next to the old entities. You will notice that PROCAD assigns this cell a reference designator of Unn as it is placed into the design database.

Check the device to insure that it was stored correctly.

When you have checked the device, you may delete the old information by accessing the **Select Entity** command from the Ribbon bar (or **[Edit]****[Select]****Entity** menu item) to return to the SELECT mode, and access the **Delete** command from the menu to Delete entities.

Delete everything from this database and access the **[File]Pack/Renumber** command to pack it before performing the next exercise of creating a template cell.

TEMPLATE CELLS

The template cell is a device which may be grouped together with other devices of the same type within a single physical package.

The most obvious example of a template cell is a 74LS00 device. This device, represented as a two-input NAND gate, may be grouped with up to three more of its kind in a single fourteen pin DIP package.

Other examples of template cells would be a resistor network, with ten resistors in a single physical package, or a switch symbol packaged in an eight switch DIP package.

The template cell is more complicated than the regular cell because of the information required during definition.

Create a 74LS00 device as a template cell, to be stored in the TUTOR library. Check to see that the active library is TUTOR, the active layer is Layer 2, and that SNAP is ON. If these parameters are not true, execute commands to make them true.

Begin by accessing the **Lines** command from the Tool bar to **Input Line**. Enter a line as illustrated in Figure 31.



Figure 31

Execute the **Arc** command from the Tool bar to input an arc as illustrated in figure 32.



Figure 32

The INPUT ARC command returns the following prompts:

Starting Point of Arc:
End Point of Arc:
Point on the Arc:

These prompts should be answered with the understanding that all arcs are drawn in a clockwise fashion.

After the arc is entered, access the **[View]Zoom In** command to ZOOM 2X around the device.

SET GHOST 50 50 <ENTER> to establish an off-grid cursor movement of 50 Mils.

SET SNAP OFF <ENTER> to allow cursor movement off the grid.

At the end of the arc, access the **Circle** command from the Tool bar to enter a circle for this device as illustrated in Figure 33. This circle should be 100 Mils in diameter.



Figure 33

The INPUT CIRCLE command returns the following prompts:

Enter Center Coordinate:
Enter Point on Circumference:

When the circle has been input execute the following commands:

SET GHOST 10 10 <ENTER>

to establish a finer off-grid cursor movement.

SET SNAP ON <ENTER>

to return cursor movement to the grid.

Input **Lines** as illustrated in Figure 34.

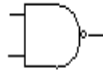


Figure 34

Now Input **Text** as illustrated in Figure 35.

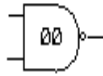


Figure 35

You have now completed the graphical definition of this symbol, and are ready to begin the CELL FORMATION command to store this symbol in the library with logical information.

Access the **[Edit][Select]Block** command to place a **SELECT WINDOW** around the device (you may also click drag the left mouse button to accomplish the same task).

PROCAD highlights the symbol and enters the SELECT mode. Execute the **[Other...]****Cell Formation** menu item to begin CELL FORMATION.

PROCAD returns the following prompts:

Cell Name:

Name this device *74LS00* and press <ENTER>.

Respond to the Cell properties dialog box prompts as follows:

Cell type: Template Cell

Enable this box, since this is a template cell.

Package Pin Numbers; Text Height:

This value is the height of text which PROCAD will input as pin numbers as this device is placed into a design database.

A template cell has variable pin numbers relative to its position in the physical package. These pin numbers are assigned by PROCAD during placement. Enter a height of 100 for this exercise.

Number of User-defined Attribute Lines:

This is the number of lines in the data attribute file. Enter 20.

Attribute Data Window Left Margin:[0]

Use the default value of zero.

Attribute Data File:

Accept the default (NONE).

Number of I/O Connections:

There are 3 I/O on this cell.

Use cell for simulation ?

Answer YES (Box checked) for this exercise.

Now click on the Okay button to close the dialog box. PROCAD will then prompt for the following information:

Cell Reference Point:

This is the point around which the device will draw when placed into a design database. Position the cursor on the top pin and click the left mouse button.

Instance_name Placement Coordinate:

This is the point at which PROCAD will place reference designators when the device is placed into a database. Click the left mouse button in the center of the symbol.

Component Value Placement Coordinate:

This may be placed in the center of the device, because the value is not used for this device. Click the left mouse button.

PROCAD will then open the Attribute Data File editor in the upper-right corner of the graphic display.

Enter the following data tokens into the Attribute Data file.

```
BOM$DESCN QUAD 2-INPUT NAND
BOM$CO_PN 7400
BOM$MF_PN SN74LS00
```

```
INS$PRFX U
```

```
RAT$PKG [PKG]IC14A
RAT$VCC1 14
RAT$GND1 7
```

```
RAT$TCNT 4
RAT$DEVA 1 2 3
RAT$DEVB 4 5 6
RAT$DEVC 9 10 8
RAT$DEVD 12 13 11
```

```
SWP$PINS 1 2
SWP$GATES A B C D
```

Notice how much more detail is required for this template cell, than was required for the regular cell you just created. The most important data tokens in this cell are the **RAT\$** tokens:

RAT\$PKG specifies the device physical package which will be used.

RAT\$VCC1 and **RAT\$GND1** allow PROCAD to automatically assign power and ground nodes to these pins during Rats Nest generation.

RAT\$TCNT tells PROCAD the Template Count used in grouping devices into a physical package. This is a very important token, and must be included in all template cells.

RAT\$DEVn (where n is a letter from A to Z) assigns variable pin numbers to actual pin locations on the device.

The **RAT\$TCNT** token must be followed by the equivalent number of **RAT\$DEVn** tokens. Notice, for this device the count is four and there are four device pin configurations listed.

Another important concept regarding the **RAT\$DEVn** pin descriptions: The pin numbers assigned must follow the actual pin definitions. Notice that these numbers are not assigned sequential values, but are instead numbered in their equivalent order.

For this cell, **DEVC** and **DEVD** are numbered according to the actual pin values Input1, Input2, and Output pins.

SWP\$PINS n1 n2 ... This keyword is used to denote that I/O #n1 and I/O #n2... can be swapped (electrically equivalent).

SWP\$GATES G1 G2 ... This keyword is used to denote that gates G1 and G2... are swappable (electrically equivalent).

Press the OK button when you have finished editing the Data Attribute file.

PROCAD returns the next prompt:

Since you responded YES to the "**Use of the Cell for Simulation**" (Previous dialog box), PROCAD then brings up the Pin Editor window, and allows you to define the three pins as follows:

P001: Type/Name	I/In1
P002: Type/Name	I/In2
P003: Type/Name	O/Out

These are the logical descriptions and names of each pin. Press OK button to exit the Pin Editor.

Note: Since you are creating a template cell, there are 2 I/O reference selections per pin.

(1) **I/O Pin #1 Connection Coordinate:**

Position the cursor at the end of the top Input pin, and click the left mouse button.

(2) **I/O #1 Packaging Pin Reference Coordinate:**

This is the location which PROCAD will place assigned pin numbers when the device is placed. Notice that the cursor moves off grid when placing this coordinate; this is to allow finer placement of the pin location.

Position the cursor slightly off the pin as illustrated in Figure 36.

Repeat these steps for both Pin #2 and Pin #3, as illustrated in Figure 36.

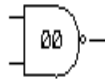


Figure 36

You have now completed the CELL FORMATION command. PROCAD saves the cell 74LS00 into the TUTOR library, and returns to the SELECT mode.

Again, execute the **[Other...]****Hold Selection** command to suspend the SELECT mode, and place this new cell into the database to insure that it has been properly stored.

Access the **[Place]****Get Cell...** command, and place the 74LS00 cell into the database. Notice as the device is placed it is assigned a reference designator of UnnA, and is assigned pin numbers 1, 2, and 3.

Place some more of these devices to see how the reference designator and pin numbers are different for each device that is added to the database.

NOTE: You do not have to suspend the SELECT mode in order to check the device every time you add a cell to a library file. However, it is wise to verify that the cell has been saved correctly before deleting the master copy, it could save you some time if an error has been made.

HIERARCHICAL CELLS

A hierarchical cell is the most complicated type of cell to create due to its depth of structure.

Cell hierarchy involves a structured approach to creation: top-down, or bottom-up. Both of these approaches start from opposite ends of the same ladder for cell formation.

The best example of a hierarchical cell is any physical package, such as the IC14A stored in the [PKG] library.

The IC14A symbol includes the assembly outline of the component at the top level; however, nested within the cell are the padmaster information, and the silkscreen. Within the padmaster is information regarding copper pad, soldermask clearance pad, drill codes, and drill targets. All of this information is stored within the IC14A symbol, yet when it is called from the library for placement in a database only the assembly outline is readily obvious.

A schematic symbol may be constructed in a hierarchical fashion as well. A D-type Flip Flop is an example of this. When the symbol is called from the library, only the top level of the device is visible; however, nested down a second and third level could be the gate and switch level information respectively.

SCHEMATIC HIERARCHY:

To understand how hierarchy is used in Schematic designs, you may define the hierarchical structure of the FLIPFLOP cell which you created as a regular cell previously.

In PROCAD, open (initialize a new) database named TESTCELL.DBF as a schematic database.

PROCAD will automatically execute MACRO CKT.CMD command. This macro defines layering and function keys for schematic type databases. However, because the macro also changes the active library from TUTOR to CKT, you must execute *CHANGE LIB TUTOR <ENTER>* command.

You must have the TUTOR as the active library, because that is the library where all work is to be stored.

Access the **Component** command from the Tool bar or **[Place]Get Cell...** menu item. Position the cursor at the center of the graphic display and place cell *FLIPFLOP* at the cursor location.

CELL EDITOR:

You will now be editing this cell to add nested, or hierarchical information to this cell (Nesting and Hierarchy are the same in PROCAD). This hierarchy will be created in a top-down fashion.

You have already created the top layer of this device in the schematic symbol of the Flipflop. Now, you will be creating the gate-level information which makes up this device.

Access the **[Library]Edit/Create Cell** command. Provide the name *FLIPFLOP* <ENTER> when prompted by the system.

PROCAD returns the message **EDIT CELL**, and enters the Cell Editor.

Notice the Status Line across the top of the graphic display. It now reads:

.....CELL: FLIPFLOP

The Cell Editor is a feature of PROCAD, which allows you to edit a single cell, from any library, without affecting your current design.

When the Cell Editor is invoked, PROCAD flags your position in the current database, as well as all parameters such as SCALE and ZOOM, and leaves the database to open the Cell Editor.

Once inside the Cell Editor PROCAD commands work as they do in the Database Editor.

The **Close**, **Save**, and **Exit** commands from the **[File]** menu leave the Cell Editor and return you to the database you were last working in.

The Cell Editor also retains many of the properties of the database which you came into the Cell Editor from. For instance, because you executed the **MACRO CKT** command in the CELL database, the Cell Editor is now established with the same layering and function key definitions as a schematic database.

The cell FLIPFLOP should now be displayed on the screen. This cell is ready to be edited at this point.

Tips: You must be careful when you edit a cell in PROCAD. The changes you make to a cell while editing will apply to every occurrence of that cell, and every database it is placed in. This is a global type editing function, and definitely could adversely affect every database previously created.

Before beginning the cell editing process, type the command `SCALE -2 <ENTER>`. This command will scale the screen to 50% to allow you greater room to work.

Make layer 40 active by selecting it from the layer selection combo box. (You may have to scroll this box to locate layer 40).

You will need to change to Layer 40, because this is one of the layers reserved for nested information. Layers 40-48 are reserved for Nesting; any hierarchical cell information should be placed on one of these layers.

Since the CKT.CMD macro set automatic Cell, wire and text placement on layers 1, 6 and 9 respectively, you should use the **[Layer]Set Auto Layers...** command to change these placement layers to default (all entities placed on current active layer).



Place six of the devices **[CKT]NAND-3** to the right of the Flipflop in the configuration illustrated in Figure 37.

Tips: Use the `GET [CKT]NAND-3 <ENTER>` command entered from the keyboard, since library **[CKT]** is not currently active.

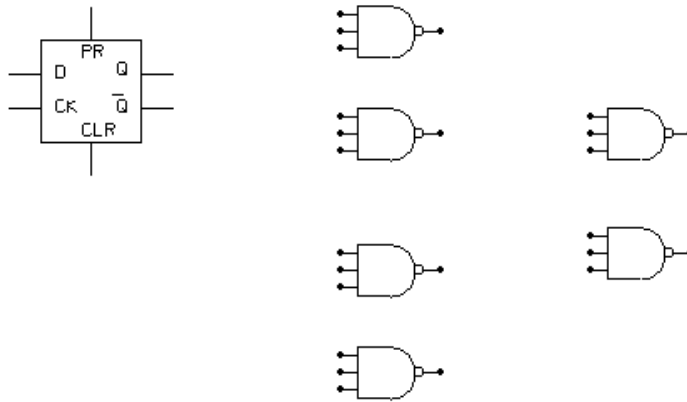


Figure 37

Now make layer 41 the current working layer. Change to this layer in order to connect these six gate devices. In hierarchical design, just as in regular schematic design, you need to keep information separated by layers for ease of viewing the design.

Connect the entire circuit as illustrated in Figure 38. using the **Wires** command.

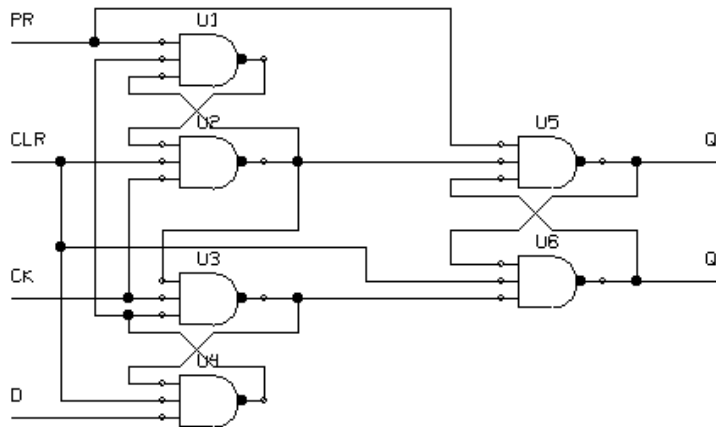


Figure 38

When interconnecting these devices, use the **[Check]Show Node** command to verify that the signals are correctly hooked up.

Also use the **CHANGE SIGNAL ([Other...][Modify]Net Name)** command to name the wires D, CK, CLR, Q-, Q, and PR, as illustrated in Figure 38.

You have now created the hierarchical information of the SN74LS74 device. This information is ready to be saved into the TUTOR library. However, you will first want to change the name of this cell, to retain the original cell in the library as well.

Access the **[File]Save As** menu item. Enter the name *74LS74* <ENTER>. This will rename the current cell to 74LS74. Check the status line to verify the name change.

PROCAD returns the message:

Edit Cell Properties (Y/N):[Y]

This means to edit the logical properties of the cell as well as the graphical. You would answer yes to this question only if you wanted to edit pin numbers, or data attributes, or some other logical property. Select *YES*.

Template Cell or Regular cell:

The Flipflop cell, which you used to create this device, was a regular cell; however, this will be a template cell. Select template cell.

Package Pin Numbers Text Height:

Type 75 here to specify a pin number height of 75 Mils. The package pin number is automatically assigned by PROCAD as this device is placed into a design database.

Number of User Defined Attribute Lines:

The default value should be [20] as previously defined.

Attribute Window Left Margin:[0]

Attribute Data File:[NONE]

Number of I/O Connections:[6]

Use cell for simulation?: YES

Now press OK to close the Cell properties dialog box.

Cell Reference Coordinate:

Click the right mouse button or Press <ESCAPE> for this selection, because the predefined value is fine in this case. All of these values were previously defined when you created this cell as a regular component named Flipflop. Some of the old values will still be acceptable, and you should only press <ESCAPE> to accept a previously defined value.

Instance Name Placement Ref Coord:**Component Value Placement Ref Coord:**

Press <ESCAPE> to accept the previously defined values for both.

PROCAD opens the Attribute Data File in a window on the left. Enter these values as data tokens **(On ProCAD V5.0 and above, these data equivalents are entered from a property sheet dialog):**

D-type Flip Flop**Created By:****Date:**

INS\$PRFX **U**
RAT\$PKG **[PKG]IC14A**
RAT\$VCC1 **14**
RAT\$GND1 **7**

RAT\$TCNT **2**
RAT\$DEVA **2 3 1 6 5 4**
RAT\$DEVB **12 11 13 8 9 10**

BOM\$DESCN **Dual D-type Flipflop**
BOM\$CO_PN **74LS74**
BOM\$MF_PN **SN74LS74**

These data tokens allow both PROCAD and PROCAP to work effectively with this device in a variety of ways.

Pay attention to the way that the tokens **RAT\$TCNT**, **RAT\$DEVA**, and **RAT\$DEVB** work together.

Press Okay when you have completed editing the Data Attribute File.

PROCAD opens the Pin Editor and allows you to alter the pin type/name which you previously defined. The predefined values should be displayed as:

P001: Type/Name	I/D
P002: Type/Name	I/CK
P003: Type/Name	I/CLR
P004: Type/Name	O/Q-
P005: Type/Name	O/Q
P006: Type/Name	I/PR

Notice how the names of the pins defined in the Pin Editor, match the user-defined signal names in the hierarchical information.

NOTE: PROCAD recognizes connectivity in hierarchical cells by virtue of the common pin and signal names.

In this cell, there is both a pin and signal which are named **D**. The hierarchical definition of this device allows a signal, which is connected to pin **D**, to pass into the nested signal which is also named **D**.

Press Okay button to accept the default pin type/name configuration.

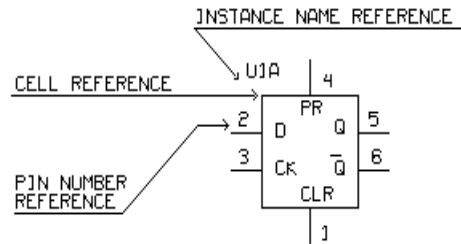
I/O #1 Connection Ref Coord:

Press <ESCAPE> to accept the default.

I/O #1 Packaging Pin Ref Coord:

This is the point at which PROCAD will automatically assign a pin number as this device is placed into a design database.

Notice that the cursor moves on a finer grid when defining this attribute. Define this point as illustrated in Figure 39.

**Figure 39**

When you have defined these coordinates for pins 1-6 PROCAD returns the message:

Save Changes to Cell (Y or N):[Y]

Press <ENTER> to accept the default, and save this new device into the TUTOR library.

PROCAD saves the device, and returns to the TESTCELL database which you were previously editing.

Place the newly created 74LS74 device into your database, to the right of the existing FlipFlop cell.

This device will be assigned a reference designator, and pin numbers as it is placed into the database. Place a couple of these devices to see how PROCAD assigns their reference designators and pin numbers.

SHOW HIERARCHY:

To view the hierarchical information within a single cell, while editing a schematic design, access the **[Library]Cell Hierarchy** (SHOW HIERARCHY) command, place the cursor on the device and click the left mouse button.

This command pushes into the hierarchy of a device so that you may view nested information.

The SHOW HIERARCHY command is not an editing tool, it is used simply to view nested information.

Access the **[File]Close** command to leave the cell hierarchy and return to the design database.

PCB HIERARCHY:

To understand PCB Hierarchy you must understand the fundamental differences between it and Schematic Hierarchy.

The biggest differences between the two types of hierarchy are as follows:

In a Schematic design, hierarchical information is logical information which affects the performance of the circuit while not being an actual part of the top-level design. In a PCB design the hierarchical information is physical information which is an integral part of the actual design.

This difference affects the methods used in establishing hierarchy within the two types of symbols.

You will recall, in the Schematic Hierarchy which you just completed, you placed all hierarchical information on Layers 40-48, or Nested Layers. This is to keep nested information outside of the actual design layers.

In PCB Hierarchy you will place the nested information on Layers 1-30, to keep all the information within the actual design layers.

In PROCAD, *OPEN PCELL* <ENTER>, to open a database named PCELL.DBF. This database should be established as a PCB type database, with MILS as units.

PROCAD automatically executes the *MACRO PKG* command to define the layers and function keys for use in a PCB database. change the active library to TUTOR (use the library selection combo box on the ribbon bar).

You are now ready to begin creating a hierarchical cell for use in a PCB design. The most common type of nested cell used in a PCB database is a physical package.

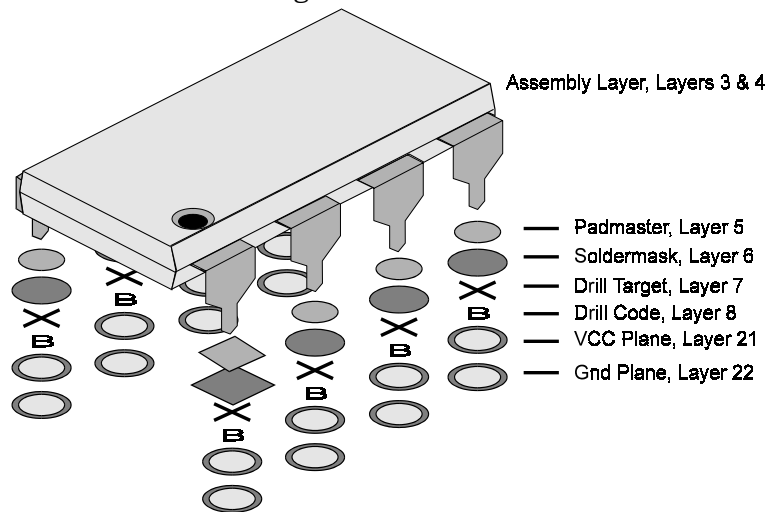
SET NEST 1 <ENTER>

To set the nesting display to level 1. Level 1 of nested information is the top-level.

Place the *[PKG]PAD* cell in the center of the graphic display.

Access the **[View]Zoom In** command, and Zoom 2X around the pad.

The PAD cell contains a variety of information which is stored on different layers. The layers used in the padstack are illustrated in Figure 40.



PAD STACKS (Ground, Voltage & Thru pads)



Figure 40

To view the various layers of information use the **SET VISIBLE** command as follows:

SET VIS 5 <ENTER>

To view the copper padmaster.

SET VIS 6 <ENTER>

To view the solder mask clearance pad.

SET VIS 7 8 <ENTER>

To view the drill target and drill code.

All of this information is stored in a single padstack. Thermal relief pads may also be included on Layers 21 and 22 for internal power and ground planes.

SET VIS ALL <ENTER>

To view all layers.

The pads which are stored in the PKG library are available for creation of new through-hole components. These pads are also utilized in all existing packages within the PKG library.

If you are creating a unique device, which does not utilize the standard padmasters you must create your own. When creating new padmasters, remember to include all information relevant to the padmaster such as copper, soldermask, and drill targets.

Tips: For faster padstack cell creation, use the automatic padstack generation feature of PROCAD by executing the [Library]Padstack Wizard... menu item. See the MAKEPAD command in the Reference section for details.

To demonstrate cell hierarchy for PCB cells you will create a 4 pin DIP package as follows:

Select working Layer 3 from the layer selection box.

Layer 3 is the Assembly Layer, where you will design the assembly outline of the new package.

SET GHOST 25 25 <ENTER>

To define a new off-grid cursor movement.

Delete the PAD cell that you've placed on the screen, using the **[Edit][Delete]Entity** command.

Access the **Lines** command from the Tool bar. Position the cursor in the center of the display and click the left mouse button to start Inputting a Line.

Create the assembly outline of the 4 pin DIP as illustrated in Figure 41.

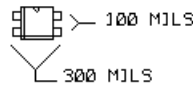


Figure 41

Place [PKG]PAD1 cell on pin 1 of this device (you may use the *GET [PKG]PAD1 <ENTER>* for faster access).

Place the cell [PKG]PADG on pin 2, [PKG]PAD on pin 3, and [PKG]PADV on pin 4.

You are now ready to store this device in the library.

Access the **SelBlk** from the Ribbon bar or **[Edit][Select]Block** menu item. Position the cursor off the device, and **SELECT WINDOW** around this device (you may also click drag the left mouse button to accomplish the same task). Now access the **[Other..]Cell Formation** command to start the cell formation process.

Name this cell IC04, and place the cell reference on pin 1. This is not a template cell, and it should have 4 I/O Connections.

Place the Instance Reference Coordinate above the device, and place the Component Value Reference Coordinate inside the device.

Allow 10 lines for the Data Attribute file, and place the following data tokens into this file:

4 Pin DIP Package
Created By:
Date:

Press Okay to exit the Data Attribute Editor.

The pin type/name attributes are not useful for PCB type cells, because they are physical devices.

Define the four pin connection points, and PROCAD will save this device into the TUTOR library.

Execute the **[Other...]Hold Selection** command (or *Cntrl-Z*) to suspend the SELECT mode.

Now call the device IC04 from the TUTOR for placement into the database. Notice that this cell is assigned the reference designator of U1.

The first thing which should be apparent to you is the fact that the pad cells which you added to this device seem to be missing. They are not missing, however, they are nested into the device.

SET NEST 2 <ENTER>

To view up to the second level of hierarchy of the database. This command, followed by **Refresh**, should make the padmasters visible.

This points out a very important concept of hierarchical design.

A cell which is placed into another cell though the CELL FORMATION command is automatically nested one-level deeper into the hierarchy.

In plain English, this means that the [PKG]PAD cells, which you used in defining the IC04 symbol, were nested into that new cell one level deeper than they started.

This occurs through the CELL FORMATION command, and applies to all cells which are used in the formation of any other cells.

GENERAL HIERARCHY:

There are two reasons to nest information into a cell: logical and graphical.

In schematic designs you want to nest logical information in a cell most often for simulation purposes. If, for instance, you wanted to simulate the performance of a Flip Flop at the gate level you would have to nest gate level information into that device.

In PCB designs you want to nest graphical information into the cells so that you can speed the screen refresh and view the complete information only when it is required.

There are also two techniques used in viewing nested information in a design database: **SHOW HIERARCHY** and **SET NESTING**.

SHOW HIERARCHY (accessible as **[Library]Cell Attribute** menu item) is used to view the cell hierarchy of an individual cell. This command does not allow any editing, it is only to view cell structure. This command is generally used to view schematic hierarchy.

SET NESTING (accessible also from the **[Other...]Constants** dialog box) allows you to view the nested information of an entire database, and edit the database at the same time. This command is generally used to view PCB hierarchy when the design is nearly complete.

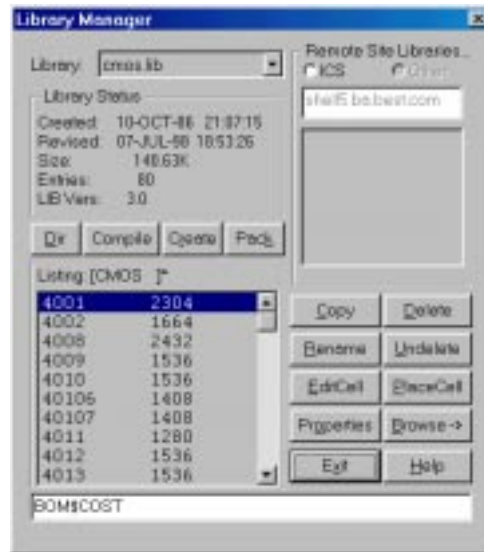
Remember, a cell stored within another cell is automatically nested one level deeper into the cell hierarchy. All other entities (lines, text, wires...) must be placed on Layers 40-48 to be nested into the cell hierarchy.

LIBRARY MANAGEMENT

The libraries are the files in which all cells must be stored. A library may contain a single cell, or up to approximately 2000 cells.

There are proper techniques to managing libraries which will help make PROCAD more efficient. There is a Library Manager utility within PROCAD to make proper management a simple task. The Library Manager is accessed by typing the following command from within a PROCAD database **LIB <ENTER>** or accessing the **Lib_Mgr** command from the Ribbon bar or **[Library]** menu.

This brings up the following dialog window:



The following commands are available within the library manager:

Create	[CR]
Compile	None
Delete	[DE]
Copy	[CO]
Rename	[RE]
Change_Lib	[CL]
Directory	[DI]
List Files	[LF]
Browse	[BR]
Disp_Attr	[DA]
Pack	[PK]
Undelete	[UN]
Edit Cell	[ES]
Place Cell	[GE]
Exit	[EX]

These commands function as follows:

[CR] is used to create a new library. The new library will be empty, and may be used to store user-defined symbols. All libraries are assigned a file extension of .LIB.

[DE] is used to Delete cells from libraries. To delete a cell you must use the following command syntax:

DE [library]cell <ENTER>

where **[library]** is the name of the library in which the cell is stored, and **cell** is actual cell name.

Note: The use of the square brackets [] acts as a delimiter of the library name from the command and cell name. If you are specifying the library name within the syntax of any command, you must bracket the name.

[CO] allows you to copy a cell in one of two ways: to another library, or to the same library under a different name.

To copy a cell from one library to another use the command syntax:

CO [library1]cell [library2] <ENTER>

where **[library1]** is the source library, and **[library2]** is the destination library.

To copy a cell from into the same library under a different name:

CO [library]cell1 cell2 <ENTER>

where **cell1** is the original cell name, and **cell2** is the new cell name.

[RE] allows you to rename an existing cell within a library. Use the syntax:

RE [library]cell1 cell2 <ENTER>

where **cell1** is the original cell name, and **cell2** is the new name. This command is very similar to the **COPY** command; however, this command simply renames a cell without duplicating it.

NOTE: Be careful renaming cells. This does not update pointers in the database which contain cell information. If you rename a device, PROCAD will not automatically update your designs.

[CL] this command allows you to change the active library.

PROCAD allows only one active library at a time; however, you may work with all libraries simply by specifying the library name upon command execution.

[DI] displays a listing of all cells in the library, and the cell's status.

To obtain a hardcopy listing of a library, you may execute the directory command, and select file output. The file generated may later be viewed and printed from DOS. You may also direct the output to your printer directly, without creating a listing file..

[LF] displays a DOS directory files listing. This is similar to the DIR command in DOS. Wild card characters "?", "*" etc are recognized.

[BR] browses through the graphical representation of cells in a library.

[DA] this allows you to view and edit a cell's Data Attribute file while in the library manager.

[PK] will PACK the library, removing all deleted cells.

NOTE: A deleted cell will remain in the library until the library is packed, it is simply unavailable for placement into a database. Once the library is packed, the cell is irreversibly gone.

[UN] allows you to recover deleted cells in a library, until the PACK command is issued.

[ES] Invoke the cell editor to edit the hi-lited cell. This command is equivalent to the **[Library]Edit/Create** cell command, using the selected cell as an argument.

[GE] Place the selected (hi-lighted) cell into the drawing. This is equivalent to the **[Place]Get Cell...** command, using the selected cell as an argument.

[EX] leaves the Library Manager to return to the design database.

The **SHOW DIRECTORY** and **DISPLAY ATTRIBUTE** commands may also be executed from the design database to achieve some of the above result.

The **PACK** and **UNDELETE** command may also be used from the design database, but operate on the database instead of the libraries.

You have now completed the study of CELL FORMATION and LIBRARY MANAGEMENT. With this information you should be capable of creating and managing even the most complex libraries.

Hardcopy Outputs

C*chapter*

5

Hardcopy Outputs

Topics Covered:

- INTRODUCTION
- PLOT
- PROPLOT
- FINAL ARTWORK

INTRODUCTION

PROCAD offers hardcopy output to a variety of devices, including wet ink plotters, and graphic printers. PROCAD also allows output to Gerber Photoplotters, and Fire 9000 Laser plotters.

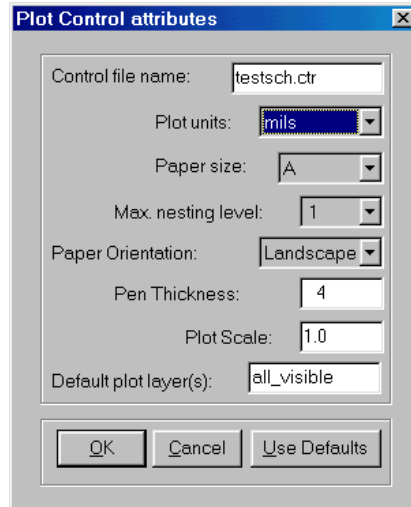
The plotter outputs are files which may be generated in the Hewlett Packard Graphic Language (HPGL), the Houston Instrument Drafting Machine Plotter Language (DM/PL), or in a standard Virtual Device Interface (VDI) format. Using PROPLOT, these files are then passed through a standard RS-232 device and cable to the plotter for hardcopy generation.

Hardcopy from a plotter can be used as a quick and cheap check plot using nylon tip pens, or can actually (in some plotters) be generated using liquid ink on mylar (or good vellum) for actual photographic reduction and use in manufacturing.

Output to graphic printers can be generated through PROPLOT using a VDI format with certain set-up parameters directing the output to a printer device through a standard parallel port (LPT1 or 2).

PLOT

To generate output from PROPLOT, you must provide a Plot Control or Setup file which must be generated from within the design database in PROCAD by using the **PLOT** command accessible from **[File][Print or Plot]Setup Ctrl file...** menu item.



PROCAD prompts you for the following information in a dialog box:

Control File Name {Filename.CTR}

This is the name of the output file. The default is the name of the database, with the extension .CTR.

Plot Scale (1.00X):

This is the scale at which the plotter will output the database.

Paper Size (A,B,C,D,E,User_def) [A]?

This represents the size of the plot paper.

Database Units (Mils,Inches,MM,CM)[Mils]?

The units of the database to output.

Paper Orientation ?

Choose between Landscape (X-axis longest) or Portrait (Y-axis longest).

Note: If you are using windows drivers in ProPLOT to generate your hardcopy, you must also set the orientation at the drivers level. Refer to PROPLOT section in the User's and Utilities manual for details.

Plot Layers (All_Visible)?

To define which layers to plot.

Remember, you must plot the correct combination of layers for output. For instance, to generate a complete output of the Component side of a PCB, you must plot Layers 5, 13, 16, & 20.

Plotting All_Visible layers relates to the defined value of the SET VISIBLE_LAYERS command. This value may not be exactly what you see displayed on your graphics screen.

Pen Thickness [4]?

This defines the thickness of the pen's tip, in the units of measure which were previously defined.

Maximum Nesting Level (1-15) [n]?

n is the defined value of the SET NEST_LEVEL command of the database. In PCB design, you will generally define the nesting level at 2.

Position the Paper Template:

PROCAD allows you to position the output onto a paper template. This provides greater flexibility at the plotter. Press <ENTER> when you have positioned the template.

Hint: You may have to Zoom Out in order to see the whole paper template. Apply the "Scaled Zoom (Screen Scale), with Scale factor -5 or -10" command before issuing the PLOT command.

Enter Plot Window:

You will define the Plot Window by entering the two opposing corners of a box. Anything which lies within the boundary of this Plot Window, will be output to the plotter. This allows you to define only certain parts of a design to be output if desired.

Plot Control File Generated.

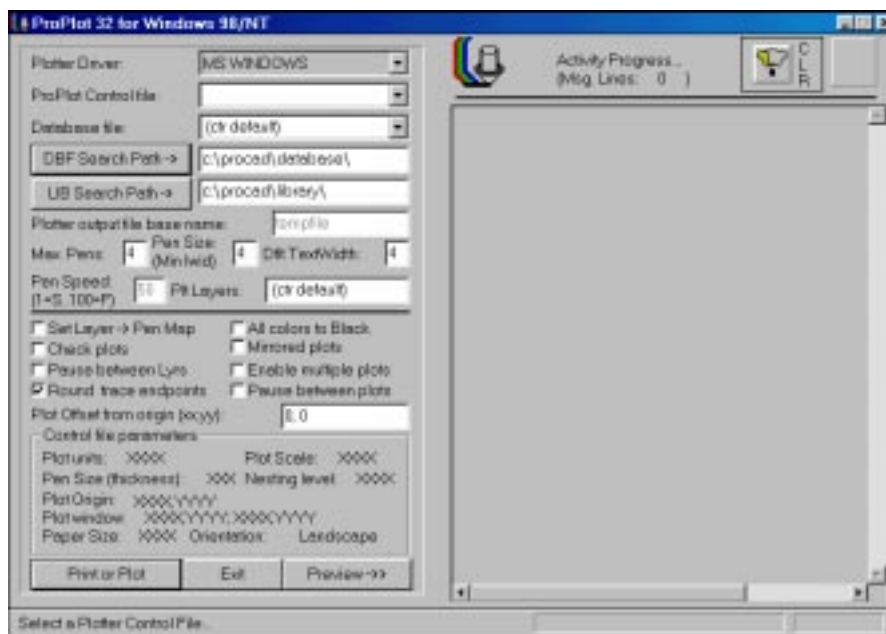
The actual generation of the Plot Control file may actually take less time than entering the preceding parameters.

The Plot Control file contains the defined parameters which PROPLOT utilizes to sort through the design database for plot generation. The actual database file, as well as all libraries utilized in the design, must be present, in the proper sub-directories defined in PROPLOT database & library search paths, for proper execution of PROPLOT.

PROPLOT

Execute PROPLOT from the Tool bar or Windows Program Manager.

Note: You may have to exit PROCAD and execute PROPLOT from the Program Manager if you do not have enough memory to execute it from within PROCAD (typically when you get MSLANGLOAD error message).



Respond to the following PROPLOT prompts as follows:

Enter Plotter Driver (WINDOWS, HPGL, DM/PL, etc):

HPGL is the Hewlett Packard Graphic Language, DM/PL is the Houston Instruments Drafting Machine Plotter Language, and WINDOWS uses the Windows installed hardcopy driver.).

Select the type of output driver which you would like to utilize.

Both the HPGL and DM/PL are available as internal drivers, or as an EXTERNAL WINDOWS driver. However, only through selections HPGL and DM/PL can you generate multiple plot files on disk. in a single PROPLOT session while processing multiple artwork layers.

The advantage of plotting to disk is that you may carry the plot file, with PROPLOT, to a distinct system to generate the hardcopy. This method of plotting frees the design workstation for use in designing.

Enter Plot Control File Name (.CTR):

This is the file generated by the PLOT command from within PROCAD.

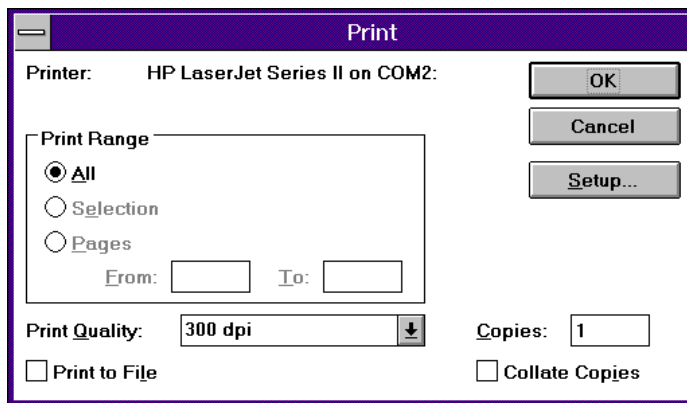
PROPLOT examines the Control file, and echoes certain parameters of the plot.

If you select either the HPGL or DM/PL drivers, you will also be prompted with the question:

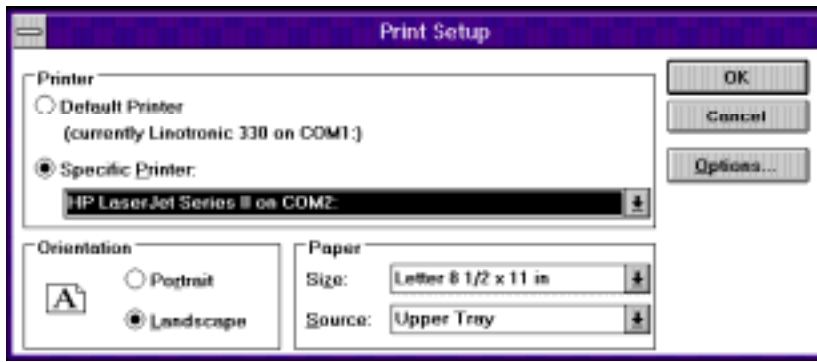
Generate Plot File or Send to Plotter?

This command allows you to tell PROPLOT to generate a Plot file to disk, or send a Plot file to the output device. In any case, if you are plotting the database for the first time, you must generate the Plot file before you can send it to the output device.

If you selected WINDOWS drivers (default), a printer/plotter specific dialog box will be displayed when you click on the Run button.



Remember to use the **Setup...** button in this dialog box to invoke a secondary dialog box from which you may select the proper paper orientation (usually landscape) to match that which you selected when you generated the plotter control file in PROCAD.



FINAL ARTWORK

PROPLOT provides you with a means of generating draft quality output for check plots, or schematics. In many cases, if care is taken, you may actually obtain production ready artwork from a PCB database, by plotting with ink on mylar. However, for the highest quality, and resolution output, you should utilize Interactive CAD Systems' Gerber plotting utility, GERPLOT.

GERPLOT is ICS' off-line Gerber Photoplotter utility, which will generate output files to disk to control either a Gerber or compatible Photoplotter, or the Fire 9000 laser plotter. Both of these plotters provide high-resolution, production ready artwork masters.

This page is intentionally left blank.

APPENDIX A

INTRODUCTION:

This command reference section is organized alphabetically by command name. A list of the commands arranged by functional groupings is presented in the "PROCAD FUNCTIONAL REFERENCE" section of the User's Guide. All Command syntax are shown in both the C and P shells. You should also read Chapters 1 of PROCAD User's Guide for a discussion of PROCAD System Overview & Command Structure.

MOUSE BUTTON ASSIGNMENTS:

ProCAD for windows supports the re-assignment of the left & right mouse buttons for entity selection or last command repeat and Escape or PopupMenu respectively. You may toggle these modes from the **[View]Preference...** dialog box. When Left mouse button is enabled for entity selection, it behaves like most windows applications that you are probably familiar with. In this mode also, the Shift-Left Button click will toggle selection of entity, and also after an entity is selected, click dragging the left mouse button again puts ProCAD in entity(s) move mode in real time (drag move).

Note: If you setup your left mouse button in entity selection mode, the middle button becomes the repeat button. If you are not using a 3 button mouse in this mode, you may NOT be able to repeat the last ProCAD command just by clicking on a mouse button.

On a three button mouse, the middle button will operate if and only if you installed the manufacturers supplied drivers for Windows. Make sure you are not emulating Microsoft mouse, since Microsoft drivers only support two mouse buttons.

KEYBOARD SHORTCUTS:

The "**PgUp**", "**PgDn**", "**Home**" and "**End**" keys performs **Zoom in (x2)**, **Zoom out (x0.5)**, **Pan** and **Refresh** screen functions respectively. These keys may be re-programmed (using the **[View]Preference...** menu item) to perform diagonal cursor movements. Also, the "**Del**" key may be used for deleting an entity. If the Del key is pressed when the mouse cursor is over a horizontal or Vertical line/wire segment in select environment, that segment only is deleted. If the cursor was on a vertex of a selected line/wire, that vertex only is deleted. If the cursor is outside the vertex or segment of a selected wire/line, the whole entity will be deleted. *Note: When the Del key is pressed while nothing is selected, ProCAD prompts for an entity to delete. Once the entity is identified, it is deleted regardless of where the cursor was placed to select it.*

Key Definitions for PCB Layout Mode (Using PKG.CMD)

ProCAD Advanced for Windows

F1 VIEW HELP TOPICS
F2 PAN
F3 REFRESH
F4 SNAP ON GRID
F5 PLACE VIA
F6 PLACE COMPONENT
F7 VIEW SOLDER SIDE LAYERS
F8 VIEW COMPONENT SIDE LAYERS
F9 VIEW CELL REF./CONN. POINTS

SHIFT F1 RE-ROUTE TRACE
SHIFT F2 SHOW SUMMARY
SHIFT F3 MOVE TO NEW LAYER
SHIFT F4 DELETE ENTITY
SHIFT F5 COPY ENTITY
SHIFT F6 MOVE ENTITY
SHIFT F7 ROTATE ENTITY 90°
SHIFT F8 DELETE VERTEX
SHIFT F9 MOVE VERTEX

CNTRL F1 STITCH TRACE
CNTRL F2 CHANGE/ASSIGN SIGNAL NAME
CNTRL F3 CHANGE/VIEW INST. ATTRIBUTES
CNTRL F4 CLOSE CURRENT WINDOW
CNTRL F5 MOVE INSTANCE REFERENCE
CNTRL F6 MOVE TO NEXT MDI WINDOW
CNTRL F7 SELECT KIND MASK
CNTRL F8 SELECT 10 X 10 GHOST GRID
CNTRL F9 SAVE WORK IN PROGRESS

NOTE: ProCAD ADVANCED FOR WINDOWS DOES NOT SUPPORT F10, SHIFT-F10, CTRL-F10 & ALT. FUNCTION KEYS. THESE KEYS ARE RESERVED FOR WINDOWS USE.

ProCAD Xtra for DOS

F1 VIEW F1-F10 KEY DEFS
F2 PAN
F3 SET VIEW
F4 REDRAW
F5 ZOOM IN
F6 ZOOM OUT
F7 SNAP GRID
F8 REFRESH
F9 SELECT ENTITY
F10 SELECT WINDOW

SHIFT F1 VIEW SHIFT F1 - SHIFT F10 DEFS
SHIFT F2 SHOW SUMMARY
SHIFT F3 MOVE TO NEW LAYER
SHIFT F4 DELETE ENTITY
SHIFT F5 COPY ENTITY
SHIFT F6 MOVE ENTITY
SHIFT F7 ROTATE ENTITY 90°
SHIFT F8 DELETE VERTEX
SHIFT F9 MOVE VERTEX
SHIFT F10 RE-ROUTE TRACE

CNTRL F1 VIEW CNTRL F1 - F10 DEFS
CNTRL F2 CHANGE/ASSIGN SIGNAL NAME
CNTRL F3 CHANGE/VIEW INST. ATTRIBUTES
CNTRL F4 CHANGE/RE-ASSIGN INST. NAME
CNTRL F5 MOVE INSTANCE REFERENCE
CNTRL F6 SELECT WORKING LAYER
CNTRL F7 SELECT KIND MASK
CNTRL F8 SELECT 10 X 10 GHOST GRID
CNTRL F9 SELECT 20 X 20 GHOST GRID
CNTRL F10 MAIN MENU

ALT F1 VIEW ALT F1 - ALT F10 DEFS
ALT F2 STITCH TRACE
ALT F3 INPUT WIRES
ALT F4 INPUT TEXT
ALT F5 PLACE VIA
ALT F6 PLACE COMPONENT
ALT F7 VIEW SOLDER SIDE LAYERS
ALT F8 VIEW COMPONENT SIDE LAYERS
ALT F9 VIEW CELL REF./CONN. POINTS
ALT F10 SAVE WORK IN PROGRESS

PCB LAYER ASSIGNMENTS

LYR	TYPE/FILL	USE	COLR	CODE
1	SLD/OFF	WORK LYR #1 DBASE	YLW	[5]
2	SLD/OFF	WORK LYR #2 CELLS	RED	[2]
3	SLD/OFF	ASSEMBLY DWG TOP	GRN	[3]
4	SLD/OFF	ASB_DWG BOTTOM	GRN	[3]
5	SLD/ON	PADMASTER	CYN	[6]
6	SLD/ON	SLDR_MSK(TOP)	GRN	[3]
7	SLD/OFF	DRILL_TARGETS	CYN	[6]
8	SLD/OFF	DRILL_CODES	RED	[2]
9	SLD/ON	SILKSCREEN (TOP)	WHT	[1]
10	SLD/OFF	SILKSCREEN (TOP)	WHT	[1]
11	SLD/OFF	CARD_OUTLINE	GRN	[3]
12	SLD/ON	CARD_DIMENSIONS	GRN	[3]
13	SLD/ON	TARGETS_& MARKS	BLU	[4]
14	SLD/OFF	NOTES_FABRICATION	CYN	[6]
15	SLD/OFF	NOTES_ASSEMBLY	GRN	[3]
16	SLD/ON	CONNECTOR_FINGERS	YLW	[5]
17	DOT/OFF	User_defined	RED	[50]
18	SLD/ON	User defined	RED	[2]
19	SLD/OFF	LOGO/DWG BORDERS	RED	[2]
20	SLD/ON	TOP SIDE TRACES[L1]		
		SMD TOP PADS	YLW	[5]
21	SLD/ON	LYR 2 VCC {L2}	[RVSE]	[5]
22	SLD/ON	LYR 3 GND {L3}	[RVSE]	[5]
23	SLD/ON	SLDR_SIDE TRACES{L4,n}		
		SMD BOTTOM PADS	BLU	[4]
24-29	SLD/ON	LAYERS #6-9 {L6-L9}	YLW	[5]
30-34		RESERVED FOR DRC		
35	SLD/ON	SMD SILKSCRN (TOP)	WHT	[1]
36	SLD/ON	SMD SLDR_MSK (BOT)		
37	SLD/ON	SMD PAST MSK (TOP)	WHT	[1]
38	SLD/ON	SMD PAST MSK (BOT)	WHT	[1]
39	SLD/ON	SMD SILKSCRN (BOT)	WHT	[1]
40-48	SLD/OFF	NESTED INFORMATION		
49	SLD/OFF	RESOLVE ERRORS		
50-98		USER DEFINED		
99	SLD/OFF	HIGHLIGHTING	WHT	[129]

LAYER COMBINATIONS FOR PCB ARTWORK GENERATION

COMPONENT SIDE - (Layers 5,13,16,20),
SOLDER SIDE - (Layers 5,13,16,23),
ASSEMBLY DWG - (Layers 3,4,10,13,15,19)
FABRICATION DWG - (Layers 7,8,11,12,13,14,19)
VOLTAGE PLANE - (Layers 13,21)
GROUND PLANE - (Layers 13,22)
SOLDERMASK - (Layers 6,13)
DRILL MASTER - (Layers 7,13)
SILKSCREEN - (Layers 9,10,13)
INNERLAYER n? - (Layers 5,13,n(24..29))

The PKG.CMD macro sets up the following default entity layer placement configuration:

ENTITY TYPE	PLACEMENT LAYER
Cells (Components)	1
All other (trace, circle, rectangle, etc)	Current active layer

SCHEMATIC LIBRARY LAYER ASSIGNMENTS

LYR	TYPE/FILL	USE	COLR	CODE
1	SLD/OFF	WORK LYR & TEXT	RED	[2]
2	SLD/OFF	SCHEMATIC SYMBOL	GRN	[3]
3	SLD/ON	SCHEMATIC SYMBOL	GRN	[3]
4	DOT/OFF	SCHEMATIC SYMBOL	GRN	[51]
5	SLD/OFF	CONNECTION DOTS	BLU	[4]
6	SLD/OFF	WIRE CONNECTIONS	YLW	[5]
7	SLD/ON	LINES(non-elect.)	RED	[2]
8	DAS/OFF	LINES(non-elect.)	RED	[34]
9	SLD/OFF	TEXT (non-elect.)	YLW	[5]
10-18		USER DEFINED		
19	SLD/OFF	LOGO/DWG BRDRS	GRN	[3]
20-29		USER DEFINED		
30-39		RESERVED		
40-48	SLD/OFF	NESTED INFORMATION		
49	SLD/OFF	RESOLVE ERRORS	RED	[2]
50-98		USER DEFINED		
99	SLD/OFF	HIGHLIGHTING	WHT	[129]

Key Definitions for Schematic Entry Mode (Using CKT.CMD)

ProCAD Advanced for Windows

F1 VIEW HELP TOPICS
 F2 PAN
 F3 REFRESH
 F4 SNAP GRID
 F5 PLACE CONNECTION DOT
 F6 PLACE OFF SHEET CONNECTOR
 F7 PLACE IN SHEET CONNECTOR
 F8 PLACE GND SYMBOL
 F9 PLACE VCC SYMBOL

SHIFT F1 RE-ROUTE WIRE
 SHIFT F2 SHOW SUMMARY
 SHIFT F3 STRETCH WIRE
 SHIFT F4 DELETE ENTITY
 SHIFT F5 COPY ENTITY
 SHIFT F6 MOVE ENTITY
 SHIFT F7 ROTATE ENTITY 90°
 SHIFT F8 DELETE VERTEX
 SHIFT F9 MOVE VERTEX

CNTRL F1 VIEW CELL REF./CONN. POINTS
 CNTRL F2 CHANGE/ASSIGN SIGNAL NAME
 CNTRL F3 CHANGE/VIEW INST. ATTRIBUTES
 CNTRL F4 CLOSE CURRENT WINDOW
 CNTRL F5 MOVE INSTANCE REFERENCE
 CNTRL F6 MOVE TO NEXT MDI WINDOW
 CNTRL F7 SELECT KIND WIRE
 CNTRL F8 SELECT KIND CELLS
 CNTRL F9 SAVE WORK IN PROGRESS

NOTE: ProCAD ADVANCED FOR WINDOWS DOES NOT SUPPORT F10, SHIFT-F10, CTRL-F10 & ALT. FUNCTION KEYS. THESE KEYS ARE RESERVED FOR WINDOWS USE.

ProCAD Xtra for DOS

F1 VIEW F1-F10 KEY DEFINITIONS
 F2 PAN
 F3 SET VIEW
 F4 REDRAW
 F5 ZOOM IN
 F6 ZOOM OUT
 F7 SNAP GRID
 F8 REFRESH
 F9 SELECT ENTITY
 F10 SELECT WINDOW

SHIFT F1 VIEW SHIFT F1 - SHIFT F10 DEFS
 SHIFT F2 SHOW SUMMARY
 SHIFT F3 STRETCH WIRE
 SHIFT F4 DELETE ENTITY
 SHIFT F5 COPY ENTITY
 SHIFT F6 MOVE ENTITY
 SHIFT F7 ROTATE ENTITY 90 DEGREES
 SHIFT F8 DELETE VERTEX
 SHIFT F9 MOVE VERTEX
 SHIFT F10 RE-ROUTE WIRE

CNTRL F1 VIEW CNTRL F1-F10 DEFS
 CNTRL F2 CHANGE/ASSIGN SIGNAL NAME
 CNTRL F3 CHANGE/VIEW INST. ATTRIBUTES
 CNTRL F4 CHANGE/RE-ASSIGN INST. NAME
 CNTRL F5 MOVE INSTANCE REFERENCE
 CNTRL F6 MOVE SIGNAL NAME REFERENCE
 CNTRL F7 SELECT KIND WIRE
 CNTRL F8 SELECT KIND CELLS
 CNTRL F9 VIEW CELL REF./CONN. POINTS
 CNTRL F10 MAIN MENU

ALT F1 VIEW ALT F1 - ALT F10 DEFS
 ALT F2 INPUT BUSS LINE
 ALT F3 INPUT WIRES
 ALT F4 INPUT TEXT
 ALT F5 PLACE CONNECTION DOT
 ALT F6 PLACE OFF SHEET CONNECTOR
 ALT F7 PLACE IN SHEET CONNECTOR
 ALT F8 PLACE GND SYMBOL
 ALT F9 PLACE VCC SYMBOL
 ALT F10 PLACE SCHEMATIC SYMBOL

The CKT.CMD macro sets up the following default entity layer placement configuration:

<u>ENTITY TYPE</u>	<u>PLACEMENT LAYER</u>
Cells (Components)	1
Wires(traces)	6
Text(Non-electrical)	9
All other (lines,circles, rect. etc)	Current active layer

DRC ERRORS LAYER ASSIGNMENT:

<u>Violation</u>	<u>Flag Layer</u>
Trace to Trace	31
Pad-to-Trace	32
Pad-to-pad	33
Shorts	34

USE OF SPECIAL KEYS:

PROCAD uses the CNTL-C and ESC keys for interrupting most PROCAD operation, eg. during screen repaint, an ESC or CNTL_C will interrupt that operation. In some instances, you might have to issue the ESC twice. CNTL-C is a harder interrupt than ESC, however, you should use it with extreme caution.

Some PROCAD command prompts enclose default values in brackets "[]" or default coordinate with a marker. If the default value in brackets is to be maintained, merely press the ESC key or <RETURN>. To maintain the coordinate default at the marker position, you must use the ESC key.

Index Section

Index

I

Index

A

ADD VERTEX, 99
Attribute Data File, 120
autorouter, 99

B

Bill-of-Materials, 55, 60
Board Limits, 76
Board outline, 76
Bus Structures, 38

C

CARD_OUTLINE, 73
Cell Editor, 132
cell formation, 115
CELL FORMATION, 119
cell hierarchy, 89
Cell Reference Coordinate, 121
Cell Reference Point, 24
CHANGE LAYER, 99
CHANGE LIB, 117
CHANGE LIBRARY, 23
CHANGE SIG, 39
Change Signal name, 33
Change Width, 44
CHANGE WIDTH, 99
CKT.CMD, 19
Close, 48
Close All, 48
Command Line, 10
component placement, 86
COMPONENT SIDE, 73
Component Value, 121
Connection Data File, 56
connection file, 85

Connectivity, 38, 88
 Common Signal Names, 36
 Common Vertexes, 36
 Hierarchical, 137
Copy, 32

D

Data Attribute files, 115
Data token, 106
data tokens, 61, 115
Database Type, 18, 70
Database Units, 18, 70
DELETE SEGMENT, 99
DELETE VERTEX, 99
Design Rule Check, 45, 107
DM/PL, 151
DOS directory, 147
DRAG, 87
DRC, 107
 CLEAR DRC, 48, 110
 Domain_check, 46, 109
 DRC Layer, 46, 108
 DRC\$PAD, 110
 Floating input check, 46
 Pd-to-Pd, 47, 109
 Pd-to-Tr, 47, 109
 Save Errors, 109
 Save Markers, 46
 Shorts, 47, 109
 Shorts_check, 46, 109
 Tr-to-Tr, 47, 109

E

Exit, 48

F

Fire 9000 Laser plotters, 151
function keys, 22, 76

G

General Hierarchy, 143
Generic Netlist, 58
GENERIC Netlist, 54
Gerber Photoplotters, 151
GERPLOT, 157
Ghost Grid, 11
ghost_grid, 89
GND, 73
Grid, 11
Grid System, 11

H

Hewlett Packard Graphic Language, 151
hierarchical cell, 131
Hierarchy
 General, 143
Hierarchy, 131
 Connectivity, 137
 PCB, 139
 Reserved Layers, 133
 Schematic, 131
Hold, 123
horizontal route, 94
Houston Instrument Drafting Machine
Plotter Language, 151
HPGL, 151

I

I/O Connections, 120
INPUT/WIRES, 29
INSHEET connector, 31
Instance_name, 121
INTERNAL layers, 73

J

Justification Coordinate, 34

K

Keepout Areas, 79

L

layering conventions, 20, 72
library management, 115
Library Manager, 144
 Pack, 147
 Printing, 147
Lines, 78

M

manual routing, 87
modifying routed traces, 98
mouse, 7
 three button mouse, 7
 two button mouse, 7
Move, 24
MOVE VERTEX, 98
Multiple Sheet Schematic, 55, 62

N

Nesting Level, 89
Net Names, 33
New, 48

O

Open, 48, 69
OPEN, 17
optimize, 86
output, 151

P

PACK, 64
PACK/R, 64
Pack/Renumber, 28
Package Pin Numbers;, 127
Packaging Data File, 56
packaging file, 81

padmaster, 141
PADMASTER, 72
padstack, 90, 140
PAN, 26
PCB Clean-up, 111
PCB Hierarchy, 139
PCB Layering Convention, 72
Pin Editor, 122
placing components, 81
PLOT, 151
Plot Control, 151
PRINT, 57
PROCAD, 6
PROCAD Database Screen, 9
PROCAD Title Screen, 7
PROCAP, 51
 Menu, 53
PROPLOT, 151
PROROUTE, 79, 99
 abort, 106
 Control file, 102
 Graphic Display, 105
 Lookout Strategy, 100
 Memory, 100
 multilayer routing, 104
 PASS, 102
 Power/Ground Bus, 100
 pre-routes, 100
 RATS NEST, 105
 routing passes, 99
 six passes, 102
 status line, 106

Q

Quick Edit, 9, 27, 35
Quick Reference Guide, 159
QUICKSET, 5

R

RAT\$DEVn, 129

RAT\$GND1, 129
RAT\$PKG, 129
RAT\$TCNT, 129
RAT\$VCC1, 129
Rats Nest, 56, 61, 81, 85, 105
RATS NEST, 73
Rats nest usage enforced, 71
reference designators, 24
regular cell, 116
relative coordinate input, 77
REROUTE, 98
RUBBERBAND, 86

S

SCALE, 10
Schematic Function Key, 23
Schematic Hierarchy, 131
Schematic Layering Convention, 20
Schematic Sheet, 19
Schematic Sheet Size, 19
Screen Display Factor, 18, 70
Screen Menu, 12
Select Filter, 31
Select Instance, 59, 63
Select Kinds, 31
SELECT KINDS, 91
SELECT mode, 123
SELECT WINDOW, 126
SET COLOR, 75
Set Layer, 28
SET MODE, 63
SET NESTING, 144
SET TRUE_WIDTH, 107
SET VIS, 90
sheet prefix, 62
SHO SIG, 42
Show Attributes, 60
SHOW HIERARCHY, 138, 144
SHOW LAYER, 75
SHOW SIGNAL command, 37

Simulatable cell, 120
simulation netlists, 55
SOLDER SIDE, 73
Status Line, 10
Step & Repeat, 40
STI B, 91
STI T, 93
STITCH, 92
 parameters, 98
 via cell, 98

T

TEMPFILE.DBF, 7
template cell, 124
Thermal relief, 141
toolholes, 80

V

value, 27
VCC, 73
VDI, 151
Vertex Recognition, 38
vertical route, 94
Virtual Device Interface, 151

W

WORK/PLACEMENT LAYER, 72

X

X-Y Coordinates, 11

Z

Zoom, 47