

User's & Reference Guide

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

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INTRODUCTION

The User's Manual provides a thorough understanding of the application of PROCAD for schematic design, netlist extraction, printed circuit board design, and special advanced topics. The special topics included in the manual provide greater detail on subjects like using surface mount components in a PCB design, Ground and Power ties in PCB and schematic designs, Multi-sheet and Hierarchical schematic designs, Building buss signals, Productivity enhancement and expert modes of operation, Creating connectors etc.

If you want specific information about a specific command, you may find it in the Reference section of this manual. You will also find specific information about the various support programs which make up the PROCAD design system in this manual.

The User's Manual is organized in the following fashion:

Installation & Configuration - This section covers the installation and configuration of PROCAD on your hard disk.

Systems Overview - An overview of the structure and concepts of PROCAD design editor. This section also includes definition of some of the terms used throughout the User's Manual.

Command Summary - A breakdown of commands which are grouped by types of tasks which are of interest when working within the PROCAD Design Editor. Specific functions such as screen display control, manipulating information within the design, and managing cell libraries is described in this section.

Quick Start Tutorial - A quick tutorial which introduces you to the process of using PROCAD as a vertical design workstation. This section covers the design of a simple Schematic, Netlist extraction and Interactive/Automatic PCB design. Some design concepts are introduced in this section.

Utilities - This section of the manual describes the many utility programs which complete the PROCAD Design System. Utilities such as PROCAP, which is used for extracting netlists from completed schematic designs, and GERPLOT, which is used to generate photo plotted artwork from a completed printed circuit board design, are discussed in detail. The section also includes topics on how to create and edit library cells (symbols), Using the design Rule checker, Creating tear-off menus, and the special topics mentioned above.

Reference - This section of the manual offers the user a complete reference of all commands available within the PROCAD Design Editor, as well as how to execute these commands from within the editor.

Appendices - The appendices include listings of library and layer conventions, as well as providing information about creating input files for some of the previously mentioned utility programs.

Installation & Configuration

BEFORE YOU START

Please take a moment and read the ICS license agreement. Fill it out and mail in your registration card. Your completed registration card entitles you to technical support, disk replacement, plus notification and discounts on future ICS products and updates.

PROCAD installation diskette contains a file named README. This file contains updated information about using PROCAD along with additional notes and changes not reflected in this manual. Review the beginning section of the **README** file for any pertinent changes to the installation procedures. **A moment spent reading the README file and this installation guide will save you hours (or may be days) of frustration !!!**

SYSTEM REQUIREMENTS

- IBM PC/AT/PS2 or compatible computer
- 80386 or higher processor
- 16M bytes RAM (32M bytes recommended)
- Math Co-processor (Optional, but recommended)
- 1 Floppy Disk Drive (Floppy distribution Version)
- 20MB Hard Disk Minimum (You need approx. 15MB to load PROCAD & Utilities)
- GRAPHICS Card & Monitor (SVGA or higher resolution strongly recommended)
- DOS 3.0 or higher (16bit Windows Version)
- MS Windows 3.1 or higher (16bit Windows Version)
- DOT Matrix printer (Optional)
- Optical or Mechanical Mouse (Optional, but strongly recommended)

PROCAD 16 for Windows requires that your MS Windows be configured for **Enhanced** mode operation (default).

INSTALLING PROCAD ON YOUR HARD DISK

Installing PROCAD PowerStation 32 for windows involves two steps.

STEP 1: PROGRAM FILES & DATA INSTALLATION

STEP 2: COPY PROTECTION (AUTHORIZATION CODE) INSTALLATION

You should use the SETUP.EXE program provided in Red installation disk#1 (Floppy version) or your CD-ROM (CD-ROM version) for installing the software. If you are installing on Windows 95 or 98, the setup program automatically installs the 32 bit version of ProCAD. If however you are installing on Windows V3.1, 3.11, NT or OS/2, you have the option of selecting 16 bit or 32

Getting Started...

bit code installation. Please note that the 32 bit code will not operate correctly on windows 3.1 or 3.11. Also on OS/2 systems, 32 bit code (WIN32) may not operate correctly. If this occurs, you should re-install the 16 bit version.

If you purchased the Authorization check (Copy-protection) bypass option or installing the CD-ROM version, you will not need to install the authorization code from the Configuration program. The buttons for installing, un-installing and Reset authorization code will be grayed (disabled) if you purchased the copy-protection bypass option.

Call Interactive CAD Systems or your dealer for details. **Note: Floppy version copy-protection bypass option is only available in USA and CANADA.**

If you did not purchase the Copy Protection bypass option, and you are installing the floppy version, you must perform Installation Step #2 (**Authorization Code Installation**). Use the **"Install Authorization"** button of the installed WCONFIG.EXE (ProCAD Configuration ICON) utility to perform the second step. You may also use the WEVMOVE.EXE program provided on the installation disk for performing the second step above (See the section on "RECALLING PROCAD" discussed later in this chapter for details).

STEP1: PROGRAM FILES & DATA INSTALLATION

Insert the installation (red label) disk in drive A or B (Floppy version), or your CD on your CD-ROM drive (CD-ROM version), and issue the following command:

A:SETUP <ENTER>

Note: You may also use the **[File]Run...** menu item of the Program Manager to execute the above command from within windows. Substitute your drive letter for "A:" above.

Respond to all installer prompts appropriately to install ProCAD executable and data files. The installer will also create the appropriate ICONS in the "ProCAD Advanced for Windows" group.

Note: The installation program will create a \WPROCAD, \WPROCAD\BIN, \WPROCAD\DATABASE and \WPROCAD\LIBRARY sub-directories on your hard disk. Your WIN.INI, Windows System files, or Windows Registry are NOT altered.

STEP2: AUTHORIZATION (SECURITY) CODE INSTALLATION (FLOPPY VERSION ONLY)

Note: The installation disk must be inserted in drive A or B, and MUST NOT be write protected before performing the Authorization (security) code installation. The installation disk is copy protected.

Getting Started...

Next, double click on the installed WCONFIG.EXE (ProCAD Configuration) ICON. Use the **"Install Authorization"** button of this utility program to perform step #2 above. (See the next section for details on the use of the Configuration utility).

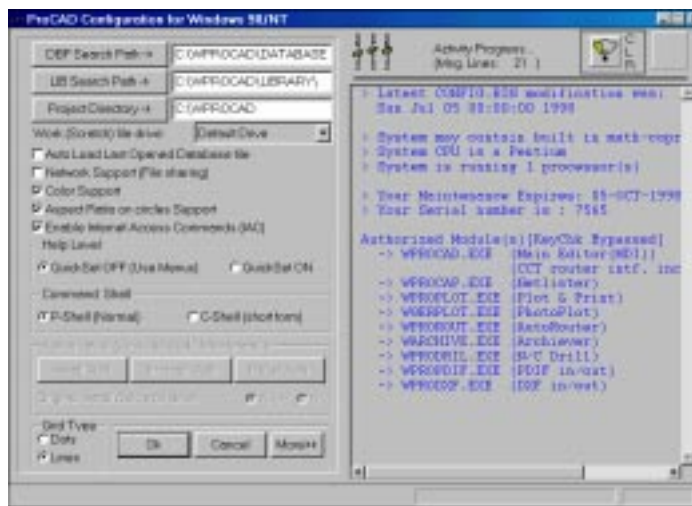
CREATING ProCAD START UP(config.bin) FILE & AUTH. MAINTENANCE

After installing PROCAD on your hard disk, you will need to configure some of its default starting parameters. The WCONFIG.EXE program is provided to aid you in this endeavor. You may also use the WCONFIG.EXE utility for performing install/un-install of security code and maintenance of security code parameters, such as resetting execution count limit (floppy evaluation versions) and restoring lost install counts. The WCONFIG.EXE is a windows program, which you should invoke by double clicking on its icon in the PROCAD PowerStation 32 group.

The WCONFIG utility will generate the necessary CONFIG.BIN start up file. This program also lists the PROCAD Utility programs which you have purchased or authorized to execute in your computer.

Note: If you attempt to start PROCAD without first generating the CONFIG.BIN file, PROCAD will automatically invoke the WCONFIG.EXE utility.

When you invoke the WCONFIG utility, the program start by displaying its copy right message. After a brief moment, the following window will be displayed:



The following is an explanation of the various prompts produced by the WCONFIG utility program:

Getting Started...

Install Authorization (Floppy version)

NOTE: If you encounter KEYCHK error 7002 when you perform any of the authorization maintenance functions (Install Authorization, Reset Authorization etc) or when you execute ProCAD or any associated utilities, you should install the provided AZMEMORY.DRV driver. Please refer to "INSTALLING AZMEMORY.DRV" section below for details.

This allows for the installation of the security authorization code onto your hard disk. You must select your source floppy drive (A or B drive radio button), and insert the **Original** installation disk (red label) onto it prior to clicking on the "Install Auth." button. You will be prompted for confirmation before the authorization code is moved onto your hard disk. **Note: The Authorization code will be installed on the same drive and directory where the Configuration program (WCONFIG.EXE) resides, typically \WPROCAD\BIN directory. Make sure the installation disk is NOT write protected.**

Un-Install Authorization (Floppy version)

This allows for the un-installation of the security authorization code from your hard disk back to the original installation floppy. You must select your source floppy drive (A or B drive radio button), and insert the **Original** installation disk (red label) onto it prior to clicking on the "Un-Install Auth." button. You will be prompted for confirmation before the authorization code is moved back to the floppy disk. **Note: The Authorization code will be un-installed from the same drive and directory where the Configuration program (WCONFIG.EXE) resides, typically \WPROCAD\BIN directory back to the original installation (red label) floppy. Make sure the installation disk is NOT write protected.**

Reset Authorization (Floppy version)

This allows for the re-initialization of some security parameters of the original installation disk, such as the execution counts (Evaluation version) and lost install counts. You must select your source floppy drive (A or B drive radio button), and insert the **Original** installation disk (red label) onto it prior to clicking on the "Reset Auth." button. You will be prompted for a reset code.

You must call Interactive CAD Systems or your dealer for the reset code. Please make available the product identification number (this will be displayed when you are being prompted for the reset code) to ICS or your dealer. The Product identification number will enable ICS or your dealer to generate the proper reset code for you.

Getting Started...

Note: You must insert the Original Installation disk (red label) in drive A or B before issuing the above command. Make sure the installation disk is NOT write protected.

WARNING: IF YOU ISSUE THE ABOVE COMMAND REPEATEDLY (SAY 10 CONSECUTIVE TIMES) WITH THE WRONG RESET CODE, A "7072" ERROR MESSAGE WILL BE GENERATED & YOUR FLOPPY DISK WILL BE BLOCKED. YOU WILL NEVER BE ABLE TO RESET ITS PARAMETERS. !!!

DBF Search path

This is the search path that PROCAD and all its utilities will use when locating a PROCAD database file. You should enter your database file search path here. You may also change the search path with the SET DATAPATH command while in PROCAD. When modified in PROCAD, the CONFIG.BIN file is NOT changed, as a result, the next time you invoke PROCAD, it will use the data path that you have specified here.

LIB Search path

This is the search path that PROCAD and all its utilities will use when locating a PROCAD library file. You should enter your library file search path here. You may also change the search path with the SET DATAPATH command while in PROCAD. When modified in PROCAD, the CONFIG.BIN file is NOT changed, as a result, the next time you invoke PROCAD, it will use the data path that you have specified here.

Project Directory

This item specifies the directory where all project related data will be stored and loaded from. Items such as ProCAP outputs, GERPLOT outputs etc will be stored in this directory by default. Note: Database and Libraries are not stored in this directory. They are stored in the directory set by the Database and Library Search paths respectively.

Work (Scratch) file drive

If you plan to use soft disk (also referred to as VIRTUAL or RAM DISK) for PROCAD generated temporary and swapper files, Select the drive letter. A "default drive" selection will cause PROCAD to use the current drive.

YOU MUST HAVE ENOUGH WORKING SPACE ON YOUR SOFT DISK TO HOLD THE TEMPORARY FILES GENERATED BY PROCAD, OR YOU MAY LOOSE YOUR DATABASE !!!!!. PROCAD REQUIRES APPROXIMATELY 128 BYTES OF DISK SPACE PER ENTITY IN THE DATABASE. DON'T USE RAM DISK IF YOU HAVE LESS THAN 1MB RAM DRIVE IN YOUR COMPUTER, OR IF YOU WILL BE OPENING A LOT OF DATABASE FILES AT THE SAME TIME (MDI VERSION).

Auto load last opened database file

Enable this box if you want PROCAD to automatically load and open the last database you were working on when you exited PROCAD. If this database could not be found, or if you are invoking PROCAD for the first time, TEMPFILE.DBF will be opened or initialized.

Network support (file sharing)

Enable this box if you plan to use PROCAD in a network environment where library file sharing is desired. If you don't plan to share library files, you should not enable this box, since PROCAD runs slightly faster if it does not have to load the over head code required for file sharing. This option is only available for Multiple Document Interface (MDI) version of PROCAD for windows.

Color Support

Enable this box if you have a color adapter and monitor installed.

Aspect ratio on circles

If you are using a non unity aspect ratio monitor and controller, circles will appear oblong when aspect ratio adjustment is turned off. When the aspect ratio for circle is turned on, circles will appear round, but will appear shorter in the Y-axis relative to the displayed grid.

NOTE: Circles & Donuts will be plotted correctly irrespective of this flag setting. This flag only affects the appearance of the display. When Aspect ratio is not preserved, Donuts will be drawn as two concentric circles on the display.

Help Level

PROCAD supports two help levels. The default, QUICKSET OFF, causes PROCAD to always prompt for coordinates. This is the preferred option if you plan to use menus. The second level, QUICKSET ON, causes PROCAD to always assume the cursor position to be the first coordinate when certain commands are issued. This method usually yields greater productivity for those who are well acquainted with PROCAD, and prefer to work mostly with function keys. See the section on QUICKSET for details.

Display Grid type

If you are using a monochrome or color adapter that does not support programmable color intensity/gray level control, you should use the dotted grid type, otherwise, you should use the line grid type. **Line grids redraws a lot faster than the dot type.**

Getting Started...

Enable Internet Access Commands (IAC)

If Internet access commands are enabled, ProCAD will allow for commands that require Internet access such as Remote library directory/downloads, Remote auto routing, etc. **If this flag is enabled and you issue an Internet access command in ProCAD without having an Internet Service Provider (ISP), ProCAD may freeze-up for several minutes while trying to make an internet connection that does not really exist.** It will eventually timeout several minutes later. You should turn this flag off if you do not have an Internet Service Provider (ISP) or you don't plan to use any Internet Access Commands in ProCAD.

If you enable Internet Access commands, you may click on the "**More>>**" button to enter your internet access properties such as your e-mail address, desired remote ftp site logon information (ftp username and password for non-ICS ftp sites), etc.

RECALLING (UN-INSTALL) PROCAD FROM HARD DISK (FLOPPY VERSION)

You can only have two copies of the working program at any one time. If you want to recall your installed system from a hard disk, you should insert the original INSTALLATION DISKETTE in drive A and either use the WCONFIG.EXE utility to un-install the security code (See the previous section), or issue the following command using the **[File]Run...** menu item of Windows Program Manager:

A:\WINMOVE C:\WPROCAD\BIN A: <ENTER>

The protection scheme will be moved from your hard disk back to the original diskette. Note that files are NOT moved. They will remain on your hard disk, but will become non-functional. To re-enable the system on hard disk, issue the command:

A:\WINMOVE A: C:\WPROCAD\BIN <ENTER>.

When recalling PROCAD from the hard disk it is a good idea to execute the recall program, "**A:\WINMOVE C:\WPROCAD\BIN A: <ENTER>**", more than once to insure that all counts of the copy protection have been removed from the hard disk. The "**un-install**" command of the WCONFIG.EXE program will give you feedback on the current (or remaining) installation counts on your original installation disk. Attempting to recall PROCAD off of the hard disk when it is not installed will not do anything except generate a harmless error message.

Note: The above command assumes your floppy drive is A and ProCAD & its utility executables (.EXE files) are installed in C:\WPROCAD\BIN directory. Make appropriate substitutions if necessary.

YOU SHOULD ONLY RECALL PROCAD TO THE ORIGINAL INSTALLATION DISKETTE !!!.

You must recall PROCAD from the hard disk before you can re-install it on other hard disks.

IMPORTANT NOTICE

KEEP THE ORIGINAL DISKETTES IN A SAFE PLACE. YOU WILL ALWAYS NEED IT WHILE TRANSFERRING PROCAD BETWEEN SYSTEMS.

NOTICE TO USERS OF THE FLOPPY EVALUATION VERSION:

The floppy evaluation version of PROCAD PowerStation 32 for Windows has **50 INVOCATION LIMIT**. This means that you may exit from PROCAD back to Windows Program Manager or DOS a total of fifty times. There are no limits to invocation of the associated utility programs, such as ProPLOT, ProCAP, etc., so long as ProCAD is still functional in normal mode. ProCAD will function in Demonstration mode (No save) after the 50th use.

You may only install the Full Function Evaluation software on one machine at a time.

INSTALLING AZMEMORY.DRV

You may not install AZMEMORY.DRV on Windows NT or OS/2. For these operating systems, you must purchase the Copy-protection removal option or install the CD-ROM version. Intl customers must contact ICS for special configuration files for NT or OS/2.

If you did not purchase the Copy-protection bypass option and you encounter keycheck error 7002, this means that you are lacking lower DOS memory due to excessive TSR's or Network Drivers (WFWG V3.11). If this occurs, you may want to install the provided ProCAD windows driver AZMEMORY.DRV by following the steps below:

OVERVIEW

AZMEMORY.DRV is provided to virtually eliminate problems if you are encountering low-memory related problems using WINMOVE or EVCHK routines (ProCAD Configuration Install, un-install or Reset commands).

This native Windows driver allocates a block of memory that will be reserved for the use of ProCAD authorization check and installation routines. Once it has been installed, it is automatically loaded whenever you start Windows.

Getting Started...

AUTOMATICALLY INSTALLING AZMEMORY.DRV

You should perform the following listed steps:

1. Go to Control Panel and double click on the Drivers icon.
2. The Drivers dialog box will then be displayed on the screen. You should then click on the Add button.
3. The Add dialog box will then be displayed on the screen. You should then double-click on Unlisted or Updated Driver.
4. The Install Driver dialog box will then ask for the disk with the driver. You should then insert the Red Installation disk (disk #1) in drive A or B and choose the drive. The files are in the root directory of disk #1 (**A:** or **B:**).
5. After choosing the appropriate drive and directory, Windows will display the drivers found. You should choose **Az-Tech Software Memory Driver** by double-clicking on this, or by selecting the driver and clicking the OK button.
6. Windows will then inform you if it has successfully installed the driver. If it was installed, Windows will ask you if you want to restart Windows. You may choose to continue or to restart Windows. The AZMEMORY.DRV will not be installed until windows has been restarted.

MANUALLY INSTALLING AZMEMORY.DRV

You may also manually install the driver by completing the following steps:

1. Copy the AZMEMORY.DRV file from the red installation disk (disk #1) to the \WINDOWS\SYSTEM subdirectory.
2. Edit the SYSTEM.INI file and add the following entries:
 - a).. In the [boot] section of the SYSTEM.INI, edit the drivers line to include AZMEMORY.DRV. If there are other entries on this line, the AZMEMORY driver should be first. This will make sure the driver is loaded first. After editing the line, it should look something like this:

```
drivers=azmemory.drv mmsystem.dll
```

- b). In the [drivers] section of the SYSTEM.INI add the line

```
AzMemory=azmemory.drv
```

3. Save the SYSTEM.INI after the specified changes have been made. You must restart Windows for the AZMEMORY.DRV to be loaded.

UNINSTALLING AZMEMORY.DRV

Uninstalling AZMEMORY.DRV is just as easy as the installation was. You must complete the following steps to remove AZMEMORY.DRV:

1. Double-click on the Control Panel icon in the Main group.
2. Double-click on the Drivers icon. A list of the installed drivers will appear.
3. Select the **Az-Tech Memory Driver** from the driver list and click on the Remove button. A confirmation window will be displayed. Click on the YES button to remove the driver.
4. The driver will not be loaded the next time Windows is started.

NOTE: This procedure only stops Windows from loading AZMEMORY.DRV. You must delete AZMEMORY.DRV from the \WINDOWS\SYSTEM subdirectory to remove the file from the hard disk.

Give Technical support a call if you are having installation difficulties.

Systems Overview

*C**hapter*

1

Systems Overview

INTRODUCTION

PROCAD is an interactive graphics editing program that allows you to create and revise drawings. The data structure of PROCAD has the intelligence required for the design and capture of electronic Schematics, Printed Circuit board and Integrated Circuit Layouts, Hybrid designs, PLD designs and a host of other electronic designs. The flexibility and editing power of the program also allows for non-electrical uses, such as mechanical, architectural and structural designs.

DATABASE

Operation of PROCAD is done within a database file, for creating or changing a drawing. PROCAD stores the drawings in a file with ".DBF" extension. The Database is the environment into which the user will actually place a design. Sometimes the words database and design will be interchangeable.

The database is the computer generated sheet of paper which the design engineer will use as a medium for his concepts and his artwork. Sometimes there may be several different designs on a single sheet of paper, and sometimes there will only be one.

Like a real sheet of paper the computer database has physical limits which must be adhered to in order to create effective designs; however, unlike the real paper, the computer database can be changed and restructured constantly, allowing the PROCAD user to revise and improve the design as he likes, taking output from plotters only when the end result is achieved.

DBU: The PROCAD database has no physical limitation until the user defines them. The database units (DBU) must be selected when first creating a database, and then the physical limitations are set.

The PROCAD database is based on a Cartesian Coordinate grid system with four quadrants. The entire area of this grid is 64,000 grid units by 64,000 grid units, or a 64K database. When selecting units to work in, the possible choices are MILS (one thousandth of an inch), INCHES, MICRON (one millionth of a meter), MM (millimeter), or CM (centimeter) with the default choice being MILS.

Once the database unit has been selected, the actual physical limitations and resolution of the database have been defined. If, for instance, the user selects a grid unit of MILS the database will have a defined physical value of 64 inches by 64 inches (or 64,000 MILS by 64,000 MILS) with a cursor movement resolution of one MIL. If the user selects a grid unit of

INCHES, the database has a physical value of 64K inches by 64K inches with a cursor movement resolution of one inch.

NOTE: It is very important for the user to be aware of these considerations when selecting database units to work with. Using MILS as the default allows the user to achieve a high resolution of cursor movement (one mil) in a large design area.

ENTITIES

All objects placed within a database or drawing are composed of one or more of the following entities:

***LINE:** A path defined by two or more points with a width of 1 or more units.

***WIRE:** A path defined by two or more points with a width of 1 or more units and a user or program assigned signal name.

***RECTANGLE:** A four-sided polygon defined by two points (opposite corners).

***POLYGON:** A n-sided closed polygon.

***AFILL/GND PLANE:** Special polygon used for automatic ground plane (copper pour) generation with user defined clearance values and net name.

***CIRCLE:** Defined by a center point and a point on the circumference.

***ARC:** Defined by a starting point, ending point and a point on the arc.

***DONUT:** A circle with width, defined by the inner and outer diameter and center point.

***TEXT:** Vectorized text defined by a starting point, size, angle and ASCII characters. True type fonts may also be used in Schematic design mode (ProCAD for Windows only).

***CELL:** Combinations of the above entities grouped together as one entity and stored in a library.

LAYERS: The PROCAD database has ninety nine layers to do design work in, all prealigned on the coordinate grid, and moving from one layer to the next is as easy as selecting a menu command or typing a few letters on the keyboard. This allows the user to create very complex multilayer designs and maintain alignment throughout the design process.

Some of these layers have special functions within the PROCAD design system, and some attention must be given to using established layering standards.

Getting Started...

To view layer information, such as name, color, and number, execute the command **SHOW LAYERS** from the command line or the **[Layer]ShoColor...** or **[Layer]Layer Attribute...** menu items.

CELLS and CELL LIBRARIES

Cells are predefined groups of entities which can be placed in the drawing. PROCAD has the ability to create and edit cells. The cells are stored in a cell library. These libraries can be edited and manipulated with library maintenance commands. When placing a cell into the database, PROCAD actually only places a reference to the cell in the database. This process requires that all libraries for that database be maintained with the database on disk. This is not difficult as usually one or two libraries will be sufficient for most of your design work.

This method of data storage also means that the user can create some very large and complicated designs with effective database memory management. For instance a device such as a two-input NAND gate may take up to 512 bytes of memory as stored in the library, while the same device placed in the database will only require 20 bytes of memory.

CELL 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$DESCN  DUAL D FLIP FLOP
    RAT$PKGO   [PKG]IC14A
    RAT$PKG1   [PCBLIB]FP14
    RAT$VCC1   14
    RAT$GND1   7
    RAT$TCNT   2
    RAT$DEVA   6 5 4 3 2 1
    RAT$DEVB   8 9 10 12 11 13
    SWP$GATES  A,B
```

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

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

The RAT\$XXXX 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. Refer to Appendix A for a list of data tokens currently supported by ICS products.

PROCAD support three category of cell types:

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 bidirectional 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.

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.

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.

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 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. You may access this command from the **[Library]Cell Hierarchy** menu item.

SET NESTING 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. You may access this variable from the **[Other...]Constants...** dialog box.

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.

Tips: Since library names are part of the database, you should NEVER use the DOS rename or copy command to rename PROCAD cell libraries. You should use the optional ARCHIVE utility for that purpose.

LINKING SCHEMATIC TO PCB DESIGNS

While most other CAD systems requires separate editors for Schematic Capture, PCB layout, etc., PROCAD is a general purpose graphics editor with intelligence for designing both Schematics, PCB layout, Mechanical drafting, etc. The editor operates in two major modes: Schematic Mode and PCB mode. The link between both modes is handled by use of KEYWORDS in the schematic components data attributes.

SCHEMATIC -> PCB LINKS

The data attribute used for linking a schematic symbol to one of four PCB decals (footprints) is the RAT\$PKGn data attribute. The format for this attribute is as follows:

RAT\$PKGn [LIBNAME]CELLNAME

Where **n**=0,1,2 or 3. If you omit "n", ProCAD assumes the default package assignment (n=0). From above, you may assign up to 4 various footprints references for the symbol.

Getting Started...

[LIBNAME]CELLNAME is the footprint cell name.

Example:

RAT\$PKG0	[PKG]IC14A
RAT\$PKG1	[PCBLIB]DIP14
RAT\$PKG2	[MYLIB]FP14X
RAT\$PKG3	[MYLIB]FP14

Note: The RAT\$PKGn data keyword must be part of the data attribute for the Schematic Symbol. You may specify these attributes in any order; They don't have to be specified sequentially or even grouped together, although for clarity, you may want to group them together.

PCB -> SCHEMATIC LINKS

The data attribute used for linking a PCB design to schematic design (back annotation) is the SWP\$PINS & SWP\$GATES data attributes on the Schematic symbol. The format for these attributes is as follows:

SWP\$GATES A,B,....
SWP\$PINS n1,n2,...

These attributes allows for specifying which pins and gates of a schematic device are swap able. See the **SWAP** command in Reference section of this manual for details.

Note: These attributes must be specified in the Schematic Symbol (NOT PCB) data attribute. You must use PROCAD's MDI feature to load both the PCB and all corresponding Schematic files before a pin or gate swap operation can be performed.

CROSS PROBING

When you use the Project feature (or MDI) of PROCAD to load your PCB and all corresponding Schematic files, you may use the **[Check]Cross Probe...** command to probe Nets, Devices etc. between your Schematic and PCB database files. (See the CROSS PROBE command in the reference manual for details)

FORWARD & BACKWARD ANNOTATION

When you use the Project feature (or MDI) to load your PCB and all corresponding Schematic files, you may use the **[File]Annotate(pcb<>sch)...** command to perform forward and backward annotation of instance names and values changes. Complete ECO changes is handled in PROCAP (Netlister utility).

VERTEX RECOGNITION SYSTEM

The database in PROCAD is a vertex recognition system which allows the user to create very large complicated designs while maintaining low memory overhead.

PROCAD stores only vertex points, cell references, and justification points as references to stacked information. This method of data storage allows you to design very large schematics and PCBs, without worrying about system limitations. Because of the vertex recognition mechanism employed by PROCAD, when using any of the select or implied select commands, you must move the select cursor to a vertex of a wire, line or polygon, justification point of text, reference origin of a cell or any of the digitized points of a circle, donut, arc or rectangle entities for proper entity selection.

MODES and WORKING ENVIRONMENTS

There is one principle operating mode in PROCAD; the **MAIN** database mode. The **MAIN** mode is for creating and editing your drawing. Within this mode there are two sub-modes. One is the **EDIT CELL** mode for creating and editing cells. The other is the **LIBRARY MAINTENANCE** mode for creating and editing libraries cells.

The **SELECT** environment provides a means of selectively editing and manipulating certain specified entities within the MAIN database or the EDIT CELL modes. Entities to be selected are identified by using the SELECT ENTITY or SELECT WINDOW (BLOCK) commands. Selected entities are highlighted in bright white color. All editing manipulation commands can be performed on the selected entities (Move, Rotate, Stretch, Copy, Delete, Show Summary, Change Layer, Mirror, etc.). When you are through with editing the selected entities, press <ESC>, and PROCAD returns to the MAIN database editing mode.

When a non select environment command is issued while in select environment, eg: INPUT/WIRE, ProCAD automatically places selected items in hold mode, exit the select environment and then execute the command in main mode. You may return to the select environment (held items) by issuing a Select Entity or Select Window command.

Note: Exiting the select environment requires pushing the Cntrl-C button twice. A single push of the <ESC> button (or right mouse button) will also exit the select environment.

Getting Started...

MOUSE BUTTON ASSIGNMENTS

The following button assignments have been made for the mouse:

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.

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 drag the left mouse button puts ProCAD in entity(s) move mode in real time (drag drop move).

Note: If you setup your left mouse button in entity selection mode, the middle mouse 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.

Users already familiar with windows convention may want to keep this mode enabled in order to have a consistent user interface between other windows applications. This is the default setting. DOS die hard users may prefer to setup the left mouse button for repeat mode.

CONTEXT SENSITIVE COMMANDS

When the left mouse button is setup for entity selection mode (default), click dragging on a vertex of a selected wire or line will automatically enter Move Vertex mode, while click dragging on a segment of a selected 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. Also you may click drag a selected cells instance name (reference designator) or instance value or a selected wires signal name to move and possibly rotate it with respect to the entity. This is equivalent to issuing a limited subset of the **CHANGE INSTANCE**, **CHANGE VALUE** and **CHANGE SIGNAL** commands respectively. In main database mode, click drag will perform a **SELECT WINDOW (BLOCK)** operation, while click drag during a Zoom In operation will perform the **SET VIEW** operation.

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. 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 the entity.

CURSOR MOVEMENT KEYS

You may use the cursor keys (the arrow keys) to control the cursor movements (See "**KEYBOARD SHORTCUTS**" above). In order to control the movement multiples of the arrow keys, the following keystroke combinations may be used:

To increase the cursor movement increments x5:

Press "INS" key.

To increase the cursor movement increments x2:

Press "Ctrl->(6)"

x3: Press "Ctrl ->(6)" twice.

:

x10: Press "Ctrl ->(6)" x9.

To decrease the cursor movement increments -1:

Press "Ctrl <-(4)" once

-2: Press "Ctrl <-(4)" twice

:

-5: Press "Ctrl <-(4)" x5

To reset the cursor movement increments to the minimum cursor movement:

Press "INS" key.

Getting Started...

See the "**SET GHOSTGRID**" command for additional information.

NOTE: These control commands only apply when using the cursor movement arrow keys (2,4,6,8), NOT the Mouse or digitizer tablet.

USE OF SPECIAL KEYS

PROCAD uses the CNTL-C and ESC keys for interrupting most operations, eg. during screen re-paint, 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.

Note: The right mouse button WILL NOT interrupt any PROCAD operation. You should use the ESC key from the keyboard for interrupting any PROCAD operation.

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 <ESCAPE> key.

QUICKSET COMMAND NOTATION

PROCAD will always assume that the current cursor position is the first coord entry for most commands. This feature is referred to as QUICKSET. If you don't want PROCAD to always make this assumption, you can turn off this feature by issuing the command SET QUICKSET OFF, and PROCAD will always prompt for the first cursor location. You should leave QUICKSET feature turned off (default) until you are quite familiar with PROCAD.

See also the Special topics section of this manual and the WCONFIG.EXE utility "Help Level" setting in the Installation Section of this manual.

USE OF TRUE TYPE FONTS

ProCAD Advanced for windows supports the Windows True Type Fonts for Schematic design. You may select the desired font from the **[Other...]Constants...** menu item dialog box.

Note: ProCAD will revert to the normal vector fonts in PCB mode. True type fonts are currently supported for all text and label entities in schematic mode only. Note also that the Gerber and Pen plot (DMP/L, HPGL etc) output formats do not support true type fonts. Since these fonts are usually proportional spaced, auto reverting to vector fonts on output devices that do not support true type fonts will make for very poor and undesirable output results. Also, the width value specified for text and labels is ignored when true type fonts are used, since these fonts have a built in width size proportional to the height and font family used.

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Command Summary

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CHAPTER TWO**Command Summary****INTRODUCTION**

This chapter gives you a way to locate PROCAD commands that you can use to do a particular task. The major sections are arranged roughly in the order that you would normally do them. Within a section, various tasks are given, together with the command(s) that you would use to do those tasks. Chapter 3 is a Quick Start hands on tutorial which allows you to make use of the commands covered in this chapter for actually performing a simple schematic entry through PCB layout.

To make more effective use of this section:

- * Locate the major topic that applies to what you want to do. Major topics are listed directly below.
- * Find the particular task of interest. For some topics, there are subsections.
- * Follow the step-by-step procedures.
- * If you need a brief summary of a command, use the HELP command to receive on-screen assistance.
- * If you need more help, look up the command in The Reference Manual Section.

SETTING SYSTEM EDITING PARAMETERS

Preparing to begin an editing session by initializing various system parameters, including:

- *System variables*
- *The color/line type map*
- *Status commands*

PROCAD ENVIRONMENT

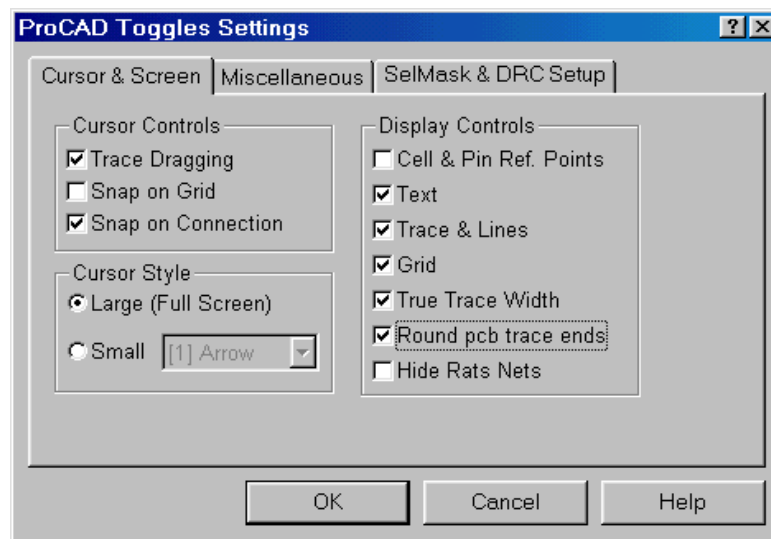
The PROCAD editor includes several parameters which may be defined in order to best configure this tool to your working habits. These parameters may be changed using two commands from the **[Other..]** menu, the **Toggle...** and **Constants...** commands.

The **TOGGLE** command allows you to access several parameters of the PROCAD editor which you may have cause to change from time to time as your design progresses. This command invokes a window of toggling parameters which may be selected by the cursor.

The **CONSTANTS** command allows you to access several parameters of the PROCAD editor which you will usually define only once when opening a new database. This command invokes a window of parameters which may also be selected by the cursor.

TOGGLE PARAMETERS

This command opens a window of program toggleable flags that could then be toggled with the mouse. This command will allow for toggling the following program flags:



PARAMETER	EQUIVALENT COMMAND
Auto Menu	SET MENU (DOS)
Auto Pan	SET AUTOPAN
Alternate Screen	SET ALTERNATE SCRN (DOS)
Cell & Pin Ref. Points	SET CELL REFERENCE
Ghost Grid Check	(None)
Grid Display	SET GRID_DISPLAY
Hide Rats Nests	(None)
Large Cursor	SET LARGE CURSOR
Logging File	SET LOG FILE
Menu Enable	SET MENU (DOS)
Online DRC	(None)

Quick Set	SET QUICKSET (DOS)
Rats nest Enforced	(None)
Round pcb trace ends	(None)
Rubber band	SET RUBBERBAND
Select Mask	SELECT KINDS
Small Cursor Style	SET CURSOR STYLE
Snap on Connection	SET SNAP CONNECTION
Snap on Grid	SET SNAP GRID
Text Display	SET TEXT DISPLAY
Trace Display	SET TRACE DISPLAY
Trace Drag	SET DRAG
True Width	SET TRUE WIDTH
Warning Bell	SET BUZZER

Note: The Toggles dialog box may include fields that do not have a corresponding command for toggling them. For these fields, you must set or clear them in the dialog window.

AUTO MENU - configures PROCAD to invoke the EDIT menu automatically when executing a select command. The EDIT menu, and its secondary EDIT2 menu, provide all of the data manipulation commands which may be used when an item or items are selected.

If you are used to working with PROCAD from the command line editor, or from the function keys interface, you will want to turn the AUTO MENU function off. However, if you are just learning PROCAD this feature will increase your productivity while decreasing the learning curve. Note: This item is always enabled in PROCAD for windows.

AUTO PAN - feature which controls the way in which PROCAD will pan the database screen. If AUTO PAN is toggled to the ON position, PROCAD will automatically pan the design as your cursor reaches the boundaries of the display area. This does not happen when the cursor reaches the upper limits of the display area due to the presence of the menus in this region. To use AUTO PAN in this direction approach the upper limits of the display area from either side of the screen.

This feature may be difficult to adjust to, or may not be effective on systems without EMS support (PROCAD *Xtra* for DOS), or on systems which use certain graphic display configurations. If you do not want to use AUTO PAN simply toggle this feature to the OFF position. PROCAD for windows users should use the scroll bar and **NOT** the Auto Pan feature.

CELL REFERENCE AND PIN NUMBERS DISPLAY - this allows you to view the reference points of all cells (and pin numbers in PCB mode) which are placed within the design. These references will allow you to quickly select a symbol in a crowded design database.

GRID DISPLAY - allows you to toggle the display of the visible placement or routing grid within the editor. It is recommended that this grid be displayed at all times. You may select Lines or Dot grid style using the WCONFIG.EXE startup utility (See the installation section for details).

LARGE CURSOR - enables the use of a large cross hair cursor for aligning multiple items in the design.

CURSOR STYLE - a series (from 0-5) of cursor types such as a cross-hair, arrow pointer, etc. This is only functional on certain types of graphical display configurations.

LOGGING FILE - creates a log file of command sequences and keystrokes which may be converted into a macro command through simple text editing.

RATS NEST ENFORCED MODE

This mode is selectable when a new PCB database is created. You may also enable or disable this mode through the "Toggles" dialog box accessible from the **[Other..]Toggles...** menu item. The default is **ON** for PCB databases. When this flag is set, you may **NOT** enter or stitch wires to a component that does not already have a net or rats net connected to it. You should select the net that you would be routing, then proceed to stitch or wire the route interactively.

When you push <ESC> to complete the route, ProCAD will verify that the new route satisfies the Net on hold, and if that is the case, the held net is deleted automatically.

NOTE: The auto net deletion only works if the net placed on hold has only 2 vertices (All optimized rat's nest satisfies this requirements since they only have 2 vertices; point-to-point).

NOTE: When rats nest enforced mode is enabled, deleting a routed trace will restore the original rats nest. Also, a warning message is issued if an attempt is made to delete or break a rats nest connection manually.

NOTE: Since a lot of checking is done to insure design integrity in this mode, it is strongly recommended that you always turn on this flag if you are designing your board from a netlist input (those from PROCAP or other third party schematic capture program).

GHOST GRID vs DISPLAY ZOOM CHECK

ProCAD for windows optionally checks the current resolution, magnification, display and ghost grid settings continuously as the design progresses. If it detects that the current ghost grid setting is not reliable for current screen magnification, a warning message is displayed and the magnification automatically adjusted. The formula used for this calculation is explained in the **CHANGE DISPLAY FACTOR** command section of the Reference section of this manual. If your cursor does not move in the increment desired (ghost or display grid settings), or your grid seems to shift at odd zoom magnifications, you should toggle this flag ON.

Note: With this flag on, SNAP-ON-DISPLAY GRID off, and you desire to work in cursor increments of 1MIL, your minimum display magnification will be 10X.

RUBBERBAND - toggles the real time, or animated movement of objects through the design database. With RUBBERBAND toggled to the OFF position PROCAD requires a starting point and an end point for editing operations like MOVE, COPY, and entering traces. With RUBBERBAND toggled ON, PROCAD allows you to enter a starting point and edit in real time.

SNAP ON CONNECTION - allows you to enter wires and traces quickly without worrying about completing actual connections. With this feature toggled ON, PROCAD will automatically connect a trace or wire to the nearest connection point in the design. This allows for rapid entry of connections into a design.

SNAP ON GRID - causes the cursor to be locked on to the visible, or display grid. With this feature turned OFF, PROCAD allows you to move the cursor in much finer increments defined by the Ghost Grid.

TEXT DISPLAY - allows the text entities in a design database to be displayed, or not displayed for a faster screen redraw.

TRACE DISPLAY - allows the trace, wire, line entities to be displayed, or not displayed for a faster screen redraw. There are also times when you may want to turn off trace display in a crowded design just to view the components, or other items in the design.

TRACE DRAG - maintains connectivity of wires and traces to components during certain editing operations such as: MOVE, ROTATE, SCALE, MIRROR etc. This feature will allow you to make changes to your design without risking connection errors caused by this process. This flag is sometimes referred to as "rubberbanding" in some other CAD systems.

TRUE WIDTH - with this feature toggled OFF all wires and traces are displayed with a width of 1 unit to speed the graphic performance of the editor. Turn this feature ON to see all traces displayed at their true width.

NOTE: PROROUTE, the automatic router, sets the TRUE WIDTH to OFF when routing a design database.

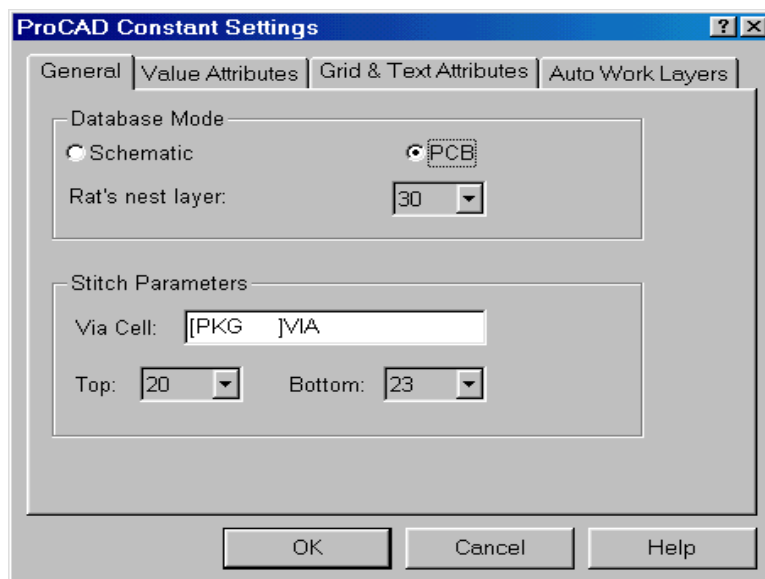
WARNING BELL - toggles the warning buzzer which sounds when an error is made during the design process.

SELECT MASK (FILTER) - allows a selection filter to be established within the design. This feature allows you to set all selection commands to function only on certain types of entities like cells, lines, text, and so on. This filter may also be set to exclude only certain items.

ON-LINE DESIGN RULE CHECKING - allows for setting real time design rule checking. When this flag is set, ProCAD performs design rule checking for clearance violations in real time. This flag should be toggled off if you notice speed degradation when placing wires into a design, or when exiting from the select environment. This flag is always off in Schematic mode.

CONSTANTS PARAMETERS

This command opens a window of program constants that can be changed or modified. This command will allow for setting or changing the following working constants:



PARAMETER	EQUIVALENT COMMAND
Layer Attribute	SET VISIBLE & SET FILL (DOS)
Layer Color	SET COLORMASK (DOS)
Auto Work Layers	SET AUTOLAYERS
Extent Limit	SET EXTENT LIMIT
Font Selection	(None)
Database Mode	SET MODE
Schematic Sheet #	SET MODE SCH
Set Pin # Display Size	(None)
Snap Connect limit	(None)
Nesting Level	SET NESTING LEVEL
Auto Save Constant	SET AUTO SAVE CONST
Fill Area Clearance	SET CLEARANCE
MSG Display Time	SET WAIT TIME
Trace width	SET WIDTH
Ghost Grid	SET GHOST GRID
Grid Increments	SET GRID_SPACINGS
Label Attribute	SET LABEL ATTRIBUTE

Note: The Constants dialog box may include fields that do not have a corresponding command for setting them. For these fields, you must enter the desired values in the appropriate fields in the dialog window.

LAYER ATTRIBUTES - In ProCAD for DOS, this menu item brings up a menu that allows you to define which layers are visible and which are filled. This is used to configure the PROCAD editor to display only the specific layers you are interested in viewing, as well as defining which layers should be filled for closed polygon objects.

This menu supports an ALL toggle under both the VLAYR and FILAYR columns which allows the entire layer grouping (1-99) to be toggled ON or OFF at one time. In order to toggle a single layer you must select either the ON/OFF attribute beneath the VLAYR column, or select the colored box beneath the FILAYR column.

*The Layer attribute dialog box is available through the **[Layer]Layer Attribute...** menu item in ProCAD for Windows.*

LAYER COLOR - In ProCAD for DOS, this menu item brings up a menu to define layer colors. This requires that you define a layer for color definition, the default provided by PROCAD is the current working layer (displayed in the Status Bar at the top of the screen).

This menu lets you define colors, linetypes, and display mode for all layers.

*The Layer color dialog box is available through the **[Layer]Layer Color...** menu item or clicking on the layer selection box on the Ribbon bar in ProCAD for Windows.*

AUTO WORKING LAYER - brings up a menu to define default placement layers for different entity types.

This menu lets you define which layers different entity types (Wires, Lines, Rectangle, Text etc) will be placed while in Schematic or PCB mode of operation.

EXTENT LIMIT - this feature allows you to define the number of pixels at which PROCAD will display cells and text entities as extents. For instance, if you don't care to see detailed outlines of small objects set this value at ten (or some similar value) and this will increase the screen redraw speed.

DATABASE MODE - defines the database as either a PCB or schematic design. This is used to change a database if you made an error when creating the design database. Do not change a schematic design to PCB mode, or vice-versa, this can cause significant problems due to connectivity.

NESTING LEVEL - allows you to define the display of hierarchical components with nested information. The concept of hierarchical components is discussed in greater detail in the CREATING COMPONENTS section of this manual.

AUTO SAVE - sets a time based on number of commands executed which PROCAD will automatically save the work in progress of your design. This value should be set at some value which approximates the value you set on your time. Do not set this number too small or PROCAD may keep interrupting your work in order to save the design. Set this value to zero in order to disable the AUTO SAVE feature.

MESSAGE DISPLAY TIME - determines the time delay used by PROCAD when displaying error messages.

TRACE WIDTH - default trace width that new traces will be defined with.

CLEARANCE VALUE - default clearance value used when AFILL/GND plane are placed (Automatic Ground Plane generation).

GHOST GRID VALUE - defines the non-visible, or small increment grid which will be used for routing wires and traces. This may also be changed using the **[Other...][Ghost Grids]** submenu.

GRID INCREMENT - defines the value of the visible grid which is used for placement of components. This may also be changed using the **[View][Display Grids]** submenu.

LABEL ATTRIBUTES - defines the default height, width and angle of text for all signal and component names, and text entities used for annotation and documentation. PROCAD uses this value when automatically assigning aliases to both signals and components. These values are also used as the defaults when you manually assign signal and component names.

RATS NEST LAYER - defines the layer where all rats nests are placed. This is typically layer 30.

SNAP CONNECTION LIMITS - defines the "hot" window for snapping cursor to connection I/O or wire vertex. If you are experiencing "Partial or incomplete Route Error" when rat's enforced flag is on, you may consider increasing the value of this variable.

STARTING AN EDITING SESSION

Beginning a session from the DOS prompt or Windows Program Manager. This section covers both creating a new database and editing an existing one.

EDITING THE DRAWING

The range of actions you can take to edit the database including:

- *Adding cells*
- *Adding entities (other than cells)*
- *Copying entities*
- *Changing existing entities (move, delete, rotate, mirror, etc)*
- *Changing attributes*

CONTROLLING THE DISPLAY

The commands that allow you to control the screen display, include:

- *Zoom in and out of the drawing*
- *Shifting the view*
- *Setting and recalling screens*

CREATING A CELL

The three methods of defining a new cell are:

- *Cell formation*
- *Automatic PadStack Cell generation*
- *The Edit Cell environment*

CUSTOMIZING THE USER INTERFACE

Creating your own interface to the system by:

- *Defining function keys*
- *Creating macro commands*

PLOTTING THE DISPLAY

Producing a hard-copy plot of a portion of the database. Use the PLOT command in PROCAD to generate a plot control file, then use PROPLOT utility to actually generate the plot.

ENDING AN EDITING SESSION

Terminating the editing session by:

- *Saving your edited database by overwriting the old file.*
- *Saving your edited database as a new database file.*

The general method of presentation is to give each step in the procedure, followed by the command(s) that allow you to do the step. For information on DOS commands, see your DOS manual. For information on WINDOWS commands, see your WINDOWS manual. For information on PROCAD commands, refer to the Command Reference Section, of the REFERENCE MANUAL.

SETTING SYSTEM EDITING PARAMETERS

Some of the editing parameters that you can change are given below, with their default values and the commands that affect them.

<u>PARAMETER</u>	<u>DEFAULT</u>	<u>COMMAND</u>	<u>MENU</u>
Color & Line Type	Varies	SET COL	[Layer]
Cursor Increment	x=10, y=10	SET GH=x,y	[Other]
Cursor Type	small	SET LAR=ON	[Other]
Digitizing Mode	Straight	SET DIG=x	[Place]
Grid Spacing	100,100	SET GRID=x,y	[Other]
Snap cursor to grid	ON	SET SN=OFF	Toggle
Working Layer	1	SET LAY=n	[Layer]
Default Library	CELLIB	CH LI=<lib>	[Library]

Getting Started...

Line and Wire Width 1 SET WI=nn Constants...

Menu ON SET ME=OFF Toggle...

Screen Scale OFF SC SC=nn [View]

Note: The **Constant** & **Toggles** command from the **[Other..]** menu is equivalent to executing the SET CONS & SET TOGG pop-up window commands respectively.

You may also construct macro commands to initialize system settings in various combinations. See the MACRO command in the Command Reference Section for an example of such a macro command.

EDITING THE DRAWING

Once you have the system variables set, you can edit a drawing. There are two main things you can do:

- * *Add entities to the drawing*
- * *Modify entities already in the drawing*

ADDING ENTITIES

There are three methods for adding entities to the drawing:

- * *Use INPUT command*
- * *Copy an entity within the drawing*
- * *Get a cell from the library (GET command)*

USING INPUT MODE [DRAWING TOOLBOX]

You can use INPUT command to add any of the following entities to the drawing by assigning one of the following input codes:

- * Lines (**L**)
- * Wires (**W**)
- * Rectangles (**R**)
- * Polygons (**P**)
- * Circles (**CI**)

Getting Started...

- * Donuts **(D)**
- * Arcs **(A)**
- * Text **(T)**
- * Cells **(CE)**
- * Connection Dots **(CO)**
- * Copper pour/GND plane/AFill polygon **(G)**

Step 1. Enter the INPUT mode.

Command: "IN/aa<RETURN>
aa == INPUT code above

Step 2. Enter information as prompted by the system.

Step 3. End the INPUT mode in certain instances.

Command: Press <ESC>

Note: You may also use the **[DRAWING]** menu (ProCAD for DOS) or Toolbar (ProCAD for windows) to accomplish the above tasks.

COPYING ENTITIES [EDIT MENU]

Step 1. Select the entity or entities to be copied using either the Select or Sel Block commands from the **[Edit]** menu (or Tool Ribbon button), or enter the following command from the keyboard:

Command: SEL E <RETURN> or
SEL W <RETURN>
then
COPY <RETURN>

or using the implied selection mode:

Command: COPY <RETURN>

Selection is implied by the current cursor location.

Step 2. Copy the entity(s).

Note: You can repeat this step as often as necessary.

Step 3. End the COPY command.

Command: Press <ESC>

MODIFYING ENTITIES [EDIT MENU]

Step 1. Select the entity or entities to be modified using either the Select or SelBlock commands from the menu, click or click drag a select window, or enter the following command from the keyboard:

Command: **SEL E** <RETURN> or
 SEL W <RETURN>

Step 2. Perform the modification. The commands that you can use to modify entities are enumerated in the reference manual. These commands are also included in the **[Edit]** menu which is invoked automatically when the select environment is entered.

Step 3. Exit SELECT Environment.

Command: Press <ESC>

THE SELECT ENVIRONMENT

The SELECT environment allows the individual the means of selectively editing the database. It is deserving of some additional explanation. An entity or entities can be placed into the SELECT environment with the use of two main commands; SELECT ENTITY (**Select**) to specify single entities, or SELECT WINDOW (**SelBlock**) to specify multiple entities. Additional entities can be added to entities selected with either command by the ADD ENTITY command. Of course, unwanted entities can be removed from the SELECT environment with the UNSELECT ENTITY (**UnSelect**) command.

While in the SELECT environment, the **[Edit]** menu will become active. You may use any of the commands in this menu for manipulating the selected entity or entities.

CONTROLLING THE DISPLAY [VIEW & LAYER MENUS]

There are two types of commands that control what you see on the screen without changing the database.

Getting Started...

* Screen View Commands: These move, redraw, or change the display scale of the viewing window.

* Layer Control Commands: These change the color and visibility of selected layers. These commands are in the **[Layer]** menu

SCREEN VIEW COMMANDS

You have a number of ways of controlling the view of the drawing. These commands which are enumerated in the reference manual can be invoked from the **[View]** menu.

LAYER CONTROL COMMANDS

You can control which layers are visible and how lines, rectangles, circles, etc. will be displayed.

ACTION

Display lines, rectangles,
circles filled/not filled
Set layers to be displayed/invisible

COMMANDS

SET FILL=n...

SET VIS=n n-n

You may also use the **[Layer]Attribute...** menu item to accomplish the above task.

CREATING A CELL

There are three ways of creating a cell which does not currently exist in the cell library:

* Using CELL FORMATION. The method is to select the entities that will make up the cell and give the CELL FORMATION command ("CELL" <RETURN>).

* Using the Edit Cell environment. You enter the environment by entering the EDITCELL command and giving a name for the cell to be created ("EDIT <cellname> <RETURN>"). All the normal editing commands are available under this environment. When you exit, the cell is saved in the cell library unless you choose otherwise. A complete explanation of the cell formation process is covered in Chapters 3-4 and 4-10.

* Using MAKEPAD command. This method is used for creating a pcb padstack cell automatically after filling in the desired size and values in a dialog style box.

USING EDIT CELL ENVIRONMENT

Step 1. Enter the EDITCELL command, EDIT <cellname> <CR> from the keyboard or **[Library]Edit/Create Cell** menu item.

Step 2. Respond to the prompts. If you omit the cell name, PROCAD will assume the cell in the drawing at the current cursor position.

Step 3. Create the geometry for the cell from any or all of the entities while in Edit Cell environment.

Commands: Any described under "USING INPUT MODE" above.

Step 4. Give the SAVE command to save the cell in the library and exit the Edit Cell environment.

Command: SAVE <RETURN>

Note: You can use the QUIT command if you don't want to save this cell. The system will prompt you for confirmation.

Step 5. Respond to the prompts. See EDIT CELL section of this manual for a description of the prompts.

MANAGING THE CELL LIBRARIES

Step 1. Enter Library Maintenance mode by executing the **LibMgr** command from the **[Library]** menu or entering the following command from the keyboard:

Command: LIB <RETURN>

Step 2. Perform the desired action.

ACTION**COMMANDS**

Copy cells from one library to another

COPY

Create a new library (empty)

CREATE

Remove cells marked for deletion

PACK

Getting Started...

Display cell attributes	DISPLAY ATTRIB
List cells in a specific library	DIRECTORY
Mark cells for deletion	DELETE
Recover cell(s) marked for deletion	UNDELETE
Rename cells in specific library	RENAME
Browse through cells in a specific library	BROWSE
Display directory of DOS files	LIST FILES
Change the default library	CHANGE LIBRARY

Step 3. Exit Library Maintenance mode.

Press <ESC>

MANAGING THE DRAWING [FILE MENU]

The following commands included in the **[File]** menu are useful for general purpose management of the database file:

<u>ACTION</u>	<u>COMMAND</u>
Recover drawing space from deleted entities	PACK/PURGE
Recover entities deleted but not permanently removed with PACK command	UNDELETE
Rename the drawing to save as a new database file (backup)	SET FILE SAVE AS
Save drawing for backup purposes without leaving the edit session	SAVE

CUSTOMIZING THE USER INTERFACE

There are several ways in which you can tailor the user interface to your own needs:

- * Defining function keys. These are normally single commands that you define during an editing session.
- * Creating macro commands. These are command sequences which you use often. A common use for a macro is to set up the initial editing conditions and parameters.
- * Creating your own Tear-Off menu structure. You may use the **DEFINE** command to create or modify the tear-off menu structure. PROCAD comes standard with a default menu file MENU.BIN which is automatically loaded when PROCAD is executed.

DEFINING FUNCTION KEYS

Step 1. Define function keys.

Command: **SET FU** nn=<string>

Step 2. Execute function key command by striking the function key.

NOTE: You can examine the current setting of a function key with the **SHOW FUNCTION KEY** command.

CREATING MACRO COMMANDS

Step 1. Create a command file using a text editor.

Command: **EDLIN** <filename.cmd>

NOTE: You must be in DOS to do this step. You cannot define new macros while in PROCAD.

See the **MACRO** command in the Command Reference Section for the syntax of the macro commands.

Step 2. Execute the command.

Command: **MACRO** {filename} <RETURN>

STATUS COMMANDS

You can interrogate the system to find out the current settings of certain system variables. You should use the **[Other...]Status Information...** menu item. Some of the variables that can be interrogated are:

<u>SYSTEM VARIABLE</u>	<u>COMMANDS</u>
Color Setting	SHOW COLORMASK
Space available & used in database	SHOW STATUS
Database units	SHOW STATUS
GHOST GRID increment	SHOW STATUS
Total entities in the drawing	Status Line
Information on selected entitie(s)	SHOW SUMMARY
Function key assignments	SHOW FUNCTION_KEY
Input digitizing mode	SHOW STATUS
Layers currently visible	SET VISIBLE_LAYERS=?
Scale Factor	SHOW STATUS
Rubber band mode	SHOW STATUS
Magnification	Status Line
Total cells in the database	SHOW STATUS
Working database file	Status Line
Working layer	Status Line (top)
Viewing screen	SHOW STATUS

PLOTTING THE DATABASE

Step 1. Set up the screen display so that the portion of the drawing that you want to plot is completely visible.

Commands: Use the **[View]Scale xN** menu item or any command described above under "Controlling the Screen Display".

Step 2. Prepare the plotter. This step varies depending on the type of plotter you have. Typical steps are:

- * Set the baud rate.
- * Load the paper.
- * Insert the pen(s).

Check the plotter manual to see how to do these things.

Note: Your system configuration file must be set up for your type of plotter. See PROPLOT in Utilities Section for a description of how to do this.

Step 3. Generate a plot control file (.CTR) for the desired part of the drawing using the PLOT command.

Command: **PLOT**

See the Reference Manual section on the PLOT Command for more detailed information.

ENDING AN EDITING SESSION [FILE MENU]

<u>ACTION</u>	<u>COMMANDS</u>
Abandon edit and choose another file to edit	RESET
Abandon edit and return to WINDOWS	QUIT (Close)
Save database and return to WINDOWS	EXIT

Utility Guide

***C**hapter*

3

UTILITIES

INTRODUCTION:

This section covers the available utilities that comprise the Interactive CAD Systems' CAD/CAE design software.

The ICS utility programs can either be invoked from within the PROCAD editor (clicking on the appropriate utility button in the Tool bar), or from windows program manager by double clicking on the appropriate utility icon. Whatever method of invocation, the utility program begins by displaying its copyright notice for a few moments, then the main utility window is displayed. In windows environment, the utility executes as a task, while PROCAD main editor is yet another task. You may switch between tasks by a simple mouse click. Refer to your Windows User's Manual if this concept is not familiar to you.

Note: If you encounter an error message when you attempt to invoke a utility program within ProCAD (typically MSLANGLOAD... error exit), this means that you do not have enough memory to invoke the utility and ProCAD simultaneously. You should completely close out PROCAD ([File]Exit), then invoke the utility from its ICON in the Program Manager.

UTILITY PROGRAM MENUS:

All ICS utilities contains a common menu control button "[-]" at the upper left corner of the main window. When you click on this menu control button, a drop down menu appears. From the drop down menu, you may **close** the application, set its **execution priority**, **Project Directory**, get **help**, or view the **About...** dialog box for information about such items as your serial number, application version and release date, registered user name & company, etc.

FOREGROUND & BACKGROUND EXECUTION:

When a utility program is invoked from within PROCAD main editor, it is executed as a background task. When however executed from Windows program manager, it will execute as a foreground task. The fore ground execution is considerably faster than background execution, for this reason, if you don't plan to continue work on PROCAD main editor, you should exit PROCAD and start your utility program from windows program manager.

EXECUTION PRIORITY:

All utilities windows contains a control menu button. As mentioned above, you may access the priority setting command from this utility menu. If you are working within PROCAD while a utility program is executing as a background task, you might want to lower its execution priority if you notice a sluggish response in PROCAD main editor. If however you are executing a utility program as a foreground task, you should increase its execution priority to the maximum possible value.

DATABASE & LIBRARY SEARCH PATH:

All utility programs provides a field for entering your database and library search paths. These data paths will be used by the utility programs when locating the database and library files respectively. You may not specify a path statement when entering file information in any of the PROCAD utility programs. Clicking on the button (browse) next to a database or library search path will allow for menu selection of the corresponding directory .

WORKING DIRECTORY:

When a utility program requests for a non Database or Library file, it will attempt to locate it in the working directory of the utility program. Also when a utility program generates outputs and reports, they will be placed in the utilities working directory. Typically the working directory for all PROCAD utilities is "\WPROCAD". You may change the working directory of each of the utility programs using the Windows file managers "[File]Properties..." command. Please refer to your windows manuals for instructions on how to change an applications working directory.

ABORTING EXECUTION:

While a utility program is in operation generating plots, reports etc, the "**STOP**" button in the Activity Progress window will become active. If you click on this button, the utility program will terminate execution after a positive confirmation, and return to Windows. You should only use this method of program termination if you want to return control to windows.

Utilities (PROCAP)

C*hapter*

3-1

PROCAP - NETLIST EXTRACTOR

INTRODUCTION

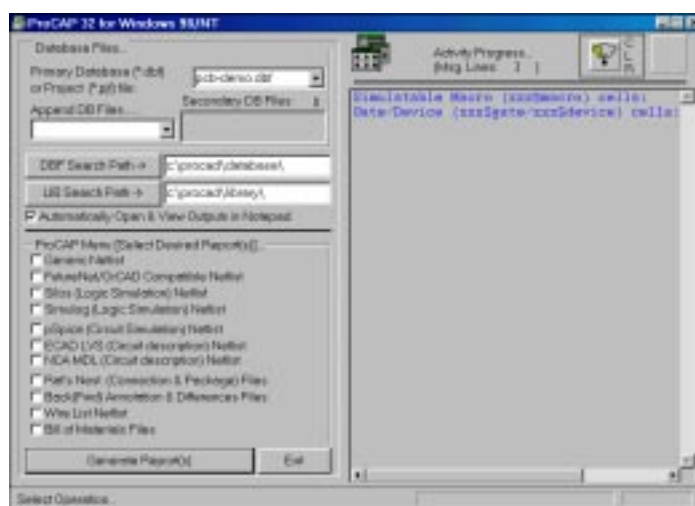
PROCAP is the Netlist Extraction utility program for PROCAD. It allows you to interface to various logic simulators. It will also generate a wire list, bill of materials, and generic netlist files. These are ASCII files which can be printed out or input to other utilities. PROCAP also generates the RAT'S NEST files used for PCB design by PROCAD.

PROCAP is a token driven extractor; that is, it searches the Attributes Data File attached to each cell placed within the database, and extracts the information pertinent to the extraction file being generated. The connectivity, of course, is extracted from the database itself.

USING PROCAP

You may invoke PROCAP from the Tool bar within the PROCAD editor, 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.

When PROCAP is invoked by either method, it displays the signon message. 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:

STEP 1 - Enter or select the file name of the database that you want to extract a netlist of, then click on the **Load DBF or Project file** button to start loading the primary database. If you are using the Project feature of PROCAD, you should specify your project file here. You should not specify a search path for the database or project file here. Database & Library search paths must be specified in the appropriate fields.

After loading the primary database, PROCAP displays the following information message in its Activity progress window.

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

STEP 2 - After loading the primary database, PROCAP will be ready to start data extraction, append more database files or re-load a new primary database file. The following screen menu will be displayed and you are prompted for the desired extraction format(s).

[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 Anotation/Diff
[W] ire	Generate Wire List
[BI] ll	Generate Bill-of-Materials

Select the appropriate report format(s) desired and click on the **"Load DBF or Project File"** button to start the extraction.

Note: While extracting a netlist, the "STOP" button will become active. If you click on this button, PROCAP will terminate execution after a positive confirmation, and return to Windows.

Note: All outputs generated will be placed in the PROCAP working directory. Typically, this would be \WPROCAD directory. Refer to your windows manual if you want to change the working directory of PROCAP or any other windows application.

Example: If you were to select the GENERIC NETLIST and click on the **"Generate Report(s)"** button, The following message would appear:

Generating GENERIC_NETLIST...

Then you will be prompted for the output file name. The default file name will be the file name shown in the brackets (original database with a ".NET" extension).

Enter Output File Name [XXX.NET]: _

After entering the desired output file name, the program will process the database information into the appropriate output.

filename <RETURN>

After the appropriate output has been generated the following message will be displayed and the PROCAP will be ready for another operation.

**File: filename.NET GENERATED.
GENERIC_NETLIST Complete.....**

STEP 3 - At this point, you have several choices:

1. Select additional netlists or lists to be extracted from the current database.
2. Input another primary database and extract desired outputs.

3. Append more database files and extract desired outputs.
4. Exit from PROCAP and return to the operating system or PROCAD.

GENERATING MULTIPLE OUTPUT FORMATS

If you enabled more than one output format (say Generic Netlist & Bill-of-materials etc), PROCAP will not prompt for output file name. The base name of your primary database will be used for output with the appropriate file extension attached as follows:

Generic Netlist	".NET"
FutureNet Netlist	".FNT"
FutureNet Pinlist	".PIN"
Silos Simulator	".SIL"
Simulog Simulator	".CKT"
Spice Simulator	".SPI"
NCA MDL Netlist	".MDL"
ECAD's LVS Netlist	".LVS"
Rats Nest	".CON & .PKG"
Back Annot Diff	".DIF"
Wire list	".WIR"
Bill of Materials	".BOM"

INPUTING/APPENDING MULTIPLE DATABASES

PROCAP can link up to thirty Schematic database files for generating the RATS NEST files for PCB layout, or the Back Annotation DIFF file.

You should enter or select the names of databases you want appended to the primary database using the "Append Database" combo box. As you append database file(s), they will be listed in the secondary database list box to the right of the append combo box. If you do not see the database files you want appended, your database search path may be incorrect. Change your database and may be your library search paths and try the operation again. You may not specify a path name while entering a database file name !. **YOU MUST ENTER YOUR SEARCH PATH IN THE APPROPRIATE DATABASE OR LIBRARY SEARCH PATH FIELD.**

PROCAP links the new database to the first one. You may then choose to generate output, or link another database to the list. Up to thirty Schematic databases may be linked in PROCAP for extraction.

Note: If you organized your PCB and Schematic files in a Project file (See PROCAD's PROJECT command), you should specify the project file in item #1 above, and PROCAP will automatically load and link all the files required for the project.

When linking Schematic databases you should take care to insure that devices are sequentially numbered in a correct fashion. You may let PROCAD number devices automatically in the first Schematic database, but all subsequent pages of the design should have the first device manually assigned to have instance number greater than the highest number used in the previous page. PROCAD will use that number as a starting instance number for that page. When manually assigning cell references, you should not use the /R option when executing the PACK command. Computer assigned nodes are correctly linked in PROCAP. See also **PACK**, **CHANGE INSTANCE**, and **CHANGE SIGNAL** commands in the Command Reference Manual.

OUTPUT FORMAT DESCRIPTION

You should also refer to Appendix A for information concerning the proper use of keywords as related to output extraction.

GENERIC NETLIST:

KEYWORDS -

** NONE **

The output format generated is self explanatory. The data in the attribute files of each cell in the database will be written to the output file. Note that the Generic Netlist only works on the top level of the drawing database. Nested cells are identified, but not expanded.

SILOS:

KEYWORDS-

GATE - Gate type, eg: .NAND
DELAY- rise/fall, rr ff
OTHER - \$ Comment/text/params.
MACRO - MACRO NAME

For a Subckt (MACRO):

```
.MACRO MACRO_NAME PIN_NAME1 PIN_NAME2
+...OTHER
```

For a GATE:

```
PIN1_WIRE_NAME GATE_NAME RISE_DELAY{rr}
+ FALL_DELAY{ff} PIN2_WIRE_NAME PIN3_WIRE_NAME
+ .... OTHER
```

For Macro expansion call:

```
(_TAG MACRO_NAME PIN1_SIGNAL_NAME
+ PIN2_SIGNAL_NAME ..... OTHER
```

SIMULOG:

KEYWORDS-

GATE - Gate type, NAND
DELAY- single delay xx
OTHER - \$ Comment/text/params.
MACRO - MACRO NAME

For a Subckt (MACRO):

```
.MACRO MACRO_NAME PIN_NAME1 PIN_NAME2 ..
+ ...OTHER
```

For a GATE:

```
PIN1_WIRE_NAME GATE_NAME DELAY
+ PIN2_WIRE_NAME PIN3_WIRE_NAME .... OTHER
```

For Macro expansion call:

```
*_TAG MACRO_NAME PIN1_SIGNAL_NAME
+ PIN2_SIGNAL_NAME ..... OTHER
```

SPICE/PSPICE & LVS:

KEYWORDS-

DEVICE	- device type, 3 char.
MODEL	- model name
OTHER	- other params, "*"comments
MACRO	- subckt "macro/gate"

For a SUBCKT-

```
.SUBCKT SUBCKT_NAME PIN1_NAME/NUMBER
+ PIN2_NAME/NUMBER ..... OTHER
```

For a DEVICE-

```
DEVICE_TYPE{2}TAG PIN1_NAME/NUMBER
+ PIN2_NAME/NUMBER ..... MODEL_NAME OTHER
```

For subckt reference call-

```
X_TAG PIN1_NAME/NUMBER PIN2_NAME/NUMBER
+ ..... SUBCKT_NAME OTHER
```

NOTE: Generated output format is slightly different from the above if LVS netlist is generated.

NCA's MDL:

KEYWORDS-

DEVICE	- device name
MACRO	- subckt name
OTHER	- "#"comments, etc.

For Define 1 -

DEVICE_NAME_TAG PIN1_NAME PIN2_NAME OTHER

For Define 2 -

```
DEFINE SUBCKT_NAME "O"(OUTPUT_SIGNAL_NAMES)
+ "I"(INPUT_SIGNAL NAMES) "B"(IN/OUT_SIGNAL_NAME) + OTHER
```

For Reference -

```
SUBCKT_NAME "O"(OUTPUT_SIGNAL_NAMES) + "I"(INPUT_SIGNAL NAMES)
"B"(IN/OUT_SIGNAL_NAME) + OTHER
```

RATS NEST FILES

KEYWORDS-

RAT\$PKGn	- RAT Package cell name
RAT\$TCNT	- RAT # of devices/package
RAT\$DEVn	- RAT Device n ordered pin #s
RAT\$VCC1	- RAT Global VCC1 pin #s
RAT\$VCC2	- RAT Global VCC2 pin #s
RAT\$VCC3	- RAT Global VCC3 pin #s
RAT\$VCC4	- RAT Global VCC4 pin #s
RAT\$GND1	- RAT Global GND1 pin #s
RAT\$GND2	- RAT Global GND2 pin #s
RAT\$NET	- RAT Global NetName and pin #s

Output Format-

The Connection data file ".CON" has \$CON in the first line to signify a connection file, and the packaging data file ".PKG" has \$PKG.

The format for the connection data file is:

```
/SIG UU,nn UU,nn UU,nn .....
```

SIG	= signal/node name
UU	= instance name
nn	= instance UU pin number

The format for the packaging data file is:

```
UU [Library]Cell_name
```

```
UU = Instance name
```

```
[Library]Cell_name = The instance foot print cell name
                    spec. specified in the
                    RAT$PKGn token (n=0,1,2 or 3).
```

WIRE LIST FILES**KEYWORDS-**

```
** NONE **
```

Output Format-

```
SIG UU,nn UU,nn UU,nn .....
+ UU,nn UU,nn ...
```

SIG	= signal/node name
UU	= instance name
nn	= instance UU pin number

BILL-OF-MATERIALS

KEYWORDS-

CO_PN - Company part number
MFG_PN - Manufacturer's part #.
VALUE - Component value if any
DESCN - Description (Up to 19 characters)

Output Format-

CO_PN	Qty	MFG_PN	DESCN	VALUE	REFERENCES
					+ more refs.

BACK ANNOTATION / DIFFERENCES FILE

KEYWORDS-

DRC\$VIA - Layers connected by the specified pad.
RAT\$XXX - Layers connected to plane (XXX=VCC1..).

Output Format-

Differences are flagged on the PCB, SCHEMATIC, or BOTH databases. The ASCII differences file generated is self explanatory.

PROCAP back annotation command compares a PCB database file against a series of schematic databases. For this command to work correctly, the primary database loaded (first database file) must be the PCB file.

After inputting the primary PCB database file, use the [I]nput command to append a list of schematic database files that made up your PCB database file. If you issue the [BA]ck annotation command without first appending the schematic database files, PROCAP will prompt for only one Schematic database file before beginning the PCB/ Schematic comparison process.

You must append Schematic databases using the [A]ppend combo box (or load a Project file) if you have more than one Schematic database file to be compared against the PCB database. PROCAP will generate a PCB net file, and proceed to read in the Schematic data base files and compare the netlists. Errors can be optionally flagged (back-annotated) to the PCB and/or Schematic database files in form of markers on the error pins in layer 35. A differences file is also generated.

The back-annotation feature of PROCAP will recognize power, ground and global nets connections made through power/ground or signal layers. These power/ground or global net pins must have the appropriate RAT token e.g.:

RAT\$XXX n1 n2 n3.....

or

RAT\$NET signal n1 n2 n3.....

where **XXX** is VCC1, VCC2, VCC3, VCC4, GND1, GND2, **signal**=Any global Net and **n1,n2,...** are the pins connected to the supply or global net.

NOTE: The rat's nest command will generate connecting trace to pins n1 n2 n3 .. above only if those pins are not connected to other signals in the schematic diagram.

PROCAP BACK/FWD ANNOTATION & DIFF CHECK

This is a ProCAP command used for checking the PCB design against the corresponding schematic sheet(s), or for forward and backward annotation of data between the PCB and corresponding schematics. A less complete version of this command is provided in ProCAD via the **[File]Annotate...** menu item.

You may [B]ack annotate instance name changes from the PCB database back to the Schematic database, or [F]orward annotate the changes from the Schematic database to the PCB database. This is a useful feature used for maintaining ECO changes that might be required during a design process.

PROCAP will generate a list of all devices that could not be Back or Forward annotated due to connectivity differences errors between the PCB and the Schematic databases. If this occurs, you must manually change those instance names using the CHANGE INSTANCE command in PROCAD. You should then run the differences [C]heck option of the Back-

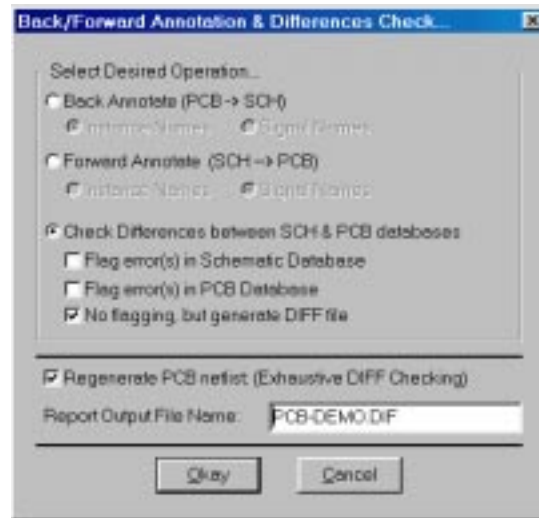
annotation/Difference command in PROCAP to determine the possible cause of the connectivity errors.

Tips: Before you start renaming the PCB or Schematic instance names, you should first run the differences check and fix any connectivity errors. The Differences check will not work properly if you have instance name differences between the PCB and Schematic databases as a result of you manually renaming some instance names or the [B]ack or [F]orward annotate command renaming some instances and leaving some un-renamed.

The Differences command checks for connectivity errors while making the assumption that the instances names are consistence between the Schematic & PCB databases.

You must load your PCB file as the primary database in ProCAD and the Schematic sheet(s) as the secondary databases before issuing this command. For best result, you should use the Project feature of ProCAD to organize your design files and just enter the project file name in ProCAP.

When this command is executed in ProCAP, the following dialog box will be displayed:



SELECTING DESIRED OPERATION:**Back Annotate (PCB -> SCH):**

Instance Names: This operation allows for back annotating Instance Names (Reference Designators or U numbers) from the PCB back to the schematic. This is useful after a PCB has been re-numbered and you desire to change the schematic reference designators to match those on the PCB. This option is also available in ProCAD using the [File]Annotate... menu item. It is preferable to use the corresponding ProCAD **[File]Annotate...** menu item if you purchased the MDI option in ProCAD.

Signal Names: This version of ProCAP does not currently support back annotating signal names from PCB to Schematics. Support for Forward annotating signal names from schematic to PCB is supported and described below.

Check Differences between SCH and PCB:

This option allows for comparison checks between the PCB and corresponding Schematic sheet(s). This is the main strength of the ProCAP ECO and integrity checking. This command is not available in the main ProCAD Editor. This function must be performed in ProCAP.

At the conclusion of this command, you may desire ProCAP to flag any errors in the Schematic Database(s) and/or the PCB database by enabling the "**Flag Errors in Schematic Database**" and "**Flag Errors in PCB Database**" check boxes respectively. Flagged pins markers are placed on layer 49 (DRC error layer).

Whether or not you flag errors in the database files, you will still be able to generate an activity report log file (differences file).

Regenerate PCB Netlist (Exhaustive Differences Check):

The "**Regenerate PCB Netlist**" check box allows for exhaustive differences check. When this box is checked, ProCAP will generate new temporary net names for **ALL** PCB nets regardless of whether they were user or machine assigned before performing the differences operation. At the conclusion of the differences function, ProCAP will re-synchronize the proper schematic net names to the temporary PCB net names. If you interrupt the operation before completion, your PCB will have totally different net names from your PCB !!! If this happens, don't panic, you may fix this problem by re-executing ProCAP and either Forward Annotating Nets Names (SCH->PCB) or re-execute the Differences check and let it conclude successfully.

The main purpose of enabling this flag is to enable ProCAP to detect broken daisy chain nets in a PCB design. If this flag is unchecked, broken daisy chain nets whose net names were user assigned such as GND, VCC, Address/data signals (A0,..D0..) etc will not be detected. The side effect of enabling this check box is that ProCAP will not be able to report location of vias or pins where two different nets may have been connected together (shorts). Real shorts will be reported as connectivity fan-out errors whose cause may be difficult to locate.

The desired approach is to perform the Differences Check with this box unchecked and fix all detected errors (shorts etc). Then perform the Difference function a final time with the box checked to detect any possible broken daisy chain error(s).

Note: This flag does not affect the schematic net list.

EXPLANATION OF DIFFERENCE ERRORS:

When ProCAP detects differences between the schematic and PCB, it reports them as "**FanOut->(P/S)Uxx:nn**", "**NodeCheck->(P/S)Uxx:nn**", or "**Un-matched SCH Net**" errors.

Example #1:

U12 1 D0 D0 10 5 FanOutChk->(S)U9:11

The above error means that device U12 pin 1 connected to PCB net D0 which is also connected to schematic net D0 does not drive the same number of devices. In PCB, U12 pin 1 drives device U9 pin 11, but in the Schematics, U12 pin 1 does not drive device U9 pin 11. In PCB, U12 pin 1 drives 10 nodes and in the schematic, U12 pin 1 only drives 5 nodes. You should check schematic device U9 pin 11 for the source of error.

Example #2:

U15 8 N0004QQ A0 8 >=3 FanOutChk->(P)U2:21

The above error means that device U15 pin 8 connected to PCB net N0004QQ and in Schematics, connected to net A0 may not drive the same number of devices. In Schematics, U15 pin 8 drives device U2 pin 21, but in the PCB, U15 pin 8 does not drive device U2 pin 21. In PCB, U15 pin 8 drives 8 nodes and in the schematic, U15 pin 8 drives at least 3 nodes (exactly 3 nodes in the sheet where it was placed in a multi-sheet design). You should check PCB device U2 pin 21 for the source of error. Also since the net names are different and the schematic net is user assigned, there may be a broken daisy chain in PCB signal A0.

Example #3:

U3B 4 VCC1 -NC- 20 0 NodeCheck->(S)U9:14

The above error means that device U3 pin 4 connected to PCB net VCC1 and in Schematics, un-connected (floating) does not drive the same number of devices. In PCB, U3 pin 4 drives device U9 pin 14, but in the Schematic, U3B pin 4 does not drive device U9 pin 14. In PCB, U3 pin 4 drives 20 nodes and in the schematic, U3B pin 4 does not drive any node. You should check Schematic device U9 pin 14 for the source of error. Also since the PCB net is VCC1 and the Schematic is floating, you should check the RAT\$VCC1 attribute for Schematic device U3B. It may be missing or not connected to the right pin. Also make sure Schematic device U9 has a RAT\$VCC1 token for pin 14. or U9 pin 14 is actually connected to VCC1.

Example #4:

U17 19 N0018QQ 4 *UnMatched SCH Net*

The above error means that device U17 pin 19 connected to PCB net N0018QQ drives 4 nodes. A corresponding Schematic Net could not reliably be located for it. This error occurs in a Multi-sheet Schematic Design where there exists a broken daisy chain for the net connect to U17 pin 19. The broken chain could be in any of the 4 nodes driven by this net. You should locate this net in the PCB and figure out where the broken daisy chain occurred.

These types of errors may be difficult to locate if the fanout number is large (4 in the above example). You should fix all "**FanOutChk->(P/S)Uxx:nn**" type errors and re-execute the differences check function to see if the "***UnMatched SCH Net***" errors still exists. If they are the only error types, you should then take the time to locate and fix them.

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C*hapter*

3-2

PROPLOT PLOTTER UTILITY

INTRODUCTION

PROPLOT is the hardcopy plotting utility program for PROCAD. It allows you to produce a hardcopy of selected portions of a PROCAD database. The hardcopy can be produced on various plotters and/or printers

The program supports plotters which utilize the HPGL and DM/PL plotter languages. In addition, plotters and printers for which there are Windows drivers are also supported.

USING PROPLOT

Before executing the program, insure that the following files are in the PROPLOT.EXE program working directory (usually \WPCAD). Please refer to your windows manual for instructions on changing an applications working directory.:

- A. The plot control file(s) generated from the database.(.CTR)
- B. The text font table file (TEXTFONT.BIN or TEXTFONT.TXT)

The database file (.DBF) from which you want to plot the information and all library files (.LIB) used in the creation of that database must be placed in the database & library search directories respectively.

DATE & TIME STAMP

PROPLOT will automatically expand the following characters enclosed in "<" & ">" brackets to their values in the plot drawing:

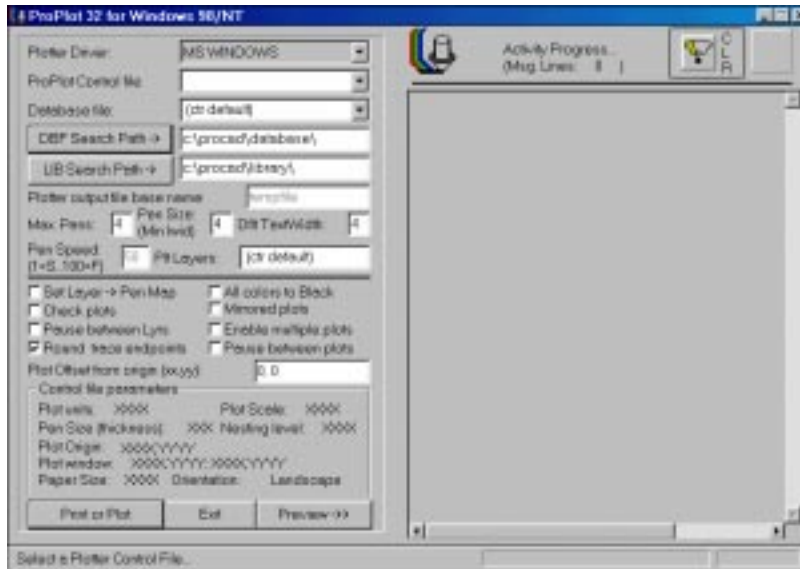
TIME stamp

- | | |
|--------|---|
| <DATE> | Will be expanded to current system date in the DD-MM-YY format. |
| <TIME> | Will be expanded to the current system time in HH:MM:SS format. |

The <DATE> and <TIME> characters must be placed in your drawing in PROCAD with the **Input/Text** command (or **Text** command from the Tool bar or **[Place]** menu.). The characters must appear as shown above. The angled brackets are not optional; they are required.

You may invoke PROPLOT from the Tool bar within the PROCAD editor, 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.

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



PROPLOT Window Control-menu Box:

The Upper left corner of the PROPLOT window is a Control-menu box [-]. This Control-menu box when invoked drops down a menu for controlling the various aspects of the PROPLOT window, and also submenus for displaying the current PROPLOT 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.

The following is a description of the prompts produced by the PROPLOT application:

"Plotter Driver"

Select the appropriate drivers for the plotter/printer that will produce the hardcopy.

Utility Programs Guide...

"Plotter Control File"

Enter or select the "**file_name.CTR**" that contains the portion or portions of the drawing that you want to plot. See the PLOT command in the Reference manual for instructions on creating the Plot Control File. This file must be present in your PROPLOT working directory (See your windows manual for instructions on how to change an applications working directory), typically \WPROCAD directory. You may not specify a path name while entering the control file name !.

"Modify or Set Layer -> Pen map"

This prompt allows you to define which pen to use with different layers (assuming your plotter supports multiple pens). A yes response will cause PROPLOT to produce the following prompt while processing your control file:

"Enter Layer/Pen Map file [NONE]: "

At this point you should enter the name of an ASCII which contains the layer/pen map (the default extension is ".DAT"). The format of the Layer/Pen map is as follows:

Layer# Pen# <CR>.

Where Layer# is the layer number (1-99), and Pen# is the corresponding pen number (1-20). You should have only one (1) entry per line.

Example:

5	3
20	6
23	8

The above example will assign pen #'s 3, 6 & 8 to Layers 5, 20 & 23 respectively.

Note: If you do not specify a Layer/Pen map file, or if the file specified is of the wrong format, You will be prompted to enter the layer/pen correspondence usage from the keyboard.

"Database File"

This prompt allows you to enter the name of the database file to be processed by the ".CTR" control file. The default is the name of the database the control file was generated from.

"Plot output file base name"

This prompt allows you to specify an output file base name. The default is the name of the ".CTR" control file, but with ".PLT" file extension (This prompt is not available if you are using Windows drivers).

"Enable multiple plots"

This prompt allows you to generate multiple plot files (or plots) from the same database and control file. You will be prompted to enter the layer combinations to use for each plot file (or plot).

Note: If you are using windows drivers, **DO NOT** direct the output to a plot file if you plan to generate multiple plot files, since windows drivers will not change the output file name for the subsequent plots generated.

"Check Plots"

This option allows you to generate check plots. If you answer YES to this prompt, all traces and lines will be drawn with one (1) database unit width. Also, area filling will be turned off for polygons, circles and rectangles.

"All colors to Black"

This option allows you to generate monochrome plots. If you answer YES to this prompt, all colors will be plotted as black. This flag is especially useful for generating schematic outputs on a laser or ink jet printers.

"Mirrored Plots"

This option allows you to generate mirrored plots. If you answer YES to this prompt, ProPlot will generate mirrored image of your database. This is especially useful for generating PCB artwork directly on transparencies for direct PCB photo-etching.

"Round PCB trace endpoints (Gerber emulation)"

This option allows you to generate rounded trace end points and corners. This closely emulates what you would see if you generated GERBER files using round trace aperture. This option is ignored if you are plotting a schematic database.

"Plot offset from origin"

If you find out that your plots are not positioned properly on your paper, you may enter an offset value for the whole drawing. You may use positive and or negative offsets for X and Y values.

"Number of Pens [1] "

Enter the number of pens which will be installed in the plotter. The pen used is calculated in the following manner (assuming you did not specify Layer/Pen map):, ie. If for example the plotter will have 6 pens installed, and the layer color within PROCAD is 16, then Pen #4 will be used to plot that layer.....

16 divided by 6 = 2 with a remainder of 4 = Pen #

Note:If a Layer/Pen map was specified, this file will be used instead.

"Pen Size [7]"

Pen thickness or width is measured in database units, enter the appropriate value. This is very important if there are any area of the drawing to be filled in solid.

"Default Text Width [7]"

The default text width is defined in database units. If a text entity in the database has a width of 0 or 1, PROPLOT uses the Default Text Width specified above for that entity.

"Pen Speed (1-100) [1] "

This is a relative indication of speed. Experiment with several speeds to find the right pen/paper/speed combination for your needs.

"Plt Layers [DEFAULT] "

Enter the layers you want plotted. The default will be those layers specified in the .CTR file.

After entering your responses to the above queries, click on the **"Print or Plot"** button to start generating hard copy output or **"Preview ->"** button to preview your plots.

"[S]end PLT file to plotter or [G]enerate PLT file ?_"

Note: The above prompt will not be displayed if the Windows driver is selected. When windows driver is selected, the appropriate MS Windows printer/plotter setup dialog box will be displayed.

At this point you have a choice of generating and/or sending a plot file. Use "S" to send a previously generated plot file to the plotter/printer. Use "G" to generate a plot file and the following prompt will appear:

The following messages and prompt will then be displayed in the Activity window:

"Database File: aaaaaaaa.DBF"

"Output File: bbbbbbbb.PLT"

When PROPLOT is finished generating the plot file, the following message and prompt will be displayed:

"File: bbbbbbbb.PLT Generated."

"Do you want to send file to plotter (Y/N) [N] ?_"

At this point you can send that same file to the plotter or printer. Or you can generate another plot file by providing a NO response. A YES response will cause the following prompt to appear:

"Plotter COM port (1 or 2) [1] ?_"

Select COM port to which your plotter will be connected.

A NO response will re-enable the PROPLOT main window for further operation.

At this point, you have three choices: Enter a new control file and generate new plots, or Quit. To Quit click on the **Exit** button.

NOTE: If you are using the Windows drivers, ProPLOT will not generate a .PLT file. ProPLOT will automatically send the database to the attached plotter (unless you choose output to a file in Windows printer setup dialog box).

Utility Programs Guide...

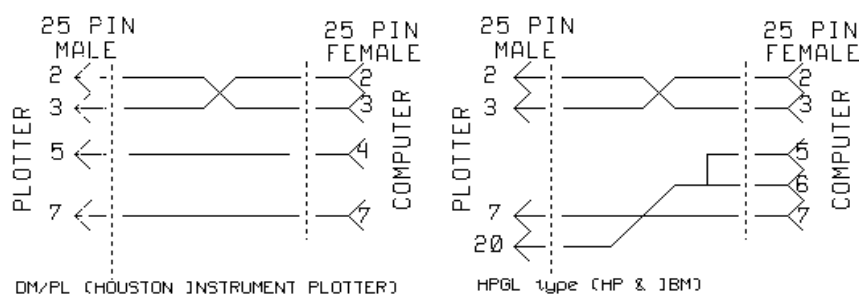
CONFIGURATION REQUIREMENTS

If you plan to use the internal HP/GL or DMP/PL drivers, your plotter should be set for the following parameters:

DATA: 8 bits, PARITY: None, STOP BITS: 1, BAUD RATE: 9600, PROTOCOL: XON/XOFF

If you are going to use the Windows drivers for your plotter/printer, you must first install them using the windows print manager or control program. (See your windows manuals for details).

The following cable configurations were used and will work with their respective plotters (other combinations might work also).



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Utilities (PROROUTE)

C***chapter***

3-3

PROROUTE UTILITY

INTRODUCTION:

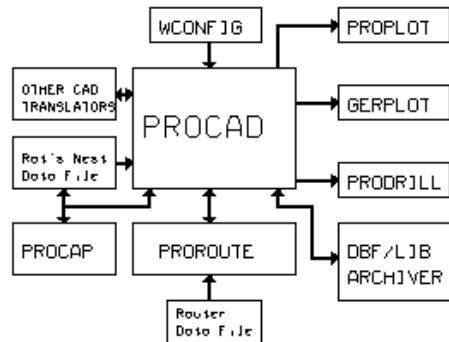
Interactive CAD Systems' PROROUTE provides complete integration of all of ICS's PROCAD line of electronic design workstation software, from schematic capture to PCB layout. Create your design, simulate it, generate the board layout, and produce documentation - all on a single personal computer workstation.

As illustrated in the overview diagram, on the next page, PROROUTE can be added as a modular package to your existing PROCAD design workstation. A schematic is created in PROCAD (or an installed third party software) and captured using PROCAP. The RATS NEST data files are utilized in PROCAD to create a board placement, and pin-to-pin connections, which may then be optimized for the most efficient layout. Preroutes, and keepout areas may also be implemented in the design utilizing the PROCAD interactive graphics editor. This placement/connection file is then passed to PROROUTE for re-entrant autorouting, and the completed PCB design passed back into PROCAD for final rework (clean-up). Once the design is ready, artwork may be generated using either PROPLOT or GERPLOT.

PROROUTE is a fully re-entrant autorouter utilizing the most sophisticated 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.

PROROUTE is an unlimited grid router (to 1 mil resolution) ensuring routes to off grid pins and connectors, and multiple traces between pads. This powerful router also has an unlimited routing trace widths and pad sizes. It will even route orthogonal, or true 45 degrees for boards up to 64"x64". The user-configurable parameters of PROROUTE and its unlimited iteration route strategy allow it to tackle even the most difficult multi-layer designs, as well as the simple single-sided. The user definable route grids, unlimited trace widths, and multiple pad layer definitions, make PROROUTE a powerful routing tool for surface mount technology as well.

THE OVERVIEW INTERACTIVE CAD SYSTEM'S PROFESSIONAL ENGINEERING DESIGN SYSTEM



MULTIPLE PASS ROUTER

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 - Power Router routes only nodes which are labelled 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 - Memory Router 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 - Lookout Router performs a series of three strategic Lookout routes on each specified trace as required. The Lookout Strategy router insures the least amount of feed-throughs on the PCB, while completing the routing of traces on the board.

INSTALLATION

If you are using the DOS version of PROROUTE, make sure that it has been properly installed over the normal demo version which is shipped as part of any PROCAD configuration. To install the DOS version of PROROUTE, insert the PROROUTE disk in drive A and issue the following command:

INSTALL A: C: <RETURN>

The above process will install the full version of PROROUTE together with its copy protection parameters.

NOTE: PROROUTE for windows is automatically installed when you installed the main editor. There is no separate PROROUTE diskette.

All files associated with the database being routed (.DBF and .LIB) must be in the PROROUTE working directory. Also be sure to check that all libraries being used in the board database include the DRC\$PADx token as discussed later in this manual. This is absolutely essential to successful routing using PROROUTE.

DOS version of PROROUTE may also be executed on a non-graphic computer with a minimum configuration of: 640K RAM, 1 Floppy Drive, Hard Disk (recommended), Math Coprocessor (optional). This low cost computer may be used to perform routing, while using the graphic workstation for editing new or existing designs.

THE DESIGN

The entire design process using PROCAD, PROCAP, and PROROUTE, should flow along these lines:

I	Schematic Design	PROCAD
II	Schematic Capture	PROCAP
III	Board Mechanics	PROCAD
IV	Board Placement and Connection	PROCAD
V	Preroute	PROCAD
VI	Auto Route	PROROUTE
VII	Clean-up	PROCAD
VIII	Check plots	PROPLOT
IX	Artwork Masters	GERPLOT

THE SCHEMATIC

When designing a schematic with PROCAD, there are no special considerations which must be maintained to use PROROUTE. All output generated from a standard PROCAD schematic design can still be applied to a new design using PROROUTE.

It is however, important to utilize the correct data tokens within the schematic symbols in order to generate the design information required by PROCAP to generate proper rats nest files for use by PROROUTE. These data tokens are not uniquely applied to PROROUTE though, and should already exist in most of your established library symbols. See Appendix-A, "Attribute Data Files" for more information on data tokens.

SCHEMATIC CAPTURE

PROCAP, the Netlist Extraction utility, is an important tool when designing a circuit to be routed by PROROUTE. PROCAP accepts a PROCAD schematic database as input (or a series of up to 30 schematics) to generate documentation output. Some of the types of output can include Netlists, Simulation Files, Bill-of- Materials, and RATS NEST.

THE RATS NEST FORMAT

The most important files for use with PROROUTE, which PROCAP will generate are the RATS NEST files. The RATS NEST, which is formatted into two standard ASCII files contain the packaging information for all components used in the schematic design, as well as signal names and pin-to-pin connections. These files are used separately to first place the physical packages on the PCB, and then make all electrical connections in a straight point-to-point fashion. It is this placement and connectivity which PROROUTE utilizes to automatically route the board layout.

The RATS NEST files may be extracted from any Schematic Editor which generates a netlist including packaging and connection data. By manually generating the RATS NEST files from external sources, you may create database files which will allow PROROUTE to autoroute designs from existing schematics, or third party software.

The RATS NEST is divided into two separate files for effective use during design. These two files are the packaging file (**.PKG**), and the connection file (**.CON**).

The packaging file, which must end with a three letter file extension of .PKG, contains information regarding the device packaging of schematic symbols for use on the PCB. The format of the .PKG file is as follows:

```
$PKG
IC1  [PKG]IC08A
IC2  [PKG]IC14A
IC4  [CELLIB]PLCC64
R1   [PKG]RC07
```

The first line in the .PKG file must begin with "\$PKG", and subsequent lines contain the packaging information. The Instance name (reference designator) must come first in a line, then the device library and package name. This allows PROCAD, upon execution of the RATS NEST command, the capability to select a device from the various packaging libraries, interactively place components on the board, and assign each physical device the instance name which is carried over from the schematic.

The format of the connection file must include the net name as defined by you, or assigned by the computer, as well as the pin-to-pin connections of the entire net. The connection file, which must end with an extension of .CON, is structured as follows:

```
$CON
/CLK      IC1,3 U2,3 U8,10
/GND      IC1,7 U2,7 U8,20
/VCC      IC1,14 U2,14 U8,40 U8,1
/VCC      U8,1 U15,16 U14,14
```

The first line in the .CON file must begin with "\$CON", and subsequent lines contain the connectivity information. The net name must be preceded by a forward slash character (/), and cannot exceed 12 characters in length (excluding the slash). What follows is simply the pin-to-pin connection of the net by device reference designator, and pin number. Multiple lines of the same signal name are permitted (see above example), provided that the signal name is repeated, and the last instance, pin_number is also carried over to the next line.

These two files may be created manually from an existing schematic design, or you may work with existing interfaces to utilize third party engineering software. PROCAD rats nest command will also accept Futurenet compatible netlist (eg. those generated from ORCAD STD).

BOARD MECHANICS

For PROROUTE to work effectively, certain considerations must be established in the PCB database at the very beginning of the layout process.

As when designing with the PROCAD interactive editor to do manual designs, you must establish a standard layering convention to be used by PROROUTE. This USER's Guide describes a proposed layering convention that is used by all ICS CAD libraries, and therefore it is recommended that this same layering convention be applied.

See Appendix-E "Libraries & Layer Assignments" for a better understanding of the layering convention utilized by ICS.

The first step in beginning any PCB layout is the construction of the actual PCB substrate which is to be used. This construction must include the assembly outline, all mechanical features of the board such as cutouts and toolholes, as well as dimensions and fabrication instructions. Once complete, the board outline may be stored in a library, as a cell, for easy recall in later projects.

A new feature of this board cell, which is required by PROROUTE, is an area limit. The area limit defines the extents of the board space within which PROROUTE is allowed to place traces. To create this limit you must understand a concept in routing known as a keepout.

DEFINING KEEP-OUT AREAS

The keep-out is an area, or pre-routed trace, which PROROUTE understands as an existing limits to the router. A keepout may tell the router not to route within a certain area, for instance a hole in the center of the substrate which will be used in mounting the PCB into a chassis. A keepout could also be any pre-routed traces which the designer wishes to place manually before using PROROUTE; for instance an ECL trace with critical (source-load-terminator) routing.

The method of establishing a keepout is to place a closed polygon (using the INPUT POLYGON command) of one (1) database unit line width. The area within the polygon is considered to be a void, not used as allowable routing area by PROROUTE. Figure 1 illustrates a void area of a PCB as defined by a keepout. The board used in Figure 1 requires a one square inch cutout for mounting purposes. The keepout defines this cutout as a non-routing area.

To establish a pre-route simply INPUT WIRE, with a selected line width, to place traces as you want them. This is usually an option only required for special traces, and should be used with care to insure the optimum routing results from PROROUTE.

The most important difference between a pre-route and a keepout is the type of line used in placement. The keepout must be placed utilizing the INPUT POLYGON, or INPUT LINE commands; while the pre-routed traces must be placed with the INPUT WIRE command. The difference here is that a wire will automatically be assigned a signal name (which may be user-assigned) and will behave as a node for the purposes of design rule check, and back annotation. The line, or polygon, does not have a signal name and will not appear in the netlist. For a pre-route, which must appear in a netlist for back annotation, a wire must be used; however, for a keepout, which should not be netlisted (and in fact will be removed before final artwork generation), a line or polygon must be used.

DEFINING KEEP-IN LIMITS

In the case of the area limit, the keepout is actually used as a means of maintaining all routes within the physical limits of the PCB. After all, the router must know the limits of the area within which it is allowed to place routes. The area limit, which outlines the PCB (usually 50 mils inside the edge) **must exist on all routing layers of the PCB**. If, for instance, you are routing a two sided PCB the area limit would exist on both the solder and component sides of the board; however, for a multilayer board the area limit must exist within all internal layers as well as the top and bottom of the board. Figure 2 illustrates an area limit on a PCB design.

Note: Keepout area used as board limits (keep-ins) must be drawn using polygon of 2 or more database units line width, and must exist on all routing layers of the PCB.

Keepouts, and Pre-routes, should only exist on those layers which it affects.

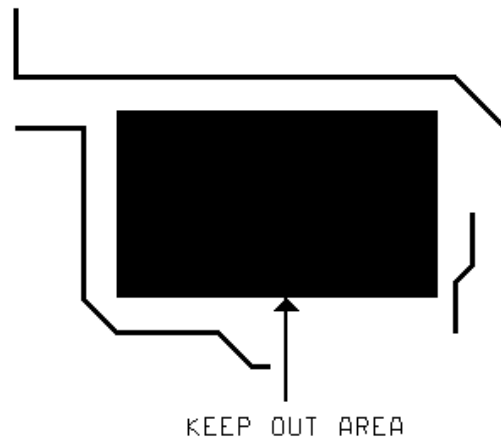


Figure 1 Keepout Void on a PCB

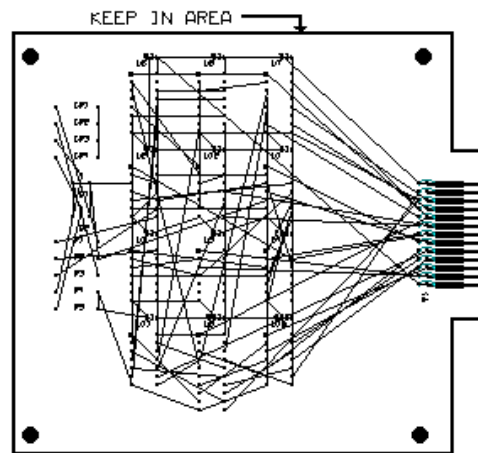


Figure 2 Area Limit of PCB

BOARD PLACEMENT AND CONNECTION

Now that the board substrate has been mechanically designed (and stored in a library if desired), the components must be placed. To do this placement simply check to insure that your present working layer is one and execute the RAT {Filename} command of PROCAD to utilize the .PKG packaging file which has been previously generated from the schematic by PROCAP.

The .PKG file contains the physical equivalents for all devices used in the schematic designs, as well as their reference designators. The execution of the RAT command within PROCAD will allow you to interactively place these components, and assign them reference designators, in an effective fashion.

Once the board components have been placed, you are ready to make the electrical connections of the circuit. To do this simply insure that your present working layer is 30 (described as the default source layer for PROROUTE), or any layer which you have chosen as a standard for RATS NEST. On this layer you will want to execute the PROCAD command RAT {Filename.CON} (make sure to include the .CON extension to apply the second RATS NEST file which contains the pin-to-pin connections of the design.

With the board placed, and all connections made, you are now able to utilize the PROCAD DRAG and RUBBERBAND commands to move components, maintaining electrical connections, to achieve what you feel is the optimum placement. This optimum placement of the board will insure a high completion rate by PROROUTE during the autorouting process.

PRE-ROUTE

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 doing this though, 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. Figure 3 illustrates a pre-routed trace on a PCB.

To place a pre-route, simply INPUT WIRE to replace the RATS NEST line, or use the REROUTE command to utilize the existing line. In either case, once the pre-route is complete, you must be sure to delete the RATS NEST line which represents that connection, or PROROUTE will automatically route the connection a second time. The INPUT WIRE command must be used to insure that the traces of the PCB will Netlist accurately for the purposes of Back Annotation.

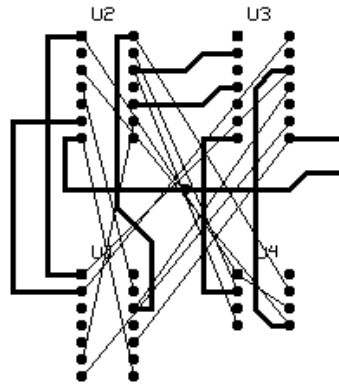


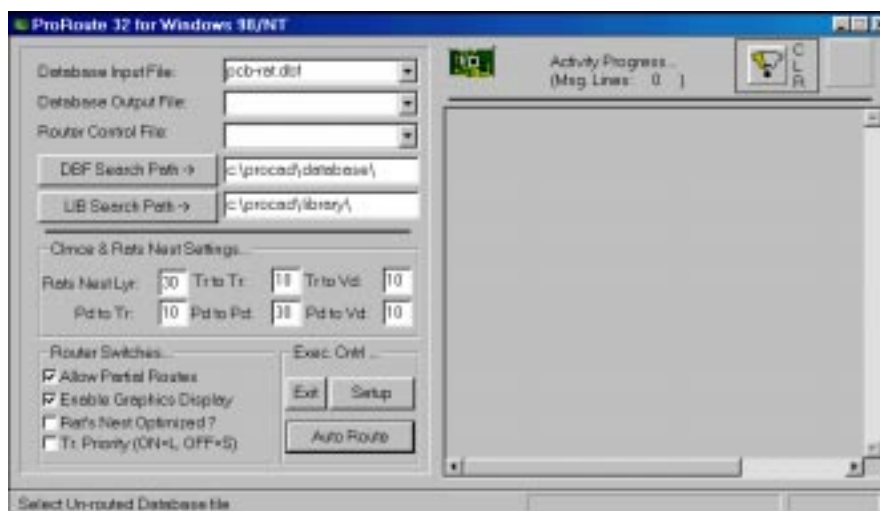
Figure 3; Prerouted Traces of a PCB

USING PROROUTE

You may invoke PROROUTE from the Tool bar (Router button) within the PROCAD editor , 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.

Tips: For interactive routing within ProCAD, you should use the [Route]Auto_Route menu item (AROUTE command) to invoke a built-in Quick route version of the ProRoute utility. Refer to the AROUTe command in the Reference Manual for details.

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



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 submenus 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.

The following is a description of the prompts generated by PROROUTE:

Database input file

This is the file which you have created that includes the board outline, area limits, component placement, pre-routed traces, and a RATS NEST of unrouted connections. This file must be located in PROROUTE database path specified in the "**Database path field**". You may not specify a database path when entering the file name.

Database output file

This is the file which PROROUTE will create at the completion of the route iteration. This file should have a different name /extension than the original file to insure back-up protection against failure. If you decide to start PROROUTE from scratch, after making a few more changes, you will need the original database to work with. This file will be placed in PROROUTE database path specified in the "**Database path field**". You may not specify a database path when entering the file name.

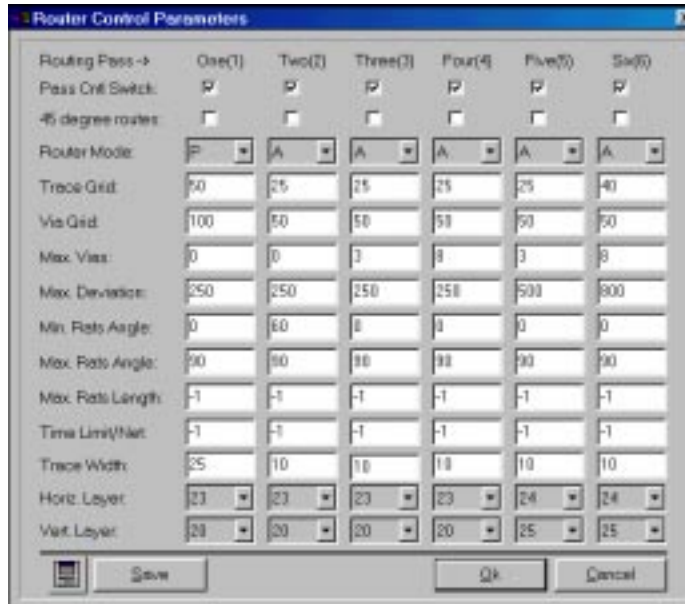
Router control file

This is a file which contains a series of parameters for each of the six allowed route passes per execution of PROROUTE. You define this .DAT file prior to executing PROROUTE. This file must be present in PROROUTE working directory. You may not specify a data path statement for this file.

Database & Library Search Paths

You must specify the proper database and library search path for the database and all libraries used in the database. PROROUTE will use these data paths when locating the appropriate files. You may not specify a path statement when entering file information in PROROUTE.

PROROUTE displays a screen of information which echo the variables that you have established for this particular route iteration. The data displayed may be edited before the routing process is started. A sample of this screen display is illustrated below.



KEYWORD DATA 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 data file is created by the user utilizing a standard line editor, or word processor in non-document mode. The data file must be a straight ASCII file, with no word processor control characters, with a single route parameter per line. The data file may include comment lines for ease of understanding by preceding each comment line with the \$ character. A series of predefined data 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.

The two types of data file parameters are PASS, which are variables required for each of the six allowable passes within one route session, and GLOBAL, 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).

In the following discussions of keywords for both PASS and GLOBAL parameters, each keyword relates to a line number of the Router Control Parameters illustrated previously.

ROUTER PASS PARAMETERS

The keywords discussed in this section are for use in every specified pass. If a pass is set ON, then these parameters will be considered by PROROUTE.

If no value is specified for a keyword within a given pass, the default will be chosen. If redundant values for a keyword, other than the default value, are to be used in multiple passes the value must be repeated for each pass definition.

BEGIN PASS N Line 1

Signifies the beginning of Pass N data definition

Example Begin Pass 4

END PASS N

Signifies the end of Pass N data definition

Example End Pass 4

To move from one Pass definition to the next does not require an END PASS statement; however PROROUTE does require an END PASS statement at the end of data definition for the final pass.

PASS N {ON/OFF} Line 2

Enables or disables a designated router pass. This keyword must appear within a BEGIN/END keyword statement.

Example PASS 2 ON
 PASS 3 OFF

MAXSegments=N

No Line

Specifies the maximum number of segments per route. The maximum number cannot exceed 23 segments. The limitation is based on a PROCAD limitation of 24 vertices within a wire.

Default: 23

Example MAXS=20

MAXVia=N

Line 7

Specifies the maximum number of vias per route. The maximum number of vias cannot exceed 8, although it is recommended that fewer vias be allowed during the early passes.

Default: 8

Example MAXV=3

NOTE: For single sided boards MAXV=0

MAXAngle=N

Line 10

Specifies the maximum absolute angle of an unrouted trace which will be routed during this pass. The value should be specified between 0 and 90 degrees.

Default: 90

Example MAXA=45

MINAngle=N

Line 9

Specifies the minimum absolute angle of an unrouted trace which will be routed during this pass. The value should be between 0 and 90 degrees.

Default: 0

Example MINA=15

NOTE: The MAXA and MINA keyword allow you to define routing priorities for each pass. For instance by establishing MAXA=45, and MINA=0, you will be telling PROROUTE to select only the traces which are primarily horizontal for routing.

MAXLength=N

Line 11

Specifies the maximum point to point length of an unrouted trace which will be routed during this pass. A value of "-1" denotes an unlimited length.

Default: -1 (unlimited)

Example MAXL=800

NOTE: This keyword allows you to establish a routing priority by length. When values of distance are specified within a keyword, it is assumed to be defined in the Database Units (DBUs) of the PROCAD database specified for routing.

MAXDeviation=N

Line 8

Specifies the maximum distance in an un-preferred direction which PROROUTE will utilize in order to route around an obstacle. If PROROUTE can not successfully route around an obstacle within the MAXD, then a through-hole will be placed when allowed.

This variable also defines the maximum deviation PROROUTE will utilize in standard routing movements, before placing a feed-through. The definition of this variable allows PROROUTE to maintain horizontal and vertical routes on separate layers of the board, which is the most effective method of board routing.

Default: 200

Example MAXD=250

GRID=N

Line 5

Establishes a routing grid search interval. The value specified is used extensively when PROROUTE changes directions in routing.

Default: 25

Example GRID=20

VGRID=N

Line 6

Establishes the via placement grid. The value specified is used extensively when PROROUTE places via. It is suggested that this value be at least two time the routing grid (2*GRID).

Default: 50

Example VGRID=75

TRACe=N

Line 13

Specifies the trace width which will be used when routing during the specified pass. While PROROUTE will use this trace width for routing, the true width flag of the database file generated will be turned off as default to speed the screen repaint process. You should use the "**SET TRUE_width=ON**" command in PROCAD to re-enable the display of true trace width. (See also the **SET TRUE_width** command in PROCAD Reference manual).

Default: 10

Example TRAC=12

HLAYer=N

Line 14

Specifies the PROCAD layer which will be used for placement of horizontal traces during the specified pass. The value specified must be between 1 and 98.

Default: 23

Example HLAY=15

NOTE: To create multilayer boards simply specify different horizontal layers for different passes.

VLAYer=N

Line 15

Specifies the PROCAD layer which will be used for placement of vertical traces during the specified routing pass. The value specified must be between 1 and 98.

Default: 20

Example VLAY=13

NOTE: To create single sided boards simply specify a common layer for HLAY and VLAY on all routing passes.

MAXTime=nn

Line 12

Sets the time limit in seconds that PROROUTE spends attempting to route a trace. A negative number demotes an unlimited time limit.

Default: UNL

Example MAXT=30

45DBend={ON/OFF}

Line 3

Enables/disables 45 degree bends.

Default: ON

Example 45DB=OFF

NOTE: The Memory Route algorithm defaults to 45DB=ON and the POWER/GROUND Route algorithm defaults to 45DB=OFF regardless of its setting for the specified pass. PROROUTE may violate trace to pad/via spacing around 45 degree bend if 45DB=ON. You should run the DRC command of PROCAD Xtra to verify and fix such violations.(See MODE keyword for more details).

BENDreduction={ON/OFF} No line

Enables/disables segment reduction optimization during routing. When BEND is set OFF routing could be as much as 50% faster; however, the resulting routes may wander and a high completion rate may not be achieved.

Default: ON

Example BEND=OFF

VIACcell=[Library]Cell_name No line

Specifies the cell name which will be used for via placements during the specified pass.

Default: [PKG]PAD030

Example VIAC=[CELLIB]VIA

NOTE: The specified cell must contain the DRC\$PADx data token (See Appendix-A, Attribute Data Files).

MODE=router mode Line 4

Selects a specific router mode. The valid router modes are:

M	Memory router
P	Power/Ground bus router
A	LOOKOUT/PROBE router

Default: A

Example MODE=M

NOTE: Each of these routing modes is discussed later in this manual.

ROUTER GLOBAL PARAMETERS

The following parameters may be stated once within the router control data file to be used globally for every pass which is set ON. It is not relevant to PROROUTE if these parameters are placed at the beginning or at the end of the router control data file.

MONOchrome={ON/OFF} No line

This token controls the colors used for displaying and editing the routing control parameters. This flag does not control the real time display of the routing process. The real time display of the routing process is controlled by the appropriate CGI/VDI drivers installed.

Default: OFF

Example MONO=ON

SLAYer=N Main Window

Specifies the router source layer. Only wires on this layer will be selected for routing by PROROUTE. The source layer is the PROCAD database layer which contains the RATS NEST of connections.

Default: 30

Example SLAY=26

PROROUTE utilizes internal design rules which are maintained during the routing process. These design rule checks (DRC) are completely user definable as follows:

TT=N

Main Window

Specifies the Trace-to-Trace air gap clearance for PROROUTE to utilize while routing. Prerouted traces will not be checked against this rule during the execution of PROROUTE.

Default: 10

Example TT=15

NOTE: The keywords for the DRC parameters are only two letters in length.

NOTE: The entire design may be verified a second time by using the PROCAD DRC facility available within PROCAD Xtra. It is highly recommended that you use the DRC command in PROCAD Xtra to verify routed results if 45DB=ON during any routing pass.

PT=N

Main Window

Specifies the Pad-to-Trace clearance which will be maintained by PROROUTE during routing.

Default: 10

Example PT=15

NOTE: In order for PROROUTE to verify the spacing of any item to a Pad, the Pad (or Device with pads) must include the DRC\$PADx token as discussed previously.

PP=N

Main Window

Specifies the Pad-to-Pad clearance which will be maintained by PROROUTE during routing.

Default: 30

Example PP=15

TK=N

Main Window

Specifies the Trace-to-Keepout clearance which will be maintained by PROROUTE during routing.

Default: 10

Example TK=15

NOTE: A Keepout is defined by PROROUTE as any void areas on a routing layer as defined by a closed polygon.

PK=N

Main Window

Specifies the Pad-to-Keepout clearance which will be maintained by PROROUTE during routing.

Default: 10

Example PK=15

You will find that certain router data files may be used repeatedly for a variety of different board designs. After running each route iteration it is wise to enter PROCAD to check on the progress of the design, and make any alterations to the existing routes which you feel might improve the design. After the power/ground route iteration you may want to establish a memory route iteration, which may only contain one active pass (all other passes being turned off. Then you may want to utilize a final general pass which routes the board using the Lookout Strategy router.

Each of these route iterations may be used in a variety of ways which will produce the most effective routing of the board layout. Variables within each route iteration may be changed to cause the router to take a new approach for the layout. You will find that a series of predefined router data files, which may be altered slightly, is all you'll need to successfully route most of your PCB layouts.

ROUTER SWITCHES

Allow Partial Routes: This switch allows for partially routed traces. If this switch is OFF, ProROUTE will restore rats nest to traces it could not route completely.

Enable Graphics Display: Enabling this box will cause PROROUTE to display the routing process in graphical form. This may slow down the routing process, but it gives you the opportunity to see routes as they are made, and terminate the routing process if you notice routing problems.

Rat's Nest Optimized ?: If you have already optimized your rats nest in PROCAD or you are re-entering PROROUTE, you should enable this box. This will cause PROROUTE to skip the rat's nest optimization phase. If you are not sure, leave this box disabled.

Trace Priority (ON=LONG, OFF=SHORT): This switch controls which trace lengths have high routing priority. If checked (ON), PROROUTE will attempt to route Long traces before short ones. For best results, you should not enable this switch, so that PROROUTE routes all short traces which have a lower priority of blocking routing channels.

PROCESSING RATS NEST

After clicking on the "**Auto Route**" button to start execution, PROROUTE returns the message "Processing RATS NEST". PROROUTE begins defining the route requirements by first breaking down the entire RATS NEST 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\$PADx 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. Therefore, you must insure that every cell in the CAD libraries of packages, pads, and vias, must include the DRC\$PADx token. This point is absolutely essential to total completion of the routing task. The format for the DRC\$PADx token is:

```
DRC$PADx Pad_size Layer(s) {pins}  
DRC$PADR Xwidth Yheight Layer(s) {pins}  
DRC$PADS Pad_size Layer(s) {pins}
```

Where:

x is either **C** or blank (Circular pad), **R** (rectangular pad) or **S** (square pad)

Pad_size is the diagonal size of the circular pad or sides of a square pad in database units.

Xwidth is the width and **Yheight** is the height of the rectangular pad in DBU.

Layer(s) is the valid layer(s) where this pad is found. If you specify layer 5 or ALL, PROROUTE will assume the pad to be a drill-through type connecting all trace layers.

{pins} are pin (I/O) numbers this pad is valid on. You must enclose the pin numbers in curly brackets. If you omit this field or specify ALL, PROROUTE will assume the pad to be used for all pins not explicitly defined with a DRC\$PADx token above.

Note: This data token should be placed in the attribute data file of all pads and devices with pads used in the database. (See also the section on Attribute Data Files).

Make sure that every cell in the CAD libraries of packages, pads, and vias, include the DRC\$PADx token in their attribute data files. This is of primary importance.

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.

While PROROUTE is processing the RATS NEST data, the screen will display the various router PASS and GLOBAL parameters as you have defined them in your router control file. You should pay close attention to this screen before the actual routing begins, to insure that the variables displayed are the ones which you wanted to define for this particular route iteration.

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 feed-throughs placed, and the percentage of completion of the entire board.

The important information on the status line is the pass being utilized, its percentage of completion, and the total percentage of completion for the 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 is desired.

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, or route another iteration of the just completed board, simply provide the database name and go from there, or click on the **"Exit"** button to exit PROROUTE.

ROUTER MODES

When using PROROUTE on a RATS NESTED board design there are three distinct types of routing strategies to utilize: Power/Ground Bus, Memory, and Lookout Strategy.

POWER/GROUND BUS ROUTER

The Power/Ground Bus router will only act on nodes with a signal name of VCCx or GNDx, where x represents either a blank (), or a single digit (1,2,3...). The purpose of this route iteration is to establish an effective Power and Ground route structure consisting of only vertical or horizontal traces. This route mode will only select for manipulation those named nodes which are either oriented in the horizontal or vertical axis. 45DBends have been established as default OFF, and MINAngle & MAXAngle variables are ignored for Power/Ground bus routing, any definitions to the contrary will be ignored by PROROUTE.

This is a T-junction router which, in order to be most effective, will tap into existing traces with a feed-through. All non vertical or horizontal nodes will be managed in a later route pass.

MEMORY BUS ROUTER

The Memory router, like the Power router, will only affect those nodes which are strictly horizontal or vertical in orientation. The purpose of this route strategy is to effectively handle all bussed arrays of nodes such as would be found in memory sections of a PCB. The Memory router will not route any nodes whose signal names are VCCx or GNDx; so, you may chose to run the Memory pass before any other route strategies.

The Memory router is the only route pass which actually consists of two internal passes (A and B), which route first the horizontal traces then the vertical. Only signals of at least 400 database units (DBUs), and no longer than the defined maximum length of selected traces (see Keyword Data "**MAXLength**".), will be affected by the memory pass.

Other route parameters which are affected during the execution of a memory pass are:

MAXVia is assumed to be zero. The memory routing is single sided and will not utilize vias.

MAXAngle, and **MINAngle**, are ignored. The memory router only affects the vertical and horizontal nodes.

MAXDeviation is assumed to equal two GRID points (GRID being defined by the user in the router control data file).

45DBend is assumed to be on. The memory router requires 45 degree bends in order to route the bussed node arrays.

BEND reduction is ignored.

Any variable definitions established during a memory pass contrary to the discussion above will be ignored, and PROROUTE will utilize internal values. All other variable definitions will be used as you have defined them.

LOOKOUT/PROBE STRATEGY ROUTER

The Lookout Strategy router will utilize all variables as you define them to create multiple pass routing strategies. The method which the Lookout router will utilize is to first try routing without any feed-throughs in either the horizontal or vertical layers. If a route can't be completed using this technique, a check will be performed to attempt a T-junction route utilizing any existing traces in the net to complete the route. Finally the Lookout router will attempt to route the node utilizing first one via, then two vias, and so on until the trace is complete. If the node can't be routed by PROROUTE utilizing any of these algorithms, the RATS NEST will be maintained in the database for manual routing, or another route pass.

One consideration for the Lookout router is to define the **MAXVias** as zero for any first passes, or for any subsequent passes with variable changes. If you do not establish the first route pass with zero vias, PROROUTE will assume that it is correct to utilize vias in routing during the current pass. If any changes are made to variable definitions for a subsequent route pass, the **MAXVia** must be reset to zero to allow PROROUTE to attempt effective routing.

See ROUTE RECOMMENDATIONS for more suggestions when working with PROROUTE.

ABORTING AND TERMINATING PROROUTE

You may abort a route session at any time by using the <ESCAPE> key, or the **"STOP"** button. When you abort a route session by hitting the <ESCAPE> key, PROROUTE give you the opportunity to modify route pass parameters and continue routing or save your results and about the route session, returning control to the main window, where you may enter a new database for routing. Aborting operation using the **"STOP"** button will return you to windows, and all routes will be lost.

When you abort a route session, through the use of the <ESCAPE> key or after all enabled passes are completed, 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 returns a message requesting an input database, either after completion of the route iteration or aborting the route session, you may terminate the execution of PROROUTE by clicking on the **"Exit"** button. At this point PROROUTE will stop execution, and return to Windows.

It is recommended that you take the time to view the routed database inside PROCAD and make any changes to the completed routes before moving into another route iteration. This recommendation is based on the understanding that PROROUTE is a design tool which will significantly improve the productivity of a designer. PROROUTE is not meant to replace a designer, and is most effective when used as a routing tool to increase the speed of the overall board layout process.

This is the benefit of the re-entrant nature of PROROUTE. You may execute PROROUTE, abort the route session at any time to make changes to the design, and begin PROROUTE again. By watching the method which PROROUTE is performing the layout of the PCB, you may be able to insure high completion rates. It is important to use the re-entrant capability of PROROUTE to work with this design tool to complete the design the way you want.

CLEAN-UP

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 feed-throughs 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.

FINAL ARTWORK

You are now ready to produce artwork masters on film or mylar, by utilizing Interactive CAD Systems' PROPLOT or GERPLOT. PROPLOT is the off-line plotting utility provided with PROCAD which will generate printer plots, ink plots on paper for checking and documentation. PROPLOT will also provide you with the capability of generating photographic ready artwork masters by using liquid ink on mylar.

GERPLOT is ICS' off-line Gerber Photoplotter utility, which will generate output files to disk to control either a Gerber Photoplotter, or the Fire 9000 laser plotter. Both of these plotters provide high-resolution, production ready artwork masters.

ROUTE RECOMMENDATIONS

This final section is a discussion of recommendations which may improve your ability to achieve consistent 100% results with PROROUTE. These recommendations are provided by beta-user's of PROROUTE, and have been used in actual route test cases.

- 1.) Route the power and ground nodes first taking advantage of PROROUTE's special POWER/GROUND BUS algorithm.

2.) Before moving into another route iteration, check the progress of the routing and make any modifications you feel are necessary. This procedure allows you to free certain channels of the design, which you feel might be used better by another route iteration.

3.) Route memory next, making use of PROROUTE's MEMORY algorithm. This algorithm will route all memory type RATS NEST connections (exclusively vertical and horizontal) to 100% completion. The MEMORY router utilizes a two-part route and will be displayed on PROROUTE's status line as Pass nA, and Pass nB.

4.) Route all remaining traces on a priority basis utilizing the Lookout Strategy router. This router utilizes three distinct algorithms to route the remainder of the board as effectively as possible. The priority basis for the Lookout Strategy router should include:

A.) Route all traces from edge connectors first, as these traces tend to be the toughest to get to.

B.) Route the direction of alignment of components first. This means that if the component placement is primarily vertical (orientation), then route the RATS NEST lines which are primarily vertical first. This may be effected by utilizing the PASS PARAMETER variables MAXAngle, and MINAngle, to limit the routed traces.

C.) Route the densest areas of the PCB first. To determine the "hot spots" in the layout simply look at the entire RATS NEST (possibly from a zoomed out view of the board) and locate the most tangled areas. These areas may be singled out for routing by transferring portions of the RATS NEST to a separate source layer for use by PROROUTE.

This presents another concept which may be utilized with PROROUTE. Selective routing of signals may be achieved by simply separating some RATS NEST wires onto a unique layer which will be used as the source layer for the router. Remember that the source layer is a GLOBAL variable in the router control file, and can only be changed by beginning a new route iteration. This does allow you the capability to selectively route certain portions of your design.

5.) To accommodate surface mount devices (SMD), which contain connection pads on one side of the board only (unlike through-hole packages, which pads go through every layer of the board) you must define a special DRC\$PAD token. The DRC\$PAD token for an SMD must include a physical layer specification other than 5. PROROUTE considers any DRC\$PAD token which specifies a layer of 5 or All to be a through-hole.

For instance, if you had an SMD mounted to the component side of the PCB (layer 20) then it must contain the token DRC\$PAD 50 20, where 50 is the physical dimension of the pad, and 20 is the layer which it exists on. PROROUTE would recognize that pad as existing only on layer 20, and could route under that pad on another layer, but would only connect to it on the specified layer.

6.) To route a single-sided board specify the HLAY and VLAY variables to be the same. This will insure that all routes are placed on a single database layer with no cross-overs. You will also have to greatly increase the value of the MAXD variable, to allow PROROUTE some room to route on. It is best to increase this MAXD value slowly, in increments, in order to achieve the best results.

7.) REMEMBER... A GOOD PLACEMENT IS THE KEY TO SUCCESSFUL ROUTING. Take the time to manipulate the RATS NEST of components and connections to insure an optimum placement. The time you take to achieve a good placement will pay you dividends during the route process.

TUTORIAL SESSION

DEMO ROUTE DATA FILE

The PROROUTE diskette includes a demonstration PCB database (ROUTE.DBF, and ROUTE.LIB) for use as a tutorial. By working interactively with PROCAD to view the PCB, and PROROUTE to perform route iterations, you can obtain a clearer understanding of how routing is performed. The ROUTE.DBF circuit includes a memory array, a variety of logic components, discreet devices, and an edge connector.

To start the tutorial session, Make sure PROROUTE is installed on your hard disk.

STEP 1- To invoke PROROUTE either by double clicking on its icon in the Program manager window, or from within PROCAD by clicking on its button in the Tool bar (Forth button).

Database input file:

STEP 2- Enter the file name of PROCAD database containing the placed components and the rats nest. For this demo, select or enter: **ROUTE .DBF**

Database Output file:

STEP 3- 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: **ROUTE1.DBF**

Enter Router Control file [ROUTER.DAT]:

STEP 4- Enter the name of the file containing the parameters to be used by PROROUTE. For this tutorial session, select or enter: **ROUTER.DAT**.

The parameters described in the keyword data file (ROUTER.DAT) are used to define both PASS and GLOBAL variables. By using this data file, the demo circuit was routed to a completion of 99.7%, on a standard 6 MHz PC/AT, within 15 minutes. This completion rate was achieved in a single, six-pass, route iteration as described 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).

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=300 allow a maximum deviation in trace placement of 300 mils. This maintains a 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

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.

begin pass 2 begin pass 2 variable definitions.

pass 2 on set pass 2 on for use by PROROUTE.

mode=m establish pass 2 as a memory route pass. The memory route pass does guarantee 100% completion of all memory type nodes oriented in either the vertical or horizontal direction. The memory route pass is the only pass which consists of two internal passes, and will be displayed as Pass 2A, and Pass 2B. The A and B internal passes concern themselves with horizontal routing, and vertical routing respectively.

maxv=0 allow no via placement during this route pass. The memory route pass defaults this variable to zero, as all memory routing is done without vias.

maxa=90 these two variables are ignored during this route

mina=0 pass (mode=m). The memory router affects only the horizontal and vertical nodes.

maxl=-1 the memory route pass will affect only those nodes which are at least 400 mils, but no greater than the defined maximum length (here maxl is defined as unlimited).

maxd=300 the maximum deviation is defaulted during this route pass (mode=m) to equal two grid units, so this maxd variable is ignored.

grid=25 define the routing and placement grid. It is highly recommended that routing grid be set at 25mils during memory routing pass. This will produce a much better routed results

trac=10 establish trace width for this pass.

bend=on bend reduction is ignored during the memory route.

45db=on 45 degree bends are required for the memory route pass to achieve 100% completion, so 45db is established on as a default during this pass (mode=m).

hlay=23 define the horizontal route layer

vlay=20 define the vertical route layer

In order to create multilayer PCBs, simply change the definition of the horizontal/vertical routing layers over the different route passes or iterations.

begin pass 3

pass 3 on

mode=a establish the Lookout Strategy router for this pass. The Lookout router will layout the remaining RATS NEST nodes as defined by all variables.

maxv=0 allow no vias during this route pass. It is important to begin each Lookout route pass (mode=a) with maximum via usage defined as 0 in order to achieve the most effective routing.

maxa=90 set maximum angle deviation of RATS NEST to 90 degrees.

mina=60 set minimum angle deviation of RATS NEST to 60 degrees. By setting MAXA and MINA to equal 90 and 60 degrees, you insure the selection of vertically oriented RATS NEST lines for routing first.

maxl=-1 set unlimited line length for route selection.

maxd=300 set maximum deviation to 300 mils.

grid=25 set routing grid to 25 mils.

trac=10 set routing trace width to 10 mils.

bend=on set bend reduction on. This variable is most important when applied to this type of routing pass (mode=a) in order to achieve the highest completion rate.

45db=on set 45db on to achieve 45 degree angle routes.

hlay=23 define the horizontal route layer.

vlay=20 define the vertical route layer.

begin pass 4

pass 4 on

mode=a

maxv=0 here again maximum via usage is defined to allow no placement of vias because, although it is still a Lookout router pass (mode=a), the variables have changed significantly. When route parameters change significantly from one pass to the next, MAXV should be maintained at zero to achieve the best results.

maxa=30 set maximum angle at 30 degrees.

mina=0 set minimum angle at 0 degrees. By setting these variables in this manner you insure the selection of horizontally oriented RATS NEST for routing during this pass.

maxl=-1

maxd=300

grid=25

trac=10

bend=on

45db=on

hlay=23

```
vlay=20  
begin pass 5  
pass 5 on  
mode=a
```

maxv=0 again maximum via usage is set to zero because of the change in variables during this pass. The variables which are changing are the maximum and minimum angle.

maxa=90 set maximum angle to 90 degrees.

mina=0 set minimum angle to 0 degrees. By setting these variables in this way you allow the selection of all remaining RATS NEST lines for routing during this pass.

```
maxl=-1  
maxd=300  
grid=25  
trac=10  
bend=on  
45db=on  
hlay=23  
vlay=20
```

```
begin pass 6  
pass 6 on  
mode=a
```

maxv=3 here maximum via usage is defined as three. This is the maximum number of vias allowed for any route requiring vias for completion. The maximum allowed by PROROUTE is 8 vias per trace; however, we have chosen three as the maximum limit for practical purposes. PROROUTE begins via usage at one via per trace and slowly expands to the limit if required.

It is because there are no changes in the variable definitions of this pass from the previous pass that MAXV is allowed to change. As a rule MAXV should only be set higher than zero when no variables change from one pass to the next.

```
maxa=90  
mina=0  
maxl=-1  
maxd=500  
grid=25  
trac=10  
bend=on  
45db=on  
hlay=23  
vlay=20
```

viacel=[pkg]pad030 set the library ([pkg]), and cell (pad030) definition for use as a via by PROROUTE. This via cell may be changed for different route strategies and iterations. The library name must be included in square brackets, and the cell name follows. It is important to insure that the cell chosen for use by PROROUTE contains the DRC\$PADx, and DRC\$VIA data tokens to maintain the established design rules during routing.

\$ 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.

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

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

pp=30 define the Pad-to-Pad clearance. In order for PROROUTE to check the clearance of a PAD, to anything, the library cell which is the pad must contain the data token DRC\$PADx. This is the same data token which is utilized by PROCAD Xtra's on-line DRC facility.

tk=10 define the Trace-to-Keepout clearance. The Keepout allows you to define regions on the PCB which may not be utilized for routing by PROROUTE.

It is suggested that you may learn more about how PROROUTE will complete the demo circuit, and other boards, by running the preceding route iteration one pass at a time. To do this simply set PASS 2 thru 6 OFF and run pass 1, when the router is finished take a look at the effects on the board design by entering PROCAD. Repeat this procedure by simply turning on and off different route passes each time.

By doing this simple exercise you will be able to get a greater understanding of how PROROUTE works, and how you can best work with PROROUTE.

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Utilities (SPECCTRA INTERFACE)

C*hapter*

3-4

CCT SPECCTRA INTERFACE

INTRODUCTION

The interface between Interactive CAD Systems' ProCAD and the Cooper & Chyan Technology (CCT) SPECCTRA Autorouter is by means of the IMPORT and EXPORT facilities of ProCAD. You use the Export command to generate the CCT SPECCTRA design file (".DSN"), and use the IMPORT command to bring back the Routes file (".RTE") from the CCT SPECCTRA autorouter.

PREPARING DESIGN FOR EXPORT TO CCT SPECCTRA

Create a PCB type database, with all components, keep outs, pre-routes (if any) and board outline defined. You should then connect all rats nests traces. This is done by using the RATS nest command in ProCAD.

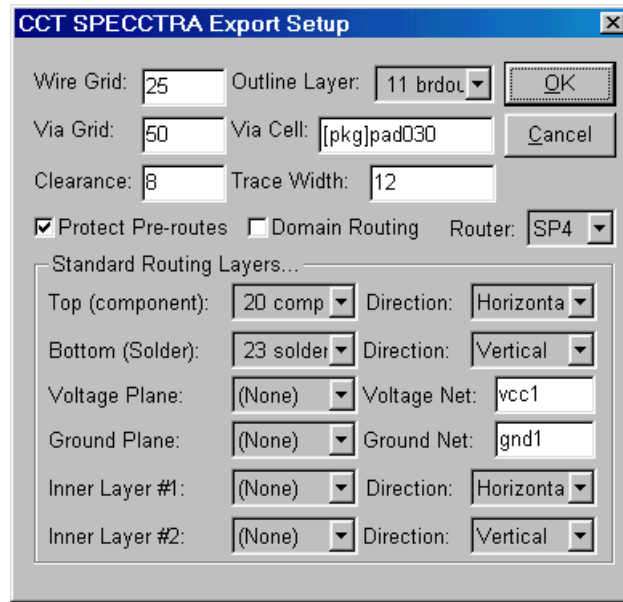
Note: All components placed on the PCB must have the appropriate DRC\$PADx... tokens specified. Please refer to the discussion of DRC\$PADx... keywords in the PROROUTE section of this manual and also Appendix A.

The board outline must be defined with the INPUT/LINE, INPUT/RECTANGLE or INPUT/CIRCLE (for circular boards) command. The outline should be placed on the appropriate outline layer (usually layer 11). If you have mounting holes or cut-outs, they should be drawn as polygons placed on the outline layer. For normal keep outs, the appropriate non-cell objects must be placed on the desired routing layer(s). If you have any pre-routes, you must use the INPUT/WIRE and/or STITCH command(s) to place them in the appropriate routing layers before exporting the database for SPECCTRA routing.

Note: The CCT Router interface does not require keep-ins as required in PROROUTE. The non-cell type object placed on the outline layer will suffice.

INVOKING THE CCT SPECCTRA EXPORT COMMAND

Once a proper database has been prepared as above, you should then invoke the **[File]Export...** menu item and select the CCT SPECCTRA design file format. You will be prompted for design file output name. This is the name of the design file that will be used by the SPECCTRA router. Provide a name (for clarity, it is best to use the name of your PCB database with ".DSN" extension). The following export dialog box will then be produced:



The following is a description of the above prompts:

Wire Grid:

This is the default trace routing grid. Please refer to your SPECCTRA manuals for details.

Outline layer:

This is the layer where the board outline is defined. The default is 11.

Via Grid:

This is the default via placement grid. Please refer to your SPECCTRA manuals for details.

Via Cell Name:

This is the name of the ProCAD cell which will be used as vias. You must provide a cell name and optionally a library name enclosed in square brackets "[]". The default is PAD030 cell from the PKG library.

Clearance:

This is the default routing clearance between traces and pads. Please refer to your SPECCTRA manuals for details.

Trace Width:

This is the default trace width. Please refer to your SPECCTRA manuals for details.

Protect Pre-Routes :

This flag when set instructs SPECCTRA not to rip-up and re-route all pre-routes placed by the user. Please refer to your SPECCTRA manuals for details.

Domain Routing:

This flag allows for routing only a desired section of the board. ProCAD will prompt for a routing window if this flag is enabled, otherwise the whole board will be routed.

Router type:

You may select between the following three CCT router types supported by ProCAD:

SP2 (2 signal layers plus 2 dedicated power layers with unlimited pins)

SP4 (4 signal layers plus 2 dedicated power layers with 4000 pins capacity).

SP10 (10 signal layers plus 2 dedicated power layers with unlimited pins).

Note: If you are using SP4 router on a large board, you may want to use the "Domain flag" to select a small routing area of no more than 4000 pins. Selecting SP10 router will expand the above dialog box to allow for additional layers property entry.

Routing layers and direction:

You should select your routing layers, preferred directions and any dedicated power and ground net names. Note: If you selected the SP10, you will have more layers displayed in the above dialog box.

Note: Un-used layers may be disabled. If you do not want to use dedicated power and/or ground layers, you should also disable them.

Press Okay to close the dialog box. ProCAD will then proceed to generate the CCT design file using the file name specified.

After the ".DSN" file has been generated, you should then invoke the SPECCTRA router, using the generated design file as input. Follow the instructions in the SPECCTRA router manual to complete the routing process.

RETURNING ROUTER RESULTS BACK TO PROCAD

When you have finished routing your design, you should generate the "Routes" file. This file contains only the routed traces and any vias generated by the SPECCTRA router. Please refer to your router manual for details.

Next, you are ready to import the route results to ProCAD.

1. Open your original ProCAD PCB design database (The file you exported to CCT SPECCTRA).
2. Invoke the **[File]Import...** menu item and select the CCT SPECCTRA format option. ProCAD will then prompt for the CCT SPECCTRA Routes file. You should select the appropriate routes (.RTE) file generated above.

ProCAD will then proceed to replace the rats nests with the proper routed traces and vias. Now you have a Routed design which you may clean up in ProCAD, and re-export for further routing if desired.

Note: Un-routed traces are retained as rats nests in the database. Further more, if for any reason routed traces are off pin (even by 1 database unit), a very rare occurrence, ProCAD will not remove the corresponding rats nest. You should manually delete such rats nest if they exist. This occurrence may also cause a false DRC "short at pad" error messages in ProCAD.

Utilities (GERPLOT)

C*hapter*

3-5

GERPLOT UTILITY

INTRODUCTION

GERPLOT is the photo plotter utility program for PROCAD. It allows you to produce a high quality photo artwork from a Gerber format compatible photoplotter. All versions of Gerber controllers are supported. GERPLOT also supports the extended number of flash apertures of the MDA 9000 laser photoplotter.

The program also generates a PROCAD compatible macro command file which is recommended as a final check before actually generating a photoplot.

USING GERPLOT

Before executing the program, insure that the following files are in the the GERPLOT.working directory, (usually \WPROCAD):

- A. The plot control file(s) generated from the database using the PLOT/G command in PROCAD (.GCF files)
- B. The text font table file (TEXTFONT.BIN or TEXTFONT.TXT)

The database file (.DBF) from which you want to photoplot the information and all library files (.LIB) used in the creation of that database must be placed in the database & library search directories.

DATE & TIME STAMP

GERPLOT will automatically expand the following characters enclosed in "<" and ">" brackets to their values in the plotted file:

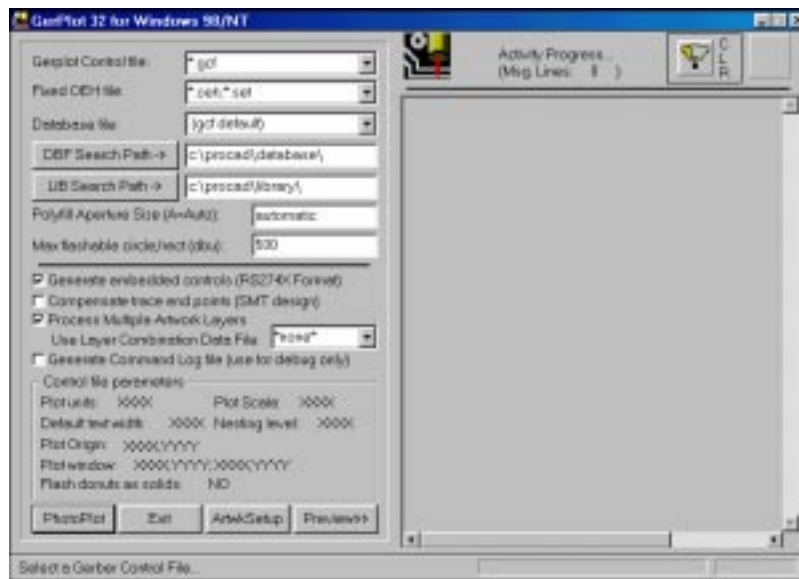
TIME stamp

- | | |
|--------|---|
| <DATE> | Will be expanded to current system date in the DD-MM-YY format. |
| <TIME> | Will be expanded to the current system time in HH:MM:SS format. |

The <DATE> and <TIME> characters must be placed in your drawing in PROCAD with the **Input/Text** command (or **Text** command from the Tool bar or **[Place]** menu.). The characters must appear as shown above. The angled brackets are not optional; they are required.

You may invoke GERPLOT from the Tool bar within the PROCAD editor, 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.

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



GERPLOT Window Control-menu Box:

The Upper left corner of the GERPLOT window is a Control-menu box [-]. This Control-menu box when invoked drops down a menu for controlling the various aspects of the GERPLOT window, and also submenus for displaying the current GERPLOT 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.

Utility Programs Guide...

The following is a description of the prompts produced by the GERPLOT application:

"GERPLOT Control File"

Enter or Select the FILE_NAME.GCF that contains the portion or portions of the drawing that you want to plot. Instructions for generating this GERPLOT Control File are provided later in this section. The control ".GCF" file must be available in GERPLOT working directory, usually \WPROCAP. You may not specify data path for the file.

"Database & Library file search paths"

Normally, the ".GCF" file contains the data paths for Database and Library files that were in effect when it was created. You may provide a new database and library search paths if you so desire.

"Database file"

The default database file is the one from which the ".GCF" file was created. This prompt allows you to override the default. The database file must be located in the Database search path defined above. You may not specify a data path for this file.

"Fixed OEH file"

You should enter the name of your fixed OEH setup file. (The format of this ASCII file is described at the end of this section). If you do not have such file, leave this field blank. GERPLOT will process your database and generate the required .SET file. If you specify an .OEH file with wrong data format, GERPLOT will process your database and generate a correct one for you.

"Compensate trace endpoints [SMT design]"

The default apertures used for GERBER photoplot line/trace draws is usually circular. This typically causes an over-extension of trace end points by 1/2 the trace width. If traces end on larger pads, this is quite alright. However, if your traces are large, and end up on small pads, such as those used for SMT designs, this may cause some problems. This prompt allows you to move the trace back by 1/2 a trace width. You will end up with gaps around the trace end points. This may be more problem than it is worth in most cases. As a rule of thumb, if you

are using square trace apertures, you should compensate trace end points, if you use circular apertures (default), **DO NOT** compensate trace end points.

"Maximum flashable circle/rect (dbu)"

This is the maximum size of circles and rectangles that can be flashed. If a rectangle or circle is greater than this value, it is drawn as a filled polygon. The default is 500 database units (dbu).

"Layer Combination data file"

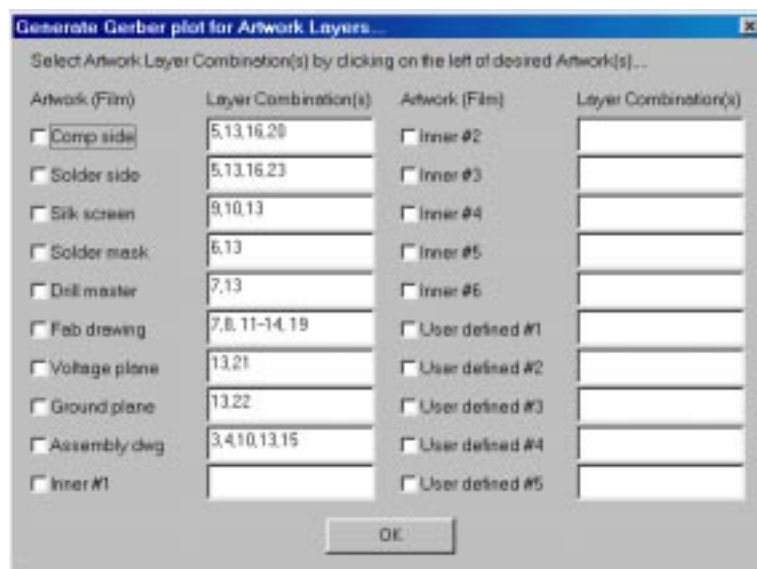
When generating multiple artworks (default), you may specify an ASCII file that contains the layer combination for the different artworks (".LCF" file). You must specify one line of data per artwork layer combination.

Example:

```
5,13,16,20
5,13,16,23
9,10,13
```

The above data in the layer combination file (".LCF") will cause GerPlot to generate 3 artwork files. The first file will have Gerber data for component side (layers 5,13,16,20), the second Gerber data file for Solder Side (layers 5,13,16,23) and the third Gerber data file for Silk Screen (layers 9,10,13). You may add comment lines to this file (first character on a line must be "\$").

If you prefer not to use a layer combination file, you may click on the "ArtwkSetup" button, and GerPlot will display the following dialog box:



The default ProCAD's **[View]Preference...** artwork combination setting is pre-loaded in the Artwork Layer Combination dialog box above. You should select (✓ check mark on) the layer combination(s) desired. You may also modify the pre-loaded layer combinations for the current GERPLOT session only.

"Polyfill Aperture size"

When polygons are filled (including automatic ground plane polygon), GERPLOT uses a pen of the above defined size for filling. An automatic selection will cause GERPLOT to analyze your database for the smallest trace size for filling. See the discussion on **"Filled polygons"** later in this section for more in depth discussion of polygon filling algorithm.

"Generate command Log file"

A Yes response to this prompt (Box checked) will cause GERPLOT to also generate the "GCF_FILENAME.CMD" file. **Note: You should leave this box unchecked unless you are trying to debug generated Gerber data file.**

"Process multiple artworks Layers"

If you want to process only the layers specified in the .GCF file, you should respond "NO" to the above prompt (Box un-checked).

The following messages will be displayed in the Activity Progress window when you click on the **"PhotoPlot"** button:

"nn Artwork(s) will be processed ..."

The following messages will then be displayed in the status line:

"Loading Database file XXXX.DBF .."

"Preprocessing data for OEH setup"

GERPLOT will load the database file from the information on the .GCF file and preprocess the data for Optical Exposure Head (OEH) setup. During the OEH setup processing phase, GERPLOT checks the database for the proper flash apertures and patterns to be used. This process usually takes a few minutes depending on the complexity of the data to be plotted. Refer to the end of this section for the criteria used by GERPLOT for selection OEH apertures.

"Do you want to modify OEH setup (Y/N) ?"

GERPLOT will display the generated OEH setup table, and prompt for any modification(s) desired.

If you are going to modify the OEH table, you should note the following points:

1. Special flash apertures and text trace aperture width cannot be modified within GERPLOT by the user. If you want to modify the special flash apertures or text trace aperture width, this should be done in the Database/Library level with PROCAD before invoking GERPLOT.

2. Modifying the OEH table may result in some entities not be plotted. You should only modify the OEH table when you have an OEH table overflow error to select the critical entities you want plotted, given the limitations on the number of OEH positions.

3. To exit OEH table modification process, respond with just a <RETURN> in response to the "OEH position" prompt.

GERPLOT will then generate the OEH setup file GCF_FILENAME.SET.

GERPLOT will then proceed to generate the GCF_FILE.GER file which contains the appropriate GERBER photo-plot data.

Note: If multiple artworks were processed, GERPLOT will generate files GCF_FILE.G01 - GCF_FILE.Gnn and GCF_FILE.C01 - GCF_FILE.Cnn (where nn=02 - 99).

When all the necessary files have been generated, the following message and prompt will be displayed:

"GERPLOT Complete..."

At this point, you have the option of entering another ".GCF" file, or clicking on the **"Exit"** button to exit the program.

SETTING ATTRIBUTES FOR SPECIAL FLASH

GERPLOT makes use of attribute data defined for cells within the plotting database to define special flash aperture. GERPLOT recognizes the following token/keywords:

**GER\$FLASH Aperture_name n1,n2
+ n3 ... nn**

Where:

GER\$FLASH is the keyword instructing GERPLOT to reserve an OEH position for a special flash aperture.

Aperture_name is the name of the aperture pattern (Up to 8 characters).

n1,n2 ... are layers where these flash patterns are valid.

Continuation lines and range specifiers are permitted. The first character of a continuation line must be a "+" followed by 1 or more blanks.

**eg: GER\$FLASH TGT9 1,2,4-8,20,30-35
 + 44,48**

Multiple GER\$FLASH statements can be defined for any given cell provided that no two GER\$FLASH statements should have the same layer assignments for the same cell. When GERPLOT encounters a valid flash pattern, it will flash that pattern, and will not process that cell any further. This capability allows for nested cells to have special flash patterns at any nesting level. As an example, a PCB layout footprint can have just the pad stacks flashed with special flash patterns while the rest of the cell will be plotted with traces, flash circles and squares.

Also a cell could be flashed with different patterns in different layers. This eliminates the need to define multiple cells for different artwork layers of the same database.

SQUARE & CIRCULAR FLASH APERTURES

GERPLOT will flash all filled rectangles and circles in the database. You do not need to define filled circles and squares as special flash patterns. If a filled circle or square is greater than 1 inch in diameter or width, GERPLOT will not flash that pattern - a slower scan line filling algorithm will be used in this case.

PROCESSING IRREGULAR FILLED POLYGON

GERPLOT will use scan line filling algorithm on all irregular filled polygons. The trace width used will be the smallest trace width defined in the database. You may also specify a "**PolyFill Aperture size**", and GERPLOT will use that aperture size instead.

WARNING: If you have large filled polygons, and are using small traces for filling, your GERBER files could be extremely large !!!.

GENERATING PLOTTER CONTROL FILE

The plotter control file (.GCF file) is normally generated in PROCAD using the /G option of the plot command: eg. PLOT/G. This command is similar to the regular PLOT command described in the Command Reference section of the Reference manual with the following differences:

1. There is no prompt for plotter paper size. This is not necessary in this context since the output medium is a photo film of various sizes depending on the photoplotter hardware capabilities. The prompt requesting for paper template positioning has also been eliminated.

2. There is no Portrait/Landscape orientation data, instead, you are asked if you want donuts flashed as circles. If your response is yes, GERPLOT will treat all donuts not defined as a cell with special flash attributes as a filled circle. The hole in the middle of the donut will be lost.

3. There is no prompt for pen thickness. This is irrelevant in this context. GERPLOT will however prompt for the trace width to be used for vector text plotting. This aperture will be used for filling polygons and for plotting lines or wires defined as 1 unit width provided that the minimum trace width defined in the database is less than 10 database units. Lines or wires defined with explicit trace widths greater than 1 unit will not be overridden by the text trace width.

FORMAT OF USER SPECIFIED OEH SET-UP FILE

The fixed OEH file is an ASCII file which you can create using any text editor or word processor in ASCII mode. The file can contain the following keywords:

TR	nn	Trace aperture of size nn
RO	nn	Round flash of size nn
RE	xWid,yHgt	Rectangular flash (width=xWid, Height=yHgt)
SQ	nn	Square flash of size nn
SP	name	Special flash (aperture name=flash)

Any line beginning with "\$" character is treated as a comment line, and will be ignored by GERPLOT.

You should only include one keyword/size definition per line. The first line will represent aperture position 1, line 2 = position 2 etc.

Example:

```
TR    10
SQ    12
RO    50
RE    10 15
```

The above example assumes a 10 unit trace at D10, 12 unit Square at D11 , 50 unit Round flash at D12 and a 10 x 15 unit Rectangular flash at D13.

You may include D-CODES if you don't want GERPLOT to map line positions to D-Codes.

Example:

```
D10  TR 10
D15  SQ 12
D20  RO 50
D12  RE 10 15
```

The above example assumes a 10 unit trace at D10, 12 unit Square at D15 , 50 unit Round flash at D20 and a 10 x 15 unit Rectangular flash at D12.

Comment lines are ignored and not counted as aperture positions.

FILES GENERATED BY GERPLOT

As mentioned above, GERPLOT will generate three files:

GCF_FILENAME.SET

This file contains the data for proper OEH setup and the necessary data required for setting up the photo-plotter hardware (eg: units, coordinate mode, etc).

GCF_FILENAME.CMD or GCF_FILENAME.Cnn

This file contains the proper PROCAD macro command data for displaying the data to be plotted in PROCAD. You should invoke PROCAD and issue the command:

MACRO GCF_FILENAME

This will cause PROCAD to display the interpreted data as would be seen by the photo-plotter. It is recommended that you view the GERPLOT interpreted data before actually generating the photo artwork.

Note: The data will be displayed in 1X scale only. References made to special flash patterns will appear as an "INPUT CELL" command with the cell name being the special flash pattern name, from library GERBER.LIB. This means that you must have GERBER.LIB cell library containing drawing information for all your referenced special flash patterns. Due to the size of data generated, you might experience database overflow error in PROCAD when attempting to view large drawings or drawings with a lot of filled polygons. Occasionally due to display resolution and consequence arithmetic truncation of display units, circles and squares may not display properly or might appear to be missing. If this occurs, you should use a text editor to examine the ".CMD" file for the presence of that circle or square to be sure that the data is not actually being dropped by GERPLOT.

Tips: ProCAD Advanced for Windows version 3.0 or higher supports GERBER import command ([Files]Import menu item). You should use this feature instead of the command file method for better accuracy. See GERBER IMPORT command in the Reference section for details.

GCF FILENAME.GER OR GCF FILENAME.Gnn

This is the actual data for the photoplotter. The data file is in ASCII with variable length records (maximum record length of 128 characters), and end of buffer character "*".

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Utilities (PRODRILL)

C*hapter*

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PRODRILL UTILITY

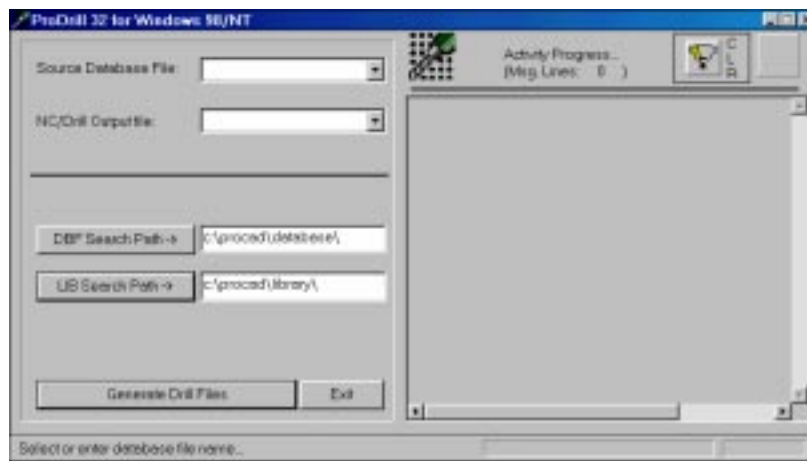
INTRODUCTION

PRODRILL is a utility program provided to generate an N/C Drill file from a PROCAD PCB database. PRODRILL currently supports the Excellon N/C format.

USING PRODRILL

You may invoke PRODRILL from the Tool bar within the PROCAD editor, 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 this concept is not familiar to you.

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



PRODRILL Window Control-menu Box:

The Upper left corner of the PRODRILL window is a Control-menu box [-]. This Control-menu box when invoked drops down a menu for controlling the various aspects of the PRODRILL window, and also submenus for displaying the current PRODRILL 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.

The following is a description of the prompts produced by the PRODRILL application:

Source Database File:

This is the name of the PROCAD PCB-type database from which the Drill file will be extracted. You may not specify a data path with the file name. The source database must be location in your Database search path.

NC/DRILL Output File:

This is the base name of the files which PRODRILL will generate. PRODRILL generates two files using the extensions **".DST"** and **".DRL"**. You may leave this field blank if the output files should be named the same as the input file with extensions **".DST & .DRL"**.

Database & Library Search Paths:

You should insure that all databases, and their supporting libraries are in their proper search paths. Data files generated by PRODRILL will be placed on PRODRILL working directory, usually \WPROCAD.

By clicking on the **"Generate Drill Files"** button, PRODRILL will process the .DBF file and generate a .DST and .DRL files.

The following messages will be displayed on the status line and Activity Progress windows when PRODRILL starts execution:

Loading Database File ...

Loading Complete ...

Generating Drill Matrix ...

DATA TOKENS

PRODRILL searches the PROCAD database, and all the supporting libraries for the DRC\$HOLE data token.

This data token must be included in all symbols which require drilled holes, such as a via cell, a DIP package, or discrete components.

The format of the token is:

DRC\$HOLE n X

Where n is the size of the drill in database units.

NOTE: Usually database units will be defined as MILS for any ProCAD PCB databases.

X is the pin number to which this token is applied. The value of **X** may be specified as a single pin, a group of individual pins, a string of pins, or all pins.

Examples of valid tokens are:

```
DRC$HOLE 36 1
DRC$HOLE 109 1 2 6 9
DRC$HOLE 45 3-11
DRC$HOLE 36 ALL
```

Notice the various ways the value of **X** may be specified.

A string of pins which exceeds the line-length of the Data Attribute file may be continued by using the plus (+) sign as follows:

```
DRC$HOLE 36 1 3 5 8-10 12 15-20
+ 22 29 31-35 37 40
```

For PRODRILL to recognize the hole locations of a multiple pin device, pin numbers must be logically assigned during the CELL FORMATION process.

If no pin numbers are logically assigned, the DRC\$HOLE token will be assigned to the cell reference coordinate.

*NOTE: A DRC\$HOLE token must be assigned to every device utilized in a PCB design which requires a drilled hole. The **DRC/H** command ([Check]Show **DRC\$HOLE graphics** menu item) in PROCAD will display the graphical interpretation of all DRC\$HOLE keywords used in the database.*

DATUM CELL

PRODRILL generates the Drill file in an optimized fashion from the lower left corner of the database. This allows the NC Drill Machine to process the board in the most efficient manner.

PRODRILL searches the database for a cell which is named **DATUM**. PRODRILL will utilize this cell's reference as the relative 0,0 coordinate from which all other drill coordinates will be extracted. By using the DATUM cell, you may effectively control the coordinates output in the Drill file.

OUTPUT FILES

There are two files output from PRODRILL, the Set-up file (.DST) and the Drill file (.DRL). These are ASCII files and will be located in PRODRILL working directory, usually \WPROCAD.

When these files have been generated PRODRILL displays the following information messages in the Activity progress window:

Drill Matrix Generated ...
Holes, # Different sizes found ...
Generating Output files ...

SETUP FILE: "filename.DST"
DRILL FILE: "filename.DRL"

At this point you may enter another PROCAD PCB database to be processed, or you may exit PRODRILL by clicking on the **"Exit"** button.

If you receive the following warning:

WARNING: NO DATUM CELL SPECIFIED
LOWER LEFT HOLE COORD. ASSUMED

PRODRILL could not locate a DATUM cell in the database file. When this happens, PRODRILL simply designates the lower-left hole coordinate as the relative 0,0 point. You should provide a dummy cell named "DATUM" in your database. This dummy cell's reference coordinate will be the 0,0 reference point.

Note: You may define the DATAUM cell in any library of your choice.

SETUP FILE:

The Set-up file contains information representing the units of measure and the drill sizes. This file is utilized by the machinist during actual fabrication of the PC board.

The following is an example of a PRODRILL Set-up file:

EXCELLON N/C Drill Data - Database: PCB-PKG.DBF Date: 01-FEB-96

Part Program : Version 1, (Format 2)
Drilling Units: MILS
Drilling Mode : ABSOLUTE
Leading Zero : PRESERVED
Trailing Zero : SUPPRESSED

Tool Position	Hole Size	Hit Count
1	36 MILS	254
2	109 MILS	2

Total Holes: 256

PRODRILL extracts drill sizes, hit count, and total number of holes.

DRILL FILE:

The Drill file includes EXCELLON NC commands, as well as X-Y coordinates of the holes to be drilled. This file is a simple ASCII data which may be transferred on disk or tape to an NC Drill machine.

An example of a Drill file follows:

```
%  
M72  
G90  
T1  
X000100Y004400  
X000100Y004300  
X000100Y004200  
X000100Y004100  
X000100Y004000
```


T2

X000100Y004500

X000100Y004900

X000100Y005200

X000100Y005400

X000100Y006000

T3

X000610Y004400

X000720Y004300

X000900Y004200

X001100Y004100

X002100Y004000

Utilities (PROPDIF)

C*hapter*

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PEDIF INTERFACE (PROPDIF)

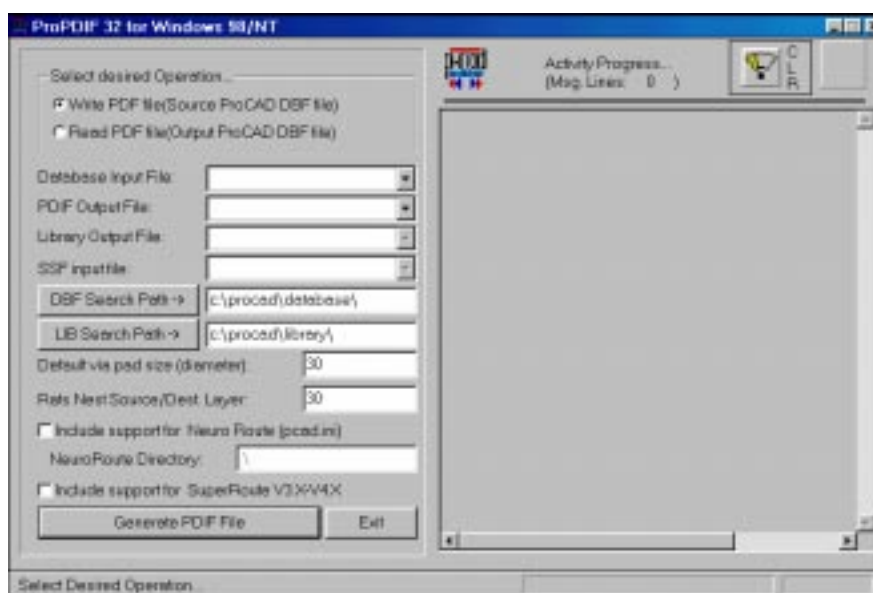
INTRODUCTION

PROPDIF (PROCAD File Transfer Program) is a utility used for generating and reading ASCII format of PROCAD databases and libraries. The ASCII data format generated and accepted is compatible to Personal CAD System's (PCAD) PDIF version 4.0. With the PROPDIF program, the user can transfer PROCAD database and libraries to other systems, including the Superoute software from Router Solutions, Inc. Also databases from P-CAD and systems that generate P-CAD compatible PDIF files such as MASSTECK/OrCAD can be converted to PROCAD's binary database format.

USING PROPDIF

You may invoke PROPDIF from the Tool bar within the PROCAD editor, 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.

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



PROPDIF Window Control-menu Box:

The Upper left corner of the PROPDIF window is a Control-menu box [-]. This Control-menu box when invoked drops down a menu for controlling the various aspects of the PROPDIF window, and also submenus for displaying the current PROPDIF 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.

The following is a description of the prompts produced by the PROPDIF application:

Database & Library Search paths

You should insure that all databases, and their supporting libraries are in their proper search paths. Non Database & Library data files generated by PROPDIF will be placed on PROPDIF working directory, usually \WPROCAD.

Writing a PDIF File from a ProCAD binary database file

If you choose the "**Write PDIF file**" option, the program will generate an ASCII PDIF format of the specified PROCAD database. The following prompts and messages will be enabled and displayed:

"Database Input file:"

Enter or select the PROCAD database file name of the design you want converted to PDIF format. You may not specify a data path for this file. You must locate this file in the your database path.

"PDIF Output file: "

Enter the name of the output file desired. The default name will be the name of the database entered above but with the extension .PDF. If you enter the output file name, you may not specify a data path for it. PROPDIF will place this file in PROPDIF working directory, usually \WPROCAD.

"Default Via pad size"

Enter the size of your via cell. The size must be in your database units (DBU), and it must be the diagonal measurement of the circular via pad.

"Rat's Nest Source/Destination Layer"

Enter the Layer number in PROCAD where rats nests signals are placed. This is usually layer 30.

"Include Support for SuperRoute V3.X-V4.X"

If you are generating file for use with the Super Router V3.X-V4.X, you should enable this box. If however, you are generating file for Max Route, or for P-CAD, this box must not be enabled.

Click on the **"Generate PDIF File"** button to start generating PDIF file. PROPDIF will then display the following messages in the status line and Activity Progress Window:

Loading <fname>
Loading Complete

If the database has not been previously resolved by PROPDIF or PROCAP since the last database save, the following additional messages will be displayed while the database is being resolved:

Resolving netlist
Netlist Resolved

PROPDIF will then process your PROCAD database and generate the PDIF compatible file. PROPDIF will also generate a pad/pin data file and Max Route MDF file. These files will have the same primary name as your PDIF output file but with the extension of .PAD and MDF respectively. The format of this file will be discussed later under the section on Files generated by PROPDIF program.

After PROPDIF has finished processing your database file, it will again be ready to [R]ead or [W]rite another PDIF file.

Reading a PDIF File & generating ProCAD binary database file

If you choose the "**Read PDIF file**" option, the PROPDIF program will read a PDIF file and generated PROCAD binary database. The following prompts and messages will be enabled and displayed:

"PDIF Input file [.PDIF] :"

Enter or select the PDIF data file name you want converted to PROCAD binary database file. You may not specify a data path for this file, it must be located in PROPDIF working directory, usually \WPROCAD.

"Library output file [.LIB]:"

Enter the name of the library file you want as a default for cell accesses, and that will be linked with the generated PROCAD binary database. If you specify a file that does not exist, PROPDIF will create one.

"SSF file name [.SSF]:"

Enter the name of your special symbol file. The format of this file is discussed in the next section. If you do not enter an SSF file name, PROPDIF will use your default pad stack cell when generating the DBF file (see the next prompt below).

"Default pad stack cell name [[PKG]PAD030]:"

Enter the name of PROCAD cell you want to use as a default pad and Via. If you are reading PDIF file generated by Autorouter programs such as Superoute or Max Route, this cell name will be used for via's generated by the auto-router. This is usually referred to as pin/via type 0 in the PDIF file.

If you entered a valid SSF file name above, PROPDIF will read this file and use pad/pin type 0 for vias. It will also display the number of valid entries found in the SSF file.

"Rat's Nest Source/Destination Layer"

Enter the PROCAD Layer number where rats nests would be placed. This is usually layer 30.

Database Output file name:

Enter the name of the output file desired for PROPDIF generated PROCAD binary database. If you specify the name of an existing file, PROPDIF will display the following warning message after clicking on OK to start execution:

**-WARNING- database file already exists.
[U]pdate, [D]elete or [E]xit (U) ?**

If you choose U, PROPDIF will use the header record contained in this file, and update the database entities per PDIF data. If you choose D, PROPDIF will delete the file and generate a new one with default header record, and entities data per PDIF file. If you choose E, PROPDIF will stop processing the PDIF file and return you to the first prompt.

Note: PROPDIF will read the PDIF file to extract the default output name contained in the "COMPONENT statement of the PDIF file. If you did not specify an output file name, PROPDIF will default to this extracted file name with ".DBF" extension added.

Click on the **"Load PDIF File"** button to start execution. PROPDIF will display series of messages as it is processing your PDIF file, and will finally generate a PROCAD binary database file.

After PROPDIF has finished processing your PDIF file, it will again be ready to [R]ead or [W]rite another PDIF file.

FORMAT OF SSF FILE:

The Special Symbol File (SSF) is an ASCII file which contains the cross reference data between Pin/Pad/Via type and the corresponding PROCAD cell name. The format is:

nn [Library]cellname

where nn is the Pin/Pad/Via type (0-255), and

[Library]cellname is the PROCAD cell to be used for that pin/pad/via type. Note that the brackets "[]" are required to signify a PROCAD source library. An example of a typical SSF file will be:

0	[PKG]PAD030
1	[PKG]PAD050
2	[MYLIB]SMTPAD25

Note: If the first non blank character on a line is non numeric, PROPDIF will treat that line as a comment.

Note: Pin/Pad/Via type above are those generated by the "W" command of the PROPDIF program's .PAD description file.

FILES GENERATED BY PROPDIF

The PROPDIF generates a PDIF file (.PDF), a Pad description file (.PAD) and Max Route pad definition file (.MDF) when the "**Write PDIF file**" option is executed. The PDF file generated is compatible to PCAD's PDIF version 4.0 data format with indentation enabled. The .PAD file generated is compatible with Superoute pad description file, and the .MDF file is compatible with Max Route pad definition file.

For a description of the MDF file format, please refer to your MAX Route users manuals (P-CAD interface section).

Pad description file format:

For each unique pad type and layer definition in the layout, there is an entry in the pad description file as follows:

t l s n

where:

- t** is the number of the pad type as defined in the PDIF file (0-255)
- l** is the layer number that is defined for the pad by the DRC\$PAD attribute of PROCAD cell. An * denotes all layers.
- s** is the shape of the pad, either S(square), C(circle), or R(rectangle). PROPDIF and PROCAD currently supports only circular pad DRC\$PAD token. If you are using Superoute, you may modify the pad shape if you are using a non-circular pad.

n is the size of the pad. For a square, this represents the length of a side, for a circle, it represents the diameter (value specified in the DRC\$PAD token). For a rectangular pad, n is 2 values representing the sides of the pad.

A semicolon or percent character ";" or "%" at the beginning of a line signifies a comment line.

LIMITATIONS OF PDIF

Almost all data created by PCAD can be accurately translated to PROCAD. However, the following limitations exist:

1. PROCAD cells are translated to PCAD components in the PDIF file. PCAD cells are however files on your disk with extension of PRT. For this reason, you should avoid using period "." as part of your cell name in PROCAD if you plan to port the data into PCAD. Some characters such as "%", "@", "." etc are reserved by PCAD, so you also avoid using these characters as part of a cell or signal name in PROCAD.

INTERFACE TO MAX ROUTE & SUPER ROUTE

PROPDIF utility may be used for generating the PDIF ASCII file required for interfacing to Max Route and Super Route. When using these Routers, you should purchase and install their Personal CAD systems interface options (P-CAD)

Step 1:

Use the PROPDIF utility to generate the PDIF file from your ProCAD database file. (PCB file with parts placed and Rat's nest connected). See page 280 above.

Step 2:

Use the Max Route extractor utility (or RPCAD utility if you are using Super Route) to read in the PDIF file generated in step 1 into your Max or Super Route.

Step 3:

Route the board using the appropriate router (Super Route or Max Route). Follow the instructions in your Router Manual.

Step 4:

Use the Max Route back annotation utility (WPCAD utility if you are using Super Route) to write a PDIF file from the routed file.

Step 5:

Use PROPDIF utility to read the PDIF file generated in step 4 into PROCAD binary database format. See page 281 above.

NOTE: PROPDIF is equivalent to the combination of PDIFIN and PDIFOUT utilities provided by P-CAD (Personal CAD Systems). All references to PDIFIN and PDIFOUT in your router manual should be interpreted as PROPDIF read and write PDIF file commands respectively.

C***chapter***

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AUTOCAD DXF INTERFACE (PRODXF)

INTRODUCTION

PRODXF (PROCAD DXF Transfer Program) is a utility used for generating and reading ASCII format of PROCAD databases and libraries. The ASCII data format generated and accepted is compatible to AutoCAD DXF file format. With the PRODXF program, the user can transfer PROCAD database and libraries to other systems capable of reading AutoCAD DXF files. Also databases from systems capable of generating DXF files can be converted to PROCAD's binary database format.

USING PRODXF

You may invoke PRODXF from the Tool bar within the PROCAD editor, 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.

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



PRODXF Window Control-menu Box:

The Upper left corner of the PRODXF window is a Control-menu box [-]. This Control-menu box when invoked drops down a menu for controlling the various aspects of the PRODXF window, and also submenus for displaying the current PRODXF 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.

The following is a description of the prompts produced by the PRODXF application:

Database & Library Search paths

You should insure that all databases, and their supporting libraries are in their proper search paths. Non Database & Library data files generated by PROPDIF will be placed on PROPDIF working directory, usually \WPROCAD.

Writing a DXF File from a ProCAD binary database file

If you choose the "**Write DXF file**" option, the program will generate an ASCII DXF format of the specified PROCAD database. The following prompts and messages will be enabled and displayed:

"Database Input file:"

Enter the PROCAD database file name of the design you want converted to DXF format.

"DXF Output file [fname.dxf]: "

Enter the name of the output file desired. The default name will be the name of the database entered above but with the extension .DXF.

"Use Vector Text Font (txtfont file defs)"

Enable this box if you want all text entities to be generated as vector lines based on your TEXTFONT.TXT file definitions. You should enable this flag if you have mirrored cells or text in your database.

Note: If you are generating DXF file from a PCB database, ProDXF will automatically enable this flag and thus use vector text fonts.

PROCAD to AutoCAD units scale factor [.001]:

Enter the multiplier scale factor to be used when translating PROCAD units to AutoCAD units. If your PROCAD units are in mils, and your desired DXF (AutoCAD) units are in inches, use a scale factor of 0.001 (0.001 inches = 1 mil). The default is 0.001

Generate File for Layers:

Enter the ProCAD layers you want to include in the generated DXF file. The default is ALL layers.

Click on the "**Generate DXF file**" button to start execution. PRODXF will then display the following messages on the status line and Activity Progress window:

Loading <fname>
Loading Complete

PRODXF will then process your PROCAD database and generate the DXF compatible file.

After PRODXF has finished processing your database file, it will again be ready to [R]ead or [W]rite another DXF file.

Reading a DXF File & generating ProCAD binary database file

If you choose the "**Read DXF file**" option, the PRODXF program will read a DXF file and generated PROCAD binary database. The following prompts and messages will be enabled and displayed:

"DXF Input file [.DXF]: "

Enter the DXF data file name you want converted to PROCAD binary database file.

"Library Output file [CELLIB.LIB]: "

Enter the name of the library file you want as a default for cells (blocks) defined in your DXF file, and that will be linked with the generated PROCAD binary database. If you specify a file that does not exist, PRODXF will create one and use it.

Note: since DXF blocks lack I/O connection information, all cells generated and stored in this library will have 0 I/O reference points. You may edit these cells in PROCAD and assign I/O reference points if you want to reconstruct connectivity information from a DXF file.

"Map DXF lines & traces to ProCAD wires"

Enable this box if you want lines & traces mapped to PROCAD wires. Note that DXF files lack connectivity information. If you choose to map lines to wires, PRODXF will automatically assign machine generated signal names to the wires generated. You may then modify these names in PROCAD.

Tips: Since wires take considerably more storage space than lines, you should only enable this option if you really want to reconstruct connectivity information from a DXF file. See also the note on library cells I/O reference information above. This option defaults to OFF.

AutoCAD to PROCAD units scale factor [1000]:

Enter the multiplier scale factor to be used when translating AutoCAD units to PROCAD units. If your DXF (AutoCAD) units are in inches and your desired PROCAD units are in mils, use a scale factor of 1000 (1 inch = 1000 mils). The default is 1000

Database Output File [fname.dbf]:

Enter the name of the output file desired for PRODXF generated PROCAD binary database. If you specify the name of an existing file, PRODXF will display the following warning message during file generation :

-WARNING- database file already exists.
[U]pdate, [D]elete or [E]xit (U) ?

If you choose U, PRODXF will use the header record contained in this file, and update the database entities per DXF data. If you choose D, PRODXF will delete the file and generate a new one with default header record, and entities data per DXF file. If you choose E, PRODXF will stop processing the DXF file and return you to the first prompt.

After answering the above questions, Click on the "**Load DXF file**" button to start execution. PRODXF will read the DXF to PROCAD layer map file (DXFLAYS.DAT) for layer mapping information. If DXFLAYS.DAT file is not found, a warning message is displayed, and PRODXF will map all DXF layers to PROCAD layer 1.

Note: If you are reading DXF files generated by PRODXF from PROCAD database, you do not need the DXFLAYS.DAT file. You should ignore the "missing DXFLAYS.DAT" warning message.

PRODXF will display series of messages as it is processing your DXF file, and will finally generate a PROCAD binary database file.

After PRODXF has finished processing your DXF file, it will again be ready to [R]ead or [W]rite another DXF file.

FORMAT OF DXFLAYS.DAT FILE

The AutoCAD DXF layers to PROCAD layers map file "DXFLAYS.DAT" is an ASCII file which contains the cross reference data between AutoCAD layers & PROCAD layers. This file should contain one item (AutoCAD layer name, PROCAD layer number) per line. Blank lines are ignored. The following example of a DXFLAYS.DAT file will map AutoCAD layers 0, SIGN, & POST to PROCAD layers 1, 20, & 23 respectively:

```
0,1
SIGN,20
POST,23
```

The format of each entry in DXFLAYS.DAT file is as follows:

AutoCAD_layer_name , PROCAD_layer_number

Where:

AutoCAD-layer_name is the AutoCAD layer name
(15 characters max)

PROCAD_layer_number is the corresponding PROCAD
layer number (1 - 99)

Note: Layers not defined will be assigned to PROCAD layer 1. Layer names are limited to 15 characters. You don't need this file if you are processing DXF files generated by PRODXF.

LIMITATIONS OF PRODXF

AutoCAD is a 3-D drawing program, and PROCAD is a 2-D Schematic Capture and PCB layout program. If you are generating drawings in AutoCAD to be ported to PROCAD, you should note the following points very carefully:

1. PROCAD is a 2-D program, all Z-axis information in a DXF file will be ignored.
2. PRODXF does not support auto-dimensioning information. Any such data in a DXF file will be ignored.
3. AutoCAD stores DONUT data as polylines with bulge information, PRODXF may not translate it accurately. You should only use circles and arcs to construct donuts if at all possible.
4. AutoCAD stores fill information per entity type (SOLID), and PROCAD stores fill information per layer attribute, irregular filled polygons in PROCAD may not be filled in a DXF output file. You should make corrections in AutoCAD after translation, or use rectangle entities in PROCAD when constructing filled polygons.

Note: traces, wires, rectangles and lines are translated correctly.

5. PROCAD does not support variable polyline start & end widths. You should avoid using such polylines in AutoCAD if you plan to port the drawing into PROCAD.
6. Certain characters eg: "*" & "?" have special meaning to PROCAD library management utility. You should avoid using these characters as part of a block name in AutoCAD, since blocks translate to cells in PROCAD.

Note: PRODXF will issue warning message when it encounters reserved characters in a DXF file.

LIMITATIONS OF DXF FILES

AutoCAD is a drawing only program, it lacks connectivity information. PROCAD files converted to DXF format will lose all connectivity information and intelligence. All signal, pins and instance names will be translated to regular text entities. If you would like to reconstruct connectivity information from a DXF file, you should choose to map traces & lines to wires. After translation to PROCAD binary database and library files, you should use PROCAD to re-attach signal names to wires, and define I/O reference points for all your cells. You should also re-attach instance names and values back to the appropriate cells.

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Utilities (ARCHIVE)

C*hapter*

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ARCHIVE UTILITY

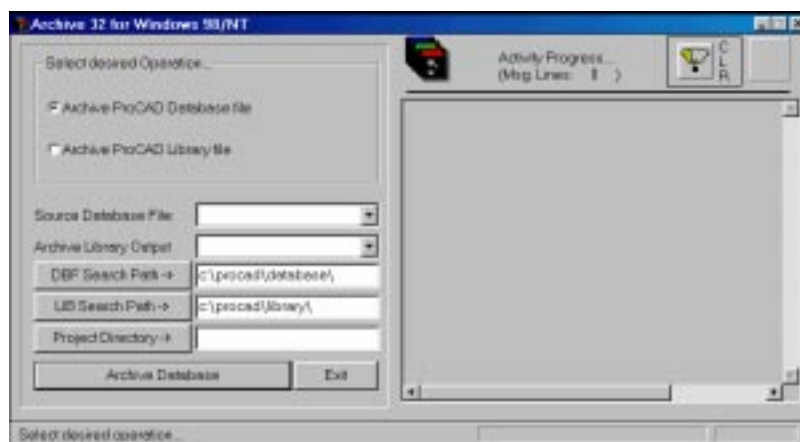
INTRODUCTION

ARCHIVE is a utility program used for combining various libraries used in a design to a single user defined library.

USING ARCHIVE

You may invoke ARCHIVE from the Tool bar within the PROCAD editor, 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.

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



ARCHIVE Window Control-menu Box:

The Upper left corner of the ARCHIVE window is a Control-menu box [-]. This Control-menu box when invoked drops down a menu for controlling the various aspects of the ARCHIVE window, and also submenus for displaying the current ARCHIVE 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.

The following is a description of the prompts produced by the ARCHIVE application:

Database & Library Search paths

You should insure that all databases, and their supporting libraries are in their proper search paths. Non Database & Library data files generated by PROPDIF will be placed on PROPDIF working directory, usually \WPROCAD.

"Archive ProCAD Database" or "Archive ProCAD Library"

If you choose the **"Archive ProCAD Library"** option, the archive utility will prompt for the **"Source library file name"** to be archived. This process updates the source library by copying all referenced nested cells from other libraries as needed.

If you choose **"Archive ProCAD Database"** option, the following series of prompts will be enabled & displayed:

"Source Database File:"

Enter or select the database file name of your completed design to be archived. You may not specify a path statement with this file. The database must be located in your database file statement defined above.

"Archive Library Output:"

Enter the name of the library that the cells used in the above database will be archived to. You may not specify a path statement with this file. The library will be place in your Library Search path defined above.

ARCHIVE will then search through your database modifying the database as needed and moving all referenced cells to the archive library. If cells used in your database are nested with nested cells from different libraries, archive will loop through those libraries and extract all referenced cells into the single archive library specified above. The original libraries are never modified.

At the conclusion of the archive process, the database date & time stamp will be modified to reflect the current values, and the archived library generated will have the current data & time for the creation and last modified field, and be attached to the database as the default library.

Note: If you plan to use the archive program on any database, you must make sure that the cells used in your design (including nested cells) have unique names in order to avoid name conflict after the archive process.

If you used the INPUT/CON command during the creation of your database, ARCHIVE will extract the cell "DOT" from the CELLIB to your archive library, however, subsequent use of the INPUT/CON command will still expect to see the cell named "DOT" from the library "CELLIB".

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Creating & Editing Cells

C*hapter*

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Creating & Editing Cells

INTRODUCTION

This section will explain the two ways of creating a cell and once the cell is created, how to edit the cell.

A cell is defined as a collection of entities grouped together and utilized as a single unit or entity. It can have attributes attached to it, or an instance name assigned to it.

Tips: For a more detailed step by step method of creating cells, you should refer to the Learners (Tutorial) guide.

CREATING A CELL

There are two ways to create a cell, both of them are available while you are in the main database environment. The first is when you are editing a database and have just finish inputting a group of entities that you want to make into a cell so that you can use it later. This method is called CELL Formation. The second method is also entered into from the main database environment. It is referred to as the EDIT CELL environment. By issuing the EDIT_CELL command, the PROCAD editor enters an EDIT CELL environment which allows you to devote an entire database to the creation of a cell.

METHOD #1 CELL FORMATION METHOD

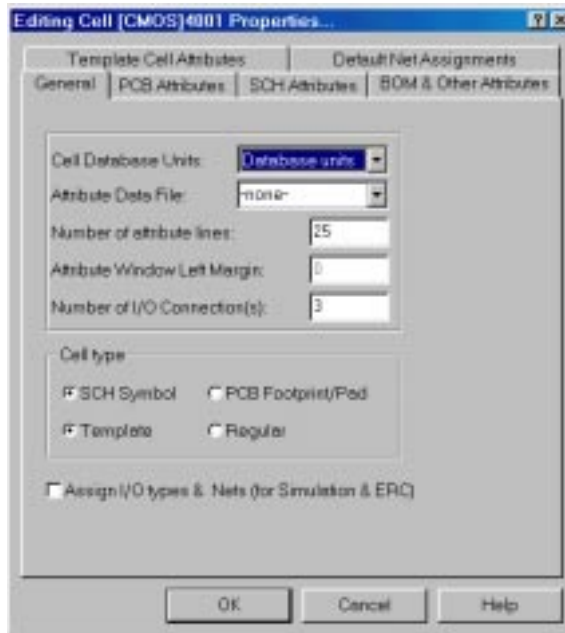
After entering into the database environment or while in the database environment, input the appropriate entities with the INPUT commands. When you are through with inputting entities for the cell you must select all of the entities involved with the SELECT command; either Entity or Window. After you have selected all of the entities to be part of the cell, issue the command: "CELL FORMATION". You will be prompted to enter the following information about the cell to complete the cell definition.

Use the <ESC> key to accept the default values displayed. You may also issue the PAN or ZOOM command whenever PROCAD prompts for coordinate entry (Prompts 8, 9, 10 & 12 below). To leave the attribute specification mode and return to the main database, you should issue a CNTL-C in response to any of these prompts.

PROMPT 1- "Cell Name:"

Enter a unique cell name, a maximum of 8-characters, by which the cell will be recalled from the library for instantiation into a database. You can use the full cell name specification ([libname] cellname) to indicate a destination library other than the current active library.

After responding to the above prompt, the following dialog box will be displayed:



The following is an explanation of the prompts contained in the "Cell Properties..." dialog box:

PROMPT 2- "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.

PROMPT 3- "Cell Type:"

Template cells are packagable devices such as NAND gate symbol. If it is desired to have PROCAD automatically assign a number of these cells to a single package with instance names such as U1A,U1B etc., select the Template cell type. If your select template cell option, PROCAD will then prompt for "**Package Pin Number(s) Text Height**". This is the text height used for displaying the package pin numbers assigned to each of these template cells as they are placed in a drawing. You may use a text height of "0" if you don't want the pin numbers displayed. Text height must be entered in database units. Please refer to the section on RATS NEST command and Appendix A for the use of proper attributes required for gate template cells.

PROMPT 4- "No. of User Defined Attribute Lines [*]:"**

This prompt allows you to attach attributes to the cell. The cell attribute file has a maximum capacity of 128 lines by 30 characters. This file is where information can be stored about the cell for use with the PROCAP Netlist Extractor. Since the netlist extractor is token driven, a complete explanation of the Attribute data File is covered in Appendix A. At this point, it will be sufficient to say that we will enter the number of lines necessary to hold the information we want to store; given the capacity. If you are not sure of the number of lines, you should enter 10. You can always increase the number of lines whenever the cell is edited. If you are editing a gate template cell, you must define the relevant tokens for the cell (see RATS NEST command & Appendix A).

PROMPT 5- "Attribute Window Data Left Margin [0]:"

The attribute editor will make all characters to the left of the specified left margin write protected. This area is usually reserved for token/keywords. You must enter "0" if you want to create new keywords or edit existing one.

PROMPT 6- "Attribute Data File [NONE]:"

PROCAD will automatically read in the file named. The file named must have an extension of .ADF. These .ADF files are ASCII files and may be edited to add repetitive information, These files, their uses and creation are more fully explained in Appendix A. For our purposes here just press accept the default: NONE.

PROMPT 7- "Number of I/O Connections [*]:"**

At this prompt, specify the number of I/O connections that the cell has, for example: a NAND Gate with 2 inputs has 3 I/O's, a 74LS121 - 14 I/O's, etc. Entering 0 will cause PROCAD to skip the invocation of the Attribute Editor.

PROMPT 8- "Assign I/O types and Nets (for ERC and Simulation)"

PROCAD needs to know if floating input checking and simulation netlist will be extracted from the component being edited. This form of netlist is only valid for schematic symbols. You should only answer enable this option if you plan to extract simulation data. If this option is enabled, PROCAD will momentarily, open a window in the upper right-hand corner of the screen after Prompt # 12. This window will have the left margin pre-filled with the following prompts:

"Pin #n TYPE/NAME "

where n=1 to number of I/O specified in prompt #7 above. You should enter the pin_type as follows: I for input, O for output & B for bi-directional followed by a / and the pin name. Netlist extractor programs like PROCAP will make use of this information while generating netlist for various simulators. Click on the **"OK"** button to exit the pin editor.

Click on the "OK" button to close the Cell Properties..." dialog box. PROCAD will then produce the following prompts:

PROMPT 9- "Cell Reference Coordinate:"

The cell reference coordinate is the reference used for inputting the cell into a database, in other words, when you input the cell into a database, it will be instantiated in precisely the same relationship to the cursor as it is at this point. Move the cursor to the point that you want to use as the reference and press <RETURN>.

PROMPT 10- "Instance-Name Ref. Coord:"

The instance name reference coordinate is the reference coordinate used as default for placing program/user assigned placement name when the cell is placed in a drawing. Move the cursor to the point that you want to use as the reference and press <RETURN>.

PROMPT 11- "Instance-Value Ref. Coord:"

The instance value reference coordinate is the reference coordinate used as default for placing user assigned component value when the cell is placed in a drawing. This field is only useful for cells that actually have values eg. Resistors, Capacitors, inductors etc. Move the cursor to the point that you want to use as the reference and press <RETURN>.

After responding to the above prompt, PROCAD will open an Attribute Editor Window.

PROMPT 12- "Press the OK button to exit the Attr. Editor"

Momentarily, a window will appear in the upper right-hand corner of the screen. While this window is open, all keyboard activity will be directed to the window text editor, for entering the attributes that you want to assign to the cell. The cursor will be displayed at the position specified in Prompt #5. Since the Netlist Extractor, PROCAP, is token driven; the first 15 character spaces can be used for token keywords. After entering all of the information necessary, press the "OK" button to exit the Attribute Editor Window. Note: This window is not displayed if you specified "0" for Prompt #7.

PROMPT 13- "I/O #n Connection Ref. Coord.:"

Position the cursor where you want the reference point for the referenced pin numbered I/O, then press <RETURN>. (This will be the point within the database that a wire will have a vertex for the netlist extractor to detect connectivity). If you are editing a gate template cell (prompt #3 above), PROCAD will momentarily toggle to the ghost grid (normally finer grid), and issue the following prompt "I/O #n Pkg Pin Ref. Coord: ". Move the cursor to the reference point where you want pin numbers displayed when the cell is placed in a drawing, and press <RETURN>.

NOTE : The above prompt 13 will be repeated for each pin, as specified in PROMPT 7. Enter "Home" for PAN or "PgUp" for Zoom IN, "PgDn" for Zoom Out, etc. while this prompt is being displayed if necessary. If you make a mistake, enter Cntrl-Z to backup one I/O reference point at a time. A Cntrl-C will return you back to the cell editing mode.

PROMPT 14-"Saving Cell [cellname] in Library [libname]"

This is actually a message rather than a prompt. It merely displays the fact that the cell is being saved in the current active or specified library.

At this point you will be in the SELECT environment and the editor will prompt for a command. Normally if the cell that was made is a part, you would enter the command DELETE to remove the entities that made up the cell, then instantiate the cell into the database. If the cell was made for a cut 'n paste application, then press <ESC> to exit the SELECT environment and return to the main DATABASE environment.

METHOD #2 EDIT CELL ENVIRONMENT

Method No. 2 is also accessed from the Main Database environment. By issuing the command **[Library]Edit/Create Cell** from the menu. When this command is issued, PROCAD effectively momentarily closes the current database and opens up an Edit Cell database. You will be required to respond to the first prompt at the beginning of the session, then you can enter any and all entities to define the graphics part of the cell. When you have completed all of the graphic input for the cell, issue the **[File]Save...** command from the menu. At this point you will be asked to respond to prompts 2 through 13 above. This method of cell creation is the most preferred as you do not have to select all of the entities that make up the cell. Upon completion of saving the cell, PROCAD closes the Edit Cell database and reopens the main database.

EDITING AN EXISTING CELL

The editing of an existing cell is essentially the same as Method No. 2. When you issue the EDIT command, specify the name of the existing cell that you want to edit. Everything will be as with the above METHOD No. 2, with the exception that PROCAD closes the main database and opens the Edit Cell database which contains the entities of the cell. After making the necessary changes issue the **[File]Save.** command. The first prompt will ask if you want to change any cell attributes (cell type, cell reference, attributes, etc). A YES response will then cause the above referenced prompts 2 through 13 to be initiated. A NO response will then cause a prompt to ask if you want to save the edited cell. A YES response will initiate PROMPT 14, while a NO response will abort the Edit Cell session, leaving the cell "as is" in the library, and return to the main database.

Note: Since this command requires a cell name in the command line, the EDIT command should be issued from the keyboard with the cell name on the command line, or invoked from within the Library Manager.

Example: EDIT 74LS00 <RETURN>

A NOTE ABOUT CELL DATA ATTRIBUTES

PROCAD communicates between the PCB and Schematic layouts by way of data keywords specified in the cell data attribute. You must enter all required attributes for effective communication between modes. Please refer to Appendix A for all currently defined keywords and their usage within the PROCAD editor. You should also refer to page #18 of this manual for the discussion on "**LINKING SCHEMATIC TO PCB DESIGNS**".

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Design Rule Checker

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Design Rule Checker

INTRODUCTION:

PROCAD supports an optional Design Rule Check program which allows you to check the minimum spacing requirements and possible trace shorts in a PCB layout. This program is an on-line DRC, which allows you to check the integrity of your PCB designs as they progress, and without ever leaving the PROCAD editor. In the schematic mode of operation, the DRC program will flag missing vertex between wires in the check layers (illegal T and crossed connections).

DRC TOKENS

The DRC program utilizes one of the DRC\$XXX tokens described in APPENDIX-A:

DRC\$PADC padDiameter thru_layer(s) {pin_number(s)}

DRC\$PADS padWidth thru_layer(s) {pin_number(s)}

DRC\$PADR padWidth padHeight thru_layer(s) {pin_number(s)}

The first format defines a circular pad, the second format a square pad and the third format a rectangular pad. The padDiameter variable is the circular pad diameter, padWidth is the width of a square or rectangular pad, and padHeight is the height of a rectangular pad. The thru_layer(s) are the layers the pad is valid on, and pin_number(s) are the pin or I/O numbers this definition applies to. *Note: the pin_number(s) must be enclosed in curly brackets "{}".* Pin numbers may be continued on the next line (the first non blank character must be a plus "+" followed by a blank).

Note also the following:

1. DRC\$PADC is equivalent to the former DRC\$PAD token.
2. If pin_number(s) are omitted, ProCAD assumes the definition is for all the pins
3. You may use the ALL clause or 5 in the layer(s) field to denote all thru layers
4. You may use the {ALL} clause in the pin number(s) field to denote all pin numbers that are not explicitly defined. Must be the part of the last DRC\$PADx token.
5. You may use multiple DRC\$PADx tokens to define various pad styles for a part

Examples:

```
DRC$PADR 50 50 5 {1}  
DRC$PADC 50 ALL {2-14}  
  
DRC$PADS 60 23 {1-3 8  
+ 14 28-40}  
DRC$PADR 60 60 23 {ALL}
```

The **DRC/V** command ([**Check**]Show **DRC\$PADx graphics** menu item) in PROCAD will display the graphical interpretation of all DRC\$PADx keywords used in the database.

Note: In order for the DRC\$PADx token to work for all pins in a device, they must be logically defined as pins during the CELL FORMATION process.

INVOKING DRC:

To invoke the Design Rule Check program, you must be inside a PROCAD PCB type database. If you are in a schematic type database, only shorts will be checked - clearance checking will be disabled. Execute the command:

DRC {drc_control_file} <RETURN>

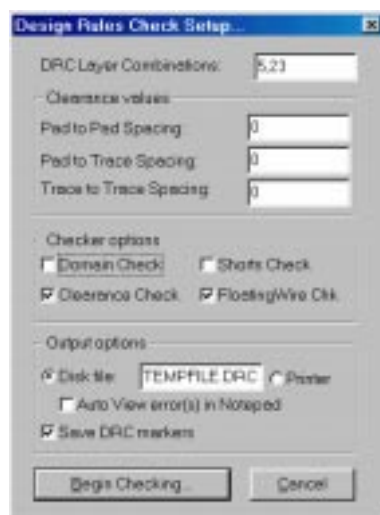
The DRC control file must be ASCII with the following format:

```
PP = nn  
PT= nn  
TT = nn
```

Where PP = pad to pad spacing, PT = pad to trace spacing & TT = trace to trace spacing. This is the same format as the PROROUTE (Auto-router) control file data. The default file extension is ".DAT".

Note: PROCAD does not prompt for the DRC control file when invoked from the [**Check**] menu. If omitted, PROCAD uses built in default settings of: PP=30, PT=10 & TT=10. You may also use the same control file used for PROROUTE (autorouter), and PROCAD will recognize TT,PT & TT values.

DRC module start by displaying the following dialog box:



Starting DRC:

You may change the default settings and then click on the "Run DRC" button to start the checker.

The following is an explanation of the prompts in the "Design Rule Check Setup" dialog box:

DRC Layer combinations: [5,23]

This has to do with what layers the DRC program will compare with each other. The default setting of 5,23 could represent the graphical layers used in creating the Solder side of a PCB layout, if the layering convention from APPENDIX-E is used.

The DRC program checks the layers which combined would make up a single side (or internal layer) of a PCB. For instance, the layer 5 contains the copper pad master from all the pad stacks used in the PKG.LIB library file, and the layer 23 contains all the traces which are found on the Solder side of the PCB. By checking layer 5, and 23 you will be checking the spacing of pads to pads (all on layer 5), pads to traces (layer 5 checked against layer 23), and traces to traces (all on layer 23). If you wanted to check the Component side of the PCB you would need to check layers 5, and 20.

Domain only check [OFF]

This is used to establish a small area within a large PCB layout which you wish to check. When Domain is toggled OFF, the entire database is checked for violations. If this is toggled on PROCAD will prompt for the Lower Left & upper right corners of the DRC window.

Shorts check [ON]

The Short checking feature can be toggled ON or OFF (default ON), to speed the DRC execution. A graphical short in a PCB or schematic database is defined as any two wires (lines are not checked by the DRC program) which cross on the same graphic layer without sharing a common vertex. Since PROCAD is a "vertex recognition" system, the DRC program assumes that two wires crossing on the same layer with common vertexes are part of a common node and will not be flagged as shorts. Shorts are flagged by small crosses of lines on layer 34, and added to the total of NET & DRC ERRORS in the STATUS information window.

This type of Short checking is useful in that it will find graphical shorts which might have been made accidentally during the design process. The DRC program may flag some "T-Junctions", or other instances, which are not shorts. To prevent the DRC program from flagging good connections, it is useful to add vertexes to the intersection of all traces which should be connected.

The Short checking capability for PCB layout also includes traces which make contact with a pad, but do not have vertexes at the reference point of the pad. If, for instance, a trace passes through the center of a pad, but does not have a vertex common with the center of the pad, it is considered a short. Another type of short is a trace which makes contact with the edge of a pad, but does not pass through the center of that pad. These features enable PROCAD to insure that the final PCB layout accurately reflects the schematic design.

Floating Inputs Check [ON]

Tells the DRC program to check and flag all floating (un-connected) schematic device input pins. This option is not available in PCB mode. You must also define your schematic device I/O's as Inputs, Outputs or Bi-directional ("Use cell for simulation" option when creating a cell), then only unconnected INPUTS would be flagged.

Clearance check [ON]

Tells the DRC program to check the clearances of pads and traces on the printed circuit board. The DRC only checks the clearances of all layers specified earlier against themselves. This option is only available in PCB mode.

There are three kinds of clearance checks provided by the DRC program:

Pd-to-Pd [30]

This is the Pad to Pad check default value, 30 database units (in most cases MILS). The default can be changed merely by typing in a new value.

The Pad to Pad check searches the design for any two pads placed within the stated distance of each other. If PROCAD finds any violations of this rule, it places a line between the two offending pads on layer 33, and keeps track of total violations in the status window.

It should be noted that pads within a cell are not checked against pads within that same cell. When you design the pad placement of a cell, you are responsible for insuring the correct spacing of pads. You may execute the DRC program during the creation of the cell to check for proper pad-to-pad spacing. Pads within a cell are checked against pads in another cell, or pads used as feedthroughs on the PCB.

Pd-to-Tr [10]

This is the default value for the Pad to Trace check. This provides PROCAD with the ability to check the minimum distance between the Pads and Traces of a PCB layout. Any violations of this minimum spacing are flagged by small lines on layer 32, and added to the total NET & DRC ERRORS in the status information window.

Tr-to-Tr [10]

Is the default value of the Trace to Trace check feature of the DRC program. This allows PROCAD to check the minimum spacing of all traces on the PCB layout. All surfaces of a trace are checked in a comprehensive manner against all surfaces of adjacent traces. Violations are flagged on layer 31, and added to the total reflected in the STATUS window.

Save Errors [ON]

The last line of the DRC window allows you to view all flags set up by the DRC as temporary or saved. If Save Errors is set to OFF, all flags of violations will be visible until the screen is REFRESHED. This makes the flags temporary. With Save Errors set to ON (the default) all flags of violations will remain on the assigned layers (31, 32, 33, 34, and 49) until the DRC program is run again or the CLEAR ERRORS command is issued.

When the DRC program is executed, the first task it performs is to delete all lines on layers 31 thru 34. This clears all previously assigned violation flags, as well as any other line entities which may be on the reserved layers.

The CLEAR ERROR command "[Check]Clear DRC Errors" is also available to remove all violation flags on layers 31 thru 34, and 49. This command also resets the total of the NET & DRC ERRORS reflected in the STATUS information window.

Output Options:

You may direct DRC errors to a disk file or your printer for hard copy record.

DRC LAYER ASSIGNMENT SUMMARY

Violation	Flag Layer
Trace to Trace	31
Pad-to-Trace	32
Pad-to-pad	33
Shorts	34
Floating inputs	49

SCHEMATIC DRC

You may use the DRC program in a Schematic type database, however, the Shorts_check and Floating inputs check are the only checking device used in these instances. Clnce_checks are toggled OFF as the default for schematic databases, and are ignored upon execution of the DRC.

Checking for shorts in a schematic database can be useful for locating all graphical shorts (lines which cross on the same layer without a common vertex) within a design. A graphical short can show you where two nodes which should be connected by a common vertex are not. However, in a schematic, it is often correct for lines to cross without a shared vertex, simply because they should not be connected. You can use this feature to check for possible shorts or opens.

ON-LINE (REAL TIME) DESIGN RULE CHECKING

You may also consider using the On-Line-DRC feature of PROCAD. This feature may be enabled or disabled via the **[Other...]****Toggles...** dialog box. You may also set default clearance & shorts checking parameters from this dialog box. When enabled, your designs will be checked in real time as traces are entered. There is however, a delay (may be considerable for large designs) when exiting the Select Environment. *Note: On-Line-DRC is not available in Schematic mode.*

SPECIAL CONSIDERATIONS

Due to the time consuming nature of DRC runs, we recommend you either run the DRC program on a domain (window), or disable some of the checking that you have checked previously or don't care to check. A zero or negative value for Trace to Trace, Pad to Trace or Pad to Pad will disable that checking. DRC checking involving pads are usually the most time consuming !.

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Special Topics

C*hapter*

3-12

Special topics

INTRODUCTION

This chapter covers the topics necessary for performing special tasks such as Ground and Power ties in a PCB and Schematic database and SMT design. An understanding of the topics covered in this section are not required for successful operation and use of PROCAD for normal designs.

This section is organized as follows:

Productivity Enhancement topic - A discussion of the QUICKSET mode of operation, and the advantages and disadvantages of the use of this feature.

Schematic topics - A discussion of the use of the power and ground macro command files and RAT\$VCCn & RAT\$GNDn tokens to generate ties to VCC and GND connections, Multi-sheet schematic design, Hierarchical top down schematic designs, Creating busses & bundles, and Creating connectors for linking signals in a schematic design to a corresponding PCB connector.

Printed Circuit Design Topics - A discussion of the use of layer mapping techniques for Surface Mount Technology design, Making automatic connection to Power and Ground plans using the voltage tie and thermal relief methods. Analog PCB design and commands necessary for generating odd shaped traces and polygons specific to analog designs are also discussed.

PRODUCTIVITY ENHANCEMENT

USING QUICKSET MODE

PROCAD Supports an advanced user mode called QUICKSET. This mode allows the user to enter commands with the current location of the cursor used by PROCAD for data entry. With QUICKSET toggled to the ON position design entry can be significantly accelerated.

To use the QUICKSET mode, select it in the Help Level area of the ProCAD Configuration Program.

With QUICKSET mode turned ON, PROCAD will not prompt for the first point of entry when executing specific commands. The following analysis should help illustrate this point:

Select and Move Operation -

QUICKSET ON	QUICKSET OFF
Select	Select
Move	What Entity
	Move
	From What Point

As illustrated, with QUICKSET turned ON, PROCAD selects the item which is located at the cursor when the SELECT command is executed. With QUICKSET turned OFF PROCAD prompts for the location of the item to select.

With QUICKSET mode turned ON, PROCAD also moves the item from the current cursor location when the MOVE command is executed. With QUICKSET turned OFF, PROCAD prompts for the coordinates to move the selected item from.

Input Line Operation -

QUICKSET ON	QUICKSET OFF
Drawing	Drawing
Lines	Lines
	Where to start

In this example PROCAD prompts for the location to begin entering a line when the **Line** command is executed. With QUICKSET turned ON however, PROCAD begins the line at the current cursor location.

The advantage in using the QUICKSET command mode is found in the greatly reduced number of keystrokes (or button clicking) that is required when entering a design. This may seem like a small advantage, but during the completion of a large design, or through extended use of the PROCAD package, a great deal of time can be saved by using the QUICKSET mode.

However, a serious disadvantage is caused by the need to know exactly where your cursor is when executing certain commands. For instance, when executing a **Zoom** command, PROCAD will zoom around the current cursor location with QUICKSET turned ON. If your cursor is not located in the correct position when executing the command, errors may occur. The increased possibility of errors occurring in the design process, due to inefficient use of the QUICKSET mode, can cause an increase in the amount of time fixing those errors. The disadvantage in using QUICKSET can actually be greater than the advantage.

It is recommended that beginning users of PROCAD use the system with QUICKSET turned OFF. This will allow you to learn the system more effectively, without making errors due to cursor positioning. As a user becomes more advanced the QUICKSET mode can offer advantages in design productivity.

NOTE: PROCAD for Windows users are strongly advised to leave QUICKSET turned OFF. Default Quickset mode may be set using the WCONFIG.EXE (PROCAD Configuration) utility.

AUTOMATIC PLACEMENT LAYER SELECTION

Recall that PROCAD has 99 layers for design. You place entities on pre-assigned layers. Normally you execute the **SET LAYER** command (or access the appropriate layer menu strip) to set the working layer, before placing the entity. The **[Layer]Set AutoLayer...** command or from the **[Other...]Constants...** dialog box allows for the automatic selection of working layer by entity type and current working mode (PCB or Schematic).

See also the SET AUTOLAYERS command in the Reference manual for more details.

PROCAD CONNECTIVITY

Recall that PROCAD achieves connectivity by means of common vertex and common signal name recognition. This powerful feature can be used to achieve easy attachment of ground and power connections to Schematic diagrams and PCB layouts.

POWER & GROUND TIES IN A SCHEMATIC DESIGN

Power and Ground signals may be attached to a schematic symbol by the following two methods:

Method 1 Using RAT\$VCCn & RAT\$GNDn tokens

This method requires that you assign the RAT\$VCCn (n=1,2..4) and the RAT\$GNDn (n=1 or 2) to the attribute data area associated with the component. For example, the [TTL]7485 symbol has 16 I/O pins, with pins 8 and 16 tied to GND1 and VCC1 respectively by the "**RAT\$VCC1 16**" and the "**RAT\$GND1 8**" tokens.

With this methodology, you do not need to connect wires to these I/O pins to establish connectivity. In order to ensure that this device is packaged to a 16 pin PCB footprint package, the number of I/O's defined for the device is 16. The "**RAT\$PKG [PKG]IC16A**" token simply establishes the name of the corresponding foot print PCB cell to use. Since pins 8 and 16 does not need wires attached to them, those points are defined to be in some area of the symbol (the center in this case) where you will not be connecting wires to.

Note: You must define the schematic symbol cell to have the same number of I/O points with the corresponding PCB package footprint cell. Unused I/O pins or those assigned by the RAT\$VCCn and RAT\$GNDn tokens may be located in the center of the symbol where wires may not be accidentally connected to.

WARNING: If a wire is attached to an I/O point which already has a RAT\$VCCn or RAT\$GNDn token, the signal name of the wire attached to those I/O points will be used, and the corresponding RAT\$VCCn or RAT\$GNDn token will be ignored. Locate RAT\$VCCn, RAT\$GNDn and unused I/O points where you will not accidentally connect a wire to !!.

Note: You can only define two ground (GND1 & GND2) and four power (VCC1, VCC2, VCC3 & VCC4) signals using the RAT\$GNDn and RAT\$VCCn tokens methodology.

Method 2 Power and Ground Signal ties

This method makes use of the common vertex and signal name recognition features of PROCAD. To assign a power or ground connection to a pin of a schematic component, you simply tie a wire named VCC1 or GND1 (your power and ground signal names) to the respective pins.

Using ProCAD PowerStation 32...

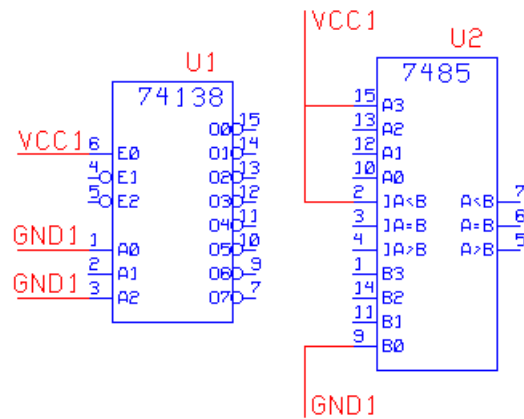


Figure D-1

In the above example (Figure D-1), the three wires connected to pins 1 and 3 of device U1 and pin 9 of device U2 are connected to the ground supply (GND1) and those tied to pin 6 of device U1 and pins 2 and 15 of device U2 are connected to the power supply (VCC1). Notice that you do not need to actually have a common trace running through those pins in order to establish the connectivity. In fact, you do not even need to display the signal names.

The example below (Figure D-2) shows the same circuit arrangement, but this time, the signal names are not shown (Use the Change Signal Name command to disable their display).

To make the appearance more conventional, the standard power and ground symbols have been drawn to the end of the respective wires. You should use the **Input/Lines** or **Input/Polygons** command to accomplish this.

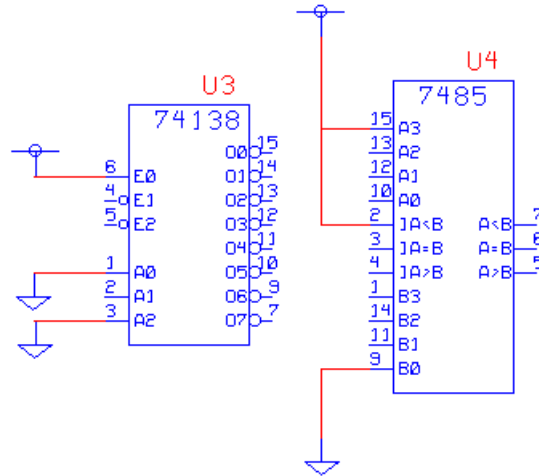


Figure D-2

Note that the connectivity in both cases were established due to the common names assigned to the common signals. The graphic symbol for power and ground does not have any significance in PROCAD. You merely place them for visual effects.

To automate the above process, a macro command file GND.CMD and VCC.CMD have been provided and assigned to function keys Alt F8 and Alt F9 respectively assuming you initialized the schematic database with the macro command file CKT.CMD.

The GND.CMD and VCC.CMD macros will place a standard ground and power symbols at the current cursor location, then the symbols are exploded, thereby making them regular entities containing a short wire named GND1 and VCC1 respectively. The appropriate graphical symbols for GND and VCC are drawn using the **Input/Lines**, **Input/Polygons** and **Input/Circles** commands.

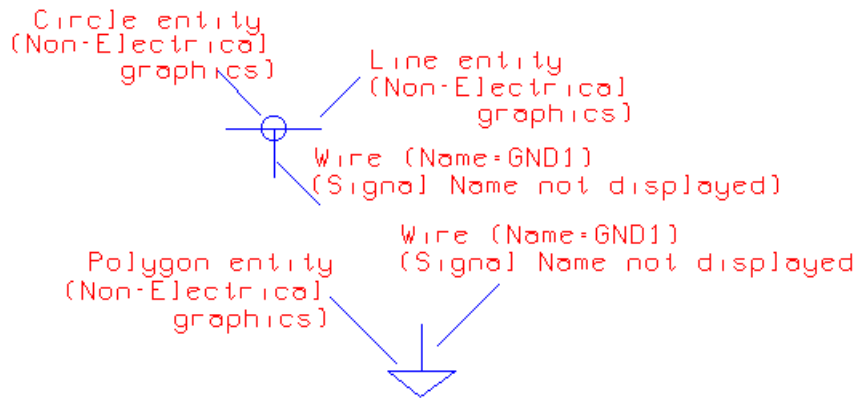


Figure D-3

Notice that the short wires do not have any signal names displayed. You must ensure that the end vertex of the wires coincides with the pins of the component or vertex of the wire that they are being connected to, in order to establish the connectivity to the destination entity by common vertex methodology.

Tips: You should create your own symbols and assign different power and ground names (eg: +5, +12, -12 Vss etc). Create a macro file similar to the GND.CMD and VCC.CMD, and assign them to your preferred function keys or to a menu item. You may also change the graphics of the supplied symbols if you choose to use a different power and ground symbol convention.

Note: If you are using the RAT\$VCCn and RAT\$GNDn tokens to make power and ground connections, your corresponding signals must be named VCCn and GNDn respectively.

BUILDING A BUS

PROCAD also offers the ability to quickly enter in a sequential array of signals such as would be found in a data signal bus. This is done through the use of the **Step&Repeat** command from the **[Edit]** menu.

The first step in building a signal bus is to route the first wire of the bus as it is required in the design. The bus signals should either be completely vertical, or completely horizontal, in order to allow effective duplication in either a horizontal or vertical fashion (see Figure D-4).

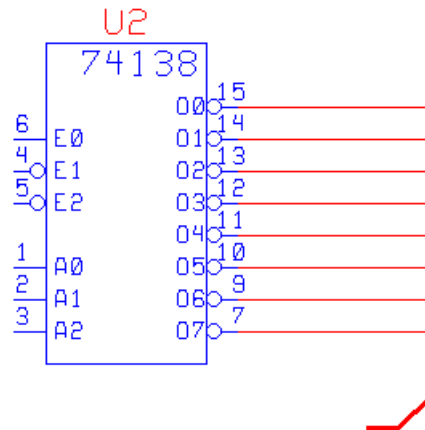


Figure D-4

Once this signal has been routed, select the signal, and then execute the **Step&Repeat** command. This command will prompt for the following parameters:

X OFFSET - the offset value in the X direction for placement of each duplication. This value can be either positive or negative for left or right offset.

Y OFFSET - the offset in the Y axis for placement of each duplication. This value can be either positive or negative for up or down offset.

X REPEAT COUNT - the number of duplications to be made along the X axis. This number should include the original item, for instance, if you want to make three additional copies of an item the X REPEAT COUNT should be set at four to include the original. This value should be set at zero if there are to be no duplications along this axis.

Y REPEAT COUNT - the number of duplications to be made along the Y axis. The count should include the original item, but should be zero if there are to be no duplications in this direction.

BUS SIGNAL BASE NAME - this is the prefix value to be assigned to every signal as the bus is defined. If you are building a data bus you would define this to be the letter D (which is the default value).

BUS SIGNAL START VALUE - this provides the beginning number of the numerical sequence for building the entire bus structure. The default value is zero.

PROCAD then asks you to confirm the values which have been entered up to this point. When this has been accomplished PROCAD asks if the signal names for the bus should be displayed. If you want the names displayed, you must provide the justification point (lower-left hand corner of the text), the size and orientation of the foundation signal. The signal bus is then entered into the design as you have defined it.

If the starting signal already has a signal name, PROCAD uses that name as the signal base name and the signal start value is assumed to be zero. If the signal already has a name and sequence number, PROCAD uses the name as the signal base name, and the value as the signal start value. Once the signals have been defined, a thick line may be added as a bundle to graphically connect bus signals on the schematic documentation. This line may be entered by using the **Line** command from the Tool bar. If you want to add thickness to this bundle line simply select the entity and use the **Change Width** command from the **[Edit]** menu to define a new line width.

The reason for using the **Line** command for representing the bundle, instead of the **Wire** command, is to prohibit signal connectivity to that bundle. Wires are logical entities in the PROCAD database, and as such receive signal names and are subject to shorts. Lines are just graphical objects with no logical attributes at all. For the purposes of documenting the connectivity of the bus signals in a graphical way, the **Line** command is best for defining a bundle.

MAKING A CONNECTOR STRIP IN A SCHEMATIC DESIGN

PROCAD provides two methods for designing connectors in a schematic drawing which will be linked to the appropriate connector in the PCB design.

Method #1 Using Template Cells

This method requires the creation of a template connector cell with a single I/O point. Include the following token in the cells attribute data:

```

INS$PRFX      J1 (Connector prefix)
RAT$PKGn      [PKG]cell (the PCB connector cell)
RAT$TCNT      n (# of pins in the connector)
RAT$DEVA      1
RAT$DEVB      2
....
RAT$DEVn      n

```

When you use this component in a Schematic design, you connect the appropriate signals to the appropriate template cell (J1A,J1B.....J1n). PROCAD will package this cell as a single PCB connector assigned the name J1.

Note: Since you only have a maximum of 26 letters in the alphabet (A-Z), you can use this method only for connectors having a maximum of 26 pins.

Method #2 Using Non-template connector Strip

This is the preferred method. It makes use of the fact that connectivity in PROCAD is by virtue of common signal name. To use this method, you create a non-template cell with the desired number of I/O points corresponding to the number of pins in the PCB connector.

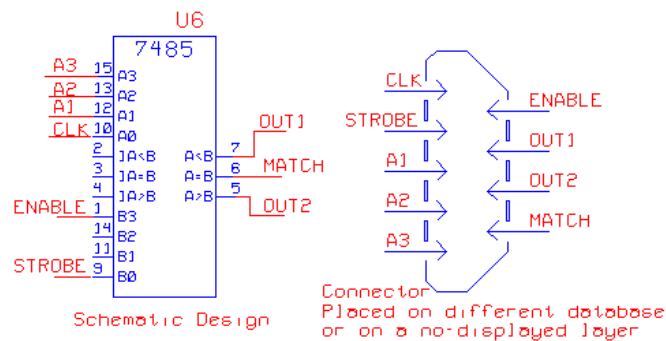


Figure D-5

Attach short named wires to the I/O points of the cell. In your schematic circuitry, the corresponding signals will be automatically hooked-up to the connector. This connector should have the RAT\$PKG token pointing to the desired PCB connector cell. The pins will be mapped one-for-one.

You may place the connector strip arrangement on a different database, and link that database with your actual schematic database in PROCAP when you are ready to generate the rat's nest or wire list files.

MULTIPLE SHEET SCHEMATICS

PROCAD allows you to design multiple sheet schematics by linking up to 30 separate schematic databases during the compilation in PROCAP, the netlist extraction program. In order to build multiple sheet schematics there are a few concepts which you should keep in mind:

When defining the schematic database you must specify which sheet of the schematic design the current database represents. The connectivity of global signals, such as VCC and GND, and signals which occur on multiple pages, such as a data bus or clock line, is based upon the recognition of common names. On and Off-sheet connectors may be employed to represent the terminations of these signals from one page to the next, but it is actually the signal name which is used to establish connectivity.

When establishing reference designators for a multiple sheet schematic design, PROCAD has the ability to name devices across multiple sheets in a sequential order. This is done with the PACK/R command from the command line editor as follows:

PACK/R=filename <RETURN>

This command tells PROCAD to renumber all devices on this schematic design using the design database specified as the starting value for all devices. For instance, if the specified file has resistors labeled R1, R2, and R3, PROCAD will automatically number resistors in the current design starting at R4.

With this command you can maintain sequential naming of all components throughout all pages of a schematic design. However, this command should only be executed as the design is completed in order to avoid inconsistencies. The best approach in using this command would

be to open each database serially, starting with the first sheet of the design, and executing this command just prior to compiling the design in PROCAP.

WARNING: The PACK/R command also renames any devices which you have manually named out of sequential order. If you want to maintain a non-sequential naming convention do not use PACK/R or you will have to manually rename components after executing the command.

Note: You may use the Project feature of ProCAD to organize you multiple sheet design into a more manageable project. See the PROJECT command in the Reference manual for details.

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.

A schematic symbol may be constructed in a hierarchical fashion for simulation purposes, when a macro cell must be broken down to a primitive level. A D-type Flip Flop could be 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.

When defining a hierarchical symbol, one with nested information, it is important to understand the concept of nesting within the PROCAD editor. This concept, nested information, can be understood in the following way:

PROCAD supports fifteen levels of nesting -

The top level is the schematic sheet level which contains the schematic symbols, sheet outlines, and any wire interconnects. The second through tenth levels of nesting contain hierarchical information that is included in cell hierarchy.

A cell, or symbol, which is used in the creation of another cell is automatically nested one level deeper inside of the new cell.

Any information which is defined on layers 40 thru 48 during the creation of a new cell will also be nested one level deeper inside of the new cell.

Using ProCAD PowerStation 32...

So, in order to nest the gate level information inside of a Flipflop, you would place the gate cell from the library into the new Flipflop cell, and connect the gates with wires routed on layer 40 (thru 48). This is done using the EDIT CELL environment when building the Flipflop symbol.

The nested information does not actually have to be contained inside the hierarchical component to be connected. PROCAD recognizes connectivity in hierarchical cells by virtue of the common pin and signal names within a cell. This is illustrated in Figure D-6 below.

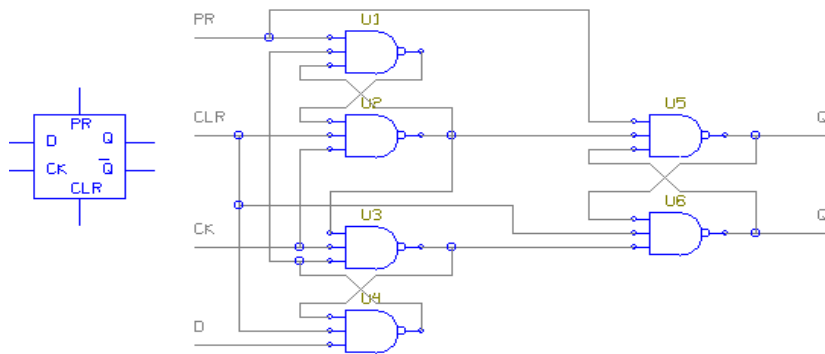


Figure D-6

Notice how the names of the pins defined in the Pin Editor, match the user-defined signal names in the nested schematic information.

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** even though they are not physically (or graphically) connected inside the cell.

PROCAD supports up to ten levels of hierarchy in order to allow complex devices such as an ALU to be nested right down to the gate level for simulation purposes. This means that a cell can be nested inside of another cell which is nested inside of another cell....

Show Hierarchy

To view the hierarchical information within a single cell, while editing a schematic design, place the cursor on the device and execute the following command:

SHOW HIER <ENTER>

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.

Type **EXIT <RETURN>** to leave the cell hierarchy and return to the design database.

POWER & GROUND TIES IN A PCB DESIGN

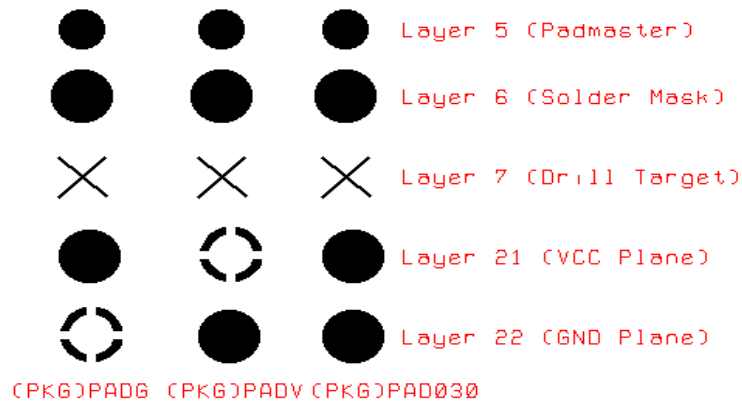
Power and Ground signals may be tied to coppered planes of a Printed Circuit Board layout by the following two methods:

Method 1 Using Copper Pour with Net ties

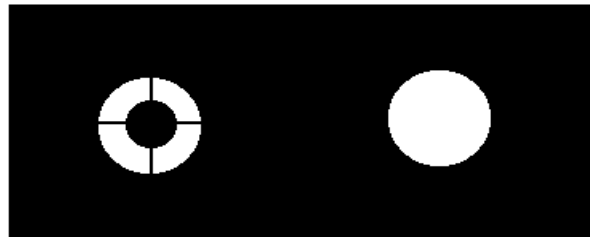
This method requires that you pour copper around your pads and traces using the Area Fill Polygon command (INPUT/G or **[Place]Copper Plane** menu item). Next, you will use the Voltage Tie command (VTIES or **[Place]Voltage/GND ties** menu item). This method though very effective, requires a considerable amount of screen refresh time. Refer to the section on AREA FILL and VTIE commands in the Reference manual for details and limitations.

Method 2 Using Thermal Relief pads

This method requires that you place thermal relief pad stacks such as [PKG]PADV and [PKG]PADG on the pins that should be connected to voltage and ground planes respectively. The structure of a typical 30 Mils thermal relief and regular pad stacks are as follows:

**Figure D-7**

You will notice that the three pads above (Figure D-7) actually has graphics information on layers 5 (Padmaster), Layer 6 (Solder mask), Layer 7 (Drill target), Layer 21 (VCC plane #1), and Layer 22 (GND plane #1). The difference between [PKG]PADG and [PKG]PADV cells is the placement layers of the thermal relief pads. On [PKG]PADG cell used for GND plane, it is located on layer 22, and on [PKG]PADV cell used for VCC plane, it is located on layer 21. These cells could be used for making the appropriate connection to the respective planes when layer 21 or 22 is plotted in reverse as shown in Figure D-8.



Layer 21 (or 22) plotted in reverse
[PKG]PADG or PADV cell used

Layer 21 (or 22) plotted in reverse
[PKG]PAD030 cell used

Figure D-8

Notice that pins connected to the regular PAD ([PKG]PAD030) does not make any connection to the voltage or Ground planes. Only pads with thermal relief makes connection to the appropriate plane.

Tips: When utilizing this method, you should include the GER\$FLASH attribute on the thermal relief pads, so that they are flashable in GERBER. If this is not possible, you should ensure that the thermal relief pads are drawn with thin traces (say 5 mils or so). See the section on GERPLOT for more details.

Method 3 Using Voltage & Ground Ties

This method requires that you place Voltage ties and Ground ties on the pins to be connected to VCC and GND planes respectively. This is accomplished as follows:

1. Place two crossed wires of about 50 Mils on layer 50 for ground tie, and on layer 51 for voltage tie. The "X" made by the two wires should have a common vertex at the crossing intersection, and they should be assigned signal names GND1 and VCC1 respectively. Use the VTIES ([Place]Voltage/Ground Ties) command.

2. Now combine layer 50 for GND and layer 51 for VCC with the padmaster layer (Layer 5) on the GERBER plot file. Generate the first GERBER file, say FILE1.GER by combining these layers.

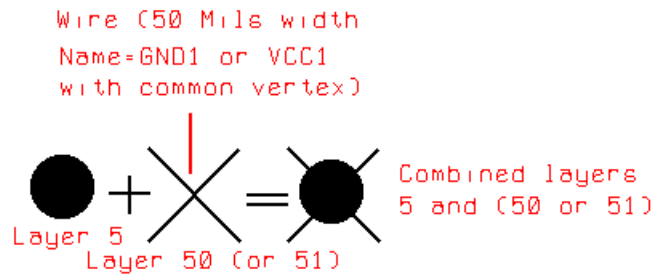


Figure D-9

3. Generate GERBER file (FILE2.GER) for layer 21 for VCC connection or layer 22 for GND connection.

4. Now have your GERBER plotting bureau combine FILE1.GER with the reverse of FILE2.GER in a multi-pass process to generate a single file as shown in Figure D-10.

Layer 21 (or 22)
plotted in reverse
and merged with layer 5
plus Layer 50 (or 51)
plotted in positive



Multipass GND or VCC plane generation

Figure D-10

Notice that the final film makes a connection to the respective power and Ground planes. The signal name assigned to the "X" wire is for use by PROCAP for the proper generation of the PCB netlist used for Back-annotation/Differences check.

WARNING: You must ensure that all your footprint pads do not contain any pads with thermal reliefs, unless you desire those pins to be connected to the respective power and ground planes.

The voltage tie method of making connection is usually preferable to the thermal relief method, since you can generate accurate netlists and Differences check. The only draw back is the fact that your GERBER service bureau must be capable of making a multi-pass plot.

Tips: To automate the above process of placing ties, create two tie cells (VTIE and GTIE) with "X" crossed wires on layers 50 and 51 respectively. Create a macro command file similar to the GND.CMD and VCC.CMD for use in placing these ties on the appropriate pins.

SURFACE MOUNT DESIGN

PROCAD provides three commands specifically designed for the placement of SMT components. These are the Set Map, Show Map, and Change Map commands.

When a cell is created, the entities making up the cell are placed on specific layer for top side artwork. In order to use the same cell for the bottom side artwork, as is typically required in SMT designs, the cell layers must be mapped to the appropriate layers for the bottom side artwork.

As an example, you may create an SMT package with the following layer configurations:

<u>Layer #</u>	<u>Usage</u>
6	Solder Mask
9	Silk Screen Legend
10	Silk Screen Text
20	SMT pads
37	SMT Paste mask

To use the same cell for bottom side placement, the layers must be mapped as follows:

<u>Top Side Layer #</u>	<u>Mapped to Bottom Side Layer #</u>	<u>Usage (Description)</u>
6	36	Solder Mask
9	39	Silk Screen Legend
10	35	Silk Screen Text
20	23	SMT Pads
37	38	SMT Paste Mask

PROCAD has three possible map groups (1,2,3). If we were to use map group 1, We must issue the following commands to set up the mapping:

```

SET MAP 1      Begin group 1 map setting
6 36           Map layer6 to layer36
9 39           Map layer9 to layer39
10 35          Map layer10 to layer35
20 23          Map layer20 to layer23
37 38          Map layer37 to layer38
<RETURN>      End group 1 map setting

```

You can use the SHOW MAP command to view your settings.

Using ProCAD PowerStation 32...

To place this same component on the bottom side of the board, you should use the **GetCell** command to place the cell, select the newly placed cell, and use the **ChangeMap ([Place][SMT]Move To Bottom** menu item) command to move it to map group 1. ProCAD will automatically mirror the cell.

You will notice a color change when you perform the above operations. This signifies that the layers have been mapped.

PROCAD AutoRouter (PROROUTE) recognizes layer mapping, and will route traces correctly to top or bottom side pads of SMT designs. When performing manual routing, you may use small ghost grid setting to achieve fine line routing.

Tips: You may use map groups 2 and 3 for buried and blind SMT vias. ProCAD will NOT automatically mirror cells moved to these map groups.

Note: Changing a cell to Map group 0 effectively disables mapping, and returns the cell to its original layers and un-mirrored.

USING PROCAD FOR ANALOG DESIGNS

PROCAD also supports several key features to allow analog PCB designs to be created. These features hinge primarily on the flexibility of the graphic editing environment within PROCAD.

The ROTATE command allows components to be placed into the design at any angle (in one degree increments). This feature allows you to position components in a more effective manner for high speed analog designs. This can also allow you to position standard packages in a radial fashion for circular boards. This is illustrated in the figure below.

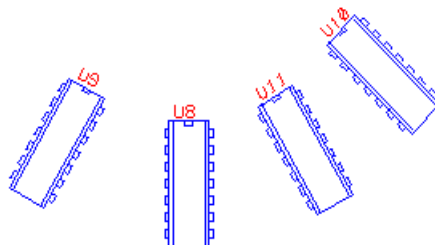


Figure D-11

PROCAD's powerful graphic editor also allows user-defined grids and trace widths in one-MIL increments. This type of resolution, coupled with the ability to route traces at any angle, allows thick analog traces to be curved for better signal performance.

Finally, PROCAD supports a POLYGON command that allows you to create large coppered areas which may be attached to ground to quiet signal noise. By coupling this feature with the ability to define specific line types, such as dashed or dotted lines, large cross-hatched areas of copper can be created to improve the performance of analog designs.

Command Reference

C***chapter***

4

About This Reference Section

INTRODUCTION

The Reference Manual offers the user a complete reference of all commands available within the PROCAD Design Editor, as well as how to execute these commands from within the editor.

PROCAD is a fully interactive Electronic Design System which provides tools for creating complete multi-sheet schematic designs, as well as fully routed multilayer printed circuit boards. The Design Editor, along with PROCAP for netlist extraction, PROROUTE for autorouting a PCB design, and PROPLOT and GERPLOT for generating documentation and fabrication ready photoplots, make-up the most complete and sophisticated CAE/CAD programs available in its price range.

If you want specific information about a specific command, you may find it in this Reference Section. You will also find specific information about the various support programs which make up the PROCAD design system in this section of the manual.

The Reference Manual is organized in the following fashion:

Command Reference - A complete alphabetical listing of all commands available within the PROCAD Design Editor. This section describes command syntax, as well as information which may be of special interest when executing a given command.

COMMAND REFERENCE

INTRODUCTION

This command reference section is organized alphabetically by command name. The following notations will be used to represent specific keystrokes:

Capital letters indicate portions of statements or commands that must be entered, exactly as shown.

Angle brackets (<>) indicate user supplied data. When the angle brackets enclose lowercase text, you must type in an entry defined by the text. When the angle brackets enclose upper-case text, you must press the key named by the text.

Braces indicate {} that the enclosed entry is optional. If omitted PROCAD will in most cases prompt for the information.

Dots (...) indicate that an entry may be repeated as many times as needed or desired.

xxx,yyy<RETURN> == Locate cursor at appropriate position and press ENTER, PUCK button or PEN DOWN switch on Tablet, Switch button on Mouse. You can also enter the absolute or relative coordinates directly from the keyboard and press ENTER. To enter relative coordinate, you must enter R before the coord offset.

eg: R 200,-500

NOTE: All other symbols are to be entered as shown.

Several of the commands can be issued directly, implying selection by the cursor position. That means that when the command is issued, the command processor will first imply the SELECT ENTITY command assuming that the cursor position is the coordinate entry, then process the actual command entered. These same commands can be issued while in the "SELECT" environment also. The SELECT environment is obtained by issuing the SELECT ENTITY, SELECT INSTANCE, SELECT SIGNAL or SELECT WINDOW command. See also the SET QUICKSET command for implying the cursor position as the first coordinate of most commands.

In the examples shown for the commands, a dotted line is used to illustrate a highlighted entity.

ADD CORNERS

PURPOSE: To automatically add a 45 degree corners to orthogonal traces.

SYNTAX: **ADD CO**rner <CR>

EXPLANATION:

This command is used for automatically adding 45 degree corners to orthogonal wires. You should select the desired trace(s), then apply the command using the above syntax or from the **[Edit][Other...]** or **[Edit][More...]** menu. ProCAD will prompt for the desired length of 45 degree segment to add. You should enter the desired segment length, and ProCAD will automatically add a 45 degree piece to all 90 degree bend corners of all the selected trace(s).

Note: You may not use this command to increase or degree the length of already placed 45 degree segments. This command only works on 90 degree turns on a wire. If the 45 degree bend desired is longer than the horizontal or vertical segment of a trace corner, that corner will be left alone (ignored).

ADD ENTITY

PURPOSE: This command is used to add additional entities to the SELECT ENVIRONMENT, either "SEL_E" or "SEL_W". Especially useful for selecting multiple, odd-shaped or widely dispersed entities.

SYNTAX: **ADD Entity**<RETURN>

EXPLANATION:

When using the SEL W command to select more than one entity and not all of the entities can be selected by a window because of location, then any entities missed by the window can be added by use of the ADD Entity command. In conjunction with the SEL E command, the ADD Entity command can be used to specify additional items for inclusion into the SELECT environment.

See also the TOGGLE SELECTION command for alternate and faster method of adding and removing entities from the select environment.

ADD VERTEX

PURPOSE: This command provides a means of adding a vertex to a selected path (line or wire).

SYNTAX : **ADD Vertex**<RETURN>

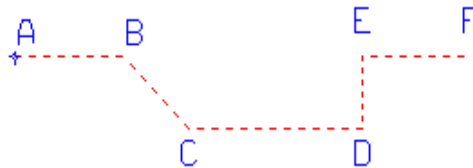
EXPLANATION:

Upon issuing the command, PROCAD will prompt you for end vertices of the segment to which you want to add a vertex. Then you will be prompted for the location of the new vertex on the specified segment.

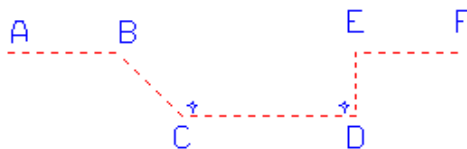
NOTE: This command can be used in the SELECT environment.

EXAMPLE:

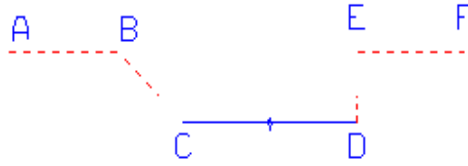
STEP1 - Place the cursor on the line or wire that you want to add the vertex to.



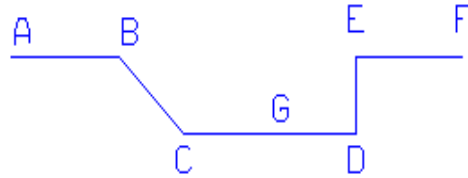
STEP 2 - Using the cursor, identify the end vertices of the segment to which you want to add the vertex.



STEP 3 - PROCAD will then change the selected segment back to its original color and prompt for the location of the new vertex.



STEP 4 - After responding with the position of the new vertex, PROCAD will return the entire entity back to its original color, the entity will contain the new vertex and may be edited just as if it had been placed with the original entity.



AREA FILL(INPUT GROUND PLANE)

PURPOSE: To generate an area fill polygon around traces, pads etc with user specified clearance value.

SYNTAX: **IN**put/**G** <CR>

EXPLANATION:

This is a special case of the entity input command. The effect of this command is to generate a polygon around already placed pads and traces (polygon pouring), or to generate a special polygon which allows for pads and traces to plow through it. You may assign a signal name to the polygon for intelligent DRC checking (electrical polygon).

This command can be used in 2 different ways;

CASE 1 (Polygon pouring): This first case is when a user has already placed pads, traces, components etc, and desire to place a filled polygon around these objects while maintaining a certain clearance. The **INPUT/G** command will produce such a polygon while maintaining the clearance value previously set by the **SET CLEARANCE** command. The polygon generated also uses the **DRC\$PAD**, **DRC\$PADC**, **DRC\$PADS** and **DRC\$PADR** tokens for determining the size of all pads and I/O pins of a component.

CASE 2 (Polygon plowing): The second case is when a user decides to place a filled polygon in an area of his design, and then later on decides to place traces, pads, or components within it while maintaining a certain clearance. The **INPUT/G** command may be used in this instance to generate such a polygon. You may then place traces, pads, components etc directly in the interior of this polygon. PROCAD will then automatically plow through the filled polygon while maintaining the clearance value set prior to the issuance of the **INPUT/G** command. You may have to refresh your screen using any of the screen refresh commands (**REFRESH**, **PAN**, **ZOOM**, **REDRAW**, etc) to see the results of the polygon plowing. You may change the clearance value simply by selecting the Area fill polygon and issuing the **CHANGE WIDTH** command (which in this case changes the clearance value of the Area fill polygon). Note: any pads or components placed in the interior of the polygon must also have the proper **DRC\$PAD**, **DRC\$PADC**, **DRC\$PADS** or **DRC\$PADR** token as in case #1 above.

Example: **IN**put/**G** <CR>

Note:

1. The Polygon generated uses a scan line algorithm to fill between traces and pads. The size of the filled polygon scans generated are such that they are at least the size of the largest trace, pad etc plus 2 times the required clearance. The effect of this is apparent when large clearances are specified, in this case, there may be some areas around traces which will be left blank. You may then use a regular INPUT/P command to generate filled polygons in these areas, or reduce the clearance value specified. You may also use multiple IN/G commands to generate different Area fill polygons covering areas of varying pads and/or trace sizes.

2. You may have as many Area fill polygon entities as you desire, but be aware that it takes considerable amount of computer time to re-paint these types of polygons. You should use regular INPUT/P command whenever possible to fill areas that were not filled completely by the INPUT/G command due to large clearance size limitations as outlined above.

3. The INPUT/G command will only generate a filled polygon on the layer which it is placed on regardless of the fill attribute on that layer, and clearances will be generated around traces which are placed on that layer only. Pads placed on the polygon will always have clearances around them regardless of the layer the pad or component I/O's are placed on. The only requirement is that the DRC\$PAD, DRC\$PADC, DRC\$PADS or DRC\$PADR token be specified for the pads and the component as a whole.

4. You should ensure that wire segments within the Area Fill polygon are completely inside the polygon. You may have to add extra vertices to long wires (breaking it up to multiple segments) to accomplish this. Partially enclosed wire segments within an Area Fill polygon may not be processed accurately.

See also the SET CLEARANCE and CHANGE WIDTH commands.

Tips: The INPUT/G command produces a special polygon entity referred to as an Area Fill Polygon. This polygon can be edited as any other regular polygon using the existing PROCAD editing commands (ADD VERTEX, STRETCH etc). The Area Fill property is lost if the polygon is edited in such a way as to make the starting vertex not equal to the end vertex (Non-closed polygon).

AUTOROUTE

PURPOSE: To Interactively route a trace or group of traces while in ProCAD editor.

SYNTAX: **AR**route<CR>

SYNTAX: **AR**route/**P**<CR>

EXPLANATION:

This command allows for the invocation of a built in version of ProRoute Autorouter for single trace or multiple trace routing. Before using this command, you should use the second syntax of the command or **[Route]Setup AutoRouter...** menu item to load a router control file and invoke the Router Control Parameters setup screen.

The screenshot shows the 'Router Control Parameters' dialog box. It has a title bar with a plus icon and a close button. The dialog is organized into columns for 'One(1)', 'Two(2)', 'Three(3)', 'Four(4)', 'Five(5)', and 'Six(6)'. The rows include: 'Routing Pass ->' (One(1) to Six(6)), 'Pass Ctrl' (checkboxes), '45 degree router' (checkboxes), 'Router Mode' (dropdowns: F, A, A, A, A, A), 'Trace Grid' (text boxes: 50, 25, 25, 25, 25, 40), 'Via Grid' (text boxes: 100, 50, 50, 50, 50, 50), 'Max. Vias' (text boxes: 0, 0, 3, 8, 3, 8), 'Max. Deviation' (text boxes: 250, 250, 250, 250, 500, 800), 'Min. Rats Angle' (text boxes: 0, 60, 0, 0, 0, 0), 'Max. Rats Angle' (text boxes: 90, 90, 90, 90, 90, 90), 'Max. Rats Length' (text boxes: -1, -1, -1, -1, -1, -1), 'Time Limit/Net' (text boxes: -1, -1, -1, -1, -1, -1), 'Trace Width' (text boxes: 25, 10, 10, 10, 10, 10), 'Horiz. Layer' (dropdowns: 23, 23, 23, 23, 24, 24), and 'Vert. Layer' (dropdowns: 20, 20, 20, 20, 25, 25). At the bottom are buttons for 'Save' (with a floppy disk icon), 'OK', and 'Cancel'.

	One(1)	Two(2)	Three(3)	Four(4)	Five(5)	Six(6)
Routing Pass ->	One(1)	Two(2)	Three(3)	Four(4)	Five(5)	Six(6)
Pass Ctrl	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
45 degree router	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Router Mode	F	A	A	A	A	A
Trace Grid	50	25	25	25	25	40
Via Grid	100	50	50	50	50	50
Max. Vias	0	0	3	8	3	8
Max. Deviation	250	250	250	250	500	800
Min. Rats Angle	0	60	0	0	0	0
Max. Rats Angle	90	90	90	90	90	90
Max. Rats Length	-1	-1	-1	-1	-1	-1
Time Limit/Net	-1	-1	-1	-1	-1	-1
Trace Width	25	10	10	10	10	10
Horiz. Layer	23	23	23	23	24	24
Vert. Layer	20	20	20	20	25	25

The Setup dialog above is similar to that used in ProRoute. Please refer to ProRoute section of the Users and Utilities manual for explanation of the router control file and different setup elements.

The Quick route command will route all currently selected rats nets traces. If the command is issued when no rats nest trace was selected, ProCAD will prompt for a net pin to route. This approach allows for routing one net at a time.

Quick route uses the default pad to pad, pad to trace, trace to trace etc clearance rules established for On-Line DRC or in the Router Control File (if defined in the file), and nets placed in the current rats nest source layer.

Note: This command is only available in the PCB mode. Schematic nets may not be autorouted. Also, you must purchase the ProRoute Router option in your standard package (Combo packs 3, 4 or 5) in order to be able to use the Quick Route feature of ProCAD.

Please refer to the section on ProRoute PCB Autorouter in the Users and Utilities Manual for details on the router requirements (DRC\$PADx tokens etc), operation and features.

CELL FORMATION

PURPOSE: To allow the formation of a cell from within the Main Database mode using entities already placed in the database.

After the desired entities have been selected:

SYNTAX: **CEL**I<RETURN>

EXPLANATION:

Issuing this command will cause the series of prompts and messages associated with the EDIT CELL command to be activated. See the EDIT CELL command and the chapter on CREATING CELLS in the User's manual for further details.

CHANGE DISPLAY FACTOR

PURPOSE: To allow user defineable grid values to be more accesible by changing the screen display factor of an existing database.

SYNTAX: **CH**Ange **D**isplay <RETURN>

EXPLANATION:

Use this command to change the screen display factor and make available new GHOST GRID values. This command is based on the understanding that:

The smallest increment of movement across the screen is one pixel, therefore the smallest value of movement at any time is the number of units assigned to a pixel.

Screen Display Factor = the number of database units assigned to a pixel on the graphic display at 1X magnification.

Minimum Delta X and Y = Screen Display Factor times the absolute value of a negative Screen Scale (or the reciprocal of a positive Screen Scale), divided by the Magnification Factor.

Using the above information it can be calculated that:

Screen Display Factor (SDF) = 10 states that 10 database units will be assigned to each pixel. So, at a Screen Scale of 1, Magnification of 1, the smallest Delta X & Y is 10.

Using the CHANGE DISPLAY command, you can now define the appropriate Screen Display Factor (SDF) to achieve the finest grid resolution you need. For example:

CHANGE DISPLAY 3 <RETURN>

There are now three database units assigned to a single pixel, so at a Screen Scale (SS) of 1, Magnification (M) of 1, the finest resolution of movement is 3 database units.

If you do not reset the Ghost Grid when you change the Screen Display Factor, the new Ghost Grid value is equal to the multiple of the smallest increment which is closest to the assigned GHOST GRID. This simply means, with a SDF=3, SS=1, and M=1, an assigned GHOST GRID of 10 will result in an actual value of 9 units. As follows:

$$(SDF \times [SS]) / M = \text{smallest movement} \\ (3 \times 1) / 1 = 3 \text{ database units}$$

GHOST GRID 10 = 9 (nearest multiple of minimum movement).

Using the same values as above, except Screen Scale (SS) is -3, you can see how the minimum movement is affected.

$$(3 \times 3) / 1 = 9 \text{ database units}$$

Remember to use the absolute value of the negative Screen Scale (SS) values (use the reciprocal of positive SS values).

GHOST GRID 10 = 9 (nearest multiple of minimum movement).

The last variable in the relationship of values described above is Magnification (M). Changing the value of M, you can see its impact on the smallest increment of movement. Using the same values as before, except with a Magnification equal to 2:

$$(SDF \times [SS]) / M = \text{smallest movement} \\ (3 \times 3) / 2 = 4.5 \text{ database units}$$

So, by zooming in a Magnification factor of 2 times, the smallest increment of movement (1 pixel on the display) is equal to 4 and one-half database units. However, it is important to remember that PROCAD is an integer based system, and will not allow fractions such as 4.5. The value 4.5 is actually rounded off as 4 on the first pixel, and 5 on the second pixel, adding up to a total of 9 every two pixels.

THE CHANGES IN MINIMUM MOVEMENT, AND THEREFORE THE CHANGES IN ALLOWABLE GHOST GRIDS OCCUR WITHOUT WARNING TO THE USER. YOU MUST BE AWARE AT ALL TIMES, BY CHANGING ANY VALUE IN THE RELATIONSHIP:

$$(SDF \times [SS]) / M = \text{SMALLEST MOVEMENT}$$

YOU WILL BE CHANGING THE VALUE OF 1 PIXEL ON THE DISPLAY.

Lastly, the actual area of the display can be calculated by using the value assigned to 1 pixel (the value of the smallest movement) and multiplying that by the number of pixels (the resolution) of the graphic monitor.

CHANGE INSTANCE ATTRIBUTE

PURPOSE: To allow for the modification of instance attribute parameters of name, placement origin, text height, width and orientation, or the association between PCB and Schematic symbols. This command may also be used to change the pin names of an instance of a template cell.

SYNTAX #1: **CH**ange **IN**stance/**L**<RETURN>

SYNTAX #2: **CH**ange **IN**stance_name<RETURN>

EXPLANATION:

The first syntax of the command (with the **/L** qualifier) is used for modifying the link between the device in the Schematic with the corresponding footprint in the PCB. If this command is issued from a schematic database, you will be prompted to enter the corresponding PCB footprint instance for the device. If issued from a PCB database, you will be prompted to enter the corresponding schematic device instance the footprint will be associated with.

Note: The link list for all devices are usually set when you issue the RATS NEST command. The CHA INS/L command is provided for those instances where you wish to modify the link. You should only use this command if you are quite familiar with ProCAD structure and how its makes its link (association) between schematic and PCB symbols.

The second syntax is used for modifying the device instance name and its display parameters of height, width, orientation and justification. You will be prompted to change instance placement parameters. A YES response will allow the prompts for Text origin, height, width and angle of the instance name. You will next be prompted for instance name. When you enter a new instance name, PROCAD checks the name for consistency with the cell instance name prefix as defined by the INS\$PRFX token when the cell was created. If an inconsistency is detected, PROCAD will issue a warning message and prompt for use of the name anyway. If you respond with a YES, and the cell is a template, PROCAD will further ask if you would like to edit the pin numbers assigned to that instance. A YES response will allow you to change the pin numbers of that instance alone.

NOTE: You must follow the same naming convention used by PROCAD in order for the Re-assign option in the PACK command to work properly, and to ensure proper RAT'S NEST file generation. (See Appendix A for instance naming format).

Tips: You may also use the Quick Edit (double clicking on the component) to accomplish this task faster, especially if you do not desire the text origin changed.

EXAMPLE:

The following examples will assume that you have instantiated (placed) a two input nand gate (NAND2) into your database and want to change the program assigned instance name and parameters.

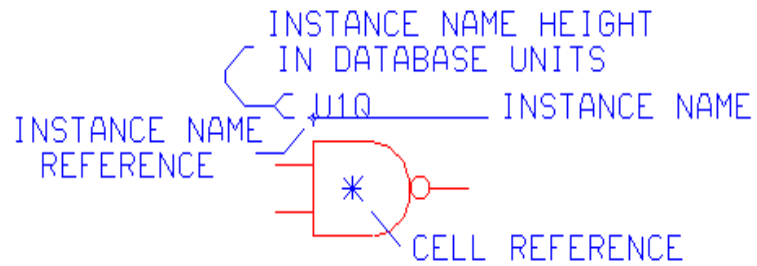
STEP 1 - By placing the cursor within the cell extents (the body of the cell) and issuing the command "CH IN", the cell reference of the selected cell will be displayed. You will be prompted to change placement parameters of the instance. A NO response will move you to STEP 3.

STEP 2- A YES response will then allow you to input the following:

- a. Instance name ref.
- b. Instance name height
- c. Instance name width
- d. Instance name angle

NOTE: Use the <ESC> key to retain the default values.

STEP 3- You will be prompted to enter the desired instance name. Press <ENTER> to leave the instance name unchanged. PROCAD will automatically re-assign pin numbers to a template cell when its instance name is changed, provided that the naming convention is consistent with those normally assigned by PROCAD.



NOTE: When the **CHANGE INSTANCE** command is issued in the select environment, ProCAD allows for position and angle changes only. The instance name, width or height may not be changed in this mode.

CHANGE LAYER

PURPOSE: To allow a selected entity(s) to be moved to a different layer.

SYNTAX #1: **CH**ange **LAY**er{=nn}<RETURN>

SYNTAX #2: **CH**ange **LAY**er/S {=nn} <RETURN>

EXPLANATION:

After the command has been issued, the selected entity will hi-lite and change to the new layer color/line - type. The second syntax (with /S qualifier) will change the layer of a trace (wire) segment, and automatically insert vias at both ends of the trace. The "/S" qualifier is ignored in Schematic mode, or if non wire or multiple entities were selected.

NOTE: This command can be used in the SELECT environment. Using the CHANGE LAYER command on cell instances only changes their placement layer, not their color/line-type. Layers 30-34, 49 and 99 are reserved for PROCAD use. (See the section on LAYER ASSIGNMENTS/RESERVED STATUS in this manual for more details).

CHANGE LIBRARY

PURPOSE: Used to change the current active working library.

SYNTAX: **CH**ange **LIB**rary{=<libname>}<RETURN>

EXPLANATION:

If the argument is omitted, PROCAD will prompt you to enter the new working library name. The current active working library is the default library for all library operations, including inputting cells into the database. This does not prevent you from inputting cells from other libraries, see the INPUT commands for further details.

NOTE: Issuing this command for a library that does not exist will allow you to create a new, empty library.

CHANGE MAP

PURPOSE: To move a placed cell to one of the mapping groups established by the SET MAP command.

SYNTAX: **CHange MAP** <CR>

EXPLANATION:

This command will move a cell to one of the three mapping groups. Mapping group 0 actually turns off layer mapping for the cell. When a cell is moved to a particular layer mapping group, the cell will be drawn with the corresponding mapped layers. If the cell is moved to map group 0, this effectively returns the cell to its natural state with no layer mapping. This command when used in conjunction with the MIRROR X/Y commands eliminates the need to design a separate cell for the top and bottom sides of an SMT design. See also the SET MAP and SHOW MAP commands for more details.

Example:

```
CHA MAP 2  
CHA MAP 0
```

The first example will cause the selected cell to use layer map group #2 as defined by the SET MAP command. The second example will turn off layer mapping for the selected cell.

LAYER MAP GROUP 0 & 1 ASSIGNMENTS

Layer map group 1 is assigned to bottom side of a printed circuit board. When cells are moved to layer map group 1 (**CHANGE MAP 1**), ProCAD will automatically mirror the reference designator of the cell. Layer map group 0 is similarly assigned to the top side of the board. When cells are moved to layer map group 0, the reference designator of the cell becomes un-mirrored.

*NOTE: Layer map groups 2 & 3 are user definable for use in buried via placement. Use the **CHANGE MAP/T** & **CHANGE MAP/B** commands to move a cell to the top & bottom sides of a board respectively, or use the **Quick Edit** feature to accomplish the same task.*

Note: This command can be issued from the `SELECT ENVIRONMENT`. Also, the `SHOW SUMMARY` command will report the current map group selected for any selected cell. See also the `SET MAP` and `SHOW MAP` commands.

CHANGE SHELL

PURPOSE: To toggle between the 2 available PROCAD command interpreter shells.

SYNTAX #1: **CH**ange **SH**ell n <RETURN>

SYNTAX#2: **XXX** n <RETURN>

EXPLANATION:

PROCAD supports 2 command interpreter shells, P & C shells. The P shell is the default and is normally a VMS DCL type command structure. The C command shell supports 2 letter mnemonic commands. To toggle between both command shells, simply enter the command XXX from either shell. To force a specific shell, enter the command XXX n, where n = P or C. (See the Quick Reference Guide for the command cross reference between both shells).

NOTE: If you plan to use the C command shell in PROCAD Xtra (DOS Version), you should re-program function key 30 to "XT 0" for proper invocation of the main menu from the mouse.

Syntax #1 above is only valid when issued from the P command shell. Syntax #2 may be issued from either the "P" or "C" shells.

CHANGE SIGNAL NAME

PURPOSE: To allow the assignment of a user specified signal name to wires. PROCAD assigns it's own signal names to wires as they are input into the database, which can be changed with this command.

SYNTAX: **CH**ange **SI**gnal_name<RETURN>

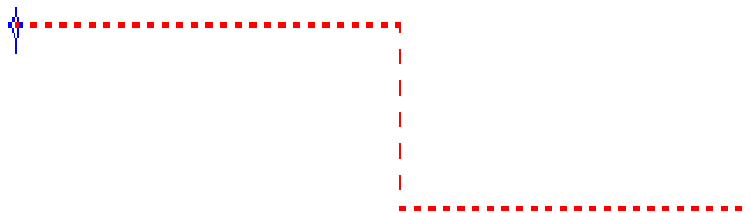
EXPLANATION:

You will be prompted "Is signal_name placement required?" A YES response will allow the prompts for Text origin, height, width, angle, and characters of the signal name to be entered and display the signal name(see Fig. 1). A NO response will only allow the characters to be entered and the signal name will be changed but will not be displayed (see Fig. 2). If the signal name of the specified wire is already user defined, PROCAD will issue a warning if the wire is connected to or shorted to any other user defined wire.

Tips: You may also use the Quick Edit (double clicking on the wire) to accomplish this task faster, especially if you do not desire the text origin changed.

EXAMPLE:

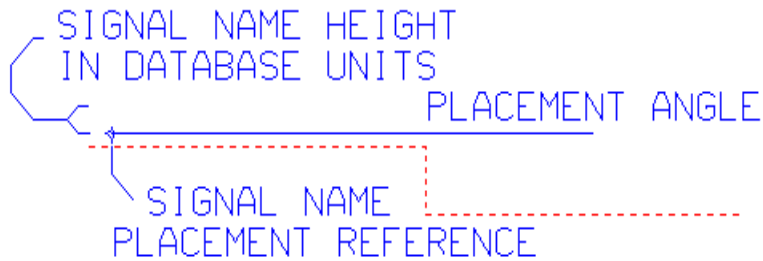
STEP 1 - Place the cursor on the wire that you want to change the signal name of:



STEP 2 - After the wire is selected, you will be asked if the signal name is to be displayed. A NO response will then only allow you to change the signal name. A YES response will prompt you for the following:

- a. Signal Name height
- b. Signal Name width
- c. Placement angle
- d. Placement reference
- e. Signal Name Text

NOTE..Use the <ESC> key to accept the default values



STEP 3 - After you have entered the Signal Name text the highlighting will be removed and the signal name will displayed.



STEP 4 - If the default (PROCAD assigned signal name) is used, it will have a characteristic "N" prefix and "QQ" suffix.



NOTE: When the **CHANGE SIGNAL** command is issued in the select environment, ProCAD allows for position and angle changes only. The signal name , width or height may not be changed in this mode.

CHANGE VALUE

PURPOSE: To assign or change the value assigned to an instance in the drawing/database.

SYNTAX: **CH**ange **V**alue <RETURN>

EXPLANATION:

This command will allow the user to change the value field of an instance. The size, angle and justification of the text can also be modified by this command. A size of 0 (zero) will disable the values assigned to the cell. The default display justification point is defined globally for the cell during the cell ref. point definition process of the CELL FORMATION or CELL EDITING command.

If the cell has the INS\$VAL or BOM\$VAL attribute specified, this value will be used as the default value for every instance utilizing the cell. The change value command will only modify the value of the selected instance. The default value specified by the INS\$VAL or BOM\$VAL command for the cell is not affected.

Note: Instance values are limited to 8 alpha-numeric values eg: 200K. The instance value is included in the status summary and Bill of Materials listing of PROCAP.

Tips: You may also use the Quick Edit (double clicking on the component) to accomplish this task faster, especially if you do not desire the text origin changed.

NOTE: When the CHANGE VALUE command is issued in the select environment, ProCAD allows for position and angle changes only. The value name, width or height may not be changed in this mode.

CHANGE WIDTH

PURPOSE: Allows you to change the width of a selected wire, line, circle, donut, polygon, Area fill clearance, or text height.

SYNTAX : **CH**ange **W**idth {=nn}<RETURN>

SYNTAX : **CH**ange **W**idth/**S** {=nn}<RETURN>

EXPLANATION:

The specified wires, lines, polygons or donut will have their width changed. Text entities will have their height changed. Area Fill (GND planes) entities will have their clearance values changed. "nn" equals the desired width/clearance in database units. The second syntax of the command (with "/S" qualifier) will change the width of a trace segment. The **"/S"** qualifier is ignored in Schematic mode, or if non wire or multiple entities were selected.

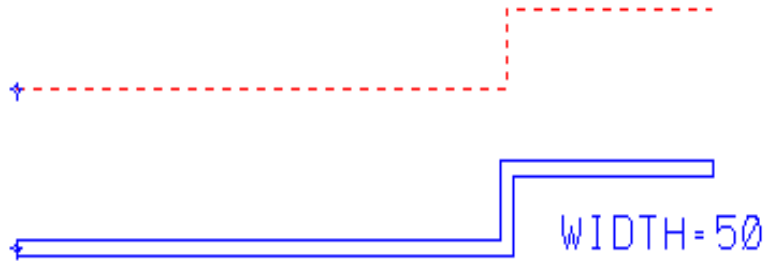
*Note: The **INPUT/WIRE** and **STITCH** commands will also allow for width change while the wire (trace) is being input into the database, if the user pushes the **"W"** key while ProCAD is prompting for the next trace coordinate.*

By using the SELECT environment, multiple wires or lines can have their widths changed by issuing just one command for the entire group.

This command can be used in the SELECT environment. When applied to circles, donuts will be generated with the specified width. Donuts are special cases of circles with widths greater than 1 unit. When applied to Area Fill polygons (polygons typically used for plowed or poured ground planes), their clearance values will be changed.

EXAMPLE:

After selecting the wire, circle, donut, polygon or line that you want to change the width of, issue the command: "CH WI 50" or "CH WI 100"



Note: The width argument is generic. When the command is issued on a single entity, PROCAD will prompt for the nature of the argument required eg: text height & width, clearance value etc.

CLEAR ERRORS

PURPOSE: Allows you to clear the special marks made by the Design Rule Check (DRC) & Back annotation programs. This command actually deletes all lines on layers 31, 32, 33, 34, 35, and 49.

SYNTAX : **CLE**ar <RETURN>

EXPLANATION:

Use this command to remove all of the spacing violation, and connectivity error flags placed into the design by the DRC and Back annotation program. This command deletes all lines on layers 31 thru 35 and 49.

NOTE: Care should be taken to insure that no design data is placed on critical layers to avoid any accidental deletions.

CLIPBOARD

PURPOSE: Allows you to copy any area of the client window to the windows clipboard in bitmap format. This command is only available in PROCAD for Windows MDI version.

SYNTAX : **CLIP**board <RETURN>

EXPLANATION:

Use this command to copy an area of the client window to the windows clipboard in windows bitmap format. PROCAD will prompt for the rectangular area to copy. Window area copied to clipboard may be pasted into other windows applications such as word processors, desktop publishers, spread sheets, etc.

NOTE: This command is only available in the MDI version of PROCAD. See also "CUT" and "PASTE" commands.

CONNECT SIGNAL

PURPOSE: To connect a signal between instances by name and pin numbers.

SYNTAX: **CON**nect {/SIG} Ua,n Ub,n Uc,n ..

EXPLANATION:

This command will cause PROCAD to connect a signal between instances Ua pin n, Ub pin n, Uc pin n etc. /SIG is optional, and if present, PROCAD will assign signal name SIG to the connecting wire.

NOTE: connecting wire generated will be placed on the current active layer and assume the current line/path width settings. This command will not connect wires to pins that already have a valid wire connected to it.

COPY

PURPOSE: Allows you to copy selected entity(s) to any chosen location. Use "CLIPBOARD" command to copy an area to the clipboard.

SYNTAX: **COPY**<RETURN>

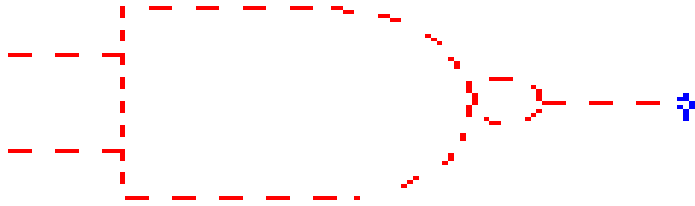
EXPLANATION:

After issuing the command, you will be prompted to specify a "copy from" reference point and then a "copy to" reference point. When you are done copying, press <ESC> to exit the COP command.

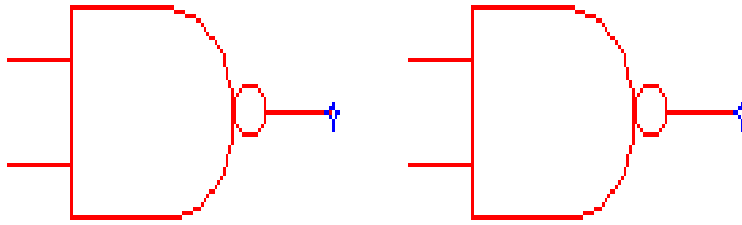
NOTE: This command can be used in the SELECT environment.. For clipboard operations, use the CLIPBOARD command. See also the "CUT" and "PASTE" commands.

EXAMPLE:

STEP 1 - After selecting the entity(s) to be copied, you will be prompted for a reference point to copy from.



STEP 2 - When you select the "copy from reference point", the entity(s) to be copied will return to their layer color. And you will be prompted for the "copy to reference point".



STEP 3 - After you have finished copying, Press "ESC", and the last copy will be highlighted, press "ESC" a second time to remove the highlighting and exit the Select environment.

COPY TO GRID

When a single component is selected and the Move Entity or Copy Entity command issued (**[Edit]Move** or **[Edit]Copy**), ProCAD automatically uses the cell reference origin as the move from or copy from point. This feature allows for the precise placement of the cell reference origin to the current snap grid (Ghost or Display Grid).

*Note: If for some reason you accidentally placed your cell when snap on grid was off, you may toggle snap on grid back on (**[Other...]****Toggles** dialog box), select the cell and then issue the **[Edit]Move** command. This will in turn snap the origin of the cell back on the current grid.*

Note: The move to grid sequence only works when the Move or Copy command is issued from the menu or entered on the command line when Quick Set is OFF. The implied move (drag move) will use the current cursor position and NOT the cell origin as the from point. Also, you may only use this feature when a SINGLE entity (cell) is selected.

CROSS PROBE

PURPOSE: To cross probe physical vs logic devices and Nets between the PCB and Schematic databases (drawings).

SYNTAX: **CRO**ssprobe <CR>

EXPLANATION:

This feature requires the MDI (Multiple Document Interface) option. The Cross Probe command may be accessed from the **[Check]Cross Probe** menu item. When invoked, a menu of options is displayed. You may select probe by signal name, node location, connections to a component or the device/footprint links. Note: All databases comprising your design must be loaded before this command is issued. You may use the **[Windows]Tile** menu item to tile all opened files, so that you may view the cross probe result instantly. For best results, PCB databases must be designed with the rats nest enforced feature in effect.

Note: Probed devices are automatically placed in select-hold mode. To remove the hi-lite from a probed device, you should issue the **De-Select All** command from the **[Edit] menu**. See the **HOLD SELECT** command for details.

CUT (Copy to temp)

PURPOSE: Allows you to copy selected entities to a temp area for subsequent pasting to another PROCAD database.

This command is only available in PROCAD for Windows MDI version.

SYNTAX : **CUT** <RETURN>

EXPLANATION:

Use this command to copy selected entities to a temporary cell area for subsequent pasting to another database. This command is most effective if issued from within the select environment. The command is equivalent to issuing a "CELL FORMATION" command using "[CELLIB]PROCAD\$\$" as the cell name. This command does not "cut" the selected entities from the database. You must also issue the "DELETE" command following the "CUT" command to obtain the customary windows "CUT" style command. Note also that this command does not use the windows clipboard. For clipboard operation, you must use the "CLIPBOARD" command.

NOTE: Since this command is a special case of the "CELL FORMATION" command, all limits and restriction placed on CELL FORMATION command applies. You may use the EDIT CELL command to modify the copied data in [CELLIB]PROCAD\$\$. See also the PASTE command, which is the companion to this command.

DEFINE MENU

PURPOSE: To define, edit or load user pop-up menu text and commands. In Windows, this command invokes the [View]Preference... dialog box for tool button and artwork view definition.

ProCAD PowerStation 32 and ProCAD Advanced for Windows:

PURPOSE: To define assignments to 7 user tool buttons and artwork work layer combinations.

SYNTAX: **[View]Preference...**

EXPLANATION:

This command allows for the definition of the 7 tool buttons provided in ProCAD and the various layer combinations for the artwork views combo box. In ProCAD for DOS, this command allows for the definition of menu items. This command actually invokes the preferences dialog box (**[View]Preferences...** menu item).



You may enter your appropriate layer(s) for each artwork view in the corresponding edit fields e.g.

Reference Guide...

For Component Side, enter layers 5,13,16,20.

Note: You may use range specifiers when entering layer combinations. The Default layer combinations are those defined in ProCAD Quick Reference and Reference Manuals.

For Tool button definitions, you must enter the appropriate "C Shell" (2-letter mnemonic) command and any parameter(s) in the provided edit control field for each of the tool buttons you wish to define.

Example: US CKT.CMD

The above example will invoke the macro file CKT.CMD. The "US" command is the "C-Shell" (2 letter mnemonic) command for MACRO, and CKT.CMD is the macro file name to invoke.

Note: For the C Shell command cross reference to standard ProCAD commands, you should refer to ProCAD Quick Reference Manual or the appropriate Help topic in ProCAD.

If you have Quick help item enabled in the Preference dialog box above, ProCAD will display the assigned command in the help view (title bar area) when you move your cursor across the appropriate tool button.

ProCAD Xtra for DOS:

PURPOSE: To define, edit or load user pop-up menu text and commands.

SYNTAX #1: **DEF**ine <CR>

DEFine menufile <CR>

DEFine/L menufile <CR>

EXPLANATION:

The define menu command allows the user to define their own tear off menu text and corresponding command line. The first syntax will allow definition or editing the current menu, the second syntax will allow the definition or editing of a user specified menu file, and the third syntax will only load a user specified menu file. The third syntax is useful for inclusion in a start-up macro command file where only menu loading and no editing is required.

DELETE ENTITY(S)

PURPOSE: Allows selected entity(s) to be deleted from the database.

SYNTAX : **DEL**ete<RETURN>

EXPLANATION:

The specified entity will be marked for deletion in the database and removed from the display.

NOTE: Deleted entities can recovered with the UNDELETE command, if the database has not been packed by PACK DATABASE command. This command can be used in the SELECT environment. See also the note on "Context Sensitive Commands" section of the User & Utilities manual.

DELETE SEGMENT

PURPOSE: This command is used to delete an identified segment of a selected line or wire.

SYNTAX : **DE**lete Segment<RETURN>

EXPLANATION:

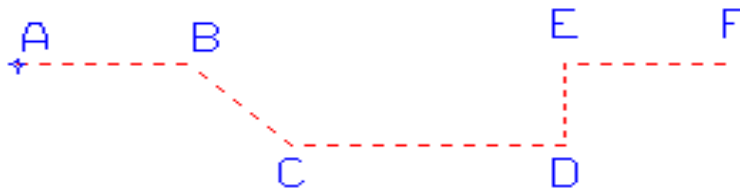
After issuing the DEL S command, you will be prompted to identify the appropriate segment by its end coordinates. The identified segment will be changed to its original color, and you will be asked for confirmation that it is the segment to be deleted. After your response, the segment will be deleted.

The remaining two segments will become two completely independent entities. In the case of a wire, two new signal names will be assigned.

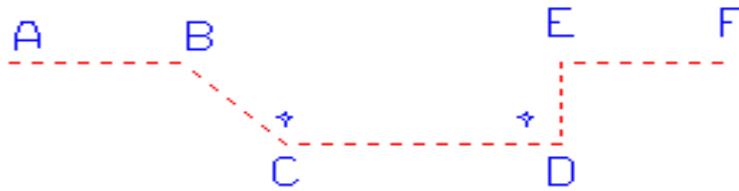
NOTE: This command can be used in the SELECT environment. See also the note on "Context Sensitive Commands" section of the User & Utilities manual.

EXAMPLE:

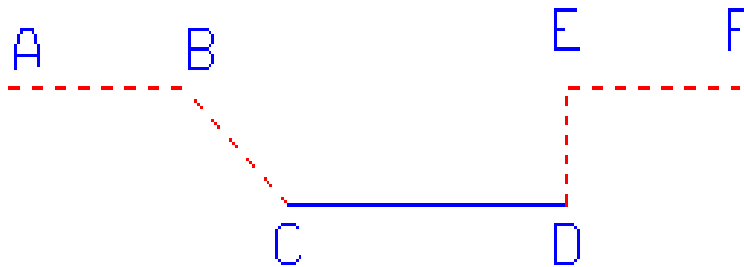
STEP 1 - Select the line or wire from which you want to delete a segment.



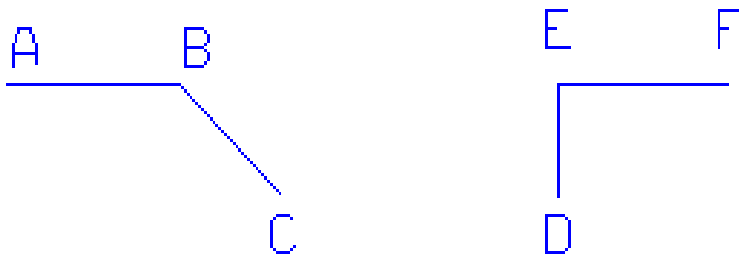
STEP 2 - You will then be prompted for the end vertices of the segment that you want to delete.



STEP 3 - The selected segment will be displayed in its normal layer color and you will be prompted to confirm that you want to delete that segment.



STEP 4 - After confirmation, the two independent segments will be displayed in their normal layer color.



DELETE VERTEX

PURPOSE: This command allows you to delete a specified vertex from a selected line or wire.

SYNTAX : **DE**lete **V**ertex<RETURN>

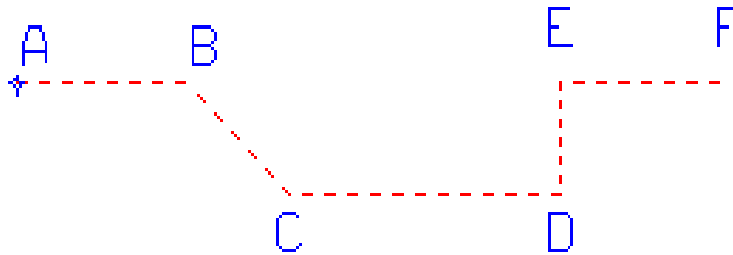
EXPLANATION:

After issuing the DEL V command, the command processor will select the nearest line or wire and delete the nearest vertex.

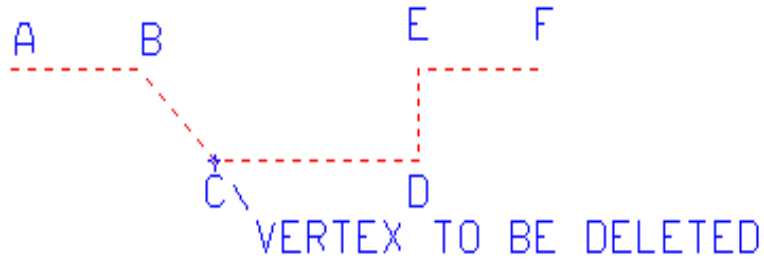
NOTE: This command can be used from within the SELECT environment. Press <ESC> to exit the SELECT environment. See also the note on "Context Sensitive Commands" section of the User & Utilities manual.

EXAMPLE:

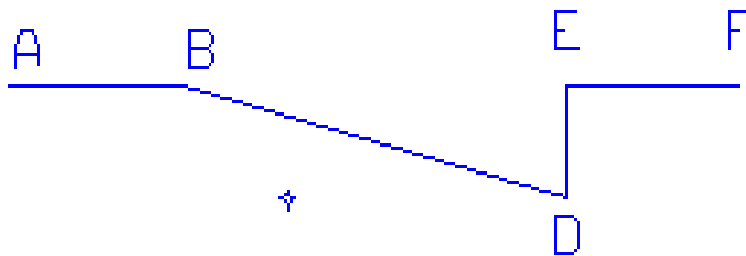
STEP1 - Select the line or wire from which you want to delete a vertex.



STEP2 - You will then be prompted for the vertex that you want to delete.



STEP3 - The selected vertex will be deleted and the selected wire or line will return to its normal layer color.



DESIGN RULE CHECK

PURPOSE: To provide an on-line means of checking the minimum spacing of traces and pads on a PCB layout. This command will also check for graphical shorts.

SYNTAX #1: **DRC** {drc_control_file} <RETURN>

SYNTAX #2: **DRC/V** <RETURN>

SYNTAX #3: **DRC/H** <RETURN>

EXPLANATION:

The first syntax of this command opens a small window of Design Rules which are fully user definable (DRC defaults will be loaded from the drc_control_file if it is specified on the command line), which are then used to check the quality of the design. Because the DRC command is on-line, you may use it to evaluate the design as it progresses, without ever leaving the PROCAD editor.

The DRC program also checks for graphical shorts between the layers specified. This feature allows you to insure that you have not routed one trace over another on the same layer.

To clear flags placed by the DRC program, you must execute the CLEAR ERRORS command.

The second syntax of the command (with the **"/V"** qualifier) will display the graphical interpretation of all DRC\$PADx keywords used in the database. The **[Check]Show DRC\$PADx** graphics menu item is equivalent to the second syntax of the DRC command above. **This command is useful for verifying proper specification of the DRC\$PADx attribute.**

The third syntax of the command (with the **"/H"** qualifier) will display the graphical interpretation of all DRC\$HOLE keywords used in the database. The graphical info is in a form of a circle of diameter corresponding to hole size and placed on layer 49. The **[Check]Show DRC\$HOLE graphics** menu item is equivalent to the third syntax of the DRC command above. **You should use this command to verify all hole position and sizes before using ProDRILL to generate Excellon drill file. Use the [Check]Clear DRC Errors & Lyr 49 Info menu item to clear the "hole " circles placed on layer 49.**

The Design Rule Checker (DRC) may be enabled for real time operation (on-line) if you are in PCB mode. You may enable or disable on-line DRC from the **[other...]****Toggles** menu item dialog box. You may also set the default clearance & shorts checking parameters from this dialog box.

Note: On-line DRC is ignored when you are in schematic mode irrespective of its settings.

See section on USING DESIGN RULE CHECKER in the User's manual and CLEAR ERRORS command for more information.

DUMP SCREEN

PURPOSE: To dump the screen to a hardcopy device.

Note: This command is not supported in ProCAD for Windows

SYNTAX: **DUMP** <RETURN>

EXPLANATION:

This command is used for dumping the image on the screen to a hardcopy device. The currently supported hardcopy devices are those supported by the DOS GRAPHICS.COM program. Prior to using this command, the dos program GRAPHICS.COM must be executed. Refer to the DOS GRAPHICS.COM program for details. NOTE: This command may not be applicable to most drivers and hardcopy devices.

EDIT CELL

PURPOSE: This command enters the EDIT CELL environment from the MAIN Database Mode to create a new cell or edit an existing cell.

SYNTAX 1: **EDIT** <cellname><RETURN>

EXPLANATION:

This command allows you to create or edit an existing cell in the current working library. After you are finished with the cell, enter the command SAVE<RETURN>. If it is an existing cell, you will be prompted for editing the cell properties. A YES response will allow you to change any of the properties associated with that cell. When you have changed all of the properties that need changing, you will be prompted to save the cell. A NO response will not allow you to edit the cell attributes. You will then be prompted to save the cell as is.

NOTE: If the cellname is omitted and the current cursor position is within a cell's extents, that cell will be selected for editing

SYNTAX 2: **EDIT** [libname]<cellname><RETURN>

EXPLANATION:

This form of the command may be used to edit a cell in a library other than the current working library.

NOTE: A question mark "?" in place of the [libname]<cellname> variable will cause ProCAD to display a menu of cells in the current active directory, from which you may select the cell to be edited. See the section CREATING & EDITING A CELL in the User's Manual for further more detailed information.

See also the MAKEPAD command for automatic padstack cell generation.

EXIT

EDIT CELL ENVIRONMENT -

PURPOSE: Allows you to exit the EDIT CELL environment or the MAIN Database Mode.

SYNTAX: **EXIT**<RETURN>

EXPLANATION:

Terminates the EDIT CELL environment - See the EDIT CELL Section of the User's Manual for more detailed information.

MAIN DATABASE MODE -

SYNTAX: **EXIT**<RETURN>

EXPLANATION:

PROCAD for DOS:

Pack and Saves the database to disk, then exits the PROCAD program.

Note: The database is packed without re-numbering any instance names.

PROCAD for Windows:

This command will prompt for automatic saving of any modified database files currently opened before returning control to windows. If a save is requested for a modified database file, the file will be saved without any form of packing or backup generation. You must manually pack and save your databases before exiting PROCAD if you want deleted entities permanently removed from the database.

See also the PACK, SAVE and EDIT CELL commands..

EXPLODE

PURPOSE: Removes the cell hierarchy one level at a time. Reduces the selected cell to it's individual entities. Expands a copper pour (AFILL) polygon to individual scan line traces.

SYNTAX : **EXP**lode<RETURN>

EXPLANATION:

CASE #1 (A cell was selected):

After issuing the EXPLODE command, the selected cell will return to it's original color (layer) as it is exploded. What was once a single entity (cell) will now be a group of entities that made up the cell originally. Each entity can be manipulated at will. The hierarchy can be reinstated by the use of the CELL FORMATION command.

CASE #2 (A copper pour (AFILL) polygon was selected):

When exploded, the individual lines comprising the Area Fill may be edited using the normal Edit commands that are applicable to lines (Move Vertex, Add Vertex, Re-Route etc.).

NOTE: Exploded AFILL polygons takes up a lot of data storage space. They do however, refresh significantly faster than the un-explode AFILL polygon. If you already have so much entities in your database, you may not be able to explode AFILL polygons.

NOTE: This command can be used with the SELECT environment. Press <ESC> to exit the SELECT environment.

See also the EDIT CELL and CELL FORMATION commands.

EXPORT

PURPOSE: To generate ProCAD or other CAD vendors ASCII format file from a ProCAD database.

SYNTAX: **A**Export <CR>

EXPLANATION:

This command is used for converting ProCAD drawings to ProCAD or other CAD vendors ASCII format files. When issued from the **[Files]** menu or entered on the command line using the above syntax, ProCAD displays a menu of currently supported ASCII formats. You should choose your desired vendors CAD from the list, and ProCAD will generate the corresponding ASCII file. Some of the CAD vendors supported includes Protel, Tango Series II, P-CAD, AutoCAD DXF, etc.

Note: You should check the README and HELP files for new additions and any limitations and restrictions on the conversion process for your specific CAD vendors format.

GET CELL

PURPOSE: To get a cell from a library for real time placement in the drawing/database.

SYNTAX: **GET** [libname]cellname <RETURN>

EXPLANATION:

This command will allow the user to get a cell from the library for placement in the database in real time. The cell picked up from the library will move in real time with the cursor. A <CR> will place the cell in the current cursor location. This command is equivalent to the combination of the input/cell, rubberband on, rotate, mirror, and move commands. ZOOM and PAN commands can be issued while this command is active. Press <ESC> to exit the command.

Note: This command is also invoked as part of the RAT'S NEST command during package placement if the rubberband flag is on.

HELP

PURPOSE: To provide on-screen help about the use of any PROCAD command.

NOTE: If you are using PROCAD for Windows, you should obtain help from the drop down menu (Help menu item). The Windows HELP processor will be used for processing your help requests. The descriptions and usage of the Windows help is available as a HELP item in the drop down menu. The description and explanation below only applies to PROCAD for DOS.

SYNTAX: **HEL**p{=command}<RETURN>

EXPLANATION:

Entering "?", or "HELP" in place of {command} or omitting command_keyword will cause PROCAD to show a list of all available commands by their full name & syntax. PROCAD will normally ask you to press ENTER to see the next screen for multiple screen help messages, or ESC to exit the help message display. In response to this prompt, you may enter a valid command before pressing <RETURN>, and PROCAD will automatically display the help message for that command. This feature is especially useful when you are viewing the command list and find the syntax of the command desired, you can then go directly to the help message for that command just by typing the command in response to the "press ENTER or ESC" prompt.

When using tear-out menus, you may invoke a help screen for a highlighted menu item just by pressing F1 (Function key #1).

Note: The data for the help screen is an ASCII file which is converted to the usable keyed-binary format (HELP.BIN) by the PROHELP utility. You may define your own help data. See the section on PROHELP utility for more information.

The standard initialization macro files (CKT.CMD & PKG.CMD) will program F1, Cntl-F1, Shift-F1, and Alt-F1 to invoke a help screen for function keys F2 - F10, Cntl-F2 - Cntl-F10, Shift-F2 - Shift-F10, and Alt-F2 - Alt-F10 respectively.

HISTOGRAM

PURPOSE: To generate a histogram or color coded density map of channel density usage.

SYNTAX #1: **HIS hlyr vlyr <CR>**

SYNTAX #2: **HIS/D clyr olyr <CR>**

EXPLANATION:

This command will allow for generating a histogram of trace density or color coded density map.

SYNTAX #1. This first syntax above will generate a histogram. The **hlyr** and **vlyr** are the horizontal and vertical layers to consider while checking trace density. If you are using this command on a rat's nested database, you should specify 30 for both the **hlyr** and **vlyr** arguments. If you are interested in generating a histogram check for a finished or partially finished PCB design, you should enter the proper values for the horizontal and vertical layers which are typically 20 and 23 respectively if you are using our suggested layering conventions.

Example: **HIS 30 30 <CR>**

The above example will generate a horizontal and vertical histogram of trace densities for traces placed on layer 30 only (Rats Nest!).

SYNTAX #2. The second syntax will generate a color coded map of trace density within regions of the design. Only traces placed on **clyr** are considered. The screen will then be refreshed, this time only making visible the density map and entities on **lyr**.

Typically, **clyr** is your rats nest layer, and **lyr** is your component outline layer.

Example: **HIS/D 30 3 <CR>**.

The above command will generate a color coded density map for traces placed on layer 30 while only making the map and entities placed on layer 3 visible.

Note: You may use any of the screen refresh commands to re-paint your screen and eliminate the Histogram or Density map diagrams. To see a density or Histogram for your whole drawing, you may use the SCREEN SCALE command to ZOOM out so that your whole drawing is visible on the screen before issuing the HIS or HIS/D commands.

HOLD SELECT

PURPOSE: This command allows you to put the "SELECT_ENTITY" or "SELECT_WINDOW" commands in suspension, leaving the selected entity(s) highlighted, while you attend to some forgotten task. When you have finished the task(s), you can return to the "SELECT" environment by issuing the "SEL E(W)" command.

SYNTAX: **H**old<RETURN>

EXPLANATION:

This command is very useful if you have just selected some entities, and forgot to SET SNAP, SET RUBBERBAND, SET GHOST GRID, etc. Issuing this command allows you to enter those commands not available in the SELECT environment and then return to the SELECT environment.

When a non select environment command is issued while in select environment, eg: INPUT/WIRE, ProCAD automatically places selected items in hold mode, exit the select environment and then execute the command in main mode. You may return to the select environment (held items) by issuing a Select Entity or Select Window command.

Note: You may also invoke the Hold command by typing Cntrl-Z while in the SELECT ENVIRONMENT.

IMPORT

PURPOSE: To import ProCAD or other CAD vendors ASCII files.

SYNTAX: **A**Import <CR>
 AImport/**c**

EXPLANATION:

This command is used for converting ProCAD ASCII files (generated from the EXPORT command) or other CAD vendors ASCII files to ProCAD Binary database. When issued from the **[Files]** menu or entered on the command line using the above syntax, ProCAD displays a menu of currently supported ASCII formats. You should choose your desired vendors CAD from the list, and ProCAD will generate the corresponding database and libraries. Some of the CAD vendors supported includes PADS, Protel, OrCAD library, PCAD, AutoCAD DXF, etc.

The second syntax of the command will bypass the main menu and directly import the file format specified as "c", where c=P (ProCAD ASCII), G=(Gerber Import), O=(OrCAD library), T=(Tango) etc.

Note: You should check the README and HELP files for new additions and any limitations and restrictions on the conversion process for your specific CAD vendors format.

TANGO SERIES II & PROTEL AUTOTRAX IMPORT

Tango and Protel data structure does not support pointer to cells in a component library. Cells (components) placed in these drawings becomes part of the drawing. ProCAD on the other hand maintains pointer to cells in a library. During import of Tango or Protel PCB files, ProCAD attempts to recreate a unique library cell for **EVERY** component in the drawing. For large drawings, this will result in a large number of cells created (even though some of the cells may be identical). The large number of unique cells in a drawing will cause for a severe degradation of refresh speeds in large designs. You should explode cells after import (or use the **REPLACE** command to replace a group of similar cells with one unique cell) to achieve faster screen refreshes.

IMPORT (GERBER)

PURPOSE: To allow for import of Gerber data into ProCAD database.

SYNTAX #1: **AI**import/**G**<CR>

EXPLANATION:

This feature is invoked either from the **[File]Import** menu, or entered from the command line using the above command syntax, then selecting the Gerber format from the import menu.

This release of ProCAD will import RS274X Gerber Data format. The Gerber Import function could also be accessed via the **[File][Gerber plot]Import Gerber file...** menu item. When executed, the following dialog box is displayed:



GERBER FORMAT: You should use this combo drop down box to select the desired format. ProCAD supports the RS274D (Default) and the RS274X (embedded aperture) formats. If you select the RS274D format, you should provide a aperture setup file (Similar to GERPLOT ".SET" or ".OEH" file format)

Note: ProCAD supports only aperture file formats generated by GERPLOT (See the section on files generated by GERPLOT in the Users & Utilities Manual for details). If you are importing Gerber data generated from other CAD systems, you must first convert the aperture file to one of ProCAD formats (".SET" or ".OEH") or generate RS274X format from that CAD system if it supports it.

In addition to providing the aperture setup file, you should select the proper data format from the drop down combo box. ProCAD supports 2:3, 2:4 and 2:5 data formats.

Note: If you specify RS274X Gerber format, You don't need to specify input data format or provide an aperture setup file since these information are contained in RS274X Gerber format files by definition.

GERBER TRACES MAPPED TO WIRES: This check box allows for the possible reconstruction of a database with connectivity information from a Gerber file. When this box is checked, ProCAD will treat all imported traces as ProCAD wires with machine assigned net names. You could change the net names to valid nets after import. Note: The current release of ProCAD does not read a netlist file for automatic net reconstruction. This flag increases the size of database generated thus affecting the redraw speed. You should NOT check this box if you don't desire a connectivity reconstruction or DRC checking of imported Gerber data.

Note: Imported flashed pads are saved as solid circles or rectangles in ProCAD, not as Pad cells. You may not be able to perform pad-to-trace or pad-to-pad DRC operations on these entities.

ARTWORKS: You should select Gerber data file(s) and the corresponding ProCAD layer(s) where they will be placed. Note: Clicking the "Brw" (Browse) button next to a file field will allow for the selection of a file from a menu of available files.

Press the OK button to start the import process. ProCAD will import all files specified in the Artworks section and place them on the corresponding layers in the same database.

FOR BEST RESULT, YOU SHOULD IMPORT DATA TO AN EMPTY PCB DATABASE ([FILE]NEW..).

Note: If you are using special flash patterns (GER\$FLASH token in GERPLOT), you must provide a cell of the same name in library [GERBER]. See the discussion on "Setting Attributes for Special Flash" in the section on GERPLOT in the Users and Utilities Manual.

The Gerber import command should be used instead of reading the optional ".CMD" file generated in GERPLOT.

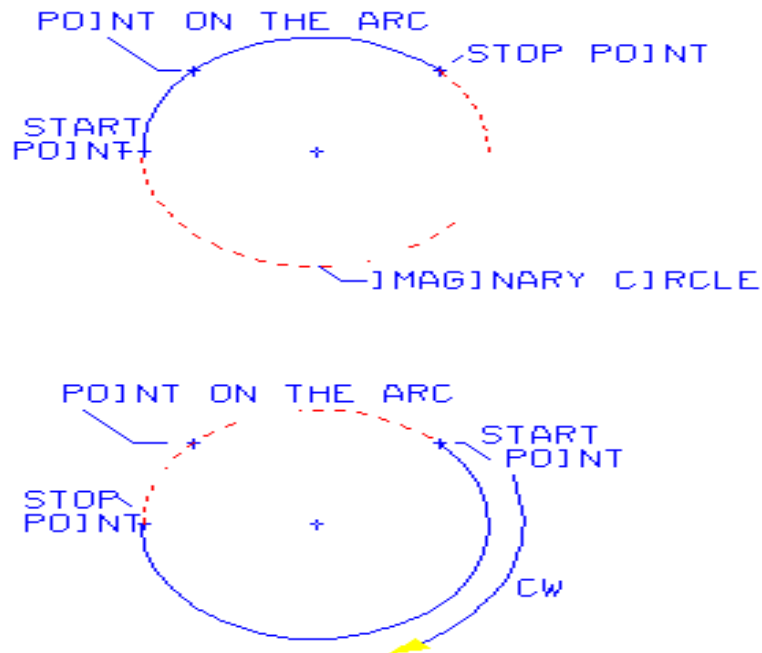
INPUT

PURPOSE: To allow the inputing of entities to the database.

SYNTAX: **IN**put/{zz}<RETURN>

EXPLANATION:

zz= **A**, for ARC, system will prompt for Start point, Stop Point and a point on the arc. An arc passing through these 3 points will be drawn in a clockwise direction. After entering the last arc, press <ESC> to terminate the INPUT mode.

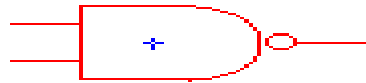


zz=**CE** <cellname> {n}, for CELL. System will prompt you for the location that you want the cell placed. n == scale factor, and must always be a positive integer; if omitted, system will not prompt for it and assume a scale factor of 1x. A placement name can be assigned, see CHANGE_INSTANCE_ ATTR command. The command IN/CE [libname]cellname {n}, can be used to input a cell from another library, other than the current default working library. (See GET_CELL command for an alternate

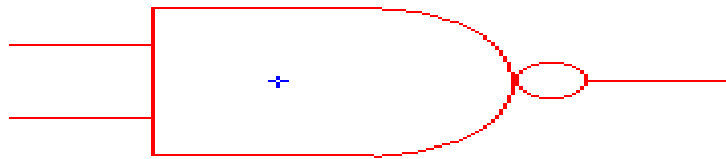
method of inputting a cell). If you omit the cell name, the system will open a window of available cells in the current default library. You may then pick a cell with your mouse or tablet from that list.

"IN/CE NAND2<RETURN>"

"IN CE NAND2"

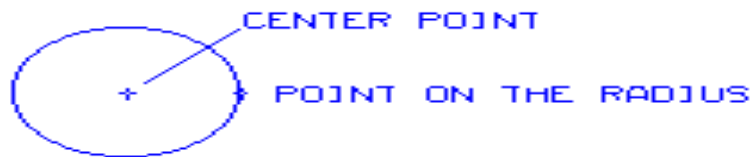


"IN CE NAND2 2"



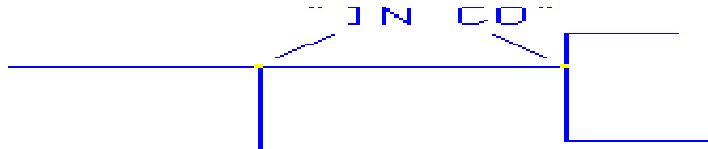
"IN/CE NAND2 2<RETURN>"

zz=**CI**, for CIRCLE, system will prompt for Center point and a point on the circumference of the circle. After entering the last circle, press <ESC> to terminate the INPUT mode. If you are operating with rubber band mode ON, the circle will be input in real time after entering the center coordinate.



zz=**CO**, for CONNECTION DOT, system will prompt for the location where the connection dot is to be placed. The program assumes that the CELLIB.LIB contains a cell whose cell name is "DOT". This is the cell that will be placed in your drawing. The input connection dot is equivalent to "IN/CE [CELLIB]DOT" and ADD Vertex (if

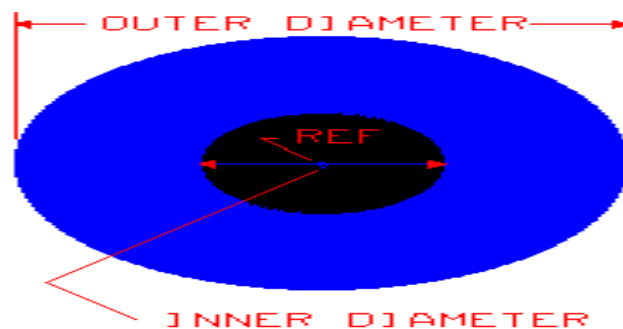
necessary) commands. Vertex will only be added to crossing horizontal and/or vertical lines to establish connectivity. This command will only add necessary vertex in Schematic mode of operation.



Note: When you are in schematic mode, starting or ending a wire on an existing horizontal or vertical wire segment ("T" connection), will cause an automatic connection dot and necessary vertex placement on the "T" junction to establish connectivity.

zz=**CU**, for CURVED TRACES, system will prompt for the starting point, end point and a point on the circumference of an arced trace. This command is similar to the ARC input mode, with the properties of wire mode in effect (trace width, signal name etc) maintained for the newly entered wire.

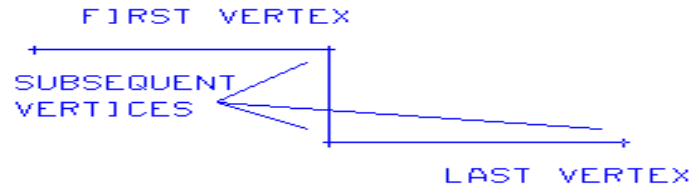
zz=**D**, for DONUT, system will prompt for Outer Diameter, Inner Diameter and placement coordinates. After entering the last donut press <ESC> to terminate the INPUT mode.



zz=**G**, for Ground/Copper plane, This is a special case of input polygon used for generating automatic area fill areas around traces, pads, etc..

See the section on **AREA FILL** command for the explanation and use of this command.

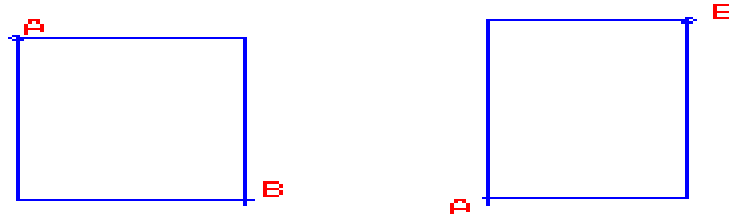
zz=**L**, for LINE, system will prompt for the beginning and subsequent vertex points. After entering the last vertex point, press <ESC> to terminate the INPUT mode. The type of line entered (straight, curved, vector or H/V orthogonal) is determined by the SET DIGITIZING MODE command. If a vertex is placed wrong, it can be removed with the UNDO command without leaving the INPUT mode. Also, the PAN and ZOOM commands can be issued while still in the input mode. If you are operating with rubber band mode ON, the line will be drawn in real time after entering the start coordinate.



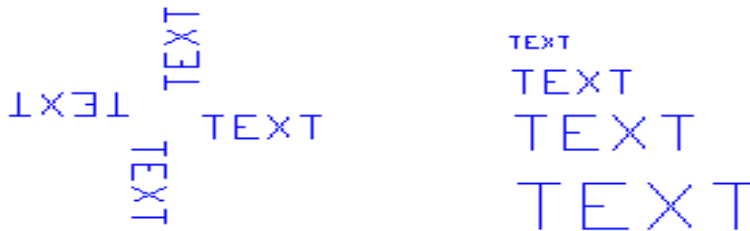
zz=**P**, for POLYGON, system will prompt for the beginning and subsequent vertex points. After entering the next to the last vertex point, press <ESC> to place the last vertex on the same coordinate as the first and terminate the INPUT mode. If a vertex is placed wrong, it can be removed with the UNDO command without leaving the INPUT mode. Also, the PAN and ZOOM commands can be issued while still in the input mode. If you are operating with rubber band mode ON, the polygon will be input in real time after entering the start coordinate.



zz=**R**, for RECTANGLE, system will prompt for: opposite corner vertex points (2), press <ESC> to terminate the INPUT mode. If you are operating with rubber band mode ON, the rectangle will be drawn in real time after entering one of the corner coordinate.



zz=**T**, for TEXT, the system will use the height, width and angle values established by the **SET LABEL** command ([**Other...**]**Constants** dialog box), and will prompt for starting point for the vectorized text. Height and Width are in database units. Pressing the <ESC> key will exit the text input mode.



USE OF TRUE TYPE FONTS

ProCAD Advanced for windows supports the Windows True Type Fonts for Schematic design. You may select the desired font from the [**Other...**]**Constants** menu item dialog box.

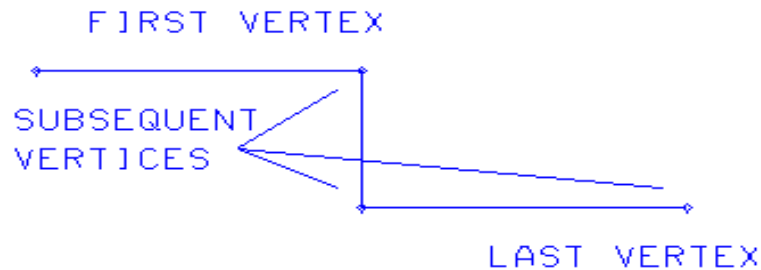
Note: ProCAD will revert to the normal vector fonts in PCB mode. True fonts are currently supported for all text and label entities in schematic mode only. Since true type fonts are usually proportionally spaced, the width argument is ignored. Note also that the Gerber and Pen plot (DMP/L, HPGL etc) output formats do not support true type fonts. Since these fonts are usually proportional spaced, auto reverting to vector fonts on output devices that do not support true type fonts will make for very poor and unpredictable output results.

zz=**W**, for WIRE, system will prompt for the beginning and subsequent vertex points. After entering the last vertex, press <ESC> to terminate the INPUT mode. PROCAD will assign an attribute (Signal Name) to each wire entered into the database. If a signal name is specified as part of the command (eg. IN/W CLK), PROCAD will assign that name to the wire being input. This attribute can be changed, see **CHANGE SIGNAL NAME** command. If a vertex is placed wrong, it can be removed with the UNDO command without leaving the INPUT mode.

The trace width of the wire being input may be changed on-the-fly if the user pushes the "W" key while ProCAD is prompting for a coordinate. In this case, ProCAD will prompt for the desired trace width, then automatically closes the current trace and then starts a new trace using the new width parameter.

The PAN and ZOOM commands may be issued while still in the input mode. If you are operating with rubber band mode ON, the wire will be drawn in real time after entering the start coordinate.

Example: IN/W STROBE (Input wire with signal name = PROBE)
IN/W (Input wire with program generated signal name)

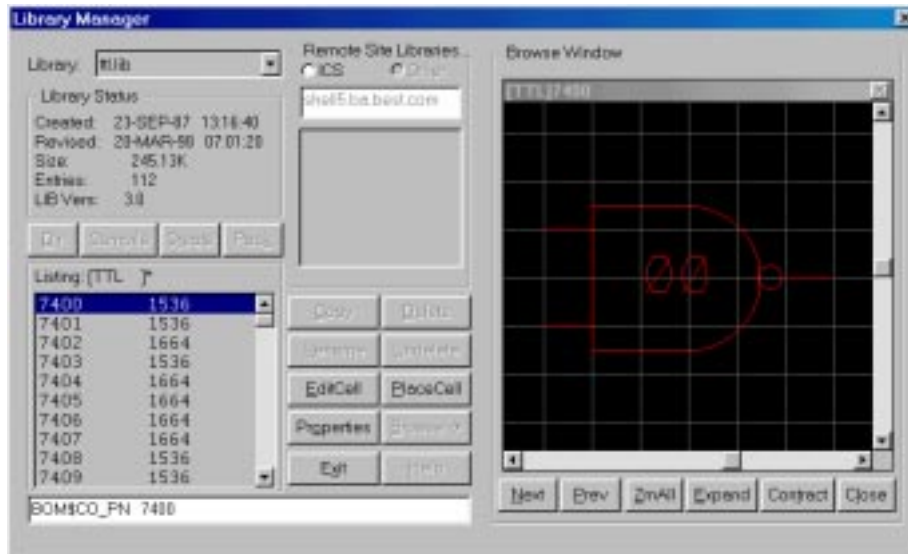


Note: To enter a bus or bundle, you should enter a single wire and use the STEP & REPEAT command to generated the other bus/bundle wires.

LIBRARY MAINTENANCE

PURPOSE: Allows you to enter the LIBRARY MAINTENANCE Mode.

SYNTAX: **LIB**rary_maintenance<RETURN>



Refer to the Library Maintenance Commands at the end of this Chapter for a discussion of this command.

MACRO

PURPOSE: To provide a means of executing a command file from within PROCAD.

SYNTAX #1 **MACro** {<file.cmd>}<RETURN>
SYNTAX #2 **MACro/P** {<file.cmd>} <RETURN>
SYNTAX #3 **MACro/H** {<file.cmd>} <RETURN>

EXPLANATION:

This command will invoke a file of command strings for execution within PROCAD. The file must be an ASCII file with the extension of .CMD. The file may contain any of the following commands or special characters as the first non-blank character on the line.

a. Any PROCAD command with or without it's argument.

b. The following special characters:

\$ == Comment. Any text following is treated as a comment and ignored by PROCAD.

% == Pause. Will cause the execution of the MACRO command file to be halted for the time specified by the WAIT value.

< == <RETURN> Represents pressing the <RETURN> key.

> == Prompt. Will cause PROCAD to display the text following the symbol as a prompt.

^ == <ESC> Same as pressing the <ESC> key.

* == Loop. Will cause the execution to loop at the begining of the MACRO file until interrupted.

Since pop-up menus and all commands that invoke the mini-text editor (SET CONSTANTS, SET TOGGLES, SHOW ATTR, EDIT-CELL, CELL-FORM pin/attribute editing etc.) will require an operator intervention, they are disabled automatically while the macro command file is being executed. You should in particular avoid using SET

CONSTANTS & SET TOGGLES commands in a macro command file. You should instead use the equivalent commands necessary for setting the appropriate program parameters.

The **"/P"** qualifier will cause PROCAD to echo the commands as they are being executed. This is useful when you are still debugging the macro command file. The **"/H"** qualifier will cause PROCAD to halt macro execution if it encounters a fatal error (error number greater than 500) during macro command execution. You may use both or no qualifier when you issue the macro command eg

MACRO/P/H VCC

When a new database file is created, ProCAD automatically executes the **CKT.CMD** or **PKG.CMD** macro file if the initialized database is a Schematic or PCB type respectively.

The QUICKSET command syntax is assumed to be off during MACRO command file execution (see SET QUICKSET command).

NOTE: The MACRO command can be interrupted by pressing <ESC> twice. When issued from the select environment, only select mode commands will be recognized. Use the HOLD SELECT command if non-select mode commands are to be issued. You can return to the select environment by including a SEL command in the macro command file. See the SET LOG FILE command for information on generating some types of .CMD files.

MAKEPAD

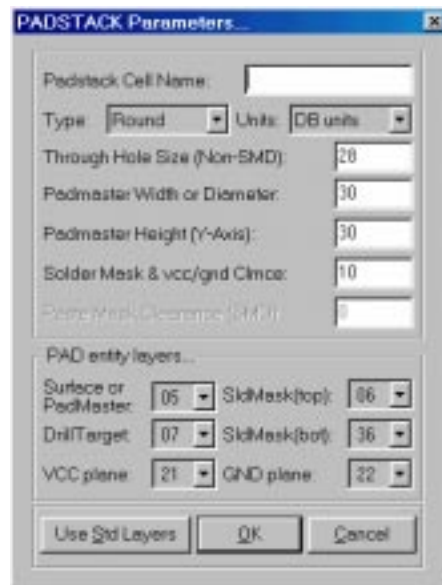
PURPOSE: To automatically generate a padstack cell.

SYNTAX: **MAK**epad <CR>

EXPLANATION:

This command may also be invoke from the **[Library]Padstack Wizard...** menu item.

When issued, this command allows for the automatic generation of a padstack cell. The following dialog box is used to collect the necessary data for the padstack cell.



Padstack Cell Name: This is the name of the cell to be generated. You may specify a library name (enclosed in square brackets) as part of the cell name

Examples:

[PCBLIB]SQPAD40
SMD2474

Note: If the specified library does not exist, ProCAD will create it. If you do not specify a library qualifier (second example), the cell will be placed on the current default library.

Type: You may select Round, Square, Oval, etc. pad styles.

Units: This is the units this padstack cell is defined in. You may select English or Metric units. Database units means that the values and dimensions of the cell will use the units of the database the cell is placed on. This allows for the flexibility of using the cell in English and Metric unit designs without automatic units conversion.

Depending on the pad type selected, you will provide the height, diameter, pad layers, etc. variables.

If you are using ProCAD standard layering conventions, you do not need to change the default PAD entity layer Settings.

Press the **Okay** button when finished, and ProCAD will generate the padstack cell with the proper attributes (DRC\$PADn, DRC\$HOLE, etc.).

Note: This command is only available in PCB mode Refer to Appendix F for explanation of padstack structure and layering conventions.

MENU

PURPOSE: This command activates the tear-off menu display.

NOTE: This Command is not supported in PROCAD for Windows. See Appendix B for PROCAD for Windows Menu items & descriptions.

SYNTAX: **MENU** {menu_number}<RETURN>

EXPLANATION

Issuing the above command without a menu number will invoke the main menu (Menu #1). To select a specific menu, issue the command with the appropriate menu number. The menu numbers are established during the definition of the menu. (See the DEFINE MENU command and Appendix B for more details) The standard menu supplied with PROCAD have the following assignments:

Menu # 1	MAIN
Menu # 2	FILE
Menu # 3	SCREEN
Menu # 4	EDIT
Menu # 5	PLACE
Menu # 6	ROUTE
Menu # 7	CHECK
Menu # 8	LAYER
Menu # 9	GRID
Menu #10	DRAWING
Menu # 11	LIBRARY
Menu # 12	MODIFY
Menu # 13	ARTWORK
Menu # 14	EDIT_2
Menu # 15	VIA/DOTS
Menu # 16	SMT menu
Menu # 17	TRACKSIZE
Menu # 18-20	Undefined

NOTE: You should re-program function key #30 to select the appropriate menu from the mouse if you do not always want to pop-up the main menu first. See also the chapter on MENUS for further details.

The above menu definition is subject to change with each new release of PROCAD. You may also re-define or create your own menu structures. (See the section on CREATING MENUS in the User's Manual).

MIRROR

PURPOSE: To allow the reflection of entities about the X and Y Axis

SYNTAX : **MI**rror X(Y)<RETURN>

EXPLANATION:

The system will prompt for a reference point about which to reflect the entity(s). Text entities are not mirrored in schematic mode. Only the reference points are mirrored. In PCB mode however, text entities are mirrored, allowing for placement of text in the solder side of a Printed Circuit Board.

NOTE: This command can be issued in the SELECT environment. Press <ESC> to exit the SELECT environment.

EXAMPLE:

STEP1 - After selecting the entity to be mirrored about the X or Y axis, you will be prompted for reference line about which the entity(s) will be mirrored.



STEP2 - After you have selected a reference point, the entity will be reflected about the straight line through that point.



Reference Guide...

MOVE ENTITY

PURPOSE: Allows you to move a selected entity(s) to a new position.

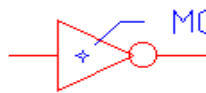
SYNTAX: **MOV**e<RETURN>

EXPLANATION:

The system will prompt for a "move from" reference point, then a "move to" reference point. After moving the entity(s) to their final position, press <ESC> to terminate the MOVE command.

NOTE: This command can be used with the SELECT environment. Press <ESC> to exit the SELECT environment. The entity(s) may be moved in real time by setting the RUBBERBAND=ON. See the SET RUBBERBAND command for details. See also the note on "Context Sensitive Commands" section of the User & Utilities manual.

DISPLAY WITH RUBBERBAND "OFF"



MOVE TO GRID

When a single component is selected and the Move Entity or Copy Entity command issued (**[Edit]Move** or **[Edit]Copy**), ProCAD automatically uses the cell reference origin as the move from or copy from point. This feature allows for the precise placement of the cell reference origin to the current snap grid (Ghost or Display Grid).

Note: If for some reason you accidentally placed your cell when snap on grid was off, you may toggle snap on grid back on ([Other...]Toggles dialog box), select the cell and then issue the **[Edit]Move** command. This will in turn snap the origin of the cell back on the current grid.

Note: The move to grid sequence only works when the Move or Copy command is issued from the menu or entered on the command line when Quick Set is OFF. The implied move (drag move) will use the current cursor position and NOT the cell origin as the from point. Also, you may only use this feature when a SINGLE entity (cell) is selected.

MOVE VERTEX

PURPOSE: To move a specified vertex on a selected line, wire, or polygon.

SYNTAX : **MOVE** Vertex<RETURN>

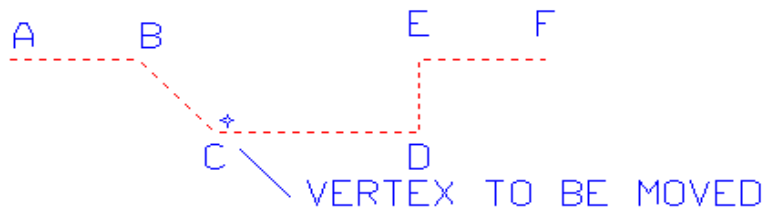
EXPLANATION:

After issuing the MOVE VERTEX command the system will prompt for the vertex to move, then it will prompt for the location to move to. Press <ESC> to terminate the MOVE VERTEX command.

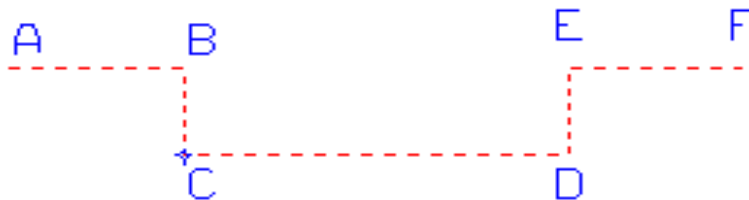
NOTE: This command can be used with the SELECT environment. Press <ESC> to exit the SELECT environment. The entity(s) can be moved in real time by setting the RUBBERBAND=ON. See the SET RUBBERBAND command for details. See also the note on "Context Sensitive Commands" section of this manual.

EXAMPLE:

STEP 1- After selecting an entity(s) to move, you will be prompted for a reference point to move from.



STEP 2- Then you will be prompted for the point to move to.



STEP3 - When the vertex is in the desired position, press <ESC> to terminate the command.



OPEN DATABASE

PURPOSE: To allow the changing of the active database.

SYNTAX **OPen** <existing.dbf><RETURN>
 OPen/New<RETURN>

EXPLANATION:

This command is used to call up an existing database or create a new database (Second syntax with "/NEW" qualifier). file for editing. When this command is issued and there is no existing database, the system will prompt for the new database's "type", "units", "screen display factor", etc. using a dialog box (PROCAD for Windows) then create a new empty database and make it the current active database. If you are working in the TEMPFILE.DBF, a new name can be assigned with the SET FILE command. If you OPEN a database from any file, other than TEMPFILE.DBF, you will be given a warning that the current active database has not been saved and any changes made will not be saved. A YES response will abort the current active database without saving it. A NO response will terminate the OPEN command so that you may save the current active database. See the SAVE command.

Note: PROCAD for Windows MDI (Multiple Document Interface) version will not abort the current active database. You may open as many as 25 database files at the same time. You may use the [Cascade], [Tile], etc commands on the [Windows] drop down menu to manage these opened database files. You may also use the Copy -> temp & Paste <- temp commands to move data between opened database files.

AUTOMATIC STARTUP DATABASE FILE

When ProCAD is invoked, it may automatically open the last database file that was loaded. If this is the first invocation, or the database file is not found, ProCAD will load or initialize the TEMPFILE.DBF database file. Use the ProCAD Configuration utility to enable or disable this feature.

AUTOMATIC STARTUP MACRO EXECUTION

When a new database file is created, ProCAD automatically executes the **CKT.CMD** or **PKG.CMD** macro file if the initialized database is a Schematic or PCB type respectively.

Note: Since setup information is always saved with a database file, automatic macro execution is disabled when an existing database file is loaded.

PACK DATABASE

PURPOSE: To remove deleted entities from the database, and/or optionally re-number instance names and forward/backward annotate the results.

SYNTAX: **PACK**/option<RETURN>

EXPLANATION:

This command removes all deleted entities from the database. After this command has been issued, no deleted entities may be undeleted unless they were deleted since the last PACK command was issued. If the "/R=nn" option is used, PROCAD will also re-assign the instance names of all the cells in the database that were generated by PROCAD or consistence with PROCAD instance naming convention, starting with instance number nn. If nn is omitted, a start value of 1 is assumed. If the option "/R=?" is used, PROCAD will prompt the user for all starting values for all prefixes in use in the database. If the option "/R=fname" is used, PROCAD will use the last+1 values of the prefixes used in database=fname as starting values for the database being packed. The SAVE or EXIT command during EDIT cell will cause an automatic PACK/R=1 command before the cell is actually saved.

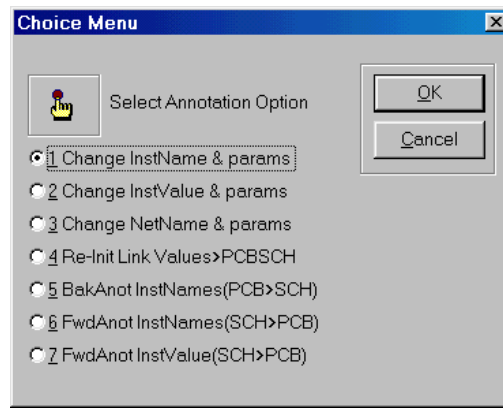
Example: PACK/R=SHEET1
 PACK/R=?
 PACK/R=200
 PACK/J
 PACK/Annotate
 PACK

The first example will pack the current database using the last+1 values of prefixes used in database SHEET1.DBF as starting values. The second example will prompt for starting values for all currently used prefixes in the database being packed. The third example will start re-numbering all prefixes starting with value 200. The forth example will pack and renumber all schematics currently loaded (project schematics). The fifth example will invoke the Annotation dialog box, and the sixth example will pack the current database without any re-numbering.

NOTE: If the "/R=nn" option is used, PROCAD will re-number all instances regardless of the prefix type starting with number nn. The option "/R=fname" is especially useful when multiple schematic sheets are being generated for use in layout.

REAL TIME FORWARD & BACKWARD ANNOTATION

The **[File]Annotate(pcb<->sch)...** menu item (**PACK/Annotate** command) may be used for performing real time forward and backward annotation of instance names and values changes. When you load your project (a single PCB database and one or more schematic databases), you may use the **[File]Pack/Renumber** menu item to renumber instance names in either or both PCB and Schematic databases, and then perform forward and/or backward annotation of the results using the **[File]Annotate(pcb<->sch)...** menu command. You may also use this command to Forward annotate instance values from the Schematic to the PCB database. Note: When forward annotating instance values, the values field of the corresponding PCB footprint (decals) must be enabled to see the results displayed. The annotation changes are performed in real time. Note: This feature requires the MDI (multiple document interface) option of ProCAD for windows.



Note: The first 3 items in the dialog box above are equivalent to **CHANGE INSTANCE**, **CHANGE VALUE** and **CHANGE SIGNAL** commands respectively.

RE-INITIALIZING PCB<-> SCHEMATIC LINK VALUES

The Annotation command requires that ProCAD understand which schematic device is linked to which PCB footprint. If you started your design with the schematic capture program in ProCAD, the Rats Nest placement command establishes this link based on the schematic

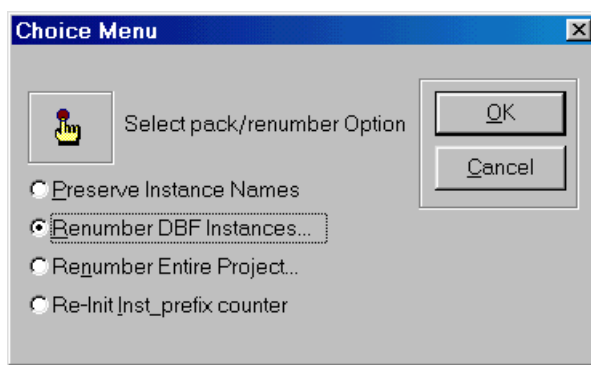
Reference Guide...

reference designators. This is usually the starting point for the link ("was") list. If you re-initialize the schematic link values (item #4 in the above dialog box), you should delete all packages in the pcb and re-issue the **RATS/P** command to establish the new ("was") links.

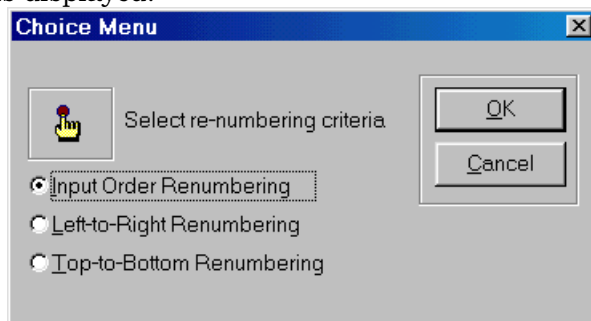
Note: If your PCB and Schematic database reference designators are numbered correctly (by manual means or whatever), you may re-initialize link values for both the PCB and Schematic databases to establish a starting "was" link list.

RENUMBERING PCB INSTANCES

You may the PACK/A ([**File**]Pack/**Renumber...** menu item) to invoke the Renumbering dialog box shown below:



When you select the second option above "**Renumber DBF Instances...**" in PCB design mode, the following dialog box is displayed:



From the above dialog box, you may choose to renumber instances in 1). Input order, which means the order the cells were placed in the database, 2). Left-to-Right or 3) Top-to-bottom. Note: You may use the **[File]Annotate(pcb<->sch)...** command to back annotate the renumbering results back to your schematic(s).

INSTANCE INDEXES RE-INITIALIZATION

You may re-initialize the subsequent starting index counters for all prefixes used in the database if this option is selected from the **[File]Pack/Renumber** dialog box. ProCAD will prompt for prefixes and starting indexes.

PAN SCREEN

PURPOSE: To allow the shifting, on a temporary basis, of selected coordinates to the center of the viewing screen.

SYNTAX: **PAN**<RETURN>

EXPLANATION:

This command allows you to move any specified point on the display screen to the center of the display. This change is temporary in nature, the REDRAW command will return the display to the "SCREEN" origin. See the SET SCREEN and SELECT SCREEN commands for further information.

Note: This command can also be issued while in the middle of various commands eg: during input/wire, Get_cell, etc commands

PASTE

PURPOSE: Allows you to paste entities from temp buffer to current database.

This command is only available in PROCAD for Windows MDI version.

SYNTAX : **PASTE** <RETURN>

EXPLANATION:

Use this command to paste entities from the temp buffer into the current database. The temp buffer must be created by the "CUT" command. This buffer is actually the [CELLIB]PROCAD\$\$ cell. This command is equivalent to the combination of "GETCELL" and "EXPLODE" commands, with the implied cell [CELLIB]PROCAD\$\$.

This command does not "paste" entities from the windows clipboard. PROCAD is a CAD/CAE program, (Drawings with intelligence), as such the customary windows "PASTE" command is not supported. For clipboard operation, you must use the "CLIPBOARD" command.

NOTE: Since this command is a special case of the "EXPLODE" command, all limits and restriction placed on PROCAD CELLS applies. You may use the EDIT CELL command to modify the data in [CELLIB]PROCAD\$\$ before issuing the PASTE command. See also the CUT command, which is the companion to this command.

PLOT DATABASE

PURPOSE: To generate the plot control file necessary for PROPLOT to create a plot file to send to the plotter.

SYNTAX: **PLO**t<RETURN>

Note: For best results, before issuing this command, you should zoom out far enough so that you can see the paper outline in step 9 below. (Scale Screen -5 is a good starting point).

EXPLANATION:

The system will prompt for various parameters and the area to be plotted. The prompts are as follows:



1. ControlFileName[dbf_name.CTR]?

This prompt allows you to give a unique name to the plot control file. Very useful if you are doing multiple plots of the same database, ie. PCB's, etc.

NOTE: If the name that you pick has been used previously, the system will prompt you to delete the previous file. A NO response will allow you to enter another name for the plot control file.

2. Plot Scale [1.0]?

Allows you to determine the scale at which the plot will be done. Plotting at a scale of less than one (1), requires that the scale be entered as 0.x, ie half scale would be 0.5.

3. Paper Size (A,B,C,D,E,User_defined) [A]?

Allows you to define the size of the plot medium. The following sizes are used for plot positioning information:

"A" 8.5 x 11 inches

"B" 11 x 17 inches

"C" 17 x 22 inches

"D" 22 x 34 inches

"E" 34 x 44 inches

"U" User defined dimensions (PROCAD will prompt for dimensions)

4. Database Units(mils,in,cm,mm) [mils]?

This prompt allows you to change the database units in which the plot will be done.

5. Paper Orientation [Landscape]?

This prompt allows you to change the orientation of the plotted material with respect to the plotting paper. PROPLOT assumes that the longest side of the plotting paper is horizontal to the display. A Portrait response would rotate the long axis to vertical.

6. Plot Layer(s) [All_Visible]?

This prompt allows you to specify the layers that you want to plot. They may be specified in any of the following ways:

1,2,3,4,5,6,7,.....

1-3 7-10

1 2 3 6-8 11

Default All currently visible (viewing) layers

7. Pen Thickness [15]?

This prompt allows you to specify the width of the plotting pen that you are going to use. This value may also be changed during the execution of PROPLOT. The value is in database units.

8. Maximum Nesting Level (1-15) [15]?

This prompt allows you to specify the maximum level of nesting that you will plot from the database. The level shown in the []'s is the present nesting level of the database. If you want to plot what is displayed, then you would normally accept the default setting.

9. Position plot paper template

Please see note at the beginning of this section.

At this prompt the cursor will be replaced by a scaled red rectangle, representing the plottable area of your specified sheet size and orientation. Simply move the rectangle as you would your cursor so that the area that you want plotted is properly positioned. When the rectangle is in the proper position, press <ESC> to leave it in that position. At this point your cursor will return to normal.

10. Enter plot window(lower left coord.):

At this prompt move your cursor to the lower left position at which a window generated from that coordinate set will encompass the area to be plotted. Press <ENTER> to set the coordinates. At this point, if the cursor is moved, a window will appear with the upper right hand corner at the present cursor position.

11.Enter plot window(upper right coord.)

At this point there will be a window dynamically displayed on the screen that originates from the previously entered coordinates and terminates with the current cursor position. Move the cursor to a position such that the area to be plotted is completely within the dynamic window. When that has been done, press <ENTER> to enter the coordinates.

At this point a message will inform you that a Plot Control File has been written. If you want to plot the file, issue the command "EXIT" to leave the PROCAD program. When you have returned to DOS/WINDOWS, execute the PROPLOT program. You may also execute the PROPLOT program from within PROCAD for window by clicking on its button in the Tool bar. Refer to the section on PROPLOT for further information.

PROJECT

PURPOSE: To load or create a project file. This command requires the Multiple Document Interface (MDI) option of ProCAD for Windows.

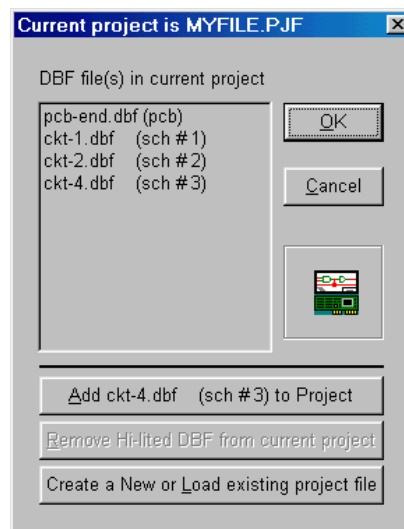
SYNTAX: **PRO**ject<CR>

SYNTAX: **PRO**ject/L<CR>

EXPLANATION:

This command allows for creating or loading of project file. A user typically organizes a design project in a project file. This file would contain the name of the databases that make up the project. Under normal use, a project would be made up of a single PCB database file and all the schematic database files (up to 24) that make up the PCB design.

When the project command is issued without the "/L" qualifier, ProCAD automatically goes into project management. The Project management dialog box allows for creating a project file, adding the current database to a project file, or removing database(s) from the project file etc.



The second syntax (with the **/L** qualifier) opens a menu of project files that you have previously created. You may select a project from the list, and ProCAD will automatically load all the database files comprising the project. Note: You must use the first syntax to create a project file before you can load it.

Note: Opening a database file (new or old) does not automatically add it to the project file. You must use the Project manager **"Add Current DBF"** button to add the currently active database to the project file. Similarly, closing a database file does not remove it from the project file. You must use the **"Remove Database"** button of the project managers dialog box to remove the hi-lighted database from the project file.

Note: Project files are ASCII format files with a ".PJF" extension stored in the current ProCAD working directory (typically \WPROCAD). Each line in the project file contains ProCAD database file to load, its type (Schematic or PCB) and the sheet number if it is a schematic database file. You may edit this file with a regular word processor in non-document mode, however, we strongly recommend you use the ProCAD Project manager command to create and maintain this file.

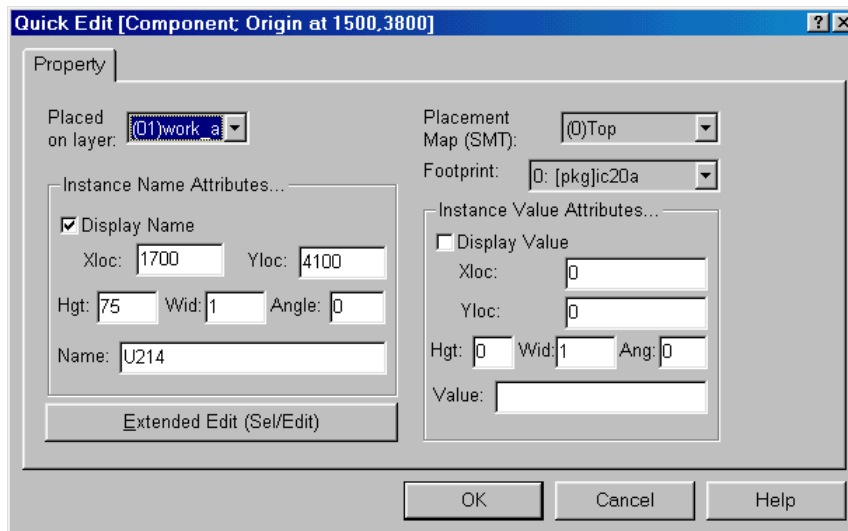
QUICK EDIT

PURPOSE: To allow for quick edit of some parameters of an entity in a dialog box.

SYNTAX: **QEdit** <CR>

EXPLANATION:

This feature is invoked either from the **[Other...]** menu, entered from the command line using the above command syntax or double clicking 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.



Example of Quick Edit Dialog box for Component editing

Note: For faster operation, you should invoke this command by double clicking on the desired entity. This was the intention of this command. See also the note on "Mouse button assignments" section of the User's manuals for further details.

QUIT

PURPOSE: To abandon the current activity without updating any files. To exit hierarchy viewing.

SYNTAX: **QUIT**<RETURN>

EXPLANATION:

If this command is issued from the EDIT CELL mode, a YES response will return you to the MAIN Database mode. The cell that was being edited will not be updated. If the command is issued from the MAIN Database mode, a YES response will cause you to exit from PROCAD and return to the operating system.(DOS Version).

The Windows version of PROCAD will close the current window (database), but will not return control to Windows Program Manager. You must use the EXIT command if you want to leave PROCAD and return to the Program Manager. The **[File]Close** menu item in PROCAD for windows actually executes this command.

The QUIT command will exit heirarchy viewing mode without asking for confirmation. In all cases the system will prompt for confirmation of the abort.

RATS-NEST

PURPOSE: To place cell instances and connections in .PKG and .CON files generated by PROCAP in a database.

SYNTAX: **RA**ts_nest={filename} <RETURN>

EXPLANATION:

Use the **RA** command of PROCAP to generate rats nest data files from your finished schematics. These files are used in conjunction with this command to generate the corresponding physical layout database. If you omit the file name extension above, PROCAD assumes a default extension of ".PKG" which will cause the placement of corresponding layout foot_print cells. You must enter the ".CON" extension to the file to cause the automatic RATS_NEST connections to be placed between the foot_print cells. If the filename is omitted from the above command line, PROCAD prompts (through a dialog box) for the file type to load; Connection file, Package file, FutureNet/OrCAD netlist or just to optimized already placed rat's nest connection wires. This form of the command offers the most flexibility.

To use the command correctly, the user must first design their schematics with the appropriate keywords attached to all the cells used in the schematic as follows:

RAT\$PKGn [library]cell_name

This keyword instructs PROCAP to use the named cell from the named library as the footprint cell to be used for physical layout. If the library qualifier "[]" is omitted, PROCAD will attempt to use the currently active library when the RATS NEST command is issued.

eg: RAT\$PKG0 [PCBLIB]IC14PIN

ALTERNATE FOOTPRINT ASSIGNMENT

ProCAD for windows supports alternate footprint (decals) reference for placed schematic symbols. This feature is accessible via the Quick Edit feature (double clicking on a component). You must assign multiple RAT\$PKGn tokens to your schematic symbol before you use the quick edit feature to access various footprint choices for the symbol. The format for the RAT\$PKGn token is as follows:

RAT\$PKGn [LIBNAME]CELLNAME

Where n=0,1,2 or 3. If you omit "n", ProCAD assumes the default package assignment (n=0). From above, you may assign up to 4 various footprint references for the symbol.

RAT\$VCCn pin_number (n=1,2,3 or 4)

RAT\$GNDn pin_number (n=1 or 2)

These keywords instruct PROCAD to generate global power & ground connections to the pin_numbers specified. Note: Signal name assigned to power or ground traces will be VCC1, VCC2GND1 or GND2.

*eg: RAT\$VCC1 14
RAT\$GND1 7*

RAT\$TCNT n

This keyword instructs PROCAD and PROCAP to assign n devices to the same package. This keyword is only recognized if the cell is defined as a template. (see EDIT CELL command)

eg: RAT\$TCNT 4

**RAT\$DEVx p1 p2 p3
+ p4 p5 p6 ...**

This keyword instructs PROCAD and PROCAP to use the specified ordered pin numbers for packagable device X. X=A,B,C Continuation lines are accepted if the first character on the following line is a plus "+", followed by 1 or more blanks.

Note: You must specify enough RAT\$DEVx keywords to match the number of devices per package as defined in the RAT\$TCNT keyword above.

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

The order in which the pin numbers appear must match the order the pin reference points are defined during EDIT CELL. The above keywords may appear in any order. Template cells must not have more than 20 pins (ie 20 pins per each device in the package).

An example attributes for a 74LS02 cell with bill-of- materials information will be:

```

BOM$DESCN QUAD 2-INPUT NOR
BOM$MFG_PN SN74LS02J
RAT$PKG [PCBLIB]IC14PIN
RAT$TCNT 4
RAT$DEVA 2 3 1
RAT$DEVB 5 6 4
RAT$DEVC 8 9 10
RAT$DEVD 11 12 13

RAT$VCC1 14
RAT$GND1 7

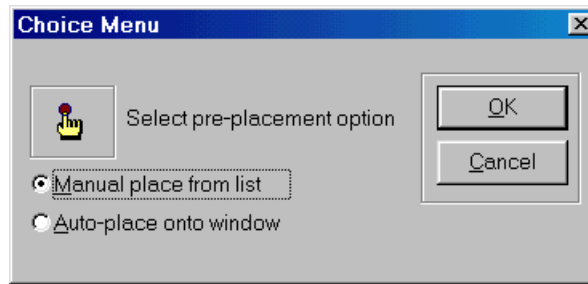
```

After the RATS NEST files have been generated in PROCAP, Open a new corresponding layout database in PROCAD and issue the following command:

RAT\$nest filename.pkg

(or access the **[Place]Rats Nest...** menu item, then select "**Load Package (.PKG) file**" from the choice menu, and load the desired package file).

ProCAD will then display the following dialog:



If you select "**Manual place from list**" option, PROCAD will prompt for the placement reference for all the cells referenced in the corresponding schematic. You can abort this command by typing CNTL-C in responds to any of the prompts. PROCAD will place the cells with the same instances name used in the corresponding schematics. Note that the foot_print instance name prefix will not be used unless the database is packed with the /R option. Also, if rubberband is ON before this command is issued, PROCAD will automatically grab the appropriate footprint cells from the library, and prompt for its placement in real time.

When placing packages from the ".PKG" file, PROCAD opens a window of required parts. You may select a device from this window while scrolling up or down the list, or you may enter the part source library and name (full device name spec). After a part is placed, it is removed from the window, this way you may not accidentally place it twice. You are free to place parts in any order desired. Note: for OrCAD/Future_Net files, PROCAD will prompt for packages to place - No window of available parts is displayed.

If you select "**Auto-place onto window**" option, PROCAD will prompt for a window (preferably off the board outline), where it will pre-place all packages sorted by size. You may then grab and place components from this window onto the desired position on the board.

Next issue the rats nest connection command as follows:

RATsnest filename.con

(or access the **[Place]Rats Nest...** menu item, then select "**Load Connection (.CON) file**" from the choice menu, and load the desired connection file).

PROCAD will automatically connect the cells together with the appropriate wire and signal names used in the corresponding schematics. If an instance is not placed in the database, and a connection to it is requested, PROCAD will issue an error message. Also, if a package of less pins than the corresponding component in the schematic is used, PROCAD will issue an error message and place the connecting wire in the instance name reference point. If this occurs, you should go back to your schematics database, correct the error and re-run PROCAP and PROCAD. You may also edit the .PKG file, make corrections and re-run the Rats nest command in PROCAD. Note: You may abort this command at any time by issuing two ESC's

RAT'S NEST TRACE OPTIMIZATION

After the .CON file is processed, PROCAD prompts for an optimization request. If optimization is requested, PROCAD will optimize the rat's nest traces for the shortest paths. You may also request for trace optimization of already placed rats nest wire if you issue the **RA**t's nest command without a filename argument (or issued from the **Place** pull down menu).

Note: ProCAD will automatically optimize rats nest if Rats Enforced flag was set ([Other...]/Toggles... dialog box).

USING FUTURE NET COMPATIBLE NETLIST

PROCAD Rats nest command will accept Future Net compatible NETLIST file. It may be necessary to provide a packaging cross reference file, since the Future Net Netlist format does not support this feature. The format of the cross reference file is described below.

USING OTHER COMPATIBLE NETLIST

PROCAD Rats nest command will accept Allegro, Calay, etc compatible NETLIST file formats. It may be necessary to provide a packaging cross reference file, since these Netlist formats may not support this feature. The format of the cross reference file is described below.

DATA FORMAT FOR PACKAGE XREF FILE

The following is the format of the cross reference file utilized by the Rats Nest command of PROCAD when processing a Future Net compatible Netlist file:

The cross reference file is an ASCII text file that contains the translation between component name and package name. This file can be created using an text editor or word processor in non document mode.

The recognized option characters in the cross reference file are :

"%" character signifies a comment line. All data on that line will be ignored.

":PACK" The ':' character is followed by a package name. The package name PACK is used for all component names between this and next package definition.

"=COMP" The '=' character is followed by a component name. The component name COMP is connected to the last given package name.

"\$PKGLIB" The '\$' character is followed by a package library name. The package library PKGLIB will be used by PROCAD to load the packages from. If no package library name is specified, the default active library will be used by PROCAD when processing the Rats Nest command.

Example of a valid cross reference file is as follows:

%This is a line with comment.

\$PKG

:IC14A

=7432

=7474

=74LS04

:IC16A

=74LS163

:IC24A

=82S181

You should use the REROUTE command to re-route the traces generated above.

RECEIVE AUTO ROUTED JOB

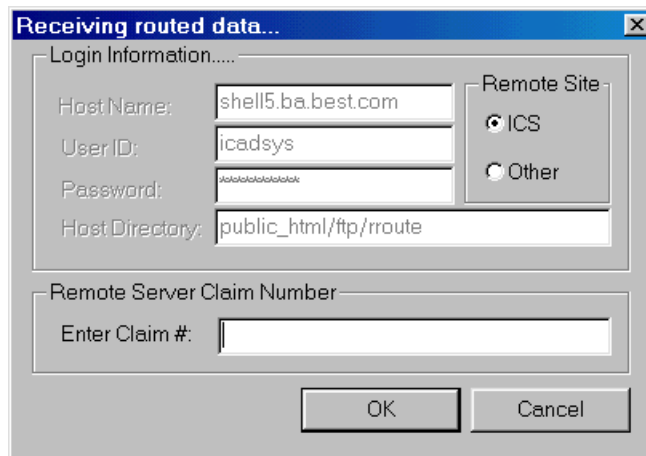
PURPOSE: To retrieve your autorouted job from our Remote internet site.

SYNTAX: SREceive<CR>

EXPLANATION:

This command may also be invoked from the **[Route][Internet AutoRoute]Receive Routed Job** menu item.

When issued, this command allows for the retrieval of routed result from a previously submitted Auto Router Job. You should open your original database (the one submitted for routing) before invoking this command.

A screenshot of a Windows-style dialog box titled "Receiving routed data...". The dialog is divided into two main sections. The top section, "Login Information.....", contains four text input fields: "Host Name:" with "shell5.ba.best.com", "User ID:" with "icadsys", "Password:" with a masked field of asterisks, and "Host Directory:" with "public_html/ftp/route". To the right of these fields is a "Remote Site" section with two radio buttons: "ICS" (which is selected) and "Other". The bottom section, "Remote Server Claim Number", contains a single text input field labeled "Enter Claim #:". At the bottom right of the dialog are "OK" and "Cancel" buttons.

Enter your claim # and click OK.

ProCAD will verify your claim # and retrieve the appropriate routed results. All your routed rats nest will be replaced with routed traces and un-routed rats nest will be left un-changed for you to route manually. A log file will also be retrieved for your records.

REDRAW

PURPOSE: To return the display to it's default settings, ie. scale, position, etc.

SYNTAX: **RED**raw<RETURN>

EXPLANATION:

This command will undo the effect of all PAN, ZOOM and SET VIEW commands issued. The results of SCREEN SCALE, SHIFT, SELECT SCREEN commands are not affected by this command.

REFRESH

PURPOSE: To redraw the display screen in place to remove any unerased bits or exclusive-or bits.

SYNTAX: **REF**resh<RETURN>

EXPLANATION:

After editing using the SELECT environment, there may be some white bits of "noise" or erased section on the display. This command will redraw the display screen without resetting the view.

REPLACE

PURPOSE: To replace a text string, a cell or group of cells that have already been placed in a drawing, while maintaining the same placement parameters.

SYNTAX: **REPl**ace <RETURN>

EXPLANATION:

The selected text string/single or group of cells will be replaced (swapped) with the new text string/cell in the drawing. When used for cell replacement, care must be exercised to make sure that Template cells are only replaced with other template cells of the same number of I/O pins, and non-template cell must have the same number of I/O pins. This command can be issued from the main database command mode, in which case, PROCAD will prompt for the location of the cell or text string to be replaced.

When issued from the select environment, a group of cells can be replaced with a single command, eg replacing all PAD030 vias with bigger PAD050 vias in a section of a board. A **group** of Text entities **cannot** be replaced with a single replace command !.

Note: PROCAD will issue an error message if an attempt is made to replace a template cell with a non-template cell, and/or a cell with different number of I/O connection pins. For Pin Swapping, you should use the CHANGE INSTANCE command. When replacing a single cell, PROCAD will also prompt for a new value to assign to the new cell.

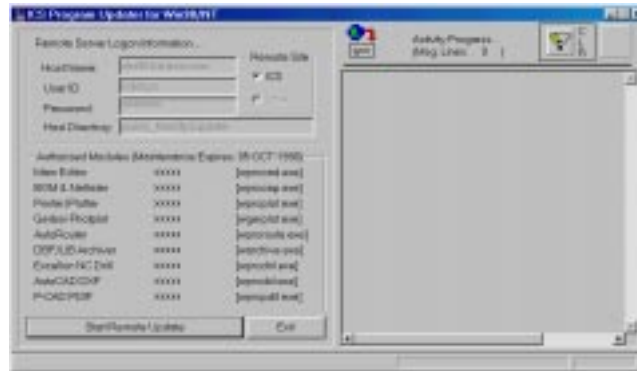
REMOTE SOFTWARE UPDATE

PURPOSE: To check and retrieve from our FTP site the latest versions of ProCAD and Associated Utilities.

SYNTAX: **RUP**date <CR>

EXPLANATION:

This command will invoke the Remote Updater Utility. Use this utility to update your ProCAD and Associated utilities to the latest versions on our FTP site.



Click the Start Remote Update button to start updating your installed ProCAD and associated utilities. You must close all related ProCAD utility programs before you start the Updater. The remote Updater will only update your programs if your current revision is older than the one on our FTP site. Your old executable will be renamed to XXXXXX.OLD.

Since this command will only update your software when necessary, you should invoke it periodically to assure that you are always using the most current revisions of ProCAD and Associated utilities. The Updater ONLY updates installed Programs, NOT your CD or floppies.

NOTE: YOU MUST HAVE A VALID AND CURRENT MAINTENANCE TO USE THIS COMMAND. THE REMOTE UPDATER WILL FAIL IF YOUR MAINTENANCE HAS EXPIRED OR YOU DON'T HAVE A REGISTERED PROCAD. CONTACT ICS OR YOUR DEALER TO PURCHASE A MAINTENANCE CONTRACT.

REROUTE

PURPOSE: To reroute a wire or line in the database.

SYNTAX: **RER**oute <RETURN>

EXPLANATION:

After a wire or line has been placed in the drawing, the routing of that wire/line can be easily modified in real time with this command. This command is equivalent to the combination of rubberband on, add a vertex and then move the newly added vertex. You can abort this command by typing CNTL_C. Note: ESC will terminate the command leaving the newly added vertex to the wire/line. You can also issue this command from the select environment for continuous re-routing of a wire/line. This command is especially useful for re-routing the traces generated by the RATS NEST command.

RESET DATABASE

PURPOSE: Allow you to quit the current database, but remain in PROCAD.

SYNTAX: **RESET**<RETURN>

EXPLANATION:

This command allows you to quit the current database, without saving any changes and remaining in PROCAD. See the QUIT command for quitting the current database without saving any changes and exiting PROCAD.

Note: In PROCAD for Windows, this command will close all active windows.

RESOLVE

PURPOSE: To resolve any connectivity errors within the database.

SYNTAX: **RESOLVE**<RETURN>

EXPLANATION:

Insures that there are no errors in the connectivity of the wires, within the database. Such errors could exist when a vertex was moved or a wire is moved for connection purposes. PROCAD will hi-lite the problem wire. PROCAD will also check for cells whose inputs are not connected to any wire or connected to wires whose signal name was program defined. If such errors are found, PROCAD will flag such inputs with markers on layer 49.

NOTE: This command should be executed periodically to check the integrity of the schematic being entered. If any errors are flagged, make necessary corrections and execute the resolve command again. In PCB mode, the resolve command will only check connectivity on layer by layer basis. Via's that make connection between layers are only recognized in PROCAP.

ROTATE

PURPOSE: To change the position of a selected entity(s) about a specified point, in 1 (one) degree increments.

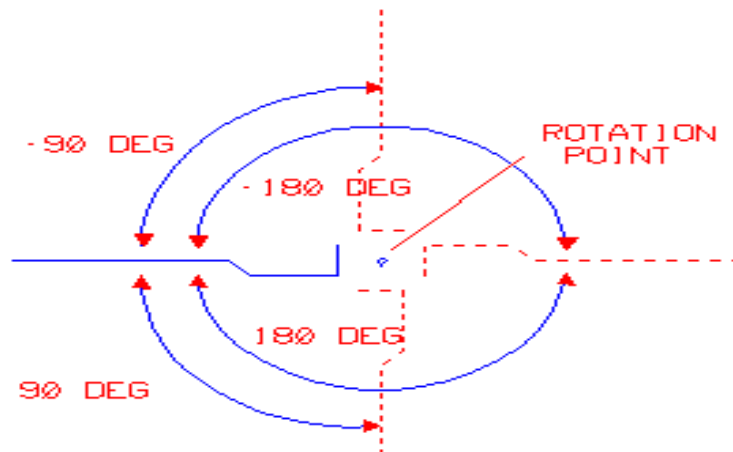
SYNTAX : **ROT**ate<RETURN>

EXPLANATION:

After issuing the command the system will prompt for a reference point about which to rotate the entity(s) and the rotation angle.

NOTE: This command can be used in the SELECT environment. Press <ESC> to exit the SELECT environment. It may also be issued while entities are being moved or copied.

EXAMPLE:



SAVE DATABASE

PURPOSE: To provide a means of periodically saving all editing changes made to the database.

SYNTAX: **SAVe** <RETURN>
 SAVe/ALL <RETURN>
 SAVe/AS <RETURN>

EXPLANATION:

This command has 2 possible qualifiers. When issued without a qualifier, the current database is saved under its current name. No backup file is generated.

When issued with the "**ALL**" qualifier, all currently opened database files in the MDI (Multiple Document Interface) version of PROCAD will be saved under their original names. Again, this command will not generate any backup files. The "**ALL**" qualifier is ignored by the SDI (Single Document Interface) and DOS versions of PROCAD.

Using the "**AS**" qualifier allows for saving the current database under a user supplied name. This is very useful for generating backups. The "**SAVE/AS**" command is identical to issuing the **SET FILE=<filename>** command followed by a non qualified **SAVE** command.

The un-qualified SAVE command is very useful for saving the current database under it's present name, then using SET FILE=<filename> to change the current database name to make some "what if" changes then comparing the two databases.

Tips: The SAVE AS command offers the option to convert the database to pre version 3.0 format (ProCAD for DOS V8.XX, ProCAD for Windows V1.XX and V2.XX formats). When this option is chosen, all text widths revert to one database unit. You should exercise this option if you must edit, view or plot this database in older versions of ProCAD. In addition to converting the database, you should also edit all cells used in the drawing where text and reference designators have widths greater than one database unit and save then in ProCAD for windows V2.XX cell format.

NOTE: The SAVE (qualified or un-qualified) command has the same effect as the EXIT command, when in the EDIT CELL enviroment. See also the SET AUTOSAVE command.

SCALE ENTITY

PURPOSE: To scale a selected entity(s), both positively and negatively.

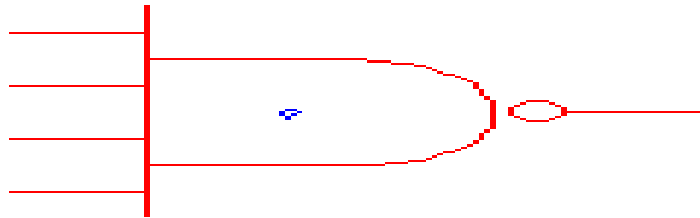
SYNTAX : **SCALE** Entity<RETURN>

EXPLANATION:

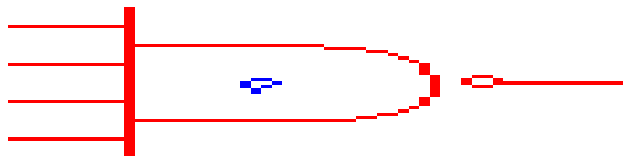
After the entity(s) is selected, the system will prompt for a reference point to scale from and the scale factor.

NOTE: Cells cannot be scaled below their 1X size. This command can also be used in the SELECT environment. Press <ESC> to exit the SELECT environment.

EXAMPLES:



"Scale Entity -3



SCALE SCREEN

PURPOSE: To allow the setting of the display scale.

SYNTAX: **SCale SCreen**={nn}<RETURN>
 SCale SCreen=OFF<RETURN>
 SCale SCreen=ON<RETURN>

EXPLANATION:

This command allows the setting of the area of the display. For example if the display has a resolution of 640x350 pixels, database units equal "mils", then the display area would be 640x350 mils (6.4" x 3.5"). By scaling the screen to a -2, the display area would increased by a factor of 4 (12.8" x 7.0") although the resolution is still 640x350 pixels. Conversely by scaling the screen to a +2, the display area would be decreased by a factor of 4 (3.2" x 1.75") again the resolution would remain at 640x350 pixels.

NOTE: The command SCALE SCREEN=ON set the screen scale to the previous setting.

You should issue this command before issuing the PLOT or PLOT/G command to generate a plotter control file. See the PLOT command.

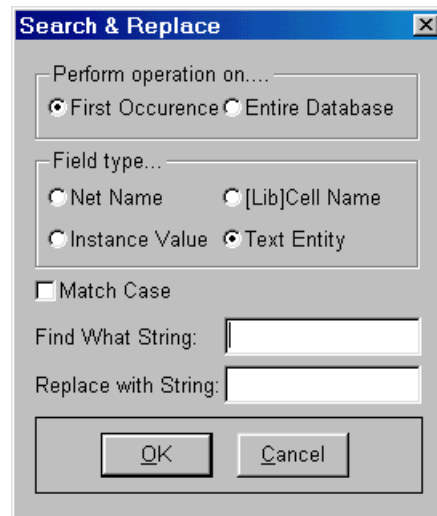
SEARCH

PURPOSE: To Search and Replace Text Strings, Cell Name and Value fields and Net Names globally in a database.

SYNTAX: **SEArch**<CR>

EXPLANATION:

The **[Edit]search & Replace** menu item or above command entry from the keyboard allows the user the ability to globally search and replace text strings (with Case Sensitive option), Full cell name spec (Cell Name and Library qualifier), Cell values, and Net (Signal) names used in design globally. When the command is issued, ProCAD displays the dialog box for entry selection:



You should type in the Find What String (String to search for), and the Replace with String (to be replaced with). Then select one of the four fields and command scope. Press OK to execute the command.

*Note: When replacing Text Entity or Cell Names, this command acts like the **[Edit]Replace Cell/Text** menu item in global mode. When the Net Name field is selected, ProCAD performs the replace without checking for connectivity errors. You should perform the **[Check]Resolve connectivity** command to check for any errors that might have been generated. The "Match Case" option is ignored when replacing Net and Cell Names. For Text entity, the "Find What" and "Replace with" strings must be the whole text entity, NOT a word, group of words or characters in a text entity string. This is also true for Net Name, Cell Name and Instance Value fields*

SELECT ENTITY

PURPOSE: This command allows you to select an entity by location.

SYNTAX 1: **SE**lect **E**ntity {nn}<RETURN>

SYNTAX 2: **SE**lect **E**ntity/**type** {nn}<RETURN>

EXPLANATION:

nn == layer number 1 to 99, only entities on indicated layer can be selected.
Default==ALL.

If the **/type** qualifier is used, (where **type**=**L**ines, **W**ires, **C**ells, **C**ircles, **R**ect, **A**rcs or **ALL**), syntax #2 above, ProCAD will in addition to only selecting items in **nn** layer, will only select the entity type specified by the qualifier. This is equivalent to the temporary issuing of SELECT KINDS (FILTER) command with the appropriate filter set before selecting entity.

*Note: The type qualifier if supplied must be preceded by a slash "/" If you wish to include layer(s) as additional selection filter, it must appear after any type qualifier, and **must not** be preceded by a slash.*

Example:

```
SEL EN /L 1-4 20-25
SEL EN/CI ALL
```

*Note: When an entity type qualifier is supplied, the current SELECT MASK (FILTER) is ignored. If no entity type qualifier is supplied, the current SELECT MASK (FILTER) setting is used. **SELECT ENTITY /W** and **SELECT ENTITY /CE** commands are available on the tool ribbon.*

Selected entity will be highlighted and further editing can be performed. Press <ESC> to exit the Select environment.

EXAMPLES:

```
SEL E 1 3 6-10<RETURN>
```

The above command would select the closest entity on layers 1,3,6 thru 10 only.

SEL E 4<RETURN>

The above command would select the closest entity on layer 4 only.

SEL E/Wire 20<RETURN>

The above command would select the closest wire (trace) on layer 20 only.

SEL E<RETURN>

The above command would select the closest entity on any layer.

NOTE: The SELECT KINDS command can be used for further defining the selection criteria. See also the note on "Mouse button assignments" section of the User's manual for quick selection with a mouse.

SELECT INSTANCE NAME

PURPOSE: To select an entity by its instance name for editing.

SYNTAX : **SEL**ect **IN**stance={instancename}<RETURN>

EXPLANATION:

PROCAD will search the database for the named entity, move the selected entity to the center of the display, (if the entity was off the display window) where further editing can be performed. Press <ESC> to exit the Select environment.

Note: This command only works on cells that have had an instance name assigned to it.

SELECT SCREEN

PURPOSE: To recall a specific location within the database to the screen.

SYNTAX: **SEL**ect **SC**reen=n<RETURN>

EXPLANATION:

n == Screen Number 1 thru 10

This command is used to recall a preset display screen reference origin, set by the SET SCREEN command. Screen #1 is always set with 0,0 coordinates in the lower left-hand corner of the display and cannot be changed. Screen #1 is always the default screen. Allows you to move around the database much faster than using the SHIFT or PAN SCREEN commands.

NOTE: See SET SCREEN command for more details

SELECT SIGNAL NAME

PURPOSE: To select a wire by its signal name for editing.

SYNTAX : **SE**lect **SIG**nal={<signalname>} <CR>
SElect **SIG**nal={<signalname>},<Layer><CR>

EXPLANATION:

PROCAD will search the display area for the named wire, highlight it, so further editing can be performed. In the PCB mode, only wires on the current layer or specified layer (second syntax above) will be selected and highlighted. Press <ESC> to exit the select environment.

NOTE: This command works on wire/signal node tree. Portions of the signal may be off the current display area. The signal name used in the command can either be PROCAD or user assigned.

SELECT WINDOW

PURPOSE: To allow the selection of a group of entities within a specified area(window) for further editing.

SYNTAX 1: **SE**lect **W**indow {nn}<RETURN>

SYNTAX 2: **SE**lect **W**indow/**type** {nn}<RETURN>

EXPLANATION:

nn == layer number 1 to 99, only entities on indicated layer can be selected. Default == ALL.

If the **/type** qualifier is used, (where **type**=**L**ines, **W**ires, **C**ells, **C**ircles, **R**ect, **A**rcs or **ALL**), syntax #2 above, ProCAD will in addition to only selecting items in **nn** layer, will only select the entity type specified by the qualifier. This is equivalent to the temporary issuing of SELECT KINDS (FILTER) command with the appropriate filter set before selecting a window of entities.

Note: The type qualifier if supplied must be preceded by a slash "/" If you wish to include layer(s) as additional selection filter, it must appear after any type qualifier, and **must not** be preceded by a slash.

Example:

```
SEL W /L 1-4 20-25
SEL W/CI ALL
```

Note: When an entity type qualifier is supplied, the current SELECT MASK (FILTER) is ignored. If no entity type qualifier is supplied, the current SELECT MASK (FILTER) setting is used.

Selected entities will be highlighted and further editing can be performed. Press <ESC> to exit the Select environment.

EXAMPLES:

SEL W 1 3 6-10<RETURN>

The above command would select only those entities on layers 1,3,6 thru 10 only.

SEL W 4<RETURN>

The above command would select only those entities on layer 4.

SEL W/Wire 23<RETURN>

The above command would select only those wires (traces) on layer 23.

SEL W<RETURN>

The above command would select all entities on any layer.

NOTE: The SELECT KINDS command can be used for further defining the selection criteria. See also the note on "Context Sensitive Commands and Mouse buttons assignments" in the User's & Utilities manual.

SET CELL

PURPOSE: This command allows you to assign a new name to the current cell being edited.

SYNTAX: **SET CELL**{=<cell_name>}<RETURN>

EXPLANATION:

This command is particularly useful for creating cells from already existing cells, ie: cells that have the same graphics, but different names (7400, 74LS00, 74H00, 74L00, 74AS00, 74ALS00, 74HC00, 74HCT00).

EXAMPLE: Current cell being edited is: 74LS00

SET CELL=74ALS00<RETURN>

Current cellname is now: 74ALS00

NOTE: The original cell in the library is not affected. See also the SAVE/AS command.

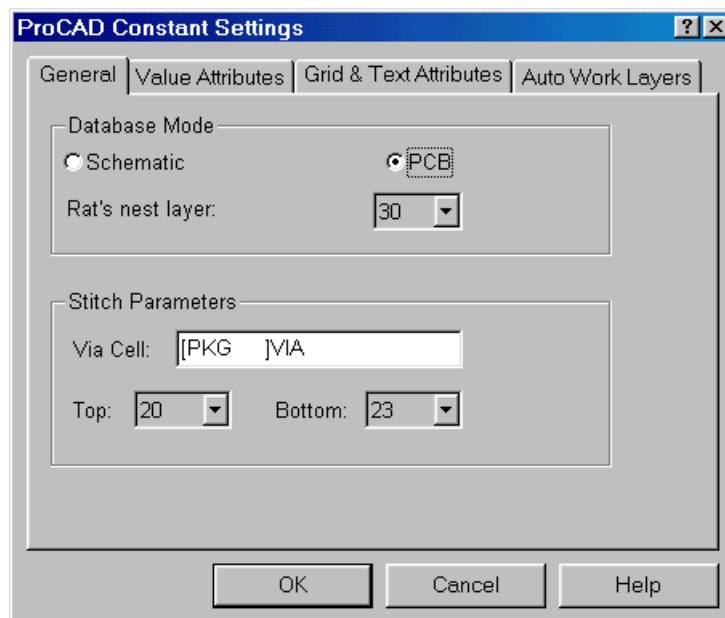
SET CONSTANT

PURPOSE: This command allows the user to configure a wide variety of system parameters which generally remain constant once they have been established for a specific design. These system parameters may also be executed from the command line.

SYNTAX: **SET CONST** <RETURN>

EXPLANATION:

The SET CONSTANT command opens a small window on the primary display which allows many different system parameters to be configured from one location. The following commands may be accessed from the SET CONSTANT window.



PARAMETER	EQUIVALENT COMMAND
Layer Attribute	SET VISIBLE & SET FILL (DOS)
Layer Color	SET COLORMASK (DOS)
Auto Work Layers	SET AUTOLAYERS
Extent Limit	SET EXTENT LIMIT
Font Selection	(None)
Database Mode	SET MODE
Schematic Sheet #	SET MODE SCH
Set Pin # Display Size	(None)
Snap Connect limit	(None)
Nesting Level	SET NESTING LEVEL
Auto Save Constant	SET AUTO SAVE CONST
Fill Area Clearance	SET CLEARANCE
MSG Display Time	SET WAIT TIME
Trace width	SET WIDTH
Ghost Grid	SET GHOST GRID
Grid Increments	SET GRID_SPACINGS
Label Attribute	SET LABEL ATTRIBUTE

Note: The Constants dialog box may include fields that do not have a corresponding command for setting them. For these fields, you must enter the desired values in the appropriate fields in the dialog window.

Note: The SET CONSTANT command cannot be issued from within a MACRO command file. You must execute the desired command from the list below to set the desired constant within a MACRO command file.

SET AUTOLAYERS

PURPOSE: To setup layers where different entity types will be placed by default.

SYNTAX: **SET AUTOLayer** {<type> <layer>} <RETURN>

EXPLANATION:

type ==A	for Arcs
CE	for Cells
CI	for Circles
CO	for Connection Dots
D	for Donuts
L	for Lines
P	for Polygons
R	for Rectangles
T	for Text
W	for Wires

layer == 0,1,2....98

The entities will be placed on the appropriate layers during INPUT and GETCELL commands. You must setup your database mode before issuing this command, since the database mode (Schematic or PCB) determines which group of layers to use. The setting of this command does not override the layers specified for the STITCH or RATSNEST commands. Specify layer 0 for entities that should be placed on the current active layer (layer selected by the SET LAYER command).

SET AUTOSAVE

PURPOSE: To set the number of commands executed in the MAIN mode of PROCAD before an automatic database save is performed.

SYNTAX: **SET AUTOSave** {nn} <RETURN>

EXPLANATION:

This command will allow the user to set the number of commands that should be execute in the MAIN mode of PROCAD before an automatic save is performed. When an automatic save is performed, PROCAD will use the current file name in effect with a ".BAK" extension added for saving the current database. This command will also use the current data path statement in effect for databases references.

Example: SET AUTOSAVE 20
 SET AUTOSAVE 0

The first example above will cause PROCAD to save the drawing after every 20 commands issued from the MAIN mode. The second example will turn off the auto save feature.

Note: The auto save constant of less than or equal to zero will turn off the auto-save feature. Use the SHOW STATUS command to view the current auto-save constant. The auto-save process generates a backup file. You must use the DOS rename command to recover the backup file in case you desire a recovery from a program crash. Commands issued in the EDIT, SELECT and LIBRARY MANAGER modes are not counted as part of the auto-save constant. The auto-save constant will be reset whenever a SAVE command is issued.

SET CLEARANCE

PURPOSE: To set the clearance value for poured or plowed area fills for subsequent INPUT/G (Area Fill) command.

SYNTAX: **SET CL**earance **nn** <CR>

EXPLANATION:

This command allows for setting the clearance value between traces, pads, etc of the generated ground/area fill planes for all subsequent **INPUT/G** command. Note that this newly set clearance value only affects subsequent **INPUT/G** commands. You may change the clearance of already placed Area fill polygon/Ground plane with the **CHANGE WIDTH** command. Use the **SHOW SUMMARY** command to obtain the current value of the fill clearance of already placed area fill polygon. Note nn must be specified in database units (DBU).

Example: **SET CLE 25 <CR>**

This will set clearance value to 25 database units (DBU) for subsequent IN/G commands.

Note: The clearance value selected directly affects the size of polygons generated to fill areas between traces and pads. This command may also be used for selecting subsequent Area Fill polygon style (Solid, Cross Hatched, etc). See also the INPUT/G and CHANGE WIDTH commands.

SET COLORMASK

PURPOSE: This command sets the line type, color, and drawing type of each layer.

SYNTAX #1: SET COLormask<RETURN>

SYNTAX #2: **SET COL**ormask {nn} {mm}<RETURN>

nn == layer number (1 - 99)

mm== colormask number as determined below:

EXPLANATION:

You may issue this command using syntax 1 or 2 above. When syntax #1 is used, A pop-up window appears which allows for the selection of color and line types from the window.

You MUST use syntax #2 is you are issuing the command from within a macro command file. When using syntax #2, the Colormask number is determined by the color - line type/fill pattern scheme explained below. The number is the decimal equivalent of a binary number. The binary number is an eight bit word with the following makeup:

BITS 0 thru 3 - Color code

BITS 4 thru 6 - Line type/Fill Pattern

BIT 7 – Exclusive-OR (Transparent) Writing

BIT	7	6	5	4	3	2	1	0
VALUE	128	64	32	16	8	4	2	1
	EX	LINE/TYPE			COLOR CODE			
	OR	FILL PAT.						

COLOR CODES

COLOR	CODE	BITS 3 thru 0
BLACK	0	0 0 0 0
WHITE	1	0 0 0 1
RED	2	0 0 1 0
GREEN	3	0 0 1 1
BLUE	4	0 1 0 0
YELLOW	5	0 1 0 1
CYAN	6	0 1 1 0
MAGENTA	7	0 1 1 1
*	8	1 0 0 0
*	9	1 0 0 1
*	10	1 0 1 0
*	11	1 0 1 1
*	12	1 1 0 0
*	13	1 1 0 1
*	14	1 1 1 0
#DARK GREY	15	1 1 1 1






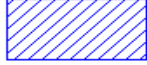



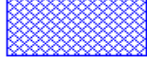


For Monochrome-

Black	0	0 0 0 0
Video	1	0 0 0 1
Video Blink	2	0 0 1 0
Bold Video	3	0 0 1 1

* - Color only available on the EGA, VGA Graphics Adaptor or color card capable of supporting 16 or more colors.

Line styles & fill pattern

LINE TYPE/FILL PATTERN CODES

LINE TYPE	FILL PATTERN	CODE	BITS 6-4
		0-31	0 0 0 or 0 0 1
		32	0 1 0
		48	0 1 1
		64	1 0 0
		80	1 0 1
		96	1 1 0

EXCLUSIVE-OR WRITING (TRANSPARENT)

Exclusive or writing means that when an entity line or wire overwrites another or itself, the display will be blank where the lines cross or overlap.

<u>WRITING</u>	<u>CODE</u>	<u>BIT 7</u>
Standard	0-127	0
Exclusive OR	128	1

EXAMPLE: To set layer 1 for a color of green, line type/fill pattern of Dash Dot/Fine X.

From the charts above:

The color RED is #2, BIT code (0010)
The line/fill code is #64 BIT code (100)
With standard writing BIT code (0)

BIT	7	6	5	4	3	2	1	0
VALUE	128	64	32	16	8	4	2	1
	EX	LINE / TYPE			COLOR CODE			
	OR	FILL	PAT.					
	0	1	0	0	0	0	1	0

$$66 = 0 + 64 + 0 + 0 + 0 + 0 + 2 + 0$$

Issue the command:

SET COLormask 1=66<RETURN>

Use the SHOW COLORMASK command to check the results.

NOTE: The Exclusive OR bit is always set for Layer 99 Colormask Code. Layer 99 is used for hi-lighting such as with the SELECT, SET VIEW, SHOW NODE etc. commands.

SET EXTENT-LIMIT

PURPOSE: To control the minimum drawing feature size of cells & text for display.

SYNTAX: **SET** Extent_limit{=nn}<RETURN>

EXPLANATION:

nn== minimum feature size in pixels.

This command only applies to the the display of cells & text entities. If the pixel size of the entity is equal to or less than the setting, only a rectangle representing the extent of the entity will be drawn on the display. The present setting of the extent can be viewed by issuing the SHOW STATUS command.

SET FILL LAYERS

PURPOSE: To allow closed polygons (Rectangles and Polygons), Lines and Wires with width > 1 unit, and Circles to be area filled on selected layers.

SYNTAX: **SET FILL**{=n}<RETURN>
 SET FILL=n1,n2,n3..n99<RETURN>
 SET FILL=n2-n10,n11...<RETURN>
 SET FILL=OFF<RETURN>

EXPLANATION:

n==layer numbers

To display the current layers which have fill enabled, issue the following:

SET FILL=?<RETURN> or
SHOW COLORMASK<RETURN>

The fill pattern is determined by the COLORMASK code. See SET COLORMASK command.

Entity filling is controlled by layer only, if moved to a none filled layer the entity will not be filled.

SET GHOST GRID

PURPOSE: This command sets the minimum movement step of the cursor, when it is not locked to grid.

SYNTAX: **SET GH**ost{=xx yy}<RETURN>

EXPLANATION:

xx ==X axis movement increments

yy ==Y axis movement increments

Since there are limits to the displaying of a small grid depending upon the display scale, this command is very useful in that it gives you a "ghost grid" with which you can precisely place entities or edit the database. The default setting is the Screen Display Factor.

NOTE: See the information at the beginning of this section on temporarily modifying the cursor movement increments.

SET GRID_SPACING

PURPOSE: This command allows you to set the spacing of the grid displayed on the screen and the display of the grid.

SYNTAX: **SET GRID**=<xxx,yyy><RETURN>
 SET GRID=ON<RETURN>
 SET GRID=OFF<RETURN>
 SET GRID=nnn<RETURN>
 SET GRID<RETURN>Toggles ON/OFF

EXPLANATION:

xxx,yyy ==number of database units per grid line in x and y direction.

nnn ==number of database units in both the x and y axis.

NOTE: See command SET SNAP for how to lock the cursor to the grid.

NOTE: When this command is issued from the "Toggles" dialog box, the state of the grid display is toggled. However, when issued from the "Constants" dialog, the spacing only is set and its visibility state is un-affected.

SET LABEL-ATTRIBUTE

PURPOSE: To set the default text height, width and angle used for all subsequent instance, signal name and text placements.

SYNTAX: **SET LABEL** {Height} {Width} {Angle} <RETURN>

EXPLANATION:

PROCAD automatically generates instance name when cells are placed in the drawing, and signal names when wires are placed. The default attribute of height, width and angle for these labels are controlled by this command. Signal names are not normally displayed as defaults, but instance names are. To disable the display of instance names generated, set the default label height attribute to 0. These parameters can also be modified by the CHANGE INSTANCE & CHANGE SIGNAL commands. The label height, width and angles are also used for default text entity parameters. You may use the **QUICK EDIT** command for modifying the height, width and angles of already placed entities.

Note: The max. height value is 500 database units, while the max. width value is 60 dbu.

SET MODE

PURPOSE: To change the database type from Schematic to PCB, or from PCB to Schematic.

SYNTAX: **SET MODE** <RETURN>

EXPLANATION:

This command allows you to define the nature of the design which you will be doing in PROCAD. If you define the database as a Schematic, the RESOLVE command and all connectivity features (i.e. SHOW NODE etc.) will work oblivious to layers. That is any common vertex will be connected to another regardless of layers.

If you define the database to be a PCB the connectivity is based on layering only. RESOLVE and all connectivity commands will work on single layers only. (See also the RESOLVE, SHOW NODE and SELECT SIGNAL commands).

Tips: Since this command will always prompt for sheet number when schematic mode is selected, it could be used for changing the sheet number of a schematic database. When PCB mode is selected, you may re-assign your rat's nest layer #.

SET NESTING LEVEL

PURPOSE: To control the display of a cell's hierarchy within the database.

SYNTAX: **SET NE**sting={nn} <RETURN>

EXPLANATION:

nn == 1 to 15

This command controls the display level of the cell hierarchy (nesting). The default value is 1. This command controls the display of nested information. PROCAD will always display data up to the level of nesting set. Entities placed on layers 40 - 48 are only displayed if the nesting level setting is higher than the nesting (hierarchy) level of those entities. This will allow you to only display the top level of the cell information by setting the nesting level to 1. Setting the nesting level to 2 would show the first level of nested information, 3 the second level, etc.

SET PIN # DISPLAY SIZE

PURPOSE: Sets the height and width of pin numbers displayed by the SHO CELL REFERENCE and SET CELL & PIN REFERENCE DISPLAY commands.

SYNTAX: **(None; Must be entered from the dialog box)**

EXPLANATION:

This entry allows for the setting of text height and width used for displaying PCB components pin numbers. This command has no effect on template cell pin numbers and pin numbers entered as text entity during cell creation. (See **SET CELL & PIN REFERENCE DISPLAY** and **SHOW CELL REFERENCE** commands).

SET VISIBLE LAYERS

PURPOSE: To allow the selective viewing of layers.

SYNTAX: **SET VISible**{=nn}<RETURN>

EXPLANATION:

nn == Layer number (1 to 99)
Can be in any of the following forms:

n1,n2,n3...
n1,n2-nx...
ALL

The default setting is ALL layers visible.

Note: To display the current visible layers, issue the command SET VISIBLE=? or SHOW COLORMASK.

SET WAIT TIME

PURPOSE: To allow the control of the display time of error messages, etc in the message area of the display screen.

SYNTAX: **SET WAIT**{=nn}<RETURN>

EXPLANATION:

nn == A positive integer approximately equal to the number of half seconds required for the WAIT TIME.

Primarily used to adjust the message display time on different machines, so that the message can be read before it is erased. Also affects the special pause character used in the MACRO command syntax. The default setting for WAIT TIME is 5 (2.5 seconds).

EXAMPLE:

SET WAIT=10

The above example will set the wait time to $10/2 = 5$ seconds

In ProCAD for windows, single line information messages are displayed in the status line area for the duration setup by the **WAIT** constant. If the wait constant is set at 0, single line messages will be displayed in a pop-up dialog box. Note: non-information error and warning messages are always displayed in a pop-up dialog box.

SET WIDTH

PURPOSE: Sets the width of lines or wires input after this command is entered.

SYNTAX: **SET WIDTH**={n}<RETURN>

EXPLANATION:

n == positive integer up to 32,767

All lines and wires input after this command is issued

will be of the width specified by "n".

NOTE: See the CHANGE WIDTH command to change the width of lines or wires already input into the database.

SNAP CONNECTION LIMIT

PURPOSE: Sets the the "hot" window where cursor may snap to I/O point or a wire vertex.

SYNTAX: **(None; Must be entered from the dialog box)**

EXPLANATION:

The "hot" window for snapping cursor to connection I/O or wire vertex may be set here. The default setting is 1/2 of the ghost grid (See SNAP ON CONNECTION Command). If you are experiencing "Partial or incomplete route error" when rat's enforced flag is on, you may consider increasing the Snap Connection limit.

FONT SELECTION

PURPOSE: Selects true type or vector fonts used for labels and text entities.

SYNTAX: **(None; Must be entered from the dialog box)**

EXPLANATION:

This entry allows for the selection of font type for text and label entities. True type fonts are only available in Schematic Design Mode. You should only use true type fonts if your output device supports it. See the caution on "**TRUE TYPE FONTS**" at the beginning of this manual.

SET DATAPATH

PURPOSE: To set the DOS path statement to be used for all PROCAD Database and Library files references.

SYNTAX: **SET DATA**path <RETURN>

EXPLANATION:

This command will allow the user to set the DOS data path statement that will be used for all database and library references. In addition, you may also use this command to set the data path for data files. The command will prompt for the data path for use with database, library and project data files. This command allows for the flexibility of putting database files in a different sub-directory from the library and executable files. The default datapaths is normally setup with the CONFIG.EXE utility provided with PROCAD. The SHOW STATUS command will also report the current database, library and project data paths in effect.

Example:

SET DATA \WPROCAD\DATABASE \WPROCAD\LIBRARY \WPROCAD

The above example will cause PROCAD to use the sub-directories \WPROCAD\DATABASE, \WPROCAD\LIBRARY & \WPROCAD for PROCAD databases, libraries and project data references respectively.

Note: The path statement is limited to 24 characters. When the PLOT and PLOT/G commands are issued, the current path data is written to the corresponding .CTR and .GCF files, and will be used by PROPLOT and GERPLOT respectively. Issue the SHOW STATUS command to view the current data path settings. Use the WCONFIG.EXE utility for setting a startup default data paths. The Project load command will also set the Project datapath to the directory the project file (.PJF) was loaded from.

SET DIGITIZING MODE

PURPOSE: This command allows you to specify the type of digitizing mode for line or wire that would be entered in the INPUT mode.

SYNTAX: **SET DIGITIZING**{=aa}<RETURN>

EXPLANATION:

Where "aa" == S for straight mode, where all angles are multiples of 45 degrees. Vertices will be snapped to the proper position.

VE for vector mode where any angle is allowed.

HO for horizontal orthogonal mode where two points are connected by a horizontal segment first, then the vertical segment.

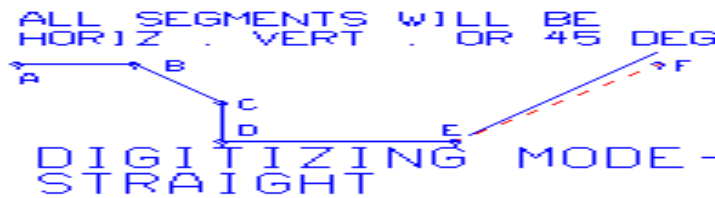
VO for vertical orthogonal mode where two points are connected by a vertical segment first, then the horizontal segment.

CU for curved trace mode where the system prompts for an arced trace starting point, ending point and a point on the arc. This input mode is similar to the ARC input command, with the current trace width (and signal name for wire) maintained.

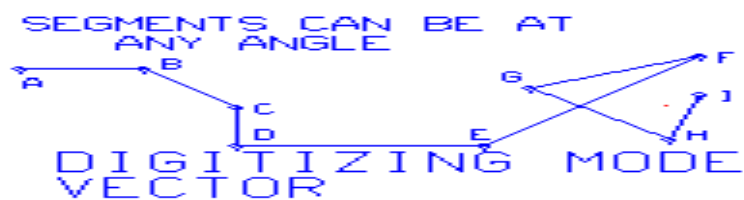
This command should be issued before an INPUT/L or INPUT/W command, to change the digitizing mode.

EXAMPLES

*SET D=S<RETURN>
IN/L<RETURN>*



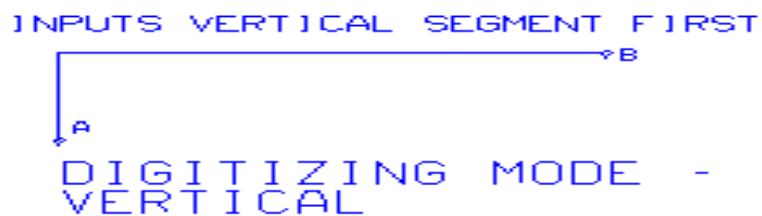
SET D=VE<RETURN>
IN/L<RETURN>



SET D=HO<RETURN>
IN/L<RETURN>



SET D=VO<RETURN>
IN/L<RETURN>



SET DISPLAY ORIGIN

PURPOSE: To set screen relative display origin.

SYNTAX: **SET** Origin <CR>

EXPLANATION:

This command allows for setting a screen origin. ProCAD will display the relative coordinate, distance and angle from the set origin on the status line field #3 as shown below:



R **xxx,yyy** L **dd** (**aa**).

Where **xxx,yyy** is the relative coordinate from the screen origin, **dd** is the distance from the screen origin and **aa** is the angle from the screen origin.

You can also invoke this command by issuing a *Ctrl+Y*. When you use a *Ctrl+Y*, ProCAD sets the screen origin to the current cursor position.

Note: You can still use the *Ctrl+X* command to set the temporary relative origin which is displayed on status line field #4 as dx: xx, dy: yy. The xx,yy is always reset to 0,0 when you click the left mouse button or press the ENTER key.

SET FILE

PURPOSE: This command allows you to assign a new name to the current active database during editing.

SYNTAX: **SET FILE**=<file name><RETURN>

EXPLANATION:

This command is very useful for doing minor changes to database and yet maintain the original database, ie. change the COLORMASK several ways, change components, move entities around, change values, etc.

EXAMPLE:

Current database file is EXAMPLE.DBF

SET FILE=EXAMPLE1<RETURN>

Make appropriate changes....

SAVE<RETURN>
OPEN EXAMPLE<RETURN>
SET FILE=EXAMPLE2<RETURN>

Make appropriate changes....

EXIT<RETURN>

You would now have three(3) databases:

EXAMPLE.DBF
EXAMPLE1.DBF
EXAMPLE2.DBF

NOTE: The original database has not been altered. See also the SAVE/AS command.

SET FUNCTION KEY

PURPOSE: Allows you to assign a command string to any one of 40 (33 for PROCAD for Windows) function keys.

SYNTAX: **SET FUNCTION** nn{=string}<RETURN>

EXPLANATION:

nn == key number 1 thru 40
 1 to 10 == F1 thru F10
 11 to 20 == Shift_F1 thru Shift_F10
 21 to 30 == Ctrl_F1 thru Ctrl_F10
 31 to 40 == Alt_F1 thru Alt_F10 (DOS vers. only)

ProCAD PowerStation 16/32 for Windows support the following extended function keys:

Function Key	ProCAD function #
F11	31
F12	32
Shft F11	33
Shft F12	34
Ctrl F11	35
Ctrl F12	36

{string} == any PROCAD command with or without its argument

By assigning command strings to function keys, you can customize many of your most often used command series to single keystrokes.

For example: You would like to set the COLORMASK codes to your own particular settings.

1. Create a Macro command file "COLOR.CMD" with the COLORMASK codes that you want. ie.

```
SET COL 1=2
SET COL 2,3=3
...
SET COL 99=129
```
2. Open a database by issuing the command "OPEN database"
3. SET FU 11= MACRO COLOR <RETURN>

Reference Guide...

4. Press Shift_F1 to invoke the above color setting macro command.

You may issue the SHOW COLOR command to see the new COLOR settings.

NOTE: Refer to SHOW FUNCTION KEY command to check current settings.

NOTE: PROCAD for windows does not support Alt-F1 - Alt-F10, (function codes 31-40), F10, Ctl-F4, Ctl-F6, Ctl-F10 and Shift-F10. keys. These keys are reserved for use by the Windows operating system., and not available to PROCAD.

SET LAYER

PURPOSE: Sets the current working layer.

SYNTAX: **SET LAY**er={nn}<RETURN>

EXPLANATION:

nn == 1 to 99

This command allows you to change the current working layer for inputting entities. The currently selected layer is displayed on the top status line at all times. You can use the CHANGE LAYER command to move entities to different layer after they have been placed in the drawing.

Note: You may also select an active layer with the mouse from the layer selection bar at the bottom of the screen (DOS version) or the Layers pull down combo-box in the ribbon bar (Windows Version).

SET MAP

PURPOSE: To set alternate layer mapping groups for SMT designs when the same cell is used for both the top and bottom sides of a design involving SMT devices.

SYNTAX: **SET MAP** map_grp, orig_lyr, mapped_lyr <CR>

EXPLANATION:

This command will allow the user to map the original layer of all entities comprising a cell to alternate layers. In the above syntax, map_grp is a number between 1 and 3. For any map group chosen, the layer mapping from the original defined layer can be set. This command in conjunction with the MIRROR X/Y command allows for the use of the same cell for both the top side and bottom side of an SMT design, simply by moving the cell to an alternate map layer group when placed at the bottom side of the board.

Example: SET MAP 1
 20,25
 5, 8
 15,26

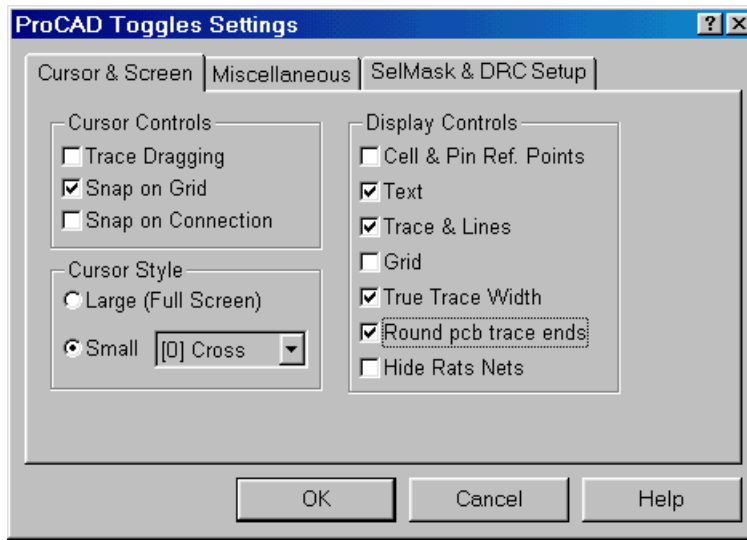
In the above example, a cell designed with entities in layers 20, 5 and 15 will be drawn as a cell with entities in layers 25, 8 and 26 respectively when the cell is moved to layer map group #1 (using the CHANGE MAP command). The color and fill attributes of layers 25,8 and 26 will be used for the drawing.

Note: This command does not actually change the layers where the original entities of the cell were placed, only the mapping is changed when the cell is plotted or drawn on the screen. See also the SHOW MAP and CHANGE MAP commands. This command in conjunction with the MIRROR X/Y command could be used for placing the same cell both at the top and bottom sides of an SMT design.

SET TOGGLES

PURPOSE: This command allows the user to configure a wide variety of system parameters which generally remain constant once they have been established for a specific design. These system parameters may also be executed from the command line as follows:

SYNTAX: **SET TOG**gles <RETURN>



EXPLANATION:

The SET TOGGLES command opens a small window on the primary display which allows many different system parameters to be toggled from one location. The following commands may be accessed from the SET TOGGLES window.

Reference Guide...

PARAMETER	EQUIVALENT COMMAND
Auto Menu	SET MENU (DOS)
Auto Pan	SET AUTOPAN
Alternate Screen	SET ALTERNATE SCRN (DOS)
Cell & Pin Ref. Points	SET CELL REFERENCE
Ghost Grid Check	(None)
Grid Display	SET GRID_DISPLAY
Hide Rats Nests	(None)
Large Cursor	SET LARGE CURSOR
Logging File	SET LOG FILE
Menu Enable	SET MENU (DOS)
Online DRC	(None)
Quick Set	SET QUICKSET (DOS)
Rats nest Enforced	(None)
Round pcb trace ends	(None)
Rubber band	SET RUBBERBAND
Select Mask	SELECT KINDS
Small Cursor Style	SET CURSOR STYLE
Snap on Connection	SET SNAP CONNECTION
Snap on Grid	SET SNAP GRID
Text Display	SET TEXT DISPLAY
Trace Display	SET TRACE DISPLAY
Trace Drag	SET DRAG
True Width	SET TRUE WIDTH
Warning Bell	SET BUZZER

Note: The Toggles dialog box may include fields that do not have a corresponding command for toggling them. For these fields, you must set or clear them in the dialog window.

Note: The SET TOGGLES command cannot be issued from within a MACRO command file. You must execute the desired command from the list below to toggle the desired parameter within a MACRO command file.

HIDE RATS NEST

PURPOSE: Controls the viewing of rats nest wires in the database.

SYNTAX: (None; Must be entered from the dialog box)

EXPLANATION:

This command controls the display of rats nests. You must enable or disable this flag from the [Other]Toggles "Hide Rats Nest" dialog item. There is no keyboard entry for this command.

When this flag is ON, rats nests are only displayed when they are selected. This feature is useful when manually routing a crowded board.

*Note: This command is ignored if TRACE DISPLAY IS OFF (See **SET TRACE DISPLAY COMMAND** below).*

SET AUTOPAN

PURPOSE: To enable or disable the automatic screen panning when the cursor is moved to the edge of the screen.

SYNTAX: **SET AUTOPan** {ON/OFF} <RETURN>

EXPLANATION:

This command allows for enabling or disabling the automatic panning feature of PROCAD. When enabled, the screen will be automatically panned when the cursor moves to any of the 3 edges (top, left or right) or 4 corners of the screen. The cursor position causing the pan will be moved (panned) to the center of the screen. The screen pan is a re-draw type for PROCAD for DOS and a combination of screen move and redraw for PROCAD for Windows.

Example: SET AUTOP ON AutoPan enabled.
 SET AUTOP OFF AutoPan disabled

Note: When issued without any arguments, the state of the AutoPan will toggle between On & OFF. This command can also be toggled from the SET TOGGLES pop-up window. The state of the AutoPan mode will always be displayed on the status line. When using non-CGI drivers, AutoPanning can be achieved regardless to what command is been executed or active. For CGI drivers, autopan will be disabled when certain commands are active. Check the status line for the state of Autopan when a command is active to see if you could invoke autopanning in that mode.

SET AUTOMENU

Note: This command is only supported in PROCAD for DOS.

PURPOSE: To enable or disable automatic menu invocation when select environment is entered.

SYNTAX: **SET AUTOMENU** {ON/OFF}<CR>

EXPLANATION:

This command allows for the automatic invocation of user defined menu #4 when the select environment is entered. Users wishing to use menus will be able to automatically select menu #4 when select environment is entered. You may still issue commands from function keys when menus are active. Users who prefer command line prompt when select environment is entered should disable this command.

Example: SET AUTOM ON AutoMenu enabled.
 SET AUTOM OFF AutoMenu disabled.

Note: Use the CONFIG.EXE utility to set the default state of this command. See CONFIG utility section for more details

SET ALTERNATE SCREEN

Note: This command is only supported in PROCAD for DOS.

PURPOSE: The purpose of this command is to allow you to change the basic screen display format.

SYNTAX: **SET AL**ternate ON/OFF<RETURN>
 SET ALternate<RETURN>

EXPLANATION:

The alternate screen display format has at least the MAIN MENU displayed at all times. Issuing the command by itself will toggle back to the previous display screen format (See the Installation procedures and SETUP utility for setting up the default screen display format.)

NOTE: If you are using graphics adapter with less than 512 pixels resolution in the X-axis eg. CGA in color mode, the ALTERNATE SCREEN MODE will always be in effect.

SET CELL REFERENCE

PURPOSE: To toggle the cell reference marker display ON/OFF of all the cells in the database. In PCB mode, physical pin numbers are also displayed.

SYNTAX: **SET CELL**Ref ON/OFF <RETURN>
SET CELLRef <RETURN>

EXPLANATION:

If the CELL REFERENCE is set ON, then a marker(X) will be displayed at all cell I/O reference point within the database. In addition to placing an X at the reference points of PCB symbols, the physical pin numbers are also displayed. Use **[Other]Constance..** dialog to set the height and width of pin numbers. See SHOW REFERENCE command for displaying individual cell references.

SET CURSOR STYLE

PURPOSE: To select 1 of 6 possible graphics cursor styles.

SYNTAX: **SET CUR**sor N <RETURN>

EXPLANATION:

This command selects 1 of 6 possible cursor styles for graphics input. In the above syntax, N represents the cursor style selected as follows:

N	Cursor-style
0	Crosshair
1	Arrow
2	Mark
3	Pointing Hand
4	Palm of Hand
5	Hourglass

*NOTE: This command has no effect on the full screen cursor type. Use the drop down combo box for style selection when small cursor is selected in the **[Other]Toggles** dialog box. Available cursor styles may be different from those described above in the Windows version of PROCAD.*

SET DRAG

PURPOSE: To turn on/off cell connection dragging when a cell is moved, rotated, reflected or scaled.

SYNTAX: **SET DR**ag ON/(OFF)<RETURN>
 SET DRag <RETURN>

EXPLANATION:

When a cell is moved, rotated, reflected or scaled, the connections to that cell are also moved appropriately to maintain connectivity if the drag feature is turned on. This command allows you to enable or disable this feature. The second syntax above will toggle drag ON/OFF.

Note: This flag is forced ON when Rats Enforced Mode is ON. This prevents the accidentally breaking of connectivity.

WARNING: If a connection on the cell being dragged is placed directly over the connection of another cell and the command terminated, the next time the cell is moved, those wires of the other cell will also be picked up and dragged.

SET GRID_DISPLAY

PURPOSE: This command allows you to set the spacing of the grid displayed on the screen and the display of the grid.

SYNTAX: **SET GRID**=<xxx,yyy><RETURN>
 SET GRID=ON<RETURN>
 SET GRID=OFF<RETURN>
 SET GRID=nnn<RETURN>
 SET GRID<RETURN>Toggles ON/OFF

EXPLANATION:

xxx,yyy ==number of database units per grid line in x and y direction.
nnn ==number of database units in both the x and y axis.

NOTE: See command SET SNAP for how to lock the cursor to the grid.

NOTE: When this command is issued from the "Toggles" dialog box, the state of the grid display is toggled. However, when issued from the "Constants" dialog, the spacing only is set and its visibility state is un-affected.

SET LARGE CURSOR

PURPOSE: This command allows the user to change from the smaller cursor to the larger, full screen cursor.

SYNTAX: **SET LAR**ge=ON<RETURN>
 SET LARge=OFF<RETURN>
 SET LARge<RETURN> Toggles ON/OFF

SET LOG FILE

PURPOSE: To generate a file of all entries into the database while the LOG File is ON. Can be used to verify potential problems or suspected software bugs. It can also be used in conjunction with the "MACRO" command to input certain data that is used repeatedly. The file is standard ASCII format, so that it can be edited if you wish.

SYNTAX: **SET LO**g=ON(OFF)<RETURN>

EXPLANATION:

After this command is issued, all commands entered into the current database will be recorded in a file named "filename.LOG" (where "filename" is the name of the current working database file). This file can be very useful for information repetitively entered into the database. See MACRO command for additional details.

SET MENU

Note: This command is only supported in PROCAD for DOS.

PURPOSE: To turn the display of the menu ON or OFF.

SYNTAX: **SET ME**_{nu}=OFF(ON)<RETURN>

EXPLANATION:

MenuEnable or Disable

This command removes the menu from across the bottom of the display. The menu across the bottom of the screen is not displayed if QUICKSET is enabled or if the alternate screen format is selected.

NOTE: This does not prevent the use of the Pop-Up Menus. See MENU command for further details. This command is not available in the Windows version of PROCAD.

SET QUICKSET

PURPOSE: To toggle the QUICKSET feature ON or OFF.

SYNTAX: **SET Q**_{uickset} ON/OFF<RETURN>
 SET Q_{uickset}<RETURN>

EXPLANATION:

The QUICKSET feature allows the user to utilize the current cursor position as the first coordinate entry in all commands where you would normally be prompted for the first coordinates. For example, the entity manipulation commands; ie MOVE, COPY, ROTATE, MIRROR, etc; would use the current cursor position at the time the command was issued for the coordinates to perform the function. The default condition is ON. This command should enhance your productivity noticeably. New users are advised to turn off this feature until they become familiar with PROCAD.

Note: In PROCAD for DOS, when QUIKSET is on, the menu strip at the bottom of the screen is not displayed.

PROCAD for Windows users are strongly urged to TURN OFF QuickSet, since using menus when Quickset is on can be very awkward.

SET RUBBERBAND

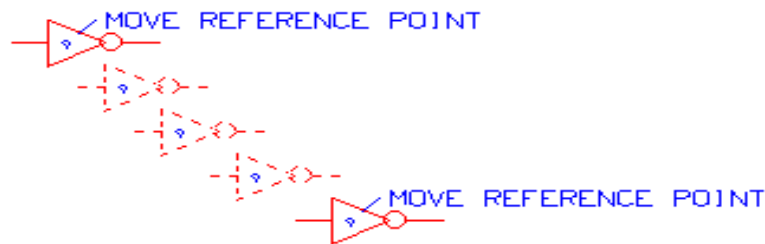
PURPOSE: To allow the dynamic movement of entities.

SYNTAX: SET RUBBERband=ON(OFF)<RETURN>

EXPLANATION:

This command will cause the movement of entities; cells, wires, lines, rectangles, vertices etc, to take place in real time. This command also affects the inputting of lines, wires, rectangles, polygons; in that entity is dynamically displayed while it is being input.

Note: The rubberband flag also affects the operation of the RATS NEST package placement command.



SET SNAP-ON-CONNECTION

PURPOSE: To allow control of the ability to snap a wire beginning & end point to the nearest connection point of a cell/wire.

SYNTAX: **SET SNAP**Connection<RETURN>
 SET SNAP OFF/ON<RETURN>

EXPLANATION:

The feature is toggled ON or OFF when the command is issued. When inputting wires, and not locked to grid, this feature will snap the first & last vertices to the nearest I/O reference point of the nearest cell or wire vertex.

Note: Snapping to cell I/O is attempted first before snap to a wire vertex is attempted. The default snap window is equal to 1/2 a ghost grid. You may change the snap window with the SET SNAPLIMITS command ([Other...]Constants dialog box).

SET SNAP-ON-GRID

PURPOSE: This command allows you to lock the cursor to the grid.

SYNTAX: **SET S**nap=ON(OFF)<RETURN>

EXPLANATION:

This command will cause the cursor movement to be locked to current grid resolution. See the section at the end of this section on modification of cursor movement. It is strongly recommended that you have snap to grid on when defining I/O reference points of cells or when inputting wires.

SET TEXT DISPLAY

PURPOSE: To allow turning off the display of text, to speed up the screen redraw time.

SYNTAX: **SET T**ext ON(OFF)<RETURN>

EXPLANATION:

When the Text Display is turned off, a warning will be issued if an attempt is made to INPUT TEXT. After the command is issued, any command that causes the display screen to be redrawn will cause the text to become undisplayed.

SET TRACE DISPLAY

PURPOSE: Controls the viewing of wires and lines in the database.

SYNTAX: **SET TR**ace=ON/OFF<RETURN>

EXPLANATION:

This command controls the display of wires. When issued the system will issue a warning if a wire is input to the database and the wire will not be visible. Whenever a display refresh is executed after this command is issued, all wires/lines will become invisible.

SET TRUE WIDTH DISPLAY

PURPOSE: To enable or disable the display of true trace widths.

SYNTAX: **SET TRU**e={ON/OFF}<RETURN>
 SET TRUe<RETURN>

EXPLANATION:

This command will toggle the display of true trace widths in PROCAD. When enabled (ON), PROCAD will display the true widths of all traces. When disabled (OFF), PROCAD will only display the center line of all traces in the database regardless of their true widths. This has the effect of speeding up screen repaint operation significantly.

EXAMPLE:

SET TRU=OFF

The above command will turn off the display of true trace widths.

Issuing the command without any arguments will toggle the state of the true width display flag.

Note: This command does not affect the output generated by PROPLOT or GERPLOT. The default state of this flag is ON, except when the database is run through PROROUTE.

SET BUZZER

PURPOSE: To allow you to turn on/off the warning buzzer, which is sounded for all error messages and warnings.

SYNTAX: **SET BUZZER**={ON/OFF}<RETURN>

EXPLANATION:

This command merely allows you to disable the error / warning buzzer (bell) which sounds with all error or warning messages.

SELECT KINDS

PURPOSE: To provide a means of specifying the types of entities that can be selected.

SYNTAX: **SELECT Kinds**{=zz,zz,...zz} <RETURN>

EXPLANATION:

zz ==	A	for Arcs
	CE	for Cells
	CI	for Circles
	CO	for Connection Dots
	D	for Donuts
	L	for Lines
	P	for Polygons
	R	for Rectangles
	T	for Text
	W	for Wires
	ALL	for all of the above.

NOTE: This command affects all commands that use the implied selection environment. See also the SELECT ENTITY and SELECT WINDOW commands.

ROUND PCB TRACE ENDS

PURPOSE: To provide rounded trace end points on screen when a PCB design is viewed.

SYNTAX: (None; Must be set/cleared in dialog box)

EXPLANATION:

When this flag is set, ProCAD will display PCB traces with rounded end points. This closely emulates what is generated in GERBER when round apertures are used.

Note: This flag will cause about 10% slower screen refreshes. This flag does not affect your PROPLOT outputs. PROPLOT "Round trace endpoints" flag must also be set if you desire this feature in your prints/plots.

GHOST GRID CHECK

PURPOSE: To provide for continual checking of ghost grid vs display magnification.

SYNTAX: (None; Must be set/cleared in dialog box)

EXPLANATION:

When this flag is set, ProCAD checks the current resolution, magnification, display and ghost grid settings continuously as the design progresses. If it detects that the current ghost grid setting is not reliable for current screen magnification, a warning message is displayed and the magnification automatically adjusted. The formula used for this calculation is explained in the **CHANGE DISPLAY FACTOR** command section of this manual.

Note: If your cursor does not move in the increment desired (ghost or display grid settings), or your grid seems to shift at odd zoom magnifications, you should toggle this flag ON.

RATS NEST ENFORCED

PURPOSE: Set a mode requiring the presence of a rats nest to a device pin before interactive routing can be performed on that pin.

SYNTAX: **(None; Must be set/cleared in dialog box)**

EXPLANATION:

This mode is also selectable when a new PCB database is created. The default is **ON** for PCB databases. When this flag is set, you may not enter or stitch wires to a component that does not already have a net connected to it. You should select the net that you would be routing, then proceed to stitch or wire the route interactively. If no rats net was selected before the Stitch or Input/Wire command is issued, ProCAD will prompt for it before proceeding with the command.

When you push <ESC> to complete the route, ProCAD will verify that the new route satisfies the Net on hold, and if that is the case, the held net is deleted automatically.

NOTE: The auto net deletion only works if the net placed on hold has only 2 vertices (All optimized rat's nest satisfies this requirements since they only have 2 vertices; point-to-point). The RATS NEST ".CON" load command will automatically optimize all rats nest if rats enforced flag was set.

NOTE: When rats nest enforced mode is enabled, deleting a routed trace will restore the original rats nest. Also, a warning message is issued if an attempt is made to delete or break a rats nest connection manually.

TIPS: If you want to completely re-route a previously routed net, you should delete the whole net, (ProCAD will automatically re-establish the route with a rats nest when rats nest enforced mode is in effect). Next you should select this newly entered rat's nest, put it on hold (CNTL-Z in Select mode), then proceed to use the stitch or input wire command to generate a new route for it.

ONLINE (REAL TIME) DRC

PURPOSE: To enable or disable PCB design rule checking in Real time.

SYNTAX: **(None; Must be set/cleared in dialog box)**

EXPLANATION:

This mode controls the amount of checking performed in real time as the design progresses. When enabled, ProCAD checks newly entered PCB trace for design rule (spacing, shorts etc) violation. DRC is also performed when a trace or device is edited (moved, rotated, etc) in select environment. You may also set your desired clearance values (pad-to-pad, pad-to-trace, trace-to-trace, etc) from this dialog box..

Note the following: 1) Online design rule checking could cause a lengthy pause when you exit the Select environment. 2) Online DRC is ignored in schematic mode.

See also the discussion on DESIGN RULE CHECKING (Chapter 4-11).

SET SCREEN

PURPOSE: To record various display references.

SYNTAX: **SET SC**reen=nn<RETURN>

EXPLANATION:

This command will assign a screen number (1 to 10) to a display window that has been positioned with the SHIFT command. Provides a very handy way of moving around the database.

NOTE: Only the SCREEN origin coordinates are saved. Use the SEL SCREEN command to recall the saved screen view. This command is not supported in scaled screen mode.

SET VIEW

PURPOSE: To allow a variable display view to be set.

SYNTAX: **SET VIEW**<RETURN>

EXPLANATION:

The system will prompt for the lower left and upper right-hand coordinates of the area of the display that you want to fill the display screen with. This command is also referred to as WINDOW ZOOM.

SET VIEWPORT

PURPOSE: To provide a means of seeing the entire database in a window.

SYNTAX: **SET VIEWPort**<RETURN>

EXPLANATION:

The system will prompt for the lower left and upper right-hand coordinates of the area of the display that you want to fill with the entire database. The defined window will become the active working window. The original display view will be redrawn upon exiting the viewport. To close the viewport issue the command: SET VIEWPort=OFF <RETURN>, and to enter the last defined view port, enter the command SET VIEWPort=ON. When the SET VIEWPORT command is issued without the ON or OFF clause, PROCAD will toggle the viewport off if it was currently opened, or will prompt for view port window coordinates if no view port was currently opened.

Note: In PROCAD for Windows, a view port is automatically closed when window causes a repaint operation (e.g. Tile, Cascade, Re-size, re-painting destroyed window area, etc.).

SHIFT

PURPOSE: Shifts the view display from within the database.

SYNTAX: SHIfT<RETURN>

EXPLANATION:

This command will prompt you for two coordinates (1. SHIFT from and 2. SHIFT to) and it will offset the display window by those coordinates. This establishes a new origin for the display screen. This view is not affected by the REDRAW command. The position can be recalled easily, record the position by the SET SCREEN command then recalling the screen display with the SELECT SCREEN command.

Note: In PROCAD for windows, clicking on the windows scroll bars actually executes this command.

SHOW ATTRIBUTES

PURPOSE: This command is used to view/edit the attributes of a selected cell.

SYNTAX 1: **SHOW** Attributes<RETURN>

EXPLANATION:

These attributes are displayed in a pop-up window in the upper right-hand corner of the display screen. If changes are made, you will be prompted to save the changes. Changes made and saved affect the named cell and not just the cell instance.

SYNTAX 2: **SHOW** Attribute [libname]cellname<RETURN>

EXPLANATION:

This version of the command allows you to display and/or edit the attributes of a named cell. This syntax allows you to choose a cell which may not be on the current drawing.

NOTE: This command does not allow you to extend the number of lines in the attribute data file. To modify the number of lines reserved for the cell attributes, you must use the EDIT CELL command, and respond with a new number of lines to prompt #3. (See also the section on Creating & Editing Cells (Chapter 4-10).

SHOW COLORMASK

PURPOSE: To display the color, line type and fill status of the 99 layers in the database.

SYNTAX: **SHO**w **COL**ormask <RETURN>

EXPLANATION:

Issuing this command will display the layer attributes, 12 layers at a time, in a window in the upper left-hand corner of the display. Press <RETURN> or <ESC> to remove the window and exit the command.

Note: The current active layer attribute is always displayed on the status/message line on the screen

SHOW CONNECTIONS

PURPOSE: To hi-lite all traces connected to a given component

SYNTAX: **SHOW CON**nections <CR>

EXPLANATION:

This command will allow for selecting a component, then all wires making valid connections to the component will be hi-lited. You may press <ESC> to remove the hi-lite. This command is equivalent to the repeated issuance of **SHOW NODE** command on all pins of the selected component, as such all the rules that applies to the **SHOW NODE** command applies to the **SHOW CONNECTION** command. This command is especially useful when you want to verify that all required connections are made to a component before using PROCAP to generate a netlist.

SHOW DIRECTORY

PURPOSE: To provide a means of viewing the directory of the current active or any cell library.

This command is not supported in PROCAD for Windows. If issued from PROCAD for Windows, this command will map to the Library Maintenance command.

SYNTAX: **SHOW DIR**ectory <RETURN>
 SHOW DIRectory=[libname]<RETURN>

EXPLANATION:

With this command, the directory of any library can be viewed without entering the LIBRARY MAINTENANCE environment. The directory will be displayed in a window opened in the upper left-hand corner of the display. Omitting the [libname] will imply that you want to see the directory of the current active cell library. The /OUT= (for generating hardcopy of the directory listing) and wild card syntax are accepted by this command. See the DIR command in the Library Maintenance section for more details and command options.

SHOW DISTANCE

PURPOSE: This command allows you to measure the distance between two points.

SYNTAX: **SHO**w **DIS**tance <RETURN>

EXPLANATION:

The message area across the bottom of the display screen will display the straight line distance between point #1 and #2, and delta x and delta y

SHOW FUNCTION KEY

PURPOSE: To allow viewing the command string assigned to a particular function key.

SYNTAX: **SHOw F**unction{=nn} <RETURN>

EXPLANATION:

nn == Function Key number (1 to 40) as follows:

1 to 10 == F1 thru F10
 11 to 20 == Shift_F1 thru Shift_F10
 21 to 30 == Ctrl_F1 thru Ctrl_F10
 31 to 40 == Alt_F1 thru Alt_F10

ProCAD PowerStation 16/32 for Windows support the following extended function keys:

ProCAD function #	Function Key
31	F11
32	F12
33	Shft F11
34	Shft F12
35	Ctrl F11
36	Ctrl F12

EXAMPLE:

Assign a command string to Alt_F1

SET FU 31=MACRO COLOR.CMD

To see what command string is assigned to F31 (Alt_F1) issue the command:

SHOw FU=31<RETURN>

Message area would display:

F31=MACRO COLOR.CMD

Note: Function Key #s 10, 20, & 30-40 are not supported in PROCAD for Windows. See the SET FUNCTION KEY command for explanation of this restriction.

Reference Guide...

SHOW HIERARCHY

PURPOSE: To display the hierarchy of a selected cell.

SYNTAX: **SHOW Hierarchy**<RETURN>
 SHOW Hierarchy cellname <RETURN>

EXPLANATION:

If a cell name is specified, PROCAD will display the hierarchy of that cell. If a cell name is not specified, PROCAD will display the hierarchy of the cell at the current cursor position. To exit the SHOW Hierarchy command, use either EXIT or QUIT command. If a cell has more than 1 level of hierarchy, this command can be issued multiple times to go down the hierarchy levels. The EXIT or QUIT command will return you to the top level.

SHOW INFORMATION

PURPOSE: To display and optionally save information such as Nets, Layers, Components etc used in a design..

SYNTAX: **SH**ow **INF**o {/qualifier}<CR>

EXPLANATION:

This command may also be invoked from the **[File]Report...** menu item.

When issued, this command allows for the display of Nets used in a design (**/N** qualifier), Layers used in a design (**/L** qualifier), Components used in a design (**/C** qualifier) or Dump cell(s) in the active library to a defined area in the current database [**/D** qualifier] . When the qualifier is omitted, ProCAD prompts for it.

These command qualifiers (except **/D**) will display results in a window. The displayed data may be saved to a file and later printed or input into a spread sheet or other applications.

The **[Files][Report]Dump cells to Database...** menu item (**/D** qualifier above) will dump a group of cell or all cells from a library to a user specified area in the database. This command accepts wild card for cell(s) to dump. You may then print/plot the database or cut and paste to other desk top publishing programs for graphical documentation.

EXAMPLE:

SHOW INFO/C

The above command will display all components in a design in a dialog window.

Note: When displaying layers used in design, ProCAD does not display layers used in a cell hierarchy. You must first use the **[Edit]Cell Hierarchy** menu item before issuing the above command with **/L** qualifier to list all layers used in a cell.

SHOW LAYERS

PURPOSE: To display the layer map and names.

SYNTAX: **SHO**w **LAY**ers <CR>

EXPLANATION:

The Show layers command will display the layer numbers and their corresponding names, fill and color attributes in a status window. The layer names are dependent on the database mode selected (PCB or Schematic). The information displayed is the same as those defined in Appendix E and G (Quick reference guide).

If you plan to define your own layer names, you must define an ASCII file with the name "LAYERS.DAT". PROCAD will search for such a file, and if found, will use the layer names contained in the file. See Appendix C for the description of the LAYERS.DAT file format.

SHOW MAP

PURPOSE: To show the current layer mapping in effect for all three map groups.

SYNTAX: **SHO MAP** <CR>

EXPLANATION:

This command opens a window on the upper left corner of the screen, and displays the color and fill attributes of all three layer mapping groups. The map setting could be modified using the **SET MAP** command.

Example:

SHOW MAP

Note: See also the CHANGE MAP and SET MAP commands for more details

SHOW NODE

PURPOSE: To highlight all wires connected to a selected node.

SYNTAX: **SHOW NO**de<RETURN>
 SHOW NOde layer<RETURN>

EXPLANATION:

This command hi-lites all wires connected to and/or has the same signal name as the selected wire. In the PCB mode, only wires on the current layer or specified layer (second syntax above) will be checked and hi-lited. Press <ESC> to remove the highlighting.

SHOW POINT

PURPOSE: To show the database coordinates of a selected point.

SYNTAX: **SHO**w **P**oint <RETURN>

EXPLANATION:

The X and Y database coordinates of the specified point will be displayed in the message area across the bottom of the display area.

SHOW REFERENCE

PURPOSE: This command allows you to display the cell references (and pin numbers in PCB mode) of a selected cell only.

SYNTAX: **SHO**w **R**eferences <RETURN>

EXPLANATION:

The cell references will be displayed in the color assigned to the layer upon which the cell was placed. Use the SET CELL REFERENCE command to enable cell references (and pin numbers in PCB mode) display for all cells in the design.

SHOW SIGNAL

PURPOSE: This commands allows you to highlight a wire selected by it's signal name for viewing purposes only.

SYNTAX #1: **SHOW SIGNAL**=`{signal_name}`

SYNTAX #2: **SHOW SIGNAL**=`{signal_name},layer`

EXPLANATION:

In the PCB mode, only wires on the current layer or specified layer (second syntax) will be checked and hi-lited. No manipulation of the entity can be done with this command. The highlighting can be removed by pressing <ESC>.

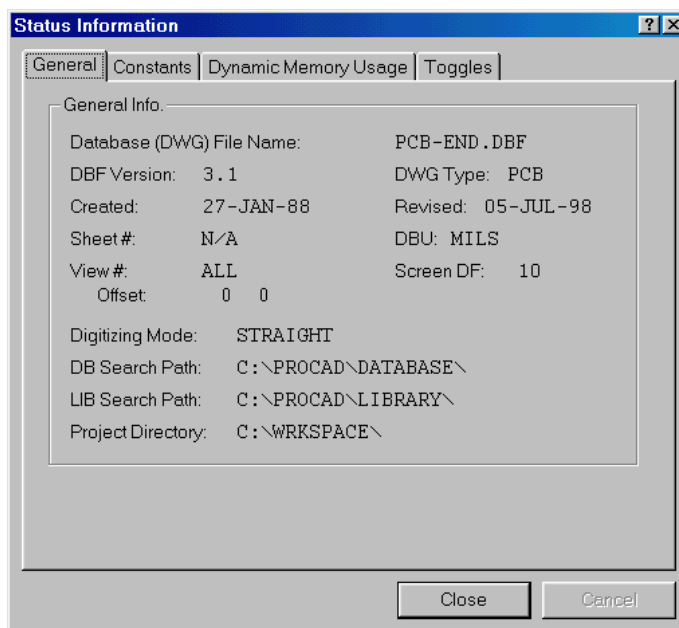
SHOW STATUS

PURPOSE: To view the database status information and defaults.

SYNTAX: **SHO**w **ST**atus <RETURN>

EXPLANATION:

This command allows the user to view the status/default information of the current database in a window displayed in the upper left-hand corner of the display.



SHOW SUMMARY

PURPOSE: To allow the inquiry of the information available on a specified entity(s).

SYNTAX: **SHO**w **SU**mmary <RETURN>

EXPLANATION:

The selected entity will be hi-lited, a readout of information will be displayed in the message area across the bottom of the display or in a dialog box, and the high-lighting removed.



Example Summary (Info.) report for a cell type entity

NOTE: This command can be issued in the SELECT environment. Press <ESC> to exit the SELECT environment.

SHOW UNCONNECTED PINS

PURPOSE: To hi-lite all pins on a device NOT connected to a trace

SYNTAX: **SHOW UN**connections <CR>

EXPLANATION:

This command will allow for selecting a component, then hi-lite all pins of the component not connected to any trace. The hi-lite is a marker of the cell placement layer color. You may refresh the screen to remove the hi-lite. The markers are not saved in the database. This command checks the cell attribute for RAT\$GNDn and RAT\$VCCn tokens and don't hi-lite those pins as un-connected.

This command is especially useful when you want to verify that all required connections are made to a component before using PROCAP to generate a netlist

Note: Since markers are placed directly on the pin(s), you should turn OFF Cell & Pin Reference display before issuing this command to enhance visibility.

STEP & REPEAT

PURPOSE: To replicate an array of cells, wires (for buss generation) or group of entities.

SYNTAX: **STEP** Xoffset,Yoffset,Xcnt,Ycnt <RETURN>

EXPLANATION:

The Step & Repeat command is used for replicating an array of cells or a group of entities. To replicate a group of entities, the SEL WINDOW command must be used to select the group of entities to be replicated, prior to issuing the above command. To replicate a single cell (or entity), the command may be issued directly, and the entity selected prior to being replicated.

The Xoffset and Yoffset are the distances to maintain between entities replicated in the X and Y directions respectively. The Xcnt and Ycnt are the number of times the entities are copied to form the matrix in X and Y directions respectively. The system will prompt for these parameters if they are omitted. A confirmation of the entered variables will be required before the replication process begins.

GENERATING SIGNAL BUSS:

When the selected entity to be replicated is a wire, and either the Xcnt or Ycnt is 0, PROCAD assumes that a bus signal is to be created. If the wire has a signal name of the following format:

BBnn

Where BB is an alpha character(s), and nn is a numeric character(s), PROCAD will assume BB to be the bus signal base name and nn the starting index. Replicated wires will have base name BB with the index incremented sequentially by 1. If the signal name format is not as stated above, PROCAD will prompt for bus signal base name and starting index. PROCAD will also automatically enable the display of replicated bus signal name if stepped wire had its signal name display enabled or if the user wishes to enable it. Note: The replication process may be aborted at anytime by typing <ESC> or CNTL-C.

STITCH

PURPOSE: To invoke the wire/trace input mode with the ability to toggle trace input between layer pairs and add feed throughs.

SYNTAX: **STI**tch/P top_Layer1 bottom_Layer2 [Library]Viacell
STItch {Top/Bottom} {signame} <CR>

EXPLANATION:

The Stitch command will enable inputting wires into the database with the ability to change layers, drop a feed-thru and continue in the input mode

SYNTAX # 1

The first syntax is used for defining the layer pairs and the via cell name to be used. The "/Parameter" option **MUST** be included in the command for proper operation. Top_Layer1 and Bottom_layer2 are the two working layers used by the stitch command, and [library]viacell is the Via cell name spec. The library qualifier must also be included.

Tips: You may also use the [Other]Constants... dialog entry to set the stitch parameters.

The STI/P command without any arguments will display the current setting of the stitch parameters.

Also, if the library qualifier is omitted, PROCAD will assume "PKG.LIB".

Note: The default layers are 20 & 23, and feedthru cell is [PKG]VIA

STI/P 20 23 [PKG]PAD050

SYNTAX # 2

The second syntax of this command actually invokes the input wire mode, using the layer specified in the command line as the starting layer. STI T & STI B will start the stitch operation from the top (first layer) & bottom (second layer) respectively. You can also specify layer numbers in place of T or B, provided that the layer specified must be one defined by the STI/P command. If a signal name is included in this command,

PROCAD will assign this signal name to the wire being input, otherwise, PROCAD will assign machine generated signal name to the wire being input. If STItch command is issued without any arguments, the stitching operation will start from the last layer used for the command.

STI T PROBE <RETURN>
STI 23 CLK <RETURN>
STI <RETURN>

When in the input mode, and PROCAD is prompting for point, you could enter V (or VIA), and PROCAD will automatically insert a feedthrough at the last input point, close the current wire being input, change layer and continue in the input mode. You may also enter "P" or "Z" to Pan or Zoom the screen respectively. To terminate the stitch command, simply press ESC.

Note: The STI T/B signame is a special case of the input wire command, and as such, all the features and limitations of the input wire command is applicable. In particular, you may enter "P" or "Z" to Pan or Zoom the screen while stitching a trace.

STRETCH

PURPOSE: To move a specified segment of a selected line, wire, polygon, or rectangle, or to stretch text height, circle diameter, or arc size.

SYNTAX: **ST**retch <RETURN>

EXPLANATION:

After selecting the entity to be stretched, the system will prompt for the coordinates of the segment to be stretched. The system will position the cursor in the center of the segment to be stretched. By moving the cursor, the segment position will be changed when <RETURN> is pressed.

When this command is applied to a circle or an arc entity, the size of the circle/arc is stretched. When applied to text entity, the height of the text is stretched; the width remains unchanged.

Press <ESC> to exit the STRETCH mode.

NOTE: It may be very helpful to execute this command with the SET RUBBERBAND=ON. See also the note on "Context Sensitive Commands" section of the User & Utilities manual.

This command can be used in the SELECT environment. Press <ESC> to exit the SELECT environment.

ProCAD will analyze your board and displays the approximate routing cost based on the number of holes and/or routing layers. This estimate is based on a 100% routing completion. If less than 100% of the board is routed, the routing charge will be calculated as follows:

$$\text{cost} = (\text{nPcnt} * \text{estCost}) / 100$$

Where **nPcnt** = Actual Percentage completion, **estCost** = The estimated cost reported by ProCAD. There is a minimum \$25.00 charge for all submitted jobs for remote routing. Please provide a credit card number and expiration date, then click the "Agree to terms" button to submit the job.

After you submit your job for remote routing, ICS high performance router, typically the CCT SPECCTRA SP10+ will be used to route the board. When the job has been routed, a claim # is automatically e-mailed to the e-mail address specified above (when the job was submitted). You should then use the **[Route][Internet AutoRoute]Receive Routed Job** menu item to retrieve the results. An invoice will also be forwarded to you by e-mail at the conclusion of your job. If you prefer not to submit your Credit Card # on-line, please call us during normal business hours with the proper payment information and we'll issue you a claim number.

Note: If you want to provide a "do list" for your routing job, it must be valid CCT SPECCTRA commands. You may also provide a default "Do list" file named DEFAULT.DO placed in your ProCAD working directory. ProCAD searches for this file and pre-loads the "do-list" edit control dialog if found. If you don't specify a do-list, the remote router defaults will be used. Please consult Cadence for SPECCTRA do-list format and commands.

NOTE: Please make sure you setup Microsoft Exchange or Outlook on your Windows. ProCAD uses your Windows E-mail client for Remote Job Submission Notification. Failure to do this could result in routing delays.

SWAP

PURPOSE: To swap pins in a component or swap gates within a PCB footprint.

SYNTAX #1: **SWAp** <CR>

EXPLANATION:

ProCAD for windows now supports pin and gate swapping of PCB database components, with instant back annotation. This feature requires the MDI (Multiple Document Interface) option. The Pin and Gate Swap command may be accessed from the **[Edit][More..]Swap...** menu item, (or if you are already in select environment, from the **[Other...]****Swap** menu item). When invoked, you select the desired swap option (Pin or Gate). You will be prompted to select the from gate or pin. You should then point to the appropriate source pin (or any pin of a source gate). ProCAD will then search the pre-loaded schematic database for the swapping criteria of the component. Then the swap choices will be hi-lighted. For pin swap, all swappable pins will be hi-lighted in red, and the source pin in white. For gate swap, the source gate pins will be hi-lighted in white and swappable gates pins will be hi-lighted in different colors, (each gate group hi-lighted in the same color). *Note: If you only have one choice, ProCAD will not hi-light the pins; swapping will be automatically performed.*

If you have more than one choice, you will be prompted to select the swap pin (or pin of a swap gate). You must select from the hi-lighted pins. ProCAD will then perform the swap operation and back annotate the results to the pre-loaded schematic sheet(s).

For instant visual effect, you should **"tile"** all loaded files before performing the swap operation.

The structure of the swap command dictates the following requirements:

1. You must define attributes for your schematic symbol which instructs ProCAD of swap-able pins and gates.

The data attribute keyword (token) for Pin Swap is:

SWP\$PINS n1 n2 ...

Where **n1 n2 ...** are pins of the symbol that can be swapped. If the symbol is a template cell, **n1 n2 ...** are the actual symbol I/O numbers. For non template cells, I/O numbers are the same as pin numbers.

The data attribute keyword (token) for Gate Swap is:

SWP\$GATES A,B,...

Where **A, B,..** are the swappable gates of the package. Note that only packaged cells (template cells) can be swapped within the same physical package.

Example:

The swap keyword for a 7400 device will be:

SWP\$PINS 1 2
SWP\$GATES A B C D

The above example permits ProCAD to swap the two input pins of the NAND gates, and be able to swap any of the 4 NAND gates within the package.

*NOTE: You may use as many **SWP\$PINS** and **SWP\$GATES** tokens as are needed for your different groups. You may also use continuation lines ("+" as the first character on a succeeding line) for the same groupings.*

2. Since the swap information is always extracted from the Schematic Symbol, You must load all the schematic sheets with the PCB file (Multiple document mode), before invoking the Pin or Gate swap command.

TOGGLE SELECTION

PURPOSE: To toggle the selection state of ProCAD entities.

SYNTAX: **T**Select <CR>

EXPLANATION:

This command is used for toggling the selection state of any ProCAD entity. ProCAD will select or deselect the entity at the current cursor location. This command is intended for ProCAD for windows users using the **Shift-Left Mouse button click** to perform the command.

Note: Although you may enter the command from the keyboard, it is strongly recommended you use the short cut method (Shift-Left mouse button click). You must setup the left mouse button for entity selection mode ([View]Preference... menu item) in order to use the short cut method. This is the default ProCAD configuration mode.

UNDELETE/UNDO

PURPOSE: To recover entities that have been deleted or undo last edits.

SYNTAX: **UNDelete**=n <RETURN>
 UNDelete=n1,n2..n3 <RETURN>
 UNDelete=All <RETURN>
 UNDelete=LAST <RETURN>
 UNDelete=? <RETURN>

EXPLANATION:

n == Layer number (1 to 99)

Undeleting "ALL" will restore all entities deleted since the last PACK DATABASE command was issued. Undeleting the "LAST" entity will recover the entity deleted by the last DELETE command. Undeleting "n" will recover only those deleted entities that were placed on the designated layers. Undelete ? will perform an Undo of the last edits.

Note: Successive UNDELETE LAST is not supported

UNSELECT

PURPOSE: This command is used to remove all selected entities from the "SELECT" mode, only while in the "HOLD" condition.

SYNTAX: **UN**Select <RETURN>

EXPLANATION:

This command allows you to remove the hi-lighting from previously selected entities that remain hi-lited while you are in the SEL HOLD condition.

UNSELECT ENTITY

PURPOSE: This command is used to remove unwanted selected entity(s) from the "SELECT" environment.

SYNTAX: **UN**Select **E**ntity <RETURN>

EXPLANATION:

When using the SELECT WINDOW command, and you "select" more than you have intended, unwanted entities can be selectively removed from the select list.

*NOTE: See the note on "Mouse button assignments" in the User's manual and **TOGGLE SELECTION** command for quicker method of performing this command.*

VOLTAGE TIES

PURPOSE: To automatically tie PCB pads to a voltage or ground planes.

SYNTAX: **VTie** <CR>

EXPLANATION:

This command is accessible from the **[Place]Voltage/GND ties...** menu item, or entered from the command line using the above syntax. This command causes ProCAD to place tie lines to pads of the desired signal. You will be prompted for the desired signal, tie placement layer and the width and length of the desired tie line in a dialog style box. ProCAD automatically deletes the corresponding rats nests after the ties are made.

Note: You should issue this command after your rats nest are placed, and before manual or auto routing of the remaining signals. You may use the copper pour command to generate the voltage/ground plane.

Since this command specifically asks for net name to tie and the corresponding layer, you may use it to tie not just power and ground pads, but virtually any signal to whatever layer plane desired.

ZOOM

PURPOSE: To allow close-up editing and viewing of both a specified magnification and display positioning.

SYNTAX #1: **ZO**om=n <RETURN>

SYNTAX #2: **ZO**om=? <RETURN>

SYNTAX #3: **ZO**om=L <RETURN>

EXPLANATION:

n == integer magnification factor (+ or -)

This command implies position by cursor location in that the position of the cursor will be moved to the center of the display and the display will be magnified from that position. Display can be reset by issuing the REDRAW command.

The second syntax of the zoom command (n=?) will perform a ZOOM ALL function. ProCAD will automatically select the appropriate zoom and scale factors in order to fit the current drawing on the current display window.

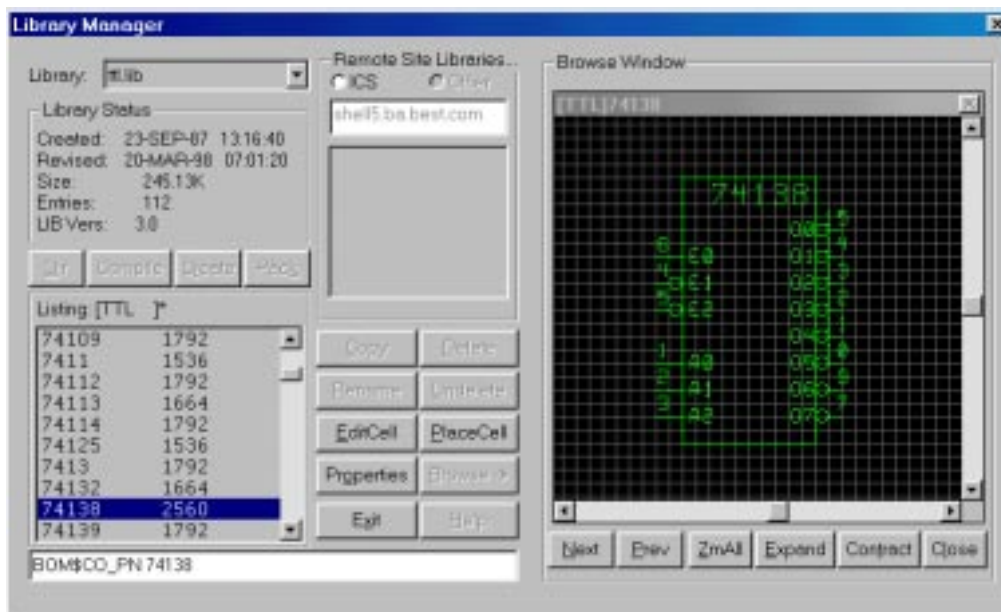
The third syntax of the zoom command (n=L) will restore the last zoomed screen (previous display). This is especially useful when you accidentally over or under zoom or when you want to pan quickly between to areas of your design.

NOTE: The view cannot be zoomed below a scale factor of 1x. Use the SCREEN SCALE command to overcome this limitation.

The ZOOM and PAN commands can also be issued while in the INPUT WIRE, GET CELL, DEFINING CELL I/O etc commands

LIBRARY MAINTENANCE COMMANDS

The LIBRARY MAINTENANCE mode of PROCAD provides a very powerful tool for managing cell libraries.



CELL NAME SPECIFICATION:

The full cell name specification is [library_name]cell_name. If the [lib_name] is omitted the command processor assumes the [lib_name] of the current active library. This affects all LIBRARY MAINTENANCE commands.

The use of "wildcard" specifiers is supported as follows:

? == any character in that position will meet the requirements.

* == any characters in all remaining positions will meet the requirements.

EXAMPLE:

DE 74?? <RETURN>

This command would mark for deletion all cells with a cell name of 7400 through 74ZZ (four character cell names that begin with 74).

DE 54* <RETURN>

This command would mark for deletion all cells with a cell name beginning with 54.

Note: The square brackets "[" must be used to designate a library as a prefix to a cell name eg: [PKG]IC14A means cell IC14A from library PKG.

WARNING: Since library names are part of the database, you should NEVER use the DOS rename or copy commands to rename PROCAD cell libraries. You should use the optional ARCHIVE utility for that purpose.

BROWSE

PURPOSE: To browse through a cell library displaying the graphical representation of all or selected cells.

SYNTAX: **BR** [libname]cells <CR>

EXPLANATION:

This library manager command will allow the user to display the graphical representation of a cell or group of cells in a given library. If the argument for the command is omitted, PROCAD will prompt for a cell name. Wild card characters "*" & "?" are recognized by this command. The current visible layers, color mask and nesting levels are used by this command for displaying cells.

EXAMPLE:

BR 74*
BR [PKG]IC??C

CHANGE LIBRARY

PURPOSE: To allow the active library to be changed while in LIBRARY MAINTENANCE.

SYNTAX: **CL**{=<newlib>} <RETURN>

EXPLANATION:

This command will only allow you to change active libraries, not create one as CHANGE LIBRARY command will do in the MAIN Database mode.

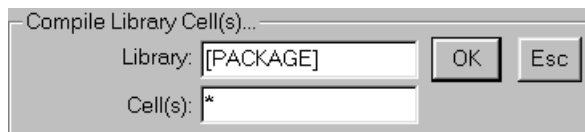
Note: The active library is only changed in the Library Maintenance mode. When you exit the Library Maintenance mode, the previous active library is restored.

COMPILE LIBRARY CELL(S)

PURPOSE: To generate DRC\$PADx, DRC\$HOLE and DRC\$VIA keywords for footprint cells based on keywords in the padstacks cells used for pads.

EXPLANATION:

This command accessible from ProCAD's Library Manager will generate the proper DRC\$PADx, DRC\$HOLE and DRC\$VIA attributes for packages from information on the padstack cells used for designing the package (footprint) cell. When issued, this command will prompt for cell(s) to be compiled in the current library. You can use a wild card (*, ?) to specify a group of cell(s) or the whole library. The default is the currently hi-lited cell.



NOTE: This command will only generate keywords for top level cells that are designed with padstack cells with the corresponding DRC\$XXX keywords. If the padstack cell does not have DRC\$XXX attributes defined, the top level cells DRC\$XXX keywords (if any) are left unchanged.

COPY

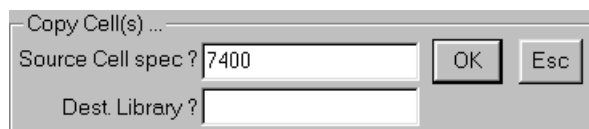
PURPOSE: This version of the "COPY" command is for use within the LIBRARY MAINTENANCE utility. It is used for copying cells from one library to another.

SYNTAX: **COPY** [fromlib]cellname [tolib] <RETURN>
 COPY [fromlib]oldname [tolib]newname<RETURN>

EXPLANATION:

The first command syntax illustrates the use of the COPY command to copy a cell from one library to another library.

The second command syntax illustrates the use of the copy command to create a new symbol with newname in either the current library, or another library.



You can also use the wildcard characters (*,?).

COPY [fromlib]??ALS [tolib] <RETURN>*

COPY [fromlib]54 [tolib]74* <RETURN>*

Note: The copy command will not copy nested cell data. Only the top level data will be copied to the specified library. You should use the Optional ARCHIVE Utility program to accomplish this task.

CREATE

PURPOSE: Allows you to create a new library, while in the LIBRARY MAINTENANCE mode.

SYNTAX: **CR**eat={<new libname>} <RETURN>

EXPLANATION:

A new, empty library file will be created.

DELETE

PURPOSE: This command allows you to delete a specified cell in a specified library.

SYNTAX: **DE**lete {<cellname>} <RETURN>
 DElete [libname]<cellname> <RETURN>

EXPLANATION:

Upon issuing the command the specified cell(s) will be marked for deletion. They may be recovered with the UNDELETE command prior to a PACK command being issued.

NOTE: Wildcard specifiers are allowed in the cellname and not the library name.

DIRECTORY

PURPOSE: To provide a listing on the display of the cell directory of the current library.

SYNTAX 1: **DI**rectory cellname <RETURN>
 DIrectory [library]cell <RETURN>

EXPLANATION:

This command will provide a directory listing of the specified cell(s). Wild card specifiers can be used for the cell name. If cellname is omitted, all cells in the current or specified library will be listed.

SYNTAX 2: **DI/OUT**{=<filename>} <RETURN>
 DI/OUT=filename [lib]cell <RETURN>

EXPLANATION:

This form of the DIRECTORY command will generate a file of the cell names contained within the current library. This ASCII file can be printed out in DOS using the PRINT command.

See also the SHOW DIRECTORY command in the MAIN DATABASE mode

DISPLAY ATTRIBUTES (PROPERTIES)

PURPOSE: This command allows you to view/edit the attributes of a selected cell.

SYNTAX: **DA** {cellname} <RETURN>
 DA [libname]cellname <RETURN>

EXPLANATION:

PROCAD will display the attributes assigned to the selected cell in a window in the upper righthand portion of the screen. All entries and/or changes made will be saved when the attribute editor is exited.



EXIT

PURPOSE: The purpose of this command is to allow you to exit the LIBRARY MAINTENANCE mode.

SYNTAX: **EX** <RETURN>

EXPLANATION:

Upon issuing this command, PROCAD will exit the Library Maintenance mode and return to the Main Database.

Note: Pressing ESCAPE has the same effect as the EXIT command.

LIST FILES

PURPOSE: To display the directory of DOS files in the current or specified directory.

SYNTAX: **LF** filename <CR>

EXPLANATION:

This library manager command will allow the user to display the file names in the current or specified DOS directory. This is equivalent to the DIR command in DOS. Wild card characters are accepted. A null filename will cause PROCAD to display all the file names in the current directory.

EXAMPLE:

*LF *.LIB
LF *.DBF*

PACK

PURPOSE: To remove cell marked for deletion from the library.

SYNTAX: **PK** <RETURN>

EXPLANATION:

The current active library will be packed; have all cells marked for deletion removed thereby conserving disk space. The cells will also be sorted in ascending order in the library after packing.

Note: This process may take a relatively long time, and should be done during a break of some sort when it could be left unattended.

REMOTE LIBRARY DIRECTORY & DOWNLOAD

PURPOSE: To View and optionally download libraries from our FTP site.

EXPLANATION:

This command is accessible from the Library Manager. While in the Library Manager dialog, select ICS under "Remote Site Libraries" group. ProCAD will dial your RAS (internet access) if necessary and display all libraries currently available on our FTP site. To download any of the libraries listed under the "Remote Site Libraries" list box, hi-lite and double click on the desired library. If your maintenance is current, ProCAD will proceed to download the library into your current library path directory. You do not need maintenance to view the list of Libraries (NOT Library Cells) available on our FTP site.

NOTE: YOU MUST HAVE A VALID AND CURRENT MAINTENANCE TO DOWNLOAD LIBRARIES FROM OUR FTP SITE. THE REMOTE LIBRARY DOWNLOADER WILL FAIL IF YOUR MAINTENANCE HAS EXPIRED OR YOU DON'T HAVE A REGISTERED PROCAD. CONTACT ICS OR YOUR DEALER TO PURCHASE A MAINTENANCE CONTRACT.

IF YOUR ISP IS DOWN OR ICS FTP IS UN-ACCESSABLE, PROCAD WILL ATTEMPT THE CONNECTION AND EVENTUALLY TIME OUT WITH AN ERROR MESSAGE. THIS MAY TAKE A FEW MINUTES, SO DON'T PANIC AND RE-BOOT YOUR SYSTEM.... PROCAD HAS NOT FROZEN FOR GOOD!!!!

RENAME

PURPOSE: To allow the renaming of specified cells.

SYNTAX: **RE**name{<oldname>=<newname>} <CR>

EXPLANATION:

This command does allow the use of the global characters ? and *.

EXAMPLE:

RE 74=54*<RETURN>*

All cells begining with 74 will be renamed to begin with 54.

UNDELETE

PURPOSE: To recover cells marked for deletion.

SYNTAX: **UN**delete{=<cellname>}<RETURN>

EXPLANATION:

The deleted cell specified will be returned to an active status.

Note: Caution should be exercised here to insure that there is not another cell within the library that has the same name.

EDIT CELL

PURPOSE: To invoke the cell editor for editing the currently hi-lited cell.

SYNTAX: **ES**<RETURN>

EXPLANATION:

ProCAD closes the library manager dialog box, and enters cell editing mode using the currently hi-lited cell as argument.

PLACE CELL

PURPOSE: To place the currently hi-lited cell into the database.

SYNTAX: **G**Cell<RETURN>

EXPLANATION:

ProCAD closes the library manager dialog box, and loads the currently hi-lited cell into the database. This is equivalent to issuing the GET CELL command in main mode using the hi-lited cell as argument.

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Attribute Data Files

A *ppendix*

A

APPENDIX A

ATTRIBUTE DATA FILES

INTRODUCTION:

The Attribute Data File that can be attached to each cell in a library is primarily for the purpose of attaching prefix keywords to the cell for utility programs such as the netlist extractor PROCAP. For instance, when PROCAP extracts a netlist for a specific simulator, or a bill-of-materials; it will look into the Attribute Data File of each cell within the database for specific keywords which will indicate information to be included within the netlist or bill-of-materials.

FILE ORGANIZATION:

The Attribute Data File is organized as an ASCII file with 30 character columns per line, and a maximum of 128 lines per file. Keywords must begin in column 1, and can be a maximum of 15 characters. The following is an example:

XXX\$KKKKKKKKKKKK

XXX == Minimum of three characters from the following list must be used for PROCAP to properly extract the appropriate netlist or bill-of-material information.

SILos	- SILOS Logic Simulator
SPIce	- SPICE,HSPICE & LVS CktSim
SIMulog	- SIMULOG Logic Simulator
MDL	- NCA Master Design Lang.
BOM	- Bill-of-Material Info.
RAT	- Rats Nest & Pkg Info.
INS	- Instance name prefix
DRC	- Design Rule Checker, Autorouter & N/C drill Info.
SWP	- Pin & Gate Swap Info.

\$ == Required separator between the simulator type and keyword.

XXXXXXXXXXXX == Keyword that is a maximum of 11 characters in length. A blank must follow the keyword. The following keywords are reserved.

RAT\$PKGn	-RAT Package cell name
RAT\$TCNT	-RAT # of devices/package
RAT\$DEVn	-RAT Device n ordered pin numbers
RAT\$VCC1	-RAT Global VCC1 pin #
RAT\$VCC2	-RAT Global VCC2 pin #
RAT\$VCC3	-RAT Global VCC3 pin #
RAT\$VCC4	-RAT Global VCC4 pin #
RAT\$GND1	-RAT Global GND1 pin #
RAT\$GND2	-RAT Global GND2 pin #
BOM\$CO_PN	-BOM company part number
BOM\$MFG_PN	-BOM mfg part number
BOM\$DESCN	-BOM part description
INS\$PRFX	-Instance name prefix
DRC\$HOLE	-N/C DRILL Hole size
DRC\$PAD	-DRC &PROROUTE token circ. for pad size
DRC\$PADC	-DRC &PROROUTE token for circ. pad size
DRC\$PADS	-DRC &PROROUTE token for square padsiz
DRC\$PADR	-DRC &PROROUTE token for rect. pad size

DRC\$VIA	-DRC Feedthrough layers by via pad.
SWP\$PINS	-Schematic symbol, swappable device pins..
SWP\$GATES	-Schematic symbol; swappable device gates.
XXX\$DELAY	-Gate delay -SILOS <rise> <fall> -SIMULOG <prop delay>
XXX\$DEVICE	-SPICE device name -NCA MDL device name
XXX\$GATE	-SILOS gate name -SIMULOG gate name
XXX\$MACRO	-SILOS macro name -SPICE subckt name -SIMULOG macro name -NCA MDL subckt name
SPI\$MODEL	-SPICE model name
XXX\$OTHER	-SILOS comment/text/param -SPICE other parameters -NCAMD L comment/text/param -SIMULOG comment/text/param

NOTE: Additional keywords will be added from time to time.

Some suggested additional uses (documentation purposes only) of the Attributes Data Files are:

DESCRIPTION: 4-2 input nand gate
 CREATED_BY: G.L. MULLIN
 CREATED_DATE: 2-22-1996
 LAST_UPDATE: M TRAN 3-3-98
 SOURCE: RCA CMOS DB SDC-250C pg 190

Any type of information can be added to the Attribute Data Files. Only if the keyword structure explained previously is properly followed will the proper information be extracted by PROCAP.

SPECIFIC USES OF KEYWORDS

SPICE Applications:

The prefix for SPICE is "SPI", and must prefix all keywords to be recognized by PROCAP as being SPICE information.

The following keywords are used for SPICE:

SPI\$DEVICE	Device type
--------------------	-------------

Note: If SPI\$DEVICE keyword is used then one of the two following keywords must also be present (NOT BOTH!).

SPI\$MODEL	Model name (transistors, diodes etc)
-------------------	--------------------------------------

SPI\$VALUE	Value of components (Resistors, Capacitors etc)
-------------------	---

SPI\$VALUE	% (Will cause PROCAP to use the value field specified for the instance in the database)
-------------------	---

Additional keywords -

SPI\$OTHER	Other parameters, values, comments etc)
-------------------	---

SPI\$MACRO	SubCkt name
-------------------	-------------

Reference Guide...

SILOS Applications:

The prefix for SILOS is "SIL", and must prefix all keywords to be recognized by PROCAP as being SILOS information.

The following keywords are used for SILOS:

SIL\$GATE	Gate type, eg .NAND
SIL\$DELAY	Rise_delay Fall_delay
SIL\$MACRO	Macro name
SIL\$OTHER	Comment/text/params

SIMULOG Applications:

The prefix for SIMULOG is "SIM", and must prefix all keywords to be recognized by PROCAP as being SIMULOG information.

The following keywords are used for SIMULOG:

SIM\$GATE	Gate type, eg NAND2
SIM\$DELAY	Prop_delay
SIM\$MACRO	Macro name
SIM\$OTHER	Comment/text/params

NCA MDL Applications:

The prefix for NCA MDL is "MDL", and must prefix all keywords to be recognized by PROCAP as being NCA MDL information.

The following keywords are used for NCA MDL:

MDL\$DEVICE	Device name
MDL\$MACRO	Subckt name
MDL\$OTHER	Comment/text/params

Bill-of-Materials Applications:

The prefix for Bill-of-Materials list is "BOM" and must prefix all keywords to be recognized by PROCAP as being a Bill of Materials information.

The following keywords are used for Bill-of-Materials:

BOM\$CO_PN	Company part number (Up to 12 characters)
BOM\$DESCN	Component description (Up to 19 characters)
BOM\$MFG_PN	Manufacturers part # (Up to 9 characters)

At least one of the above keywords must be specified for a component to be recognized by the BOM processor. Unspecified keyword fields will be left blank in the BOM output list.

Rat's Nest & device packaging Applications:

The prefix for Rat's Nest and device packaging application is "RAT" and must prefix all keywords to be recognized by PROCAD & PROCAP as being such an information.

The following keywords are recognized:

RAT\$PKGn [library]cell_name

Where **n=0,1,2 or 3**. This keyword attached to a schematic symbol instructs PROCAP to use the named cell from the named library as the foot-print (decal) cell to be used for physical layout. If the library qualifier "[" is omitted, PROCAD will attempt to use the currently active library when the RATS NEST command was issued. You may specify 4 choices of decals per schematic symbol. Use the Quick Edit feature of ProCAD to select one of the four possible choices for a placed schematic symbol.

eg: RAT\$PKG0 [PKG]IC14A
 RAT\$PKG1 [PCBLIB]FP14
 RAT\$PKG2 [PCBLIB]FP14X

RAT\$VCCn pin_number (n=1,2,3 or 4)

RAT\$GNDn pin_number (n=1 or 2)

Reference Guide...

These keywords instruct PROCAP to generate global power/ground connections to the pin_numbers specified. Note: Signal name assigned to power/ground traces will be VCC1,VCC2GND1 or GND2.

eg: RAT\$VCC1 14
 RAT\$GND1 7

Note: You may specify multiple pin numbers per token, eg RAT\$GND1 5 7 10 24

RAT\$NET signal p1,p2...

This keyword instructs PROCAP to generate global net connections to pin numbers p1,p2... of the device. Where signal=Netname, p1,p2... corresponding default pins to assign to signal. This keyword is similar to the RAT\$VCCn and RAT\$GNDn above, except that the assigned signal name is not necessarily GND1.... or VCC1..... Use this keyword to assign any signal name to the corresponding pins.

eg: RAT\$NET +5V 6 7 18
 RAT\$NET CLK1 10 12

Note: Multiple keywords with different net names and/or continuation lines are acceptable.

RAT\$TCNT n

This keyword instructs PROCAD/PROCAP to assign n devices to the same package. This keyword is only recognized if the cell is defined as a template. (see EDIT CELL command)

eg: RAT\$TCNT 4

RAT\$DEVx p1 p2 p3
+ p4 p5 p6 ...

This keyword instructs PROCAD/PROCAP to use the specified ordered pin numbers for packagable device X. X=A,B,C ...

Note: You must specify enough RAT\$DEVx keywords to match the number of devices per package as defined in the RAT\$TCNT keyword above. Continuation lines are accepted if the first character on the following line is a plus "+", followed by 1 or more blanks.

eg: **RAT\$TCNT 4**
 RAT\$DEVA 1 2 3
 RAT\$DEVB 4 5 6
 RAT\$DEVC 10 9 8
 RAT\$DEVD 13 12 11

The order in which the pin numbers appear must match the order the pin reference points are defined during EDIT CELL. The above keywords may appear in any order. Template cells must not have more than 20 pins (ie 20 pins per each device in the package).

An example attributes for a 74LS02 cell will be:

```
RAT$PKG0 [PKG]IC14A
RAT$PKG1 [PCBLIB]FP14
RAT$TCNT 4
RAT$DEVA 2 3 1
RAT$DEVB 5 6 4
RAT$DEVC 8 9 10
RAT$DEVD 11 12 13

RAT$VCC1 14
RAT$GND1 7
```

Using FutureNet/OrCAD Compatible Netlist:

PROCAD Rats nest command will accept Future Net compatible NETLIST file. It may be necessary to provide a packaging cross reference file, since the FutureNet Netlist format does not support this feature.

Note: to generate the appropriate futureNet compatible netlist from ORCAD STD, use the following ORCAD netlister command:

NETLIST infile outfile /s futurenet /k /n /p)

Data Format for Package Cross Reference file:

The cross reference file is an ASCII text file that contains the translation between component name and package name. This file can be created using a text editor or word processor in non document mode.

The recognized option characters in the cross reference file are :

"%" character signifies a comment line. All data on that line will be ignored.

":PACK" The ':' character is followed by a package name. The package name PACK is used for all component names between this and next package definition.

"=COMP" The '=' character is followed by a component name. The component name COMP is connected to the last given package name.

"\$PKGLIB" The '\$' character is followed by a package library name. The package library PKGLIB will be used by PROCAD to load the packages from. If no package library name is specified, the default active library will be used by PROCAD when processing the Rats Nest command.

Example of a valid cross reference file is as follows:

%This is a line with comment.

\$PKG

:IC14A

=7432

=7474

=74LS04

:IC16A

=74LS163

:IC24A

=82S181

Instance Name Applications:

The prefix for instance name assignment applications is "INS". PROACD currently recognizes only two tokens for this application:

INS\$VAL nn

This will cause PROCAD to load nn as the default value whenever the cell is input into the database.

INS\$PRFX UU

This is a 1 or 2 character prefix used by PROCAD when assigning instance names to cell placement. If this prefix is not defined for a cell, PROCAD will use a default prefix "U".

PROCAD uses the following format for assigning instance names to cells:

UUnnnX

UU is a 1 or 2 character prefix defined above. (The default is "U")

nnn is up to 3 digits number from 1 to 999, leading zeros suppressed.

X is a suffix. X is normally left blank for non template cell. For template cells, X is an alpha character "A - Z", depending on the number of template cells per package (RAT\$TCNT).

NOTE: You must follow these conventions when assigning your own names to cell instances in order for the PACK/R command to work correctly.

INS\$PRFX ?

If the above token is used for a cell, PROCAD will suppress the generation of instance name for that cell. This is useful for feedthrough pad cells, board outline cells etc.

Device Comments and Notes:

DEV\$CREATED Name

Reference Guide...

Where Name is the name of the creator of the cell. Name is limited to 16 characters. spaces are allowed.

DEV\$DATE Datestring.

Where Datestring = date the cell was created. Datestring is limited to 16 characters. Spaces are allowed.

DEV\$MODIFIED Datestring.

Where Datestring = date the cell was last modified. Datestring is limited to 16 characters. Spaces are allowed.

Design Rule Check Applications:

The Design Rule Check tokens concern the actual physical design characteristics of the Printed Circuit Board layout, as well as the connectivity of the finished PCB.

DRC\$PAD, DRC\$PADC, DRC\$PADS DRC\$PADR are the keywords for physical spacing characteristics of the PCB layout. These tokens are used as follows:

DRC\$PAD XX nn {pin numbers}
DRC\$PADC XX nn {pin numbers}
DRC\$PADS XS nn {pin numbers}
DRC\$PADR XW XH nn {pin numbers}

XX represents the diameter of the pad in database units.

XS represents the sides of the square in database units.

XW represents the Width of a rectangular pad in database units.

XH represents the Height of a rectangular pad in database units.

nn represents the layer number which that pad is on.

{pin numbers} are pin (I/O) numbers this token is valid on. If omitted, all pins not explicitly defined with a DRC\$PADx token will be assumed. Note: you must include the curly brackets when defining pin numbers. They may be continued on the next line (the first non blank character must be a plus "+" followed by a blank).

These tokens are also for use in physical devices because the DRC program does not contain any nesting capability. Therefore a sixteen pin IC needs to have the DRC\$PADx token for clearance at all its connection points, but it only needs that token stated once ... not sixteen times.

So a feedthrough which is 30 mils in diameter on padmaster layer 5 would have the data token:

```
DRC$PADR 30 30 5 {1}
DRC$PAD 30 5 {all}
```

or

```
DRC$PADC 30 5 {all}
```

While a surface mount rectangular pad on layer 23 of 10 mils wide by 15 mils high will have the data token:

```
DRC$PADR 10 15 23 {all}
```

A 40 pin DIP package with 60 mil pads at all 40 pin locations on layer 5 would have the token DRC\$PAD 60 5, only once.

Note also the following:

1. If pin number(s) are omitted, ProCAD assumes the definition is for all the pins
2. You may use the ALL clause or 5 in the layer(s) field to denote all thru layers
3. You may use the {ALL} clause in the pin number(s) field to denote all pin numbers that are not explicitly defined. Pin numbers must be the part of the last DRC\$PADx token field.
4. You may use multiple DRC\$PADx tokens to define various pad styles for a part

Examples:

```
DRC$PADR 50 50 5 {1}  
DRC$PADC 50 ALL {2-14}  
  
DRC$PADS 60 23 {1-3 8  
+ 14 28-40}  
DRC$PADR 60 60 23 {ALL}
```

The **DRC/V** command (**[Check]Show DRC\$PADx graphics** menu item) will display the graphical interpretation of all DRC\$PADx keywords used in the database.

DRC\$VIA is the keyword for connectivity in a PCB layout. Use this token to make a logical connection between traces on different layers of a PCB through a pad or feedthrough, as follows:

DRC\$VIA n1 n2 n3

nn represents all of the layers which this particular device connect to.

For instance a trace on the component side (layer 20) connects to a trace on the solder side (layer 23) through a 30 mil feedthrough. This feedthrough would require the token DRC\$VIA 20 23.

The DRC\$VIA token, like the DRC\$PAD token, can be used to encompass the connectivity of an entire device such as a 24 pin DIP, provided all pins on the dip connect to the same layers.

The connectivity of the DRC\$VIA token is not correctly reflected with the resolve command inside the PROCAD editor. SHOW NODE, or SHOW SIGNAL, will show connectivity in a PCB trace only on the layer specified. PROCAP uses the DRC\$VIA token to generate an accurate netlist of the PCB layout, and uses the token when generating a list of differences between the schematic and the PCB with the BACK ANNOTATION command.

PROROUTE (PCB Autorouter) Applications:

PROROUTE PCB Autorouter only recognizes the DRC\$PAD token as described above.

DRC\$PADx Padsizes Layer {pin numbers}

Padsizes is the diagonal size of the pad in database units, and Layer is the valid layer where this pad is found. If you specify layer 5 or ALL, PROROUTE will assume the pad to be a drill-through type connecting all trace layers.

Note: PROROUTE version 1.0B and earlier versions does not support square or rectangular pads. You must approximate pad size using DRC\$PADC or DRC\$PAD tokens. See also the discussion on DRC\$PADx keywords under the "Design Rule Check Applications" above.

PRODRILL (N/C Drill Utility) Applications:

PRODRILL N/C drill program only recognizes the DRC\$HOLE token as described above.

DRC\$HOLE Hole-size n1 n2 n3

Holesizes is the diagonal size of the hole in database units, and n1 n2 n3 etc. are the corresponding pins. Continuation lines can be used with this token, and multiple tokens with different hole sizes and pins can also be specified for any given cell.

The **DRC/H** command ([Check]Show **DRC\$HOLE graphics** menu item) will display the graphical interpretation of all DRC\$HOLE keywords used in the database.

Pin & Gate Swap Applications:

These keyword defines which pins or gates within a schematic symbol may be swapped (electrically equivalent).

The data attribute keyword (token) for Pin Swap is:

SWP\$PINS n1 n2 ...

Reference Guide...

Where **n1 n2 ...** are pins of the symbol that can be swapped. If the symbol is a template cell, **n1 n2 ...** are the actual symbol I/O numbers. For non template cells, I/O numbers are the same as pin numbers.

The data attribute keyword (token) for Gate Swap is:

SWP\$GATES A,B,...

Where **A, B,..** are the swapable gates of the package. Note that only packaged cells (template cells) can be swapped within the same physical package.

Example:

The swap keyword for a 7400 device will be:

SWP\$PINS 1 2
SWP\$GATES A B C D

The above example permits ProCAD to swap the two input pins of the NAND gates, and be able to swap any of the 4 NAND gates within the package.

NOTE: You may use as many **SWP\$PINS** and **SWP\$GATES** tokens as are needed for your different groups. You may also use continuation lines ("+" as the first character on a succeeding line) for the same groupings. See also the SWAP command for more information.

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Standard Menus

A *ppendix*

B

APPENDIX B

STANDARD MENU STRUCTURE

INTRODUCTION

The menus available for use with PROCAD for Windows are of the "pull down" variety. The menu items and placement adheres to the strict standard defined for all windows based applications. If you are familiar with windows, you should find the layout of menu items already familiar.

The following is a listing and description of the menu items and associated commands:

(1) File	<i>Menu for file and I/O operations</i>
New...	<i>Open a new database file</i>
Open...	<i>Open an existing database file</i>
Close	<i>Close current database</i>
Save	<i>Save current database file to disk</i>
Save As	<i>Save current database to a new file</i>
Save All	<i>Save all opened database files to disk</i>
Run...	<i>Load and execute a ProCAD or third party Utility Program</i>
Load Project...	<i>Load a project file</i>
Project Mgr...	<i>Invoke the project manager</i>
Import...	<i>Import ProCAD & Other CAD ASCII files</i>
Export...	<i>Export ProCAD & Other CAD ASCII files</i>
Pack/Rnmbr	<i>Remove deleted entities from database</i>
Annotate	<i>Fwd/Back annotate instance names and/or values(pcb<>sch)</i>
Print or Plot	<i>Generate plot ctrl file. Execute Proplot</i>
Gerber plot	<i>Generate Gerber control file. Execute Gerplot</i>
N/C Drill	<i>Load and execute ProDrill Excellon Utility</i>
Reports...	<i>Generate statistical report file</i>
DBF&LIB path	<i>Set database & library search path</i>
Execute Cmnd	<i>Execute a macro command file</i>
Exit	<i>Exit ProCAD.</i>

- (i) **Run...** *Submenu for executing ProCAD Utility & Third party pgms*
 - Archive** *Execute the DBF/LIB Archiver Utility*
 - NetChk** *Execute NetChk third party utility*
- (ii) **Print or Plot** *Submenu for generating plot control file or invoking ProPlot*
 - Setup Ctrl File** *XXX.CTR file generated*
 - Generate Hardcopy** *Invoke ProPlot utility*
- (iii) **Gerber Plot** *Submenu for generating Gerber ctrl file or invoking GerPlot*
 - Setup Ctrl File** *XXX.GCF file generated*
 - Generate Plot file** *Invoke GerPlot utility*
- (iv) **Report...** *Submenu for generating statistical reports*
 - Nets in Database** *Generate Nets Names Used Report*
 - Components in Database** *Generate Comps Used Report*
 - Layers Used in DBF** *Generate Layers Used Report*
 - Netlist & Bill of Materials** *Generate Reports Using ProCAP Utility*
- (2) **Edit** *Menu for editing selected item(s)*
 - Undo** *Undo the effect of last edit(s)*
 - Repeat** *Repeat last command executed*
 - Copy->Clipboard** *Copy an area to clipboard.*
 - Copy->temp** *Copy selected item(s) to temp cell*
 - Paste<-temp** *Paste temp cell into database*
 - Copy** *Copy selected item(s)*
 - Rotate** *Rotate selected item(s)*
 - Move** *Move selected item(s), Segment or Vertex*
 - Delete** *Delete selected item(s), Segment or Vertex*
 - Undelete** *Recover deleted last or all entitie(s)*
 - Select** *Select item(s) for editing*
 - De-Select** *Un-select item(s)*
 - Find** *Find and select a device or Net for editing*
 - Search & Replace...** *Search and Replace user specified entity fields (globally)*
 - Re-Route** *Re-Route a selected trace/wire*
 - Add Vertex** *Add a vertex to selected trace*
 - Mirror_X** *Mirror selected item(s) about X-Axis*
 - Mirror_Y** *Mirror selected item(s) about Y-Axis*
 - Info Summary** *Display vital info. on selected item(s)*

- (i) **More...** *Second level menu of editing commands*
Add to Select *Add entity to select environment*
Hold select *Hold selected entities in select mode*
Change Layer *Move selected item(s) to a new layer*
Changewidth *Change trace width or text hgt or Afill clearance of selected item(s)*

Change Seg Lyr *Move selected trace segment to a new layer*
Change Seg wid *Change select trace segment width*
Step&Repeat *Make multiple copies of selected item(s)*
Stretch *Stretch a surface of a selected trace*
Add 45° Corner *Add 45° corners to selected orthogonal trace(s)*
Cell Form *Make a cell composed of selected item(s)*
Explode Cell *Remove 1 level of hierarchy of selected cell(s)*
ChangMap *Change placement layer map group environment*
Replace *Replace selected cell(s) or text*
Swap(P/G)... *Swap Pins or Gates of selected cell*
Scale Entity *Contract or Expand selected item(s).*

- (3) **View** *Menu of Screen view operations (zoom, regen etc).*
Pan *Pan the screen per user selected point*
Refresh *Regenerate screen*
Zoom All *Zoom design to fit all items on screen*
Zoom Last *Restore last (previous) screen*
Zoom In *Zoom IN for tighter views*
Zoom Out *Zoom OUT for wider view*
Zoom X *Zoom screen IN or OUT per user defined*
SetView *Zoom IN per user defined window*
Scaled Zoom *Scale screen IN or OUT*
View Port *Define or turn off a view port*
Display Grids *Submenu of selectable display & ghost grids*
Artwork views *Submenu for viewing artwork layer combinations*
Preference... *Set user working environment preference*

- (i) **Artwork Views** *Submenu for viewing artwork layer combination*
CompSide *View component side (layers: 5,13,15,20)*
SoldSide *View solder side (lyrs: 5,13,16,23)*
SilkScrn *View silkscreen (lyrs: 9,10,13)*
SoldMask *View Soldermask (lyrs: 6,13)*
DrilMstr *View drill master (lyrs: 7,13)*

- | | |
|-----------------|---|
| FabDrwng | <i>View fab. dwg (layers: 7,8,11,12,13,14,19)</i> |
| VolPlane | <i>View voltage plane (lyrs: 13,21)</i> |
| GndPlane | <i>View ground plane (lyrs: 13,22)</i> |
| AsmbDrwg | <i>View Asmb dwg (layers: 3,4,10,13,15,19)</i> |
| All | <i>View all layers (1-99)</i> |
- (ii) **Display Grids** *Submenu of selectable display grids*
- | | |
|--------------------|---|
| ON OFF | <i>Toggle visible grid</i> |
| 10 | <i>Set 10 units visible grid</i> |
| 20 | <i>Set 20 units visible grid</i> |
| 25 | <i>Set 25 units visible grid</i> |
| 50 | <i>Set 50 units visible grid</i> |
| 100 | <i>Set 100 units visible grid</i> |
| User Select | <i>Set visible grid (1-32767) units</i> |
- (4) **Place** *Menu for component placement into DBF*
- | | |
|------------------------|--|
| Get_Cell | <i>Get a cell in real time from the library</i> |
| RatsNest | <i>Place components & traces from a Rats Nest file</i> |
| Step&Rpt | <i>Make multiple copies of selected item(s)</i> |
| Text | <i>Place vectorized text (Ht, Angl & Origin pt)</i> |
| Arc | <i>Place non electrical arcs</i> |
| Copper Plane | <i>Place auto copper plane around traces & pads.</i> |
| Voltage/gnd tie | <i>Place ties to a copper plane</i> |
| Vias/conn dots | <i>Place vias or connection dots.</i> |
| Surface Mount | <i>Submenu of surface mount operations.</i> |
- (i) **Vias/Conn Dots** *Submenu of selectable via sizes & connection dots*
- | | |
|--------------------|---|
| 30 | <i>Place 30 units via in a design</i> |
| 50 | <i>Place 50 units via in a design</i> |
| 75 | <i>Place 75 units via in a design</i> |
| 100 | <i>Place 100 units via in a design</i> |
| 125 | <i>Place 125 units via in a design</i> |
| 200 | <i>Place 200 units via in a design</i> |
| User Select | <i>Place user selected via cell in a design</i> |
| ConnDot | <i>Place a conn dot in a schematic design</i> |
- (ii) **Surface Mount** *Submenu of SMT related functions*
- | | |
|-----------------------|---------------------------------------|
| Move to Top | <i>Move SMT device to Top side</i> |
| Move to Bottom | <i>Move SMT device to Bottom side</i> |
| SetMapGroup | <i>Set or change layer mappings</i> |
| ShowMap Gp | <i>Display current layer mapping</i> |

-
- (5) **Route** *Menu for routing wires & traces into design*
Wire *Place wires/trace segments in database*
Stitch *Stitch a trace into the database*
Curved Wires *Place curved traces in database (Elec. arcs)*
Auto Route... *Invoke ProROUTE autorouter utility*
Digitiz Mode.. *Select digitizing mode (straight, ortho etc)*
Track Sizes *Submenu of selectable trace sizes*
- (i) **Track Sizes** *Submenu of selectable trace sizes*
1 *Select 1 unit trace width for routing*
5 *Select 5 units trace width for subsequent routing*
10 *Select 10 units trace width for subsequent routing*
20 *Select 20 units trace width for subsequent routing*
25 *Select 25 units trace width for subsequent routing*
50 *Select 50 units trace width for subsequent routing*
100 *Select 100 units trace width for subsequent routing*
Variable *Select variable trace width for subsequent routing*
- (6) **Check** *Submenu of design rules check functions*
Design rule Check *DRC check (Pd-Pd, Pd-Tr, Tr-Tr etc)*
Resolve *Resolve design connectivity & netlist*
Cross Probe... *Cross probe between PCB & SCH databases*
Measure Distance *Measure point-to-point distance*
Clear Error *Clear all flagged DRC errors*
Reload Netlist *Reload netlist file*
Histogram *Generate trace congestion histogram.*
Density Map *Generate color coded trace density map.*
Show Node *Highlite all nets connected to a node*
Show Signal *Highlite nets of the specified name*
Show Ref *Show all I/O & Ref. origin of a cell*
Show DRC\$PADx graph *Show graphical rep. of DRC\$PADx's used in design*
Show Conn. *Hi-lite all connections to a device.*
- (7) **Layer** *Menu for controlling layer attributes*
Layer Attribute *Set attribute of Filled & Visibility*
Top Side *Select top side trace layer (20)*
Bottom Side *Select bottom side trace layer (23)*
Via Layer *Select via placement layer (2)*
User Select *Select active layer (1-98)*
Set Layer Color *Set layer color, pattern & line style*
Set AutoLayers *Set Auto entity placement layers*
-

- (8) **Library** *Menu for Library management functions*
- Edit/Create Cell** *Edit or Create a cell (component).*
 - Padstack Wizard...** *Auto create a padstack cell.*
 - Import OrCAD library** *Import OrCAD ASCII library.*
 - Library Manager** *Library management utility screen*
 - Change Library** *Change the default working library*
 - Cell Attribute** *Show attribute data of selected cell*
 - Cell Heirarchy** *Show hierarchy of a selected cell*
- (9) **Other...** *Menu of misc. functions*
- Status Information..** *Information on current status*
 - Select Filter...** *Set entity or block selection mask*
 - Toggles...** *Invoke a dialog of toggleable vars*
 - Constants...** *Invoke a dialog of settable constants*
 - Quick Edit...** *Invoke quick edit dialog box*
 - Switch mode(pcb/sch)...** *Switch between PCB & Schematic operation modes*
 - Display Grids** *Invoke a submenu of display grids*
 - Modify** *submenu for instance mod. functions*
- (i) **Modify** *Submenu for changing instance and signal attributes.*
- Instance Name** *Modify/ Change instance name attribute.*
 - Instance Value** *Modify/ Change instance value attribute.*
 - Signal Name** *Modify/ Change signal name attributes*
- (10) **Window** *Menu of window control functions.*
- Tile** *Tile opened databases on the screen*
 - Cascade** *Cascade opened databases on the screen*
 - Arrange Icon** *Arrange opened dbf icons on the screen*
 - Close** *Close current database file & window*
 - Close All** *Close all opened databases & windows*
 - Drawing Tool** *Toggle Drawing tools dialog box*
 - Utility Tool** *Toggle Utility tools dialog box*
- (11) **Help** *Menu of windows help functions.*
- About** *Information about current version of ProCAD*

LAYERS.DAT file format

A *ppendix*

C

APPENDIX C

LAYERS.DAT FILE FORMAT

INTRODUCTION:

The LAYERS.DAT file is the ASCII data file which contains the names of all 99 layers available in PROCAD for both the Schematic & Printed Circuit Board (PCB) modes of operation.

SYNTAX: **SCH_Lyr_Name** **PCB_Lyr_Name**

Each line in the LAYERS.DAT file corresponds to a layer number. The first line represents layer 1, second line = layer 2 etc. The first field in a line corresponds to the Schematic Layer Name, and the second field corresponds to the PCB layer name. Fields are separated by one or more blank characters. Blank lines are significant, and will be treated as a blank layer name.

Example:

COMPS	PCBWRK
WIRES	TOPSIDE
LINES	BOTSIDE
TEXT	SLKSCRN

The first line of the above example will assign the names "COMPS" & "PCBWRK" to layer #1 of Schematic and PCB design modes respectively.

Layer names are limited to a maximum of six characters. You should define all 99 layers (99 lines in LAYERS.DAT file). Undefined layer names are default to XXXXXX.

Note: If this file does not exist in PROCAD sub-directory on your hard disk, PROCAD will use the built in default layer names.

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TEXTFONT.TXT file format

A *ppendix*

D

APPENDIX D

TEXTFONT FILE FORMAT

INTRODUCTION:

The TEXTFONT.TXT file is the ASCII description of characters which will be used by PROCAD when generating text by the INPUT/TEXT mode.

When PROCAD or PROPLOT are executed, these programs look for a file named TEXTFONT.BIN (the binary equivalent file). If this binary file does not exist, then one is created from the TEXTFONT.TXT file. This .TXT file is fully user defineable, allowing the individual users of PROCAD to create unique character sets for designing.

In order to understand the TEXTFONT file, you must imagine (or better still draw on paper) a simple grid structure of ten units in the X and Y directions. This grid should start at (0,0) and move to (9,9). This grid represents the relative area within which one character will exist in the database. When the physical height of the text is defined (100 mils for instance), then the actual size of the grid is defined for those characters.

Using grid coordinates from this simple ten-by-ten grid, you can now generate your own TEXTFONT.TXT file to create a special character set. First there are three characters which have special functions in the TEXTFONT file.

\$ - The dollar sign signifies a comment. Anything which appears on the same line after this character has no special significance to the TEXTFONT.BIN file.

98 - The number 98 means pen-up to PROPLOT, or end of text vector to PROCAD. This simply means end this portion of the character, and begin the next portion of the character.

99 - The number 99 means end of line, or end of character string. Use this to flag the end of a character line, and begin another.

Now take a look at a character in the TEXTFONT.TXT file.

18,17,98,28,27,99, \$"

The two digit numbers in this line actually represent X-Y coordinates in the simple grid. The number 18 is actually (1,8) and is the starting point of the first line. The number 17 is actually (1,7) and 98 means pen-up, or end of text vector. The numbers 28 and 27 become (2,8) and (2,7), and 99 is end of line. This simple set of numbers actually translates into two short vectors which when drawn in PROCAD represent the quotation character ("). You will notice that because 98 and 99 are reserved characters, you will not be able to create any characters which are drawn out to the 9 coordinate in the X axis. This is practical however, since the characters are generated with their imaginary grids flush against each other. This means that the 9th X coordinate of the first character is the 0 X coordinate of the second...

When creating new characters in the TEXTFONT file you should take care to place all characters on a base line of 2 in the Y axis. This will allow you to create characters which go below the base line such as lower case g,p,j, q, and y.

The textfont file allows you to create unique characters for specific applications, to be stored in the keyboard and used with PROCAD. If you only use upper case letters in documentation, then you can use all the lower case positions for special characters.

When creating the character to store in an allotted key on the keyboard, the following rule should be maintained: The first character in the TEXTFONT.TXT file is assigned to ASCII value 32 on the keyboard. All additional characters are assigned sequentially numbered ASCII values, the last character is assigned the value of 126. For a complete understanding of this ASCII value key assignment look at an ASCII value table.

ProCAD ASCII Interface Format

A ***ppendix***

E

APPENDIX E

PROCAD ASCII INTERFACE FORMAT

INTRODUCTION:

This appendix contains a description of the ASCII format for ProCAD Database. This is the format of data generated when **[File]Export...** command is used, and selecting ProCAD format. Users desiring to write programs that interface or modify ProCAD database and libraries may use the information in this appendix for that purpose.

\$		Comment line
PROCAD	Vnn.nn	Header character Vnn.nn = Database version number
BEGINVAR		Begin general variables section
DBUNITS nn		Database units (0=mils, 1=mm, 2=INCHS, 3=MICRONS, 4=CM)
DBMODE	S/P	Database mode S=Schematic, P=PCB
DBDATE	string	Database creation date stamp (DD-MMM-YY HH-MM-SS).
LIBCDATE	string	Library creation date stamp (DD-MMM-YY HH-MM-SS).
LIBMDATE	string	Library last modified date stamp (DD-MMM-YY HH-MM-SS).
ENDVAR		End of variable definition section
\$		
DEFINE [LIBRARY]CELLNAME	XORG YORG XREF YREF XVAL YVAL	
IOCNT IONMHGT ATRCNT CTYPE CUNIT		
\$	[LIBRARY]	Library name (up to 8 chars) enclosed in square brackets
\$	CELLNAME	Cell name (up to 8 alpha numeric characters)
\$	XORG,YORG	Cell definition origin coordinate
\$	XREF,YREF	Default cell instance name coordinate
\$	XVAL,YVAL	Default cell value name coordinate
\$	IOCNT	Number of I/O connection points (0-512)
\$	IONMHGT	Automatic assigned pin number name height (-1 = Non template cell)
\$	ATRCNT	Number of attribute lines
\$	CTYPE	Cell type (REGULAR or TEMPLATE)
\$	CUNIT	Cell units (0=< DBU, 1=Mils, 2=Inchs, 3=MM/10, 4=MM, 5=Microns)

XLOC1 YLOC1 XLOC2 YLOC2 PTYPE PNAME

\$ XLOC1,YLOC1 pin 1 location,
 \$ XLOC2 YLOC2 Prin ref loc (Template cells, 0 0
 \$ for Regular cells)
 \$ PTYPE (0=output, 1=input, 2=bi-dir)
 \$ PNAME pin name

.....

XLOC1n YLOC1n XLOC2n YLOC2n PTYPE PNAME [Note: n=LOCNT]

BEGINATR Begin attribute data definition section
line1 Attr. line 1 (upto 30 alpha numeric data)

.....

.....

lineN Attribute line N (N=ATRCNT)

ENDATR End of attribute data definition section

BEGINDATA Begin cell graphical & Net data
 Cell Graphical & Net data follows

ENDDATA End of cell graphical & Net data

ENDDDEF End of Cell definition section

\$

\$ PROCAD GRAHICAL & NET DATA FORMAT

\$

ENTITIES Begin Database Graphical & Net data

\$

***LINE* LINEWID VCNT LAYER**

\$ *LINE* Non-electrical polyline header identifier
 \$ LINEWIDLine width
 \$ VCNT Number of vertices (corners)
 \$ LAYER Placement layer number (1-98)

XLOC1 YLOC1 Vertex 1 coordinate

...

XLOCn YLOCn Vertex n coordinate (N=VCNT)

\$

***WIRE* LINEWID VCNT LAYER SIGNAME STATUS XLOC YLOC SIGHGT
SIGANG SIGSTAT MIRROR**

\$ *WIRE* Electrical wire (trace) header identifier
 \$ LINEWID Line width
 \$ VERTICES Number of vertices (corners)
 \$ LAYER Placement layer number (1-98)
 \$ SIGNAME Signal name (up to 12 characters)
 \$ STATUS Signal status (0=Program assigned,
 \$ Signal Name format =N0000QQ),
 \$ 1=User assigned)
 \$ XLOC,YLOC Signal name display loc.
 \$ SIGHGT Signal name display height & width
 \$ MOD(SIGHGT,512)=hgt, SIGHGT/512=wid
 \$ SIGANG Signal name display angle
 \$ SIGSTAT Signal name display flag
 \$ (0 <= OFF, 1=ON)
 \$ MIRROR Mirror status (X=Xreflection, Y=Yreflection
 \$ N=None)
 \$ XLOC1 YLOC1 Vertex 1 coordinate
 ...
 \$ XLOCn YLOCn Vertex n coordinate (N=VCNT)

***CIRCLE* LAYER CIRWID XC YC RADIUS**

\$ *CIRCLE* Circle header identifier
 \$ LAYER Placement layer number (1-98)
 \$ CIRWID Circle width if donut, 1=Regular circle,
 \$ 0=point
 \$ XC YC Circle center coordinate
 \$ RADIUS Circle Radius.
 \$

***RECTANGLE* LAYER XLOC1 YLOC1 XLOC2 YLOC2**

\$ *RECTANGLE* Rectangle header identifier
 \$ LAYER Placement layer number (1-98)
 \$ XLOC1 YLOC1 First rectangle corner coord.
 \$ XLOC2 YLOC2 Second rectangle corner coord.
 \$

***TEXT* LAYER XLOC YLOC HIGHT WID ANGLE MIRROR**

text line		
\$	*TEXT*	Text header identifier
\$	LAYER	Placement layer number (1-98)
\$	XLOC YLOC	Text lower left justification coordinate
\$	HIGHT	Text Height
\$	WID	Text width
\$	ANGLE	Text angle
\$	MIRROR	Mirror status (X=Xreflection, Y=Yreflection N=None)
\$		

***ARC* LAYER XC YC XS YS XE YE**

\$	*ARC*	Arc header identifier
\$	LAYER	Placement layer number (1-98)
\$	XC YC	Arc center coordinate
\$	XS YS	Arc start coordinate
\$	XE YE	Arc end coordinate
\$		

CELL* INAME [LIB]CNAME LAYER XLOC YLOC SCLE ANG MAP MIRROR*REFX REFY RHGT RANG VNAME VALX VALY VHGT VANG PKGINDEX**

\$	*CELL*	Cell header identifier
\$	INAME	Instance name (up to 6 chars ? = non
\$		defined eg. via cell)
\$	LIB	Source library name enclosed in square
\$		brackets
\$	CNAME	Cell name (Up to 8 characters)
\$	LAYER	Placement layer number (1-98)
\$	XLOC YLOC	Placement location
\$	SCLE	Placement scale (1=Normal)
\$	ANG	Placement angle
\$	MAP	Layer mapping group number
\$		(0=Un-mapped)
\$	MIRROR	Mirror status (X=Xreflection,
\$		Y=Yreflection, N=None)
\$		
\$	REFX REFY	Instance name reference coordinate
\$	RHGT	Instance name text height & width
\$		MOD(REFHGT,512)=hgt, REFHGT/512=wid
\$	RANG	Instance name text angle
\$	VNAME	Instance value string (up to 8 chars. ? =
\$		non defined)
\$	VALX VALY	Instance value reference coordinate
\$	VHGT	Instance value text height
\$		MOD(VHGT,512)=hgt, VHGT/512=wid
\$	VANG	Instance value text angle
\$	PKGINDEX	Footprint index (0=RAT\$PKG0[default],
\$		1=RAT\$PKG13=RAT\$PKG3)
\$		
\$	*END*	End of file

NOTES:

- (1) ***A cell must be defined before it is referenced in a drawing or cell definition section.***

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Layer Definitions & Standard Libraries

A *ppendix*

F

APPENDIX F

LAYER ASSIGNMENTS

INTRODUCTION:

This appendix contains a description of the standard layering convention used by ICS in creating the standard libraries provided with PROCAD package. The layering conventions for both PCB layout and schematic design are included in this section.

This is not meant to be used as a strict guide to live by, but merely a reflection of the ones which are recommended. Feel free to create your own layering convention, but be careful to maintain some compatibility with the existing libraries.

Some of the 99 layers which PROCAD supports are special reserved layers. Layer 99 is reserved for the select environment, and layer 49 is used by the RESOLVE command to flag floating inputs. Layers 31 through 34 are used by the Design Rule Checker (DRC) program, and layer 49 is also used by the BACK ANNOTATION program. Do not use any of those layers just mentioned because all of those layers are cleared (every line is deleted) by PROCAD in standard operation.

Some layers are used for Hierarchical design, and should only be used for that purpose (layers 40 - 48). The rest are clear for you to design with, but it is strongly recommended you maintain your designs below layer 30 to accomodate future revisions of the software.

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 BOTTTOM	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]

ARTWORK MASTERS -

The following layer combinations provide the artwork necessary to produce a Printed Circuit Board Documentation Package. Nesting level must be set to 2 or greater (See SET NESTING_LEVEL command).

For viewing on the display, or to plot reference (and pin numbers for template cells) designators the working layers 1 & 2 MUST also be included in the visible/plot layer table.

COMPONENT SIDE:

COMBINE LAYERS 5,13,16,20,.....

5	-Padmaster
13	-Targets & Marks
16	-Connector Fingers
20	-Comp_Side Traces

CIRCUIT or SOLDER SIDE:

COMBINE LAYERS 5,13,16,23,.....

5	-Padmaster
13	-Targets & Marks
16	-Connector Fingers
23	-Solder_Side Traces

VOLTAGE PLANE:

COMBINE LAYERS 13,21,.....

13	-Targets & Marks
21	-Layer 2 VCC

GROUND PLANE:

COMBINE LAYERS 13,22....

13	-Targets & Marks
22	-Layer 3 GND

SOLDERMASK:

COMBINE LAYERS 6,13.....

6	-Soldermask
13	-Targets & Marks

DRILL MASTER:

COMBINE LAYERS 7,13.....

7	-Drill Targets
13	-Targets & Marks

SILKSCREEN:

COMBINE LAYERS 9,10,13.....

9	-Silkscreen Legend
10	-Silkscreen Text
13	-Targets & Marks

INNERLAYER n:

COMBINE LAYERS 5,13,n(24..29)....

5	-Padmaster
13	-Targets & Marks
n	-Appropriate

FABRICATION DWG: (PEN PLOT ONLY)

COMBINE LAYERS 7,8,11,12,13,14,19....

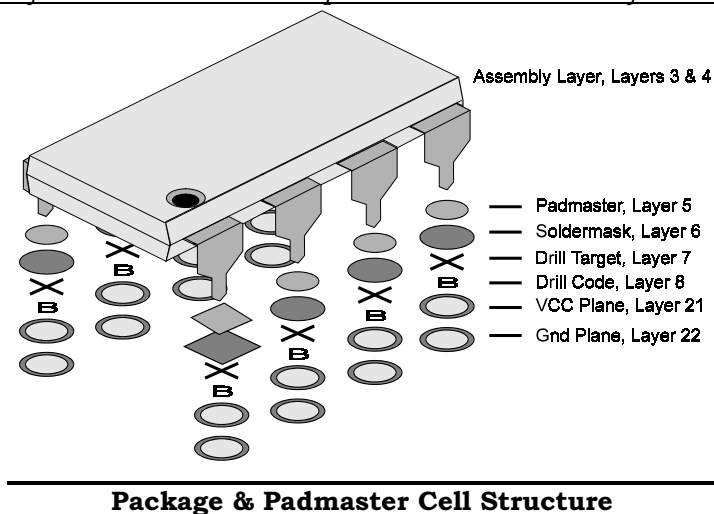
- 7 -Drill Targets
- 8 -Drill Codes
- 11 -Card Outline
- 12 -Card Dimensions
- 13 -Targets & Marks
- 14 -Fabrication Notes
- 19 -DWG Border/Logo

ASSEMBLY DWG: (PEN PLOT ONLY)

COMBINE LAYERS 3,4,10,13,15,19.....

- 3 -Assembly_1
- 4 -Assembly_2
- 10 -Silkscreen Text
- 13 -Targets & Marks
- 15 -Assembly Notes
- 19 -DWG Border/Logo

NOTE: The macro cmd file PKG.CMD will set up the above color codes for all layers.



Quick Reference Guide...

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]

Combine layers 1 - 29 to generate a pen plot drawing of your finished schematic. Nesting level must be set to 1.

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