PC Board Layout
Tools 386+
User’s Guide
PC Board Layout

Tools 386+

User's Guide
Chapter 3: Transferring from schematic to layout

The files you need ................................................................. 15
Before you begin ................................................................. 16
Keys ..................................................................................... 16
  <Enter> ............................................................................... 16
  <Ctrl> ............................................................................... 16
  Other keys ......................................................................... 16
Mouse basics ................................................................. 16
Keyboard input ................................................................. 17
Operating system command prompt ....................................... 17
Commands ........................................................................... 17
Filenames ............................................................................ 17
Designs .............................................................................. 18
Running the ESP design environment ...................................... 18
Changing to the TUTOR design ............................................. 19
  Changing the startup design .............................................. 20
Configuring DRAFT and Schematic Design Tools .................... 22
  Configuring DRAFT .......................................................... 22
  Configuring Schematic Design Tools .................................. 23
Displaying the schematic ...................................................... 24
Annotating the schematic ..................................................... 25
Viewing key fields ............................................................. 27
Creating a stuff file ............................................................ 28
Updating field contents ...................................................... 30
Checking design integrity ................................................... 31
  Configuring Cleanup Schematic ....................................... 31
  Configuring Cross Reference Parts .................................... 32
  Configuring Check Electrical Rules .................................... 33
  Running Check Design Integrity ....................................... 34
Chapter 4: Introducing Edit Layout (continued)

Setting up Edit Layout conditions ................................................................. 56
  The SET command .................................................................................. 56
  Layer ......................................................................................................... 57
  Current Settings ..................................................................................... 59
Selecting a layer ........................................................................................... 60
  LAYER ...................................................................................................... 60
  / OTHER ................................................................................................... 61
  + LAYER and - LAYER .......................................................................... 61
  * LAYER .................................................................................................. 61
Changing your view of the layout ................................................................. 62
  Zoom ...................................................................................................... 62
  Setting a zoom scale ............................................................................. 63
  Selecting a zoom window ...................................................................... 63
  WINDOW ZOOM .................................................................................... 63
Pointer movement resolution ....................................................................... 64
Using bookmarks .......................................................................................... 65
  Creating a bookmark ............................................................................ 65
  Jumping to a bookmark ......................................................................... 66
  Deleting a bookmark ............................................................................ 67
Changing the origin ...................................................................................... 68
Setting grid options ..................................................................................... 69
  Setting a grid size ................................................................................ 69
  Setting a grid divisor ........................................................................... 70
  Disabling the snap grid ....................................................................... 70
  Changing the grid color ....................................................................... 70
Saving and backing up the board file ......................................................... 71
  Updating the file .................................................................................. 71
  Saving configurations ........................................................................... 71
  Writing to another filename ............................................................... 72
  Copying a file ..................................................................................... 73
  Renaming a file ................................................................................... 73
  Deleting a file ..................................................................................... 74
Suspend to System ......................................................................................... 75
Chapter 4: Introducing Edit Layout (continued)

Macros ............................................................................................................. 76
  Creating the first macro .................................................................................. 76
  Creating the second macro ........................................................................... 79
  Running the macros ...................................................................................... 80
  Saving all macros to a file ........................................................................... 80
  Exporting a macro to a file ........................................................................... 83
  Deleting a macro from the disk ................................................................... 84
  Deleting a macro from Edit Layout ............................................................... 84
  Deleting all macros from Edit Layout ........................................................... 84
  Loading a macro from disk ........................................................................... 85
  Running a defined macro .............................................................................. 85

Summary .......................................................................................................... 85

Chapter 5: Creating board modules

About modules ................................................................................................. 87

About the library editor ................................................................................... 88
  Selecting the library editor ........................................................................... 88

Working with module libraries ......................................................................... 89
  Copying the DEMO library ........................................................................... 89
  Copying and getting a module ....................................................................... 89
  Updating the library file ................................................................................. 91
  Renaming a module ....................................................................................... 92

Displaying module information ......................................................................... 93
  INQUIRE ...................................................................................................... 93
  VERBOSE INQUIRE .................................................................................. 94
  EDIT ............................................................................................................. 95
Chapter 5: Creating board modules (continued)

Editing a module.............................................................................................. 96
Moving a module.............................................................................................. 96
Rotating a module to a specific angle............................................................. 97
Rotating a module in preset steps.................................................................. 98
Mirroring a module along the X axis.............................................................. 99
Mirroring a module along the Y axis.............................................................. 100
Mirroring a module along both the X and Y axis.......................................... 100
Flipping a module to the other side of the board......................................... 101
Moving a single module object..................................................................... 102
Moving selected objects within a group....................................................... 103
Moving an off-grid object............................................................................. 104
Moving an off-grid object back on grid......................................................... 104
Deleting and undeleting module objects....................................................... 105
Deleting objects on any layer....................................................................... 105
Deleting objects on a specific layer............................................................. 105
UNDELETE ..................................................................................................... 106
SELECTIVE ................................................................................................. 106
Permanently deleting deleted objects............................................................ 107
Exporting and importing modules.................................................................. 108
Exporting....................................................................................................... 108
Importing....................................................................................................... 109
Deleting exported modules........................................................................... 110
Creating a module.......................................................................................... 111
Starting a new module.................................................................................. 111
Setting preferences....................................................................................... 112
Changing the view of the display................................................................. 113
Drawing methods.......................................................................................... 113
Drawing the outline with 90 degree corners............................................... 113
  Adding arcs.............................................................................................. 114
  Positioning the arcs............................................................................... 116
  Deleting the 90° corners........................................................................... 117
Drawing the outline with arc corners............................................................ 119
Adding holes................................................................................................. 121
Chapter 6: Placing the TUTOR board (continued)

- Placing other board objects ................................................................................ 145
- Layer marker .................................................................................................... 145
- Board identification ...................................................................................... 146
  - Drawing the board name outline ............................................................... 146
  - Placing the board name ........................................................................ 147
- Placing a fill zone .......................................................................................... 148
  - Creating the fill zone ............................................................................. 148
  - Understanding zone/pad isolation .......................................................... 149
  - Assigning a net to the fill zone ............................................................... 150
- Viewing thermal relief .................................................................................... 151
- Placing alignment targets ............................................................................ 152
- Saving your work ............................................................................................ 153
- Summary ......................................................................................................... 153

Chapter 7: Routing the TUTOR board

- About manual routing .................................................................................... 155
- Getting started ................................................................................................ 155
  - Zooming in on the routing area ................................................................. 155
  - Highlighting a net ..................................................................................... 156
  - Displaying a ratsnest for a single pad ....................................................... 157
  - Creating a new copper tool .................................................................... 158
- Routing the board .......................................................................................... 159
  - Setting conditions ..................................................................................... 159
  - Routing the first track ............................................................................. 159
  - Routing with vias .................................................................................... 160
  - Routing with arc segments .................................................................... 161
- Performing a DRC check ............................................................................ 164
  - Running a block DRC check .................................................................. 164
  - Identifying DRC violations ..................................................................... 165
    - Using INQUIRE .................................................................................. 165
    - Using JUMP ..................................................................................... 166
    - Viewing the violated areas ................................................................. 166
Chapter 7: Routing the TUTOR board (continued)

Editing the routed board................................................................. 168
Correcting DRC errors................................................................. 168
  Drawing a new track................................................................. 168
  Deleting the offending track......................................................... 169
Running another DRC check.......................................................... 169
Deleting a stub............................................................................ 171
Deleting and undeleting a track....................................................... 171
Changing a track path................................................................... 172
Changing track width.................................................................... 173
Saving your work.......................................................................... 174
Summary ...................................................................................... 174

Chapter 8: Autorouting the TUTOR board

About autorouting.......................................................................... 175
Preparing for autorouting............................................................... 175
  Placing an autoroute zone.......................................................... 176
  Locking an existing route............................................................ 177
Setting routing conditions for a net............................................... 178
  Specifying a copper tool.............................................................. 178
  Excluding vias........................................................................ 178
Setting autorouter options............................................................ 179
  Setting an autoroute method....................................................... 179
  Setting a sweep routing direction................................................ 179
Autorouting the board................................................................... 180
  Autorouting a section of the board............................................. 180
  Autorouting the whole board...................................................... 182
    Setting a sweep window........................................................... 182
    Begin autorouting.................................................................. 183
  Via reduction........................................................................... 184
  Additional processing................................................................. 185
  Erasing all routes.................................................................... 185
Chapter 8: Autorouting the TUTOR board (continued)

Finishing the layout ................................................................. 186
  Moving reference designators ................................................. 186
  Placing an assembly outline .................................................. 187
  Placing dimensions ............................................................... 188
    Placing the first dimension ................................................... 188
    Placing the second dimension .............................................. 190
  Saving your work .................................................................... 191
  Summary .................................................................................. 191

Chapter 9: Printing and plotting the TUTOR board

About printing and plotting ......................................................... 193
Getting started ........................................................................ 193
Printing ...................................................................................... 194
  Configuring printer options ..................................................... 195
  Configuring pages ................................................................. 196
    Building the TOP COPPER LAYER page ............................... 196
    Building the BOTTOM COPPER LAYER page ....................... 198
    Building the ASSEMBLY DRAWING page ............................. 199
  Printing pages ........................................................................ 200
    Printing all pages ................................................................. 200
    Printing selected pages ......................................................... 201
  Saving a printer setup ............................................................ 203
  Loading a printer setup .......................................................... 204
Plotting ....................................................................................... 205
  Gerber (274-X) ......................................................................... 205
  HP-GL/2 ................................................................................ 206
  Postscript ............................................................................... 206
Appendix A: Phar Lap technical information

- About the CFIG386 utility ................................................................. 207
  - How to run CFIG386 ................................................................. 207
  - Error messages ........................................................................... 209
- 386|DOS-Extender command line switches ...................................... 211
  - Conventional memory switches .................................................. 212
  - Systems Call Data Buffer Switches .......................................... 213
  - Mixed mode program switches ................................................... 214
  - Stack allocation switches ............................................................ 215
  - Extended memory switches ....................................................... 216
  - Weitek 1167 switch .................................................................... 217
  - Interrupt relocation switches ..................................................... 218
  - Interrupt mapping switches ....................................................... 219
  - Paging disable switch ................................................................ 220
  - Compaq built-in memory switch .................................................. 221
  - VDISK compatibility switch ....................................................... 221
  - 80386 step B0 switch ................................................................. 222
  - EMS simulator switch .................................................................. 223
  - Address line 20 switch ............................................................... 224
  - PC and PC/XT detection switch .................................................... 225
  - 386|VMM command line switches ............................................... 226
  - Virtual memory driver switches .................................................. 227
  - Swap file location switch ............................................................ 228
  - Page replacement policy switches .............................................. 228
  - Swap file growing policy switches .............................................. 230

Index .................................................................................................. 235
Welcome to OrCAD
PC Board Layout
Tools 386+

You now have OrCAD PC Board Layout Tools 386+, a powerful yet straightforward PC board layout tool set with the capability of an engineering workstation. PC Board Layout Tools 386+ is designed with today’s high density, multi-package board engineering environment in mind.

PC Board Layout Tools 386+ features extensive autorouting capability of up to 16 simultaneous layers, and gives the PC board designer sophisticated manual routing tools. Also featured is an extensive collection of surface mount and thru-hole modules libraries, and all the utilities needed to release a board to manufacturing.

You can import an existing board file into PC Board Layout Tools 386+, then use the board as a module. This capability reduces your design time by providing access to reusable board designs.

Complex pad array layouts are easy to create in the PC Board Layout Tools 386+ module library editor. You use pad array generators to automatically lay out chip carriers, staggered pin connectors, pin grid arrays, and polar coordinate modules.

PC Board Layout Tools 386+ uses extended memory, up to the maximum available. It also uses virtual memory, so you can work with very large boards by having program and board file information temporarily stored on the hard disk that would normally be stored in memory. Board design and module complexity are limited only by the amount of available PC memory and virtual memory disk space.
PC Board Layout Tools 386+ is specifically written for personal computers using an 80386 or better microprocessor. The added capabilities of these microprocessors—chiefly larger memory capacity and greater speed—are leveraged by OrCAD's new 32-bit database and software.

PC Board Layout Tools 386+ is a completely new product, and supports most popular graphics boards, printers, and plotters. All of the PC Board Layout Tools 386+ programs, libraries, and drivers work with and support the 32-bit database.
Chapter 1: Welcome to OrCAD PC Board Layout Tools 386+

Minimum configuration

To use PC Board Layout Tools 386+, you must have an IBM PC or compatible with:

- An 80386 or faster microprocessor. A floating point coprocessor (80387, 80487) is highly recommended.
- A VGA or higher resolution display.
- Four megabytes of free RAM after DOS and all device drivers are loaded. Full autorouting capability requires eight megabytes of RAM, with sixteen megabytes recommended for increased performance.
- A hard disk with ten megabytes or more of free storage space for product installation. Additional contiguous hard disk space is needed for effective use of virtual memory. See Virtual memory recommendations.

Virtual memory recommendations

PC Board Layout Tools 386+ uses virtual memory to swap portions of program code and file data to your computer hard disk when all system RAM is used. The data swapped to disk is stored in a temporary file and read back when the data is needed. The following are recommendations for achieving optimum virtual memory performance:

- Use a disk defragmenting utility to maintain your free hard disk space as a single, contiguous area.
- Provide adequate disk space for swapping. A good rule of thumb is to multiply your system memory by 1.5 and have at least that much contiguous disk space available for the swap file.

Virtual memory for PC Board Layout Tools 386+ is dynamically allocated. This means that when system RAM is filled, the swap file increases and decreases in size, according to program demands.

Configuring virtual memory

You use CFIG386.EXE to perform custom virtual memory configurations for PC Board Layout Tools 386+. For most systems, the default configuration is acceptable.

Refer to Appendix A: Phar Lap technical information for virtual memory configuration options.
Finding the information you need

These guides accompany PC Board Layout Tools 386+:

- PC Board Layout Tools 386+ User's Guide
- PC Board Layout Tools 386+ Reference Guide
- Installation & Technical Support User's Guide
- ESP Design Environment User's Guide
- Stony Brook M2EDIT Text Editor User's Guide
- Fast Track, a quick reference for PCB II users upgrading to PCB 386+

Project-oriented design environment

PC Board Layout Tools 386+ is one part of a fully integrated Electronic Design Automation (EDA) system. The design environment means you can focus on what's important: the design. Designs are organized on a project-by-project basis, with all the design files—schematics, netlists, parts lists, simulation results, and board layouts—stored together.

The ESP Design Environment User's Guide introduces the graphical environment under which PC Board Layout Tools 386+ and the other OrCAD tool sets operate. In this environment, OrCAD tools and tool sets, such as PC Board Layout Tools 386+, are accessed via buttons. There are four OrCAD tool sets. They are:

- Schematic Design Tools
- Digital Simulation Tools
- Programmable Logic Design Tools
- PC Board Layout Tools

Buttons to access all four OrCAD tool sets display on the Design Environment screen, even if you only have one tool installed on your computer.
Beyond the basics

Once you have mastered the basics, refer to the *PC Board Layout Tools 386+ Reference Guide* for information that will help you plan and create your design. The reference guide explains how to tailor configurations to match your personal requirements, and provides detailed information about the commands and concepts of PC Board Layout Tools 386+. The *PC Board Layout Tools 386+ Reference Guide* is designed to be a continuing source of instruction and reference as you use PC Board Layout Tools 386+.

Working in the Design Environment

PC Board Layout Tools 386+ is one part of a fully integrated electronic design automation environment. The graphical design environment:

- Runs the tools within a tool set. The tools that make up PC Board Layout Tools 386+ are listed in the next section.
- Moves between tool sets without switching directories or copying files.
- Configures tools. Each tool can be configured and the configuration stored. This eliminates the need to enter command line switches every time a tool is used.
- Organizes designs by project. All files associated with a design—schematics, netlists, reports, PLD source code, simulation results, and board layouts—are stored in one location. This location is actually a directory on your computer's hard disk. Each design has its own directory containing all of the files described above.
The tools in a tool set are organized by function:

- Editors
- Processors
- Librarians
- Reporters
- Transfers

Figure 1-1 shows how these tools are organized on the PC Board Layout Tools screen.

These functions are described briefly on the pages that follow. The explanations assume you are already familiar with common electronic design terms and concepts. If you are just learning about PC board design, some terms we use to describe the tools may not be familiar to you. Don’t worry: basic, essential concepts and skills are thoroughly covered in chapters 3 through 9 of this guide. Advanced concepts are fully explained in the PC Board Layout Tools 386+ Reference Guide.

Figure 1-1. The PC Board Layout Tools screen.
Chapter 1: Welcome to OrCAD PC Board Layout Tools 386+

Editors
Editors modify or create design files. PC Board Layout Tools 386+ contains three editors:
- Edit Layout routes the layout.
- Edit File is used to create and edit text files.
- View Reference is used to review reference material supplied with PC Board Layout Tools 386+ using a text editor.

Processors
Processors are tools that subject a design file to a specific process. PC Board Layout Tools 386+ includes four processors:
- Modify Modules modifies pad shape, pad size, and drill size for modules either in a layout or in a module library.
- Create NC Drill File creates a report of drilling information, including location and drill size, for a board file.
- Reannotate Board File reannotates your board file so the modules are numbered sequentially. You can reannotate specific modules, or all modules in a board file.
- Fix Time Stamps compares the netlist file with the board file and assigns the time stamps in the netlist to the modules in the board file, based upon reference designators.

Librarians
- Make Board Template creates a custom PC board template file from a board file.
- Make Library creates a module library file from a board file.
Reporters  Reporters are tools that produce human-readable reports, but do not modify design data in any way. Reporters include:

- **Module Report** reports module locations in a PC board layout file.
- **Compare Netlists** compares an EDIF netlist with a board file and reports differences between the two.

Transfers  Transfer tools run utilities that create the files necessary for other tool sets to continue the design process. During the design process, the design database created in one tool set (such as PC Board Layout Tools 386+) is not useable by other tool sets (such as Schematic Design Tools) for much of the design process. This is because the design is not complete. The transfer is how the design database is updated so that the other tools may have access. The Transfers tools take care of intermediate steps so that you don't have to. The four transfer tools in PC Board Layout Tools 386+ are:

- **To Schematic**
- **To PLD**
- **To Digital Simulation**
- **To Main**

For example, the **To Schematic** tool does this intermediate step:

- Runs **Back Annotate**, updating the reference designators in the schematic so they match the new reference designators in the board file.
The remainder of the PC Board Layout Tools 386+ User's Guide shows how to use the tool to design a PC board by guiding you through various exercises. To create a board design, you use Edit Layout.

Each of the remaining chapters builds on the skills and concepts from the previous chapter.

The summary below describes the design concepts and skills you learn in each chapter.

Chapter 2: Installing PC Board Layout Tools 386+

In this chapter you learn how to install PC Board Layout Tools 386+.

Chapter 3: Transferring from schematic to layout

In this chapter you learn how to transfer a design from Schematic Design Tools to PC Board Layout Tools 386+. This chapter describes how to edit the TUTOR386 schematic so it contains all the information required by PC Board Layout Tools 386+. You learn how to configure Schematic Design Tools to produce a netlist, which Edit Layout uses to produce a board layout.

Chapter 4: Introducing Edit Layout

This chapter introduces Edit Layout. You learn how to change default configuration settings, change view and display options, and define and save macros.

Chapter 5: Creating board modules

Although PC Board Layout Tools 386+ provides extensive libraries, you may occasionally need a module not in any library. This chapter describes how to edit modules from within Edit Layout. In this chapter you learn how to create a new module, save the new module in a library, and export and import modules.

Chapter 6: Placing the TUTOR board

In this chapter you create a PC board layout by loading the netlist you produced in Chapter 3: Transferring from schematic to layout. You learn the basic procedures required for placing modules. You also learn how to edit module placement.
<table>
<thead>
<tr>
<th>Chapter 7: Routing the TUTOR board</th>
<th>In this chapter you route the TUTOR board. You also learn how to edit routed tracks.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Chapter 8: Autorouting the TUTOR board</td>
<td>In this chapter you use the autorouter to automatically route the TUTOR board. In this chapter you learn how to customize autorouting methods by setting routing options.</td>
</tr>
<tr>
<td>Chapter 9: Printing and plotting the TUTOR board</td>
<td>In this chapter you produce a print and a plot of the routed board on a printer and on a plotting device.</td>
</tr>
<tr>
<td>Appendix A: Phar Lap technical information</td>
<td>This appendix describes command line options for the Phar Lap DOS memory extender.</td>
</tr>
</tbody>
</table>
The installation program makes it easy to upgrade your OrCAD/PCB II board files and custom modules to PC Board Layout Tools 386+. You can choose to have all your OrCAD/PCB II board files and custom modules automatically converted during the installation process, or you can choose to convert the files manually.

**NOTE:** OrCAD/PCB II will not be accessible from the ESP design environment after PC Board Layout Tools 386+ is installed. See Fast Track for instructions on running OrCAD/PCB II from outside the ESP design environment.

To make it easy to find the instructions that apply only to systems that are being upgraded from OrCAD/PCB II, they are marked with the symbol shown at left.

If you are installing PC Board Layout Tools 386+ on a system that doesn't have any OrCAD software installed, you can ignore the instructions marked by this symbol.

**CAUTION:** If you are installing PC Board Layout Tools 386+ on a system that already contains OrCAD software, you must install new drivers. Be sure to follow the instructions given in the Installing the software section very carefully.
Installing PC Board Layout Tools 386+

To install PC Board Layout Tools 386+ you must use the new INSTALL program to copy PC Board Layout Tools 386+ to your system. This is explained in Installing the software.

Before you install

Follow these steps if you are upgrading from OrCAD/PCB II to PC Board Layout Tools 386+:

1. Back up all of your custom PC board modules. This is especially important if you have modified any OrCAD-provided modules.
2. Back up all of your OrCAD/PCB II board designs.

Installing the software

Use the INSTALL program provided with PC Board Layout Tools 386+ to install the software. Do not use an older version of INSTALL already on your hard disk.

△ NOTE: You may have problems installing the software if you have several TSR’s (terminate and stay resident programs) loaded, such as an anti-virus program. If your computer is connected to a network, running SHARE.EXE may cause installation problems. Also, problems may occur if you run the installation program as a DOS application under Microsoft® Windows™.

1. Insert the disk labeled “Install” into your computer’s floppy disk drive.
2. At the DOS prompt, enter the name of the drive the disk is in. For example, if you placed the installation disk in drive A, type A: and press <Enter>.
3. Type INSTALL and press <Enter>.

INSTALL prompts you to enter the information it needs to install the software on your system. Answer the questions and insert disks into your computer’s disk drive as requested.
If you already have any of OrCAD's Release IV or 386+ software installed on your system, be aware of these two important details:

- You must install the new INSTALL program on your hard disk, and you must install new display drivers. When INSTALL asks you if you want to install any of these, be sure to always answer YES.

- The INSTALL program asks whether or not to update the ORCADESP.DAT files in your design directories. If your system uses the directory structure recommended by OrCAD, answer YES to this question so that INSTALL will do this for you. If your directory structure does not match the directory structure recommended by OrCAD or if you answer NO to this question you will need to manually update these files using the MERGEDAT program. MERGEDAT is described in technical note #45: Updating ORCADESP.DAT files with MERGEDAT.

INSTALL asks you if you want all your OrCAD/PCB II board files and custom modules automatically converted for PC Board Layout Tools 386+.

If you select automatic conversion, the installation program uses the OrCAD environment variables listed in your AUTOEXEC.BAT file to locate and convert all OrCAD/PCB II board files and custom modules. If you select manual conversion, refer to *Fast Track* for information on file conversion commands.

You may need to manually convert files if you have OrCAD\PCB II board files or custom modules in directories that are not in any directory path defined by the OrCAD environment variables.

When the installation is complete, the DOS prompt displays.

4. Reboot your system so that any changes INSTALL made to your AUTOEXEC.BAT file can take effect.
This chapter describes the processes used to transfer a design from Schematic Design Tools to PC Board Layout Tools 386+. The schematic you will work with is TUTOR386.SCH, which is the schematic for the board layout you load in the next chapter. In this chapter, you:

- Configure the ESP design environment and Schematic Design Tools
- Configure To Layout to produce netlist TUTOR386.NET, which is used to create a circuit board in chapter 6.
- Transfer from Schematic Design Tools to PC Board Layout Tools 386+

The files you need

These files are installed on your computer when you install PC Board Layout Tools 386+, and are used in this chapter:

- TUTOR386.SCH – This is the schematic you edit to create the netlist TUTOR386.NET.
- TUTOR386.LIB – This is the schematic parts library for TUTOR386.SCH.
- TUTORORC.NET – This is an OrCAD-supplied netlist you can use if you do not want to create TUTOR386.NET.

If you do not want to perform the steps describing how to create a netlist, you can complete this chapter up to the Configuring DRAFT and Schematic Design Tools section, then skip to the next chapter.

When you begin Chapter 6: Placing the TUTOR board, you substitute TUTORORC.NET, the OrCAD-supplied netlist, for TUTOR386.NET.
Before you begin

Before you begin the exercises in this part of the user's guide, take a minute to review the conventions used in this guide and learn some operating system basics.

Keys

PC Board Layout Tools 386+ is designed to operate on a wide variety of 386 and 486 computer systems. Since many computers label their keyboard keys differently, OrCAD has adopted standards to name two of the most widely-used keys.

<Enter>

Whenever you see <Enter>, it means to press the <Enter> key on your keyboard. On your keyboard, the <Enter> key may be labeled Return.

Throughout the user's guide, you are instructed to enter text. For example, the instructions may read "Enter the filename." This means to type the name of the file and press <Enter>. If you are instructed to "Type the following characters," you should type the specified characters without pressing the <Enter> key.

<Ctrl>

Whenever you see <Ctrl> it means to hold down the <Ctrl> key and press another key. For example, if the instructions say "press <Ctrl><A>", you should hold down the <Ctrl> key and press the <A> key.

Other keys

Alphanumeric, function keys, and other special keys are shown in angle brackets.

Mouse basics

- Clicking the left mouse button is the same as pressing the <Enter> key. In this user's guide, when you are instructed to "press <Enter>," you can use either the keyboard or the mouse, whichever you prefer.

- Clicking the right mouse button is the same as pressing the <Esc> key. In this user's guide, when you are instructed to "press <Esc>," you can use either the keyboard or the mouse, whichever you prefer.
Chapter 3: Transferring from schematic to layout

**Keyboard input**

Characters that you enter are shown in bold monospace font, such as "enter tutor.bdl". This text can also be enclosed in a box:

```
tutor.bdl
```

In the example above, you enter only the characters shown in bold.

**Operating system command prompt**

In this user's guide, the operating system command prompt is shown as:

```
C:>
```

**Commands**

Commands are shown in bold type. Main menu commands are shown in uppercase letters. Other commands are shown as they appear on the menu. When you are asked to select a command, usually both the main menu command and other command are specified.

**Filenames**

Filenames can be from one to eight characters long. A filename may also have a period and an extension consisting of up to three characters. You can use either uppercase or lowercase letters when entering a filename or extension, but the operating system converts all the letters to uppercase.

Filenames and extensions usually contain only letters and numbers. However, you can use additional characters supported by the operating system. For compatibility with OrCAD's environment, use only letters (A–Z and a–z), numbers (0–9), underscores (_), number signs (#), and "at" signs (@).

Most OrCAD software works with any characters your operating system supports. Some applications used in conjunction with OrCAD software—including SPICE programs, some PCB layout programs, and some text editors—support a more limited character set. You should keep any such limitations in mind as you design and avoid using characters that are allowed by one piece of software but not another.
Designs

In the OrCAD design environment, all files pertaining to a design are stored in one directory on your disk. Putting different designs in different directories organizes your files, much as you would organize a file cabinet.

Running the ESP design environment

To run an OrCAD tool, you must first display the ESP design environment main screen. To do this, enter the command shown in bold:

```
C:> ORCAD
```

In a moment, the design environment screen displays (figure 3-1).

Figure 3-1. The ESP design environment main screen.
Changing to the TUTOR design

Before you work with any of the tools accessed from the main screen, you need to change to the TUTOR design. Remember, a design is a directory in which all the files related to a project are stored.

1. Place the pointer on Design Management Tools and click the left mouse button. The menu shown at right displays.

2. Select Execute. The dialog box shown in figure 3-2 displays.

3. Place the pointer on the design named TUTOR and click the left mouse button. This selects the TUTOR design.

4. Select OK to return to the main screen. Notice that the heading in the upper center of the screen has changed to TUTOR Design.

△ NOTE: See the ESP Design Environment User’s Guide for instructions on how to use Design Management Tools.
Changing the startup design

The ESP design environment is configured to the TEMPLATE design each time you run OrCAD tools. Since you will be working in the TUTOR design throughout this tutorial, you need to change the startup design to TUTOR. Follow these steps:

1. Select Design Management Tools.
   The menu shown at right displays.

2. Select Configure ESP.
   The Configure ESP screen displays (figure 3-3).

3. Enter TUTOR in the Startup Design entry box.

4. Select OK to save the configuration changes. The main screen displays.

△ NOTE: See the ESP Design Environment User's Guide for detailed instructions on how to configure ESP.
△ NOTE: If you do not want to create the netlist TUTOR386.NET, perform the two steps listed below and skip to Chapter 4: Introducing Edit Layout. When you load the netlist in Chapter 6: Placing the TUTOR board, select TUTORORC.NET as the netlist filename.

If you want to create the netlist, skip the two steps listed below and proceed to Configuring DRAFT and Schematic Design Tools.

1. Select PC Board Layout Tools. The menu shown at right displays.

2. Select Execute. The PC Board Layout Tools screen displays.
Before you can edit TUTOR386.SCH you need to configure DRAFT to specify the source schematic. You also need to configure Schematic Design Tools to select TUTOR386.LIB as the configured library.

Configuring DRAFT

1. Select Schematic Design Tools, then select Execute. The Schematic Design Tools screen displays (figure 3-4).

2. Select Draft, then select Local Configuration from the menu and select Configure DRAFT. The Configure DRAFT screen displays (figure 3-5).
Chapter 3: Transferring from schematic to layout

Figure 3-5. The Configure DRAFT screen.

3. Select \TUTOR386.SCH from the Files list box.


Configuring Schematic Design Tools

1. Select Draft, then select Configure Schematic Tools from the menu. The Configure Schematic Design Tools screen displays (figure 3-6).

2. Scroll to the Library Options section.

3. Select \TUTOR386.LIB from the Available Libraries list box, then select Insert. The Configured Libraries list box displays \TUTOR386.LIB.

Displaying the schematic

1. Select Draft, then select Execute from the menu. In a moment, the schematic displays (figure 3-7).

   ![Schematic Diagram]

   Figure 3-7. Partial view of the TUTOR386 schematic.

   The reference designators are not annotated and no module values are assigned. These tasks need to be done before you can create a netlist.

   You annotate reference designators by selecting Annotate Schematic from Schematic Design Tools. See Annotating the schematic.

   You assign module values to the schematic by creating a stuff file, then you select Update Field Contents to select the stuff file and insert the module values into Part Field 8 on the schematic. See Creating a stuff file and Updating field contents.

2. Select QUIT Abandon Edits to return to the Schematic Design Tools screen.
Chapter 3: Transferring from schematic to layout

Annotating the schematic

Annotate Schematic scans a design and automatically updates the reference designators of all parts in the design.

Annotate Schematic updates reference designators in the order the parts are placed in the design. You may assign all parts a new reference designator, including any manually edited parts, when annotating the design. To selectively change reference designators and leave others unmodified, use Draft's EDIT Reference Name command.


Follow these steps to annotate the reference designators on TUTOR386.SCH:

1. Select Annotate Schematic, then select Local Configuration from the menu and select Configure ANNOTATE. The Configure Annotate Schematic screen displays (figure 3-8).

```
Configure Annotate Schematic

File Options
Source TUTOR386.SCH

Processing Options
Quiet mode
Do NOT change the sheet number
Unannotated schematic
Report the last assigned reference values
Reset reference numbers to begin with 1 on each sheet of the hierarchy
Incremental annotation (only update reference designators shown as ?)
Unconditional annotation (update all reference designators)
Ignore warnings

Figure 3-8. The Configure Annotate Schematic screen.
```

2. Enter TUTOR386.SCH in the Source entry box. Do not change any of the default settings.

3. Select OK.
4. Select **Annotate Schematic**, then select **Execute** from the menu. Processing status displays in a window at the bottom of the screen. The status window closes when **Annotate Schematic** is complete.

5. Select **Draft**, then select **Execute**. The reference designators are annotated for TUTOR386.SCH, as shown in figure 3-9.

![Schematic Diagram](image)

*Figure 3-9. A partial view of the annotated schematic.*

6. Select **QUIT Abandon Edits** to return to the Schematic Design Tools screen.
Chapter 3: Transferring from schematic to layout

**Viewing key fields**

Key fields tell Update Field Contents where to look for data. A stuff file, or update file, tells Update Field Contents what module value to insert in the schematic when it finds a match.

Update Field Contents uses the preconfigured key field entry in the Update Field Contents entry box of the Configure Schematic Design Tools screen to construct a text string called a *match string*.

Key fields for Schematic Design Tools are already configured to use the schematic Part Value field and 8th Part Field. The Part Value field is used as the match string that determines what module values go in the 8th Part Field on the schematic.

Follow these steps to view the preset key fields for TUTOR386.SCH:

2. Scroll down to the Key Fields section.
3. A V displays in the Combine for Field 8 entry box for Update Field Contents. The letter V represents the value in the schematic Part Value field, and specifies the Part Value field as the match string.
4. The number 8 displays in the Module Value Combine entry box for Create Netlist. This specifies the 8th Part Field on the schematic as the field that To Layout checks for the module value when it creates a netlist.
5. Go to the top of the Configure Schematic Design Tools screen and select Cancel.
Creating a stuff file

Update Field Contents requires a stuff file. You create this text file using a text editor like M2EDIT, the text editor that comes with Schematic Design Tools. If you use a different text editor, use the comparable commands of your text editor.

⚠️ **NOTE:** Be sure to save this file as text only. Any special formatting inserted by your text editor may cause Update Field Contents to fail.

Follow these steps to create the stuff file TUTOR386.STF:

1. Select Edit File, then select Execute. The Edit File screen displays (figure 3-10).

![Edit File Screen](image)

*Figure 3-10. The Edit File screen.*

2. Enter `TUTOR386.STF` in the File to Edit entry box, then select OK. The M2EDIT screen displays.

A stuff file is composed of a list of text strings which are delimited with single quotes. The strings are separated with any number of space, tab, or return characters. A string cannot contain a single quote.

Update Field Contents looks at the strings in the stuff file in pairs, so for readability and clarity each pair is placed on a separate line in the stuff file. The first string in each pair is the match string. The second string is what Update Field Contents places into the schematic part field when it finds a match.
3. Enter the text shown below, including single quotes and upper and lower case characters. Separate the pairs on each line with two tabs.

'10k' 'RC05'
'9.1k' 'RC05'
'.01uF' 'CK05'
'100uF' 'CK05'
'47uF' 'CK05'
'470uF' 'CK05'
'22uF' 'CK05'
'74LS04' '14DIP300'
'TIL309' 'TIL309'
'22V10' '24DIP600'
'LM7805' 'TO220'
'9V' 'BAT9V'
'Mode' '8DIP300'
'Reset' 'PBTN2PIN'

4. Select Output to save the text file as TUTOR386.STF.

5. Select Exit to close M2EDIT. The Edit File screen displays.

6. Select Cancel to dismiss the Edit File screen and return to the Schematic Design Tools screen.
Updating field contents

You prepared the stuff file, now you are ready to update TUTOR386.SCH by inserting module values into the schematic 8th Part Field. Follow these steps:

1. Select Update Field Contents, then select Local Configuration and Configure FLDSTUFF. The Configure Update Field Contents screen displays (figure 3-11).

![Configure Update Field Contents screen](image)

2. Enter TUTOR386.SCH in the Source entry box.

3. Select \TUTOR386.STF from the Files list box. The filename displays in the Stuff File entry box.

4. Part Field 8 is already selected in the Field to be updated section of Processing Options. This specifies that the 8th Part Field of the schematic receives the module values recorded in the stuff file.

5. Select Set the specified field to visible. You can view the inserted module values on the schematic when this is selected.

7. Select Update Field Contents, then select Execute. Program status displays in the status window at the bottom of the screen. When the process is complete, the status window closes.

8. Select Draft, then select Execute to load the updated schematic. Module values display for each schematic part. These values are inserted in the 8th Part Field.

9. Select QUIT Abandon Edits to return to the Schematic Design Tools screen.

**Checking design integrity**

Before you create the netlist, you select Check Design Integrity to check for duplicate objects or overlapping wires in the schematic. Check Design Integrity also creates a report file, then scans the schematic for conformity to basic electrical rules, such as unused inputs and invalid connections. The three processors in Check Design Integrity are:

- Cleanup Schematic
- Cross Reference Parts
- Check Electrical Rules

Follow these steps to configure the three processors:

**Configuring Cleanup Schematic**

1. Select Check Design Integrity from the Schematic Design Tools screen, then select Local Configuration. The menu shown at right displays.

2. Select Configure CLEANUP. The Configure Cleanup Schematic screen displays (figure 3-12).
3. Enter TUTOR386.SCH in the Source entry box, then select OK. The Schematic Design Tools screen displays.

Configuring Cross Reference Parts

1. Select Check Design Integrity from the Schematic Design Tools screen, then select Local Configuration and select Configure CROSSREF. The Configure Cross Reference Parts screen displays (figure 3-13).

2. Enter TUTOR386.SCH in the Source entry box.
3. Enter \texttt{TUTOR386.XRF} in the Destination entry box. This is the filename of the report generated by this processor. You can view this file using a text editor. If a filename is not specified in Destination, the report displays in the monitor box at the bottom of the screen and is recorded in the ESP design environment redirection file \#ESP\_OUT.TXT.


Configuring Check Electrical Rules

1. Select Check Design Integrity from the Schematic Design Tools screen, then select Local Configuration and select Configure ERC. The Configure Check Electrical Rules screen displays (figure 3-14).

2. Enter \texttt{TUTOR386.SCH} in the Source entry box, then select OK. The Schematic Design Tools screen displays.

3. Select Ignore warnings, then select OK. The Schematic Design Tools screen displays.

\[\Delta\] \textit{NOTE: You select Ignore warnings because TUTOR386.SCH has unconnected pins and Check Electrical Rules will not complete successfully unless this option is selected. The preferred solution is to place a no-connect object on each of the unconnected pins in DRAFT.}
Running Check Design Integrity

1. Select Check Design Integrity, then select Execute. Processing status displays in the monitor box at the bottom of the screen. When processing is complete, the monitor box closes.

2. If errors or warnings are reported, refer to the appropriate chapters of the Schematic Design Tools Reference Guide for more information. The chapter for each of the processors is listed below:
   - Cleanup Schematic—Chapter 8
   - Cross Reference Parts—Chapter 23
   - Check Electrical Rules—Chapter 22
Creating the netlist

You use To Layout in Schematic Design Tools to run the processes needed to create the netlist and transfer your design to PC Board Layout Tools 386+.

To Layout runs three processes that update the connectivity database. These processes are INET, ILINK, and IFORM.

About INET

INET creates or updates the connectivity database for the design. See Chapter 9: Creating a Netlist in the Schematic Design Tools Reference Guide for more information about this processor.

Configuring INET

1. Select To Layout, then select Local Configuration. The menu at right displays. INET, ILINK, and IFORM are set to on by default.

2. Select Configure INET. The Configure Incremental Netlist screen displays (figure 3-15).

![Figure 3-15. The Configure Incremental Netlist screen.](image-url)
3. Enter \texttt{TUTOR386.SCH} in the Source entry box.

\textbf{About ILINK}

ILINK creates the intermediate netlist structure and the linked connectivity database. This database contains information on connectivity, parts, fields, pin typing information, and layout directives.

See \textit{Chapter 9: Creating a Netlist} in the \textit{Schematic Design Tools Reference Guide} for more information about this processor.

\textbf{Configuring ILINK}

1. Select To Layout, then select Local Configuration. The menu at right displays.
2. Select Configure ILINK. The Configure Netlist Linker screen displays (figure 3-16).

3. Enter \texttt{TUTOR386.INF} in the Source entry box.
Chapter 3: Transferring from schematic to layout

About IFORM
IFORM uses the netlist format file and an intermediate netlist structure created by ILINK to create a netlist in the format you define. EDIF is the netlist format used in this tutorial.

Configuring IFORM
Follow these steps to configure IFORM so it produces a netlist in EDIF format:

1. Select To Layout, then select Local Configuration. The menu at right displays.

2. Select Configure IFORM. The Configure Netlist Format screen displays (figure 3-17).

Figure 3-17. The Configure Netlist Format screen.
3. Enter TUTOR386 in the Source entry box.
4. Enter TUTOR386.NET in the Destination 1 entry box.
5. Select FEDIF.EXE in the Netlist Format list box. The filename displays in the Selected Format entry box.
6. Select Output pin numbers (instead of pin names) in the Format Specific Options section.
7. Select OK. The Schematic Design Tools screen displays.

Running To Layout

Now that you have configured all of the necessary To Layout processors, you are ready to run To Layout and create the TUTOR386.NET netlist.

1. Select To Layout.
2. Select Execute from the menu. INET, ILINK, and IFORM process sequentially. Processing status displays in the status window at the bottom of the screen. When To Layout is done, the netlist file is created and the PC Board Layout Tools screen displays.

Viewing the netlist

Follow these steps to use M2EDIT to display the netlist. If you have a different text editor configured, use the comparable commands of that text editor.

1. Select Edit File from the PC Board Layout Tools screen, then select Execute. The Edit File screen displays.
2. Select \TUTOR386.NET from the Files list box. The filename displays in the File to Edit entry box.
3. Select OK to display TUTOR386.NET. The netlist looks like figure 3-18.
Chapter 3: Transferring from schematic to layout

<article>

Figure 3-18. Part of the TUTOR386.NET netlist.

4. Select Exit to close M2EDIT without making any changes to the netlist file. The Edit File screen displays.

5. Select Cancel to close the Edit File screen. The PC Board Layout Tools screen displays.

Summary

In this chapter you updated schematic reference designators using Annotate Schematic. You created a module value stuff file and inserted module values into schematic part fields using Update Field Contents. You also created a netlist using the processors in To Layout.

The next chapter introduces you to Edit Layout.
PC Board Layout Tools 386+ uses the Edit Layout editor to create the board layout. As its name suggests, Edit Layout routes your layout both manually and automatically.

Edit Layout is designed to support the complete PC board layout process from netlist to high-resolution output. Edit Layout stores the information on the computer's disk as a data file.

Edit Layout saves the board file in the design in which you are working, or you can save it to another directory. The board file can have the design name and an extension of .BD1, or you may give it a different filename and extension.

In this chapter, you learn how to:

- Change default configuration settings
- Change view and display options
- Save, copy, and rename board files in Edit Layout
- Define and save macros
You configure PC Board Layout Tools to define the working board file, current device drivers, available module libraries, file paths, and default file extensions.

After completing Chapter 3: Transferring from schematic to layout, the PC Board Layout Tools screen displays (figure 4-1).

Figure 4-1. The PC Board Layout Tools screen.

Follow these steps to configure PC Board Layout Tools:

1. Select Edit Layout, then select Configure Layout Tools. The Configure PC Board Layout screen displays (figure 4-2).
2. Select a display driver for Edit Layout from the Available Display Drivers list box. The driver filename displays in the Configured Display Driver entry box. If you want to use a custom driver that is not listed, enter its filename and extension in the Configured Display Driver entry box.

3. Select a printer driver for Edit Layout from the Available Printer Drivers list box. The driver filename displays in the Configured Printer Driver entry box. If you want to use a custom driver that is not listed, enter its filename and extension in the Configured Printer Driver entry box.

△ NOTE: You must select a printer driver if you want to print the TUTOR board in Chapter 9: Printing and plotting the TUTOR board.
4. Enter the path for your TUTOR design directory in the Library Prefix entry box. The module libraries you use in this tutorial are stored in TUTOR.

If you use the default directory structure, you enter C:\ORCAD\TUTOR\*.MLB in the Library Prefix entry box.

If your TUTOR design directory is located in a different path you must enter the correct path, using the format shown in the example above.

△ NOTE: It is important that you enter the proper path so you can follow the steps in this tutorial.

5. Select TUTOR386.MLB from the Available Libraries list box, then select Insert. The filename displays in the Configured Libraries list box. You need TUTOR386.MLB configured when you load a netlist in Chapter 6: Placing the TUTOR board.
Chapter 4: Introducing Edit Layout

Δ NOTE: If a module library is not configured in the Configured Libraries list box, then all module libraries in the Available Libraries list box are automatically configured.

6. Enter the path for Board file prefix and Netlist prefix if they are different from the default directory paths.

7. Enter the path for Temp file prefix if you want to change its default path. This specifies the location of the temporary work file used by Edit Layout.

8. Change the file filters in the Filter Options area if you use different extensions for the listed file types.

9. If you want to change the directory path and filename of the virtual memory swap file, enter a new path and filename in the Directory and File entry boxes in the Virtual Memory Options area.

Δ NOTE: If you have partitioned drives, or multiple drives, you should place the virtual memory swap file on the partition or drive that has the largest amount of contiguous free disk space.

10. Change the path and filename of the Edit Layout template file in the Template entry box if you want to specify a different template file. A template file provides default settings for a new Edit Layout work session. See the PC Board Layout Tools 386+ Reference Guide for additional information on template files.

11. Scroll to the top of the configuration screen (or press <Home>) and select OK to return to the PC Board Layout Tools screen.
Configuring Edit Layout

1. Select Edit Layout. The menu at right displays.

2. Select Local Configuration, then select Configure PCB386. The Configure Edit Layout screen displays (figure 4-4).

3. Select .\DEMO.BD1 from the Files list box. This specifies which board file automatically loads when you run Edit Layout. Files displayed in the Files list are in the current design directory, and are selectively listed by entering a unique filename prefix or extension in the Prefix/Wildcard entry box.

4. If you want to reverse the function of the left and right mouse buttons in Edit Layout, select Left hand mouse operation.

△ NOTE: In this manual, all references to the left and right mouse buttons are based on the assumption that Left hand mouse operation is selected.

5. Select OK to save the changes and return to the PC Board Layout Tools screen.

Figure 4-4. The Configure Edit Layout screen.
Now that you have selected the DEMO board, you are ready to begin learning about Edit Layout.

1. Select Edit Layout. The Edit Layout menu displays.

2. Select Execute.

Edit Layout is now running, and the DEMO board displays. The Edit Layout work area is larger than the layout, so only part of the work area is visible. You can think of the current screen as a window into the larger work area.

Use the mouse to move the pointer around the layout. When the pointer reaches the edge of the screen, the display automatically pans to expose adjacent areas of the layout.

Press the up, down, left, or right arrow keys (the keys on the main keyboard, not the arrow keys on the numeric keypad) to move the pointer one space at a time. The size of the space is determined by the use of a snap grid and the zoom factor.

Very precise routing and object placement is possible using keyboard keys and combinations of the ZOOM and SET Grid Size commands. See Changing your view of the layout later in this chapter for a description of ZOOM commands, and Setting grid options for a description of SET Grid Size.
**Edit Layout command basics**

Menus guide you from step to step in *Edit Layout*. *Edit Layout* organizes commands and program options using menus, command lines, and dialog boxes. You select a command or option by either clicking the mouse or pressing a key.

See the *PC Board Layout Tools 386+ Reference Guide* for complete command descriptions.

**Displaying the main menu**

Press <Enter> or click the left mouse button to see the main menu (shown at right). Press <Esc> or click the right mouse button to dismiss the main menu and return to the *Edit Layout* screen.

There are several ways to select and use a command. You can use the methods shown in table 4-1 in any combination. The method you use is a matter of personal preference.

<table>
<thead>
<tr>
<th>Using the keyboard</th>
<th>Using the mouse</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>To highlight a menu command</strong></td>
<td>Press the up and down arrow keys to slide the highlighting over the command.</td>
</tr>
<tr>
<td><strong>To select a highlighted menu command</strong></td>
<td>Press &lt;Enter&gt;.</td>
</tr>
<tr>
<td><strong>To select any command</strong></td>
<td>Press the highlighted letter in the command name.</td>
</tr>
</tbody>
</table>

*Table 4-1. Using the keyboard or mouse to select a menu command.*
**The command interface**

**Edit Layout** responds to a command by either performing the command's function or displaying another menu, a command line, or a dialog box.

**Menus**

All menus look and work just like the main menu. Press <Esc> or click the right mouse button to return to the menu or command line that called the current menu. Follow these steps to familiarize yourself with these processes:

1. Press <Enter> to display the main menu.
2. Select QUIT. The menu shown at right displays.
3. Press <Esc> to dismiss the menu.

**Command lines**

Command lines are a series of command names listed across the top of the screen. When a command line displays, you can move the pointer around the work area and select a command by pressing the character on the keyboard that corresponds to the highlighted character in the command name.

If you prefer selecting a command with the mouse, rather than typing the highlighted character in the command name, press <Enter> or click the left mouse button to display a menu containing the same commands.

Press <Esc> or click the right mouse button to return to the menu or command line.

1. Press <Enter> to display the main menu.
2. Select BLOCK. A command line displays across the top of the screen. Part of the BLOCK command line is shown below.
3. Press <Esc> to return to the main menu level.
Dialog boxes

Dialog boxes are used to select options, add, revise, or delete selectable items, and enter keyboard data for program control. Dialog boxes give Edit Layout enormous power and flexibility. A dialog box can be displayed and used to change an item in the board file even while you are performing another task, such as moving a group of objects or manually routing.

Dialog boxes can be accessed from other dialog boxes, creating a flexible, hierarchical command structure. For example, the Edit Pad dialog box is displayed for a module pad that needs to be changed. The Module Properties dialog box can be accessed from there, which is used to change the pad’s parent module properties, like text visibility or text size.

Dialog box items

Dialog boxes in Edit Layout may contain these items:

**Button**

A button performs a task or branches to another dialog box. Click the left mouse button on it to perform the button's function.

In Edit Layout, a button can be either active or inactive, depending on the action selected in the dialog box. Active buttons in the Edit Layout screen display as shadowed, three-dimensional rectangles. Inactive buttons display as gray-filled rectangles. Selecting an item in the dialog box may change an inactive button to active, or an active button to inactive.
This manual shows all buttons as black outlined rectangles with black labels. Refer to Edit Layout to identify inactive buttons.

\[\text{NOTE: There are two buttons that are common to almost all dialog boxes: OK and Cancel. Select OK to close the dialog box and incorporate any changes. Select Cancel to close the dialog box without incorporating any changes.}\]

If the change made in the dialog box involves the addition or deletion of items, the Cancel button becomes a Close button, signifying that changes made in the dialog box cannot be reversed or undone. This is different from the ESP design environment, where the Cancel button does not change to Close.

**List box**

A list box contains a list of selectable items. Place the pointer on the item and click the left mouse button to highlight and select the item.

Scroll buttons accompany a list box when the list of items is longer than the area displaying them. See **Scroll buttons** in this section.

**Files**

<table>
<thead>
<tr>
<th>#esp_out.txt</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>2macros.mac</td>
<td></td>
</tr>
<tr>
<td>4bit.sch</td>
<td></td>
</tr>
<tr>
<td>alt-s.mac</td>
<td></td>
</tr>
<tr>
<td>bench93.sch</td>
<td></td>
</tr>
<tr>
<td>boardbak.000</td>
<td></td>
</tr>
<tr>
<td>boardbak.001</td>
<td></td>
</tr>
<tr>
<td>clean.sch</td>
<td></td>
</tr>
<tr>
<td>cmoscpu.sch</td>
<td></td>
</tr>
<tr>
<td>dclock.bd1</td>
<td></td>
</tr>
<tr>
<td>dclock.lib</td>
<td></td>
</tr>
<tr>
<td>dclock.sch</td>
<td></td>
</tr>
</tbody>
</table>
**Droplist box**  
A droplist box contains a list of selectable items that are viewed by placing the pointer in the selected item window or on the droplist button and clicking the left mouse button. Place the pointer on a droplist item and click the left mouse button to highlight and select the item.

```
Drill Diameter

0.032

Closed droplist box.
```

```
Drill Diameter

0.035
0.022
0.024
0.030
0.032
0.035
0.059
0.138

Open droplist box.
```

Scroll buttons accompany a droplist box when the list of items is longer than the area displaying them. See Scroll buttons in this section. When scroll buttons accompany a droplist box, you can use the <Page Up> and <Page Down> keys to scroll the list up or down one window-full at a time. You use the up and down arrow keys to scroll the list up or down one item at a time.

To close an opened droplist box without making a selection, select the droplist button or press <Esc>.

**Check box**  
A check box is an option that can be enabled or disabled. Place the pointer on the check box and click the left mouse button to select it. Highlight the check box to enable the option. Select the check box again to remove the highlight and disable the option.

In this manual, an enabled check box is represented by a square outline with an ‘X’ in it, and a disabled check box is represented by an unfilled square outline. See the example shown above.
Radio buttons are used in lists of mutually exclusive items: only one button can be active at a time. To activate a button, place the pointer on it and click the left mouse button.

An entry box is a field that accepts typed characters or a field with characters in it that can be changed.

Place the pointer in the entry box and click the left mouse button. A vertical bar appears at the pointer location. This is the text cursor, which you can move left or right within the character string by pressing the left and right arrow keys.

Press <Backspace> and <Delete> to delete the character to the left and right of the cursor, respectively.

Press <Ctrl><Backspace> and <Ctrl><Delete> to delete all characters to the left and right of the cursor, respectively. Press <Alt> <Backspace> to delete all characters in the entry box, regardless of cursor position.

Press <Enter> or the left mouse button to accept changes in the entry box. Press <Esc> or the right mouse button to undo any changes in the entry box.

You use scroll buttons to view a list of items that is longer than the window in which they are displayed.

Move the pointer to the scroll button and click the left mouse button to perform the button’s task.

You can repeat scroll button tasks by holding down the left mouse button while selecting the scroll button.

Each scroll button is described on the next page:
Scrolls the list up by one item at a time.

Scrolls the list up by one “window-full” of items at a time.

Scrolls the list down by one “window-full” of items at a time.

Scrolls the list down by one item at a time.
**How command names are shown in this guide**

In this guide, main menu command names are shown in bold uppercase letters. Other command names are shown with just the first letter capitalized. When you are asked to select a command, usually both the main menu command name and other command names are specified.

For example, the statement "Select PLACE Text" means "Select PLACE from the main menu, and select Text from the PLACE menu."

Where the context is clear, though, the main menu command is not specified. For example, if the PLACE menu already displays, and you are asked to select the Text command, the instruction is simply "Select the Text command."

**Returning to the main menu level**

To return to the main menu level from any menu or command line in Edit Layout, press <Esc> as many times as necessary until no menu displays in the upper left corner of the screen, or until the main menu command line displays. At this point, the main menu displays if you press <Enter>.

To return to the main menu from a dialog box, select OK, Cancel, or Close, as needed, until no dialog box displays.

You can use these keyboard shortcut keys for the OK and Cancel buttons:

- Press <Home> and the pointer jumps to the OK button
- Press <Ctrl><Home> and the pointer jumps to the Cancel or Close button
- Press <Enter> or click the left mouse button to select the button under the pointer
Setting up Edit Layout conditions

Now that you understand how Edit Layout's menus, command lines, and dialog boxes operate, take some time to become familiar with some of the commands that govern the way Edit Layout displays and maintains layouts.

The SET command

1. Press <Enter> to display the main menu.
2. Select SET. The Global Options dialog box (figure 4-5) displays.

![Global Options dialog box](image)

Figure 4-5. The Global Options dialog box.

Using the selections in the Global Options dialog box, you control features such as object appearance, selectability, cursor style, and grid size. Note the options that are enabled. These settings determine how the DEMO board displays.

You can set Edit Layout's global options to match your personal preferences, or tailor them to suit the requirements of the board design.
Chapter 4: Introducing Edit Layout

Most of the options in this dialog box are enabled or disabled through the use of check boxes. Other options are changed by selecting an item in a droplist box, or by editing a value in an entry box.

Layer and Current Settings display additional options for Edit Layout.

The Global Options dialog box is accessible from many other dialog boxes during an editing session, so you can alter global settings at any point in the design process.

Layer

You select Layer to specify the number of layers that a board uses, which layer is the current active layer, and the display color for each layer.

1. Select Layer in Global Options. The Layer dialog box displays (figure 4-6).

![Layer dialog box](image)

Figure 4-6. The Layer dialog box.

2. Select Copper Colors/Enables/... to display the Copper Colors/Enables/... dialog box (figure 4-7).
Figure 4-7. The Copper Colors/Enables/... dialog box.

This dialog box sets the number of enabled layers on a board, and specifies how the layers are considered during autorouting.

The Component Copper and Solder Copper layers are enabled in the Layer Enabled column, which specifies that the DEMO board is a two layer board and does not use any internal copper layers.

3. Select Cancel to close the Copper Colors/Enables/... dialog box and return to the Layer dialog box.

You specify the two routing layers in Copper Pairs. When you place a via while routing on one of the selected layers in Copper Pairs, you continue routing on the other selected layer.

The selections in Copper Pairs correspond to the positions of the copper layers listed in the left column of Current Layer.

The Component Copper and Solder Copper layers are the only layers enabled in Copper Colors/Enables/..., so they are the only valid selections in Copper Pairs.
Chapter 4: Introducing Edit Layout

Note that Component Copper is selected in Current Layer, which sets it as the current working layer.

4. Select CLOSE to close the Layer dialog box and return to Global Options.

Current Settings

When you create a new object, Edit Layout applies default options and properties to it. You set these default options and properties through the use of the Current Object Settings dialog box.

1. Select Current Settings in Global Options. The Current Object Settings dialog box displays (figure 4-8).

   ![Figure 4-8. The Current Object Settings dialog box.](image)

   This dialog box establishes default settings for all new objects placed in Edit Layout. You select an editor button to display the editor dialog box. You select a radio button to display in Current Values the current settings for the selected object.

2. Select Text. The current text values display in Current Values, as shown in figure 4-9.
Selecting a layer

Edit Layout has many commands that select a particular layer, or that toggle through specified groups of layers. Follow these steps to learn about the ways you can select layers:

1. Display the main menu and select LAYER. The Layer dialog box displays.

2. Select SilkScreen Component in Current Layer, then select OK. SilkScreen Component displays at the bottom of the screen, indicating it is the current layer.
Chapter 4: Introducing Edit Layout

/ OTHER

Selecting / OTHER activates one of the two copper layers selected in Copper Pairs in the Layer dialog box. Follow these steps to toggle between the Component Copper and Solder copper layers:

1. Select / OTHER from the main menu. The current layer changes from SilkScreen Component to Component Copper.

2. Select / OTHER. The current layer changes to Solder Copper.

3. Select / OTHER again to toggle the current layer back to Component Copper.

+ LAYER and - LAYER

You select + LAYER to incrementally select each enabled copper layer, from the lowest copper layer (Component Copper) to the highest copper layer (Solder Copper).

You select - LAYER to incrementally select each enabled copper layer, from the highest copper layer (Solder Copper) to the lowest copper layer (Component Copper).

Follow these procedures to learn about + LAYER and - LAYER:

1. With Component Copper as the current layer, select + LAYER to set a higher enabled copper layer (Solder Copper) as the current layer.

2. Select - LAYER to set a lower enabled copper layer (Component Copper) as the current layer.

The + LAYER and - LAYER commands are useful when you are manually routing between external and internal copper layers on a multilayer board.

* LAYER

Select * LAYER to set All Layers as the current layer. With All Layers selected, you can select any object on any enabled layer.

* LAYER is useful when you need to edit many objects on a board and the objects are on different layers.
Changing your view of the layout

Edit Layout provides many options for displaying board layouts and specific objects on the board.

ZOOM

Edit Layout can display layouts at many different magnification scales. You change the view size using the ZOOM command. The layout can be zoomed in or out to magnify or reduce its visible image.

When Edit Layout is zoomed out, you can see a large portion of the layout. Zooming in enlarges a small portion of the layout and displays more details. You can zoom in to draw intricate areas of your layout with exacting detail and then zoom out to look at the finished layout.

To zoom out and see more of the work area on the screen at one time, follow these steps:

1. Select ZOOM from the main menu. The menu shown at right displays.
2. Select Out. A reduced view of the layout displays.
3. Experiment with the scale using In, Out, and the numeric zoom scales.

The numeric values in the ZOOM menu represent the number of mils per displayed pixel. A zoom scale of 1 is then 1 pixel=1 mil (.001 inch). A scale of 5 is then 1 pixel=5 mils (.005 inch). A scale of .01 is 1 pixel=.01 mil (.00001 inch), or 100 pixels=1 mil.

The zoom scale range in Edit Layout is from .01 (maximum magnification) to 100 (minimum magnification).

The current zoom scale is displayed at the bottom of the screen, to the right of the pointer coordinates.
Chapter 4: Introducing Edit Layout

Setting a zoom scale

You can set the zoom scale to any value from 0.01 to 100.
Follow these steps to display the entire DEMO board:

1. Select ZOOM Set Scale. The Set Zoom Scale dialog box displays (figure 4-10).

2. Enter 13 in the Scale entry box, then select OK. Pan the display, if necessary, until the entire board is visible.

Selecting a zoom window

Follow these steps to zoom in on a selected area of the TUTOR board:

1. Select ZOOM Window.

2. Place the pointer at (1.8000", 1.5000"), then click the mouse or press <Enter>. The menu at right displays.

3. Select Window Zoom. A bounding box displays as you move the pointer. This box represents the zoom window.

4. Move the pointer to (4.5000", 3.5000") and select Window Zoom End. The display magnifies, filling the screen with the selected area.

WINDOW ZOOM

The same window zoom capability is available from the main menu when you select WINDOW ZOOM.

1. Place the pointer at (2.0000", 1.7000") and select WINDOW ZOOM.

2. Move the pointer to (3.2000", 2.5000") and select Window Zoom End. The display changes to show the new zoom window.

3. Select ZOOM Set Scale, then enter 13 in the Scale entry box.

4. Select OK. The entire DEMO board displays.
The maximum pointer movement resolution with **Stay On Grid** disabled is one ten-thousandth of an inch (0.0001 inch), or 0.1 mil, at the zoom levels shown in table 4-2.

Hold down the `<Ctrl>` key and press an arrow key to move the pointer five grid spaces at a time. Hold down the `<Alt>` key and press an arrow key to move the pointer in that direction to the edge of the current window.

The following table lists pointer movement resolution for the zoom levels in the **ZOOM** menu. The first entry in the table (Zoom level 0.01 to 0.1) is not in the **ZOOM** menu, but is included in the table to show the zoom range for maximum pointer movement resolution. The resolution is shown in decimal inches, and is achieved using the arrow keys and `<Ctrl>` arrow keys with **Stay On Grid** disabled.

<table>
<thead>
<tr>
<th><strong>Zoom level</strong></th>
<th><strong>Arrow key resolution</strong></th>
<th><strong>&lt;Ctrl&gt; Arrow key resolution</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td>0.01 to 0.1</td>
<td>0.1 (.0001 inch)</td>
<td>0.5 (.0005 inch)</td>
</tr>
<tr>
<td>1</td>
<td>1 (.001 inch)</td>
<td>5 (.005 inch)</td>
</tr>
<tr>
<td>2</td>
<td>2 (.002 inch)</td>
<td>10 (.010 inch)</td>
</tr>
<tr>
<td>3</td>
<td>3 (.003 inch)</td>
<td>15 (.015 inch)</td>
</tr>
<tr>
<td>4</td>
<td>4 (.004 inch)</td>
<td>20 (.020 inch)</td>
</tr>
<tr>
<td>5</td>
<td>5 (.005 inch)</td>
<td>25 (.025 inch)</td>
</tr>
<tr>
<td>6</td>
<td>6 (.006 inch)</td>
<td>30 (.030 inch)</td>
</tr>
<tr>
<td>7</td>
<td>7 (.007 inch)</td>
<td>35 (.035 inch)</td>
</tr>
<tr>
<td>8</td>
<td>8 (.008 inch)</td>
<td>40 (.040 inch)</td>
</tr>
<tr>
<td>9</td>
<td>9 (.009 inch)</td>
<td>45 (.045 inch)</td>
</tr>
<tr>
<td>10</td>
<td>10 (.010 inch)</td>
<td>50 (.050 inch)</td>
</tr>
<tr>
<td>20 T</td>
<td>20 (.020 inch)</td>
<td>100 (.100 inch)</td>
</tr>
<tr>
<td>50 F</td>
<td>50 (.050 inch)</td>
<td>250 (.250 inch)</td>
</tr>
<tr>
<td>100 H</td>
<td>100 (.100 inch)</td>
<td>500 (.500 inch)</td>
</tr>
</tbody>
</table>

*Table 4-2. Pointer movement, by arrow key, for each zoom level.*
Chapter 4: Introducing Edit Layout

Using bookmarks
A bookmark is a reference point that Edit Layout places on the board layout. You assign a unique name to the bookmark, then use the JUMP command to move the pointer to the selected bookmark.

A bookmark name can be up to 40 characters long, and can contain spaces and special ASCII characters. Assigning a descriptive name to a bookmark makes it easy to recognize.

Creating a bookmark
1. Select SET. The Global Options dialog box displays.
2. Enable Show BookMarks, then select OK.
3. Move the pointer to where you want the bookmark placed.
4. Select = BOOKMARK. The Bookmark dialog box displays (figure 4-11).

```
~--------------Bookmark--------------------~
OK   Cancel   BookMarks   DRCs   Delete

BookMarks
Origin

Color

Figure 4-11. The Bookmark dialog box.
```

Note that the BookMarks list box already contains a bookmark named Origin. This bookmark is automatically created when you start a design and it cannot be deleted.

Also, note the two radio buttons labeled BookMarks and DRCs. These selections regulate what types of bookmarks display in the list box. Select BookMarks to display all bookmarks in the list box.
The **DRCs** button works the same way. A DRC is a special type of bookmark (**DRC** stands for **Design Rule Check**) that **Edit Layout** places on the board design where it detects a design error. See **Chapter 7: Routing the TUTOR board** for more information on DRC. You select **DRCs** to display all DRC markers in the list box.

5. Enter the following name in the **BookMarks** entry box:

   *My first bookmark*

6. Select a bookmark color from the color list. Organizing your bookmarks by color helps you locate them on the display.

7. Select **OK** to accept the entries and close the dialog box. The symbol shown at right displays at the pointer location.

   The letter B in the symbol indicates it is a bookmark. The converging lines point to the location on the board that is referenced by the bookmark.

   ![Bookmark Symbol](image)

   **NOTE:** Bookmarks always display at the same size in **Edit Layout**, regardless of the current zoom scale.

---

**Jumping to a bookmark**

You use **JUMP** to select a bookmark name from a list in a dialog box. The pointer jumps to the selected bookmark location when you close the **Jump To** dialog box.

Use the following steps to jump to the bookmark you just placed:

1. Move the pointer away from the bookmark.

2. Select **JUMP**. The **Jump To** dialog box displays (figure 4-12).
Figure 4-12. The Jump To dialog box.

3. Select My first bookmark from the BookMarks list box, or enter the name in the entry box below the list box. The name entered in the entry box must exactly match the bookmark name, including capitalization and spaces. The name highlights in the list box, and the bookmark's coordinates appear in the X and Y entry boxes.

4. Select OK. The pointer moves to the bookmark location.

You can also jump to a known set of coordinates by entering them in the X and Y entry boxes and selecting OK.

Deleting a bookmark

You may want to delete a bookmark at some stage of the design process. The reference may not be required any longer, and you want to delete the bookmark to reduce the number of displayed objects.

Follow these steps to delete a bookmark by selecting its name:

1. Select BOOKMARK. The Bookmark dialog box displays.

2. Select My first bookmark in the BookMarks list box, or enter the name in the entry box. The name highlights in the list box.

3. Select Delete. The name disappears from the list box and the bookmark is deleted from the board.

4. Select Close to close the dialog box.
NOTE: You can also delete any bookmark by placing the pointer on the bookmark symbol and selecting DELETE from the main menu. Using this method, you can undelete the bookmark by selecting UNDELETE.

Changing the origin

The ORIGIN command resets the current pointer position as (0.0000", 0.0000"). Edit Layout reports locations in relation to the new origin until you change it again.

Follow these steps to locate the origin and reposition it.

1. Select JUMP from the main menu. The Jump To dialog box displays.
2. Select Origin in the BookMarks list box. The selection highlights and displays in the entry box, and the coordinates for the bookmark display in both the X and Y entry boxes.
3. Select OK. The pointer jumps to the origin.
4. Move the pointer to (0.7000", 0.5000") and select ORIGIN. The origin bookmark, and position (0.0000", 0.0000), are at the new pointer location.
5. Move the pointer to (−0.7000", −0.5000"), the far upper left corner of the display, and select ORIGIN to reposition the origin.

NOTE: The ORIGIN command is accessible from many menus and command lines in Edit Layout. You can set the origin even while you perform tasks such as moving objects or manually routing.
Chapter 4: Introducing Edit Layout

Setting grid options

While working on a layout, it is important to properly space and align objects on the board. This is done by using a grid.

You use the grid options in the Global Options dialog box to set up a grid. These options set grid spacing and appearance, and whether or not to stay on grid.

Setting a grid size

Grid Size defines the space between points on a snap grid. Grid Size also sets the routing grid for autorouting. The allowed range for the Grid Size entry box is from one tenth of a mil (0.0001 inch) to 33 inches.

1. Select SET to display the Global Options dialog box.

2. Enter 0.050000 in the Grid Size entry box to specify a 50 mil grid, then select OK.

3. Move the pointer to the origin and use the arrow keys to move the pointer one grid space at a time. Notice that the coordinates in the lower left change in 50 mil increments.

\[\text{NOTE: All entry boxes in Edit Layout have predefined allowable ranges for values. If you enter a value that is beyond the allowed range for the entry box, the entry is not accepted and the following message appears in the upper left part of the screen:}\]

Allowed Range: \(<\text{min value}>\) to \(<\text{max value}>\)
Setting a grid divisor

Grid Divisor establishes minor grid divisions for both the Grid Size and Via Grid Size options.

1. Select SET to display the Global Options dialog box.
2. Enter 3 in the Grid Divisor entry box. Grid Divisor establishes minor grid divisions for both Grid Size and Via Grid Size.
3. Select OK.
4. Move the pointer to the origin, then use the arrow keys to move the pointer. Note that the 50 mil grid is divided into three 16.7 mil grid spaces.
5. Select SET to display Global Options.
6. Enter 0.025000 in Grid Size and a 1 in Grid Divisor to return the grid to its original setting.

Disabling the snap grid

1. Disable Stay On Grid, then select OK.
2. Use the arrow keys to move the pointer around the display. The pointer movement is not constrained to the grid.
3. Select SET to display Global Options.
4. Enable Stay On Grid, then select OK. The pointer moves on grid when you press the arrow keys.

△ NOTE: Leave Stay On Grid enabled unless you have a compelling reason to work off grid.

Changing the grid color

1. Select SET to display Global Options.
2. Select a grid dot color in Dots, then select OK. Zoom in until you see the grid dots displayed with the selected color.

△ NOTE: If you want the grid dots to be invisible, select black in Dots.
Saving and backing up the board file

When you work on a layout for a long time, it is important to save your work on disk periodically as a precaution against power failures and other unexpected events.

These procedures show you how to save, copy, and rename board files.

Updating the file

To update the board file, follow these steps:

1. Select QUIT. The QUIT menu at right displays.

2. Select Update Board File. Edit Layout saves the file to the same file name in the current design.

Update Board File also creates a backup file, which is the last saved version of the board. The backup file has a .BAK extension, and is also in the current design.

Saving configurations

Configurations are automatically saved with the board file when you select QUIT Update Board File or QUIT Write Board File. When you reload the file into Edit Layout, all previously defined configurations for the board are used.
Writing to another filename

To save the board to a different filename, follow these steps:

1. Select QUIT Write Board File. The Write Board File dialog box displays (figure 4-13).

   ![Figure 4-13. The Write Board File dialog box.](image)

   DEMO.BDI is highlighted in the Files list box, and displays in the list entry box. The current drive, working directory, and any subdirectories of the current working directory also display in the dialog box.

2. Enter WRITE.BDI in the entry box beneath the Files list box, then select OK. WRITE.BDI is written to the current working directory. The Edit Layout screen displays.

   ![Diagram](image)

   NOTE: You can use QUIT Write Board File to create incremental backups of your board file. Use the procedures described above and change the filename each time you save.
Copying a file

Follow these steps to copy the file you just saved, WRITE.BD1, to another filename:

1. Select QUIT Write Board File. The Write Board File dialog box displays and WRITE.BD1 is selected in the Files list box.

2. Select Copy. The Copy File dialog box displays (figure 4-14) and WRITE.BD1 is selected in the list box.

3. Enter COPY.BD1 in the Copy To: entry box, then select OK. The file is copied and the Write Board File dialog box displays. COPY.BD1 displays in the Files list box.

4. Select Close to dismiss the Write Board File dialog box.

Renaming a file

Follow these steps to rename COPY.BD1:

1. Select QUIT Write Board File. The Write Board File dialog box displays.

2. Select COPY.BD1 in the Files list box, then select Rename. The Rename File dialog box displays (figure 4-15).
3. Enter `RENAME.BD1` in the Rename To: entry box and select OK. The Write Board File dialog box displays, and `RENAME.BD1` displays in the Files list box.

4. Select Close to dismiss the dialog box.

**Deleting a file**

After a design is complete, you may wish to delete some of the older versions of the board file. Follow these steps to delete `WRITE.BD1` and `RENAME.BD1`:

1. Select QUIT Write Board File. The Write Board File dialog box displays.

2. Select `RENAME.BD1` in the Files list box, then select Delete. The file is deleted from the disk and removed from the Files list box.

3. Select `WRITE.BD1` and select Delete. The file is deleted and removed from the Files list box.

4. Select Close to dismiss the dialog box.

△ **NOTE:** You can also copy, rename, and delete a file by selecting QUIT Initialize Board File, then selecting the file in the Initialize to Board File dialog box and selecting Copy, Rename, or Delete.
Most of the file maintenance tasks previously described, as well as running other programs, can be executed from the DOS command line while Edit Layout is suspended in the background.

Follow these steps to access the DOS command line from Edit Layout:

1. Select QUIT Suspend to System. The Edit Layout screen disappears and the DOS prompt displays.
   A right arrow bracket (>) is added to the end of the DOS prompt, indicating that Edit Layout is suspended in the background.

2. Enter EXIT to close the DOS editing session and return to Edit Layout.

Suspend to System is also available from many of the dialog boxes in Edit Layout.

△ NOTE: Suspend to System is operational only on computers that have a floating point coprocessor, such as an 80387, or built-in coprocessing functions.
Macros

Macros record virtually anything you do in Edit Layout, so you can automate many repetitive tasks and speed up your work. You assign a macro to a key or combination of keys, then press the key to execute the recorded macro. See the PC Board Layout Tools 386+ Reference Guide for a list of valid keys.

Macros in Edit Layout are relative-event data captures. This means that a macro executes its commands relative to the current pointer location, rather than from the original location when the macro was created.

You can record macros using either menu selections or keystrokes. A macro created using keystrokes runs faster because there are fewer events to process when the macro is played back.

The following steps describe how to create two macros. The first macro is assigned to the <Alt><S> keys, and records the commands to draw an outline and place a text string inside the outline. The second macro is assigned to function key <F1>, and places a line of text on the screen.

Creating the first macro

1. Select SET to display the Global Options dialog box.
2. Enter 0.100000 in Grid Size to change the grid size to one tenth of an inch. Be sure Stay On Grid is enabled, then select OK.
3. Move the pointer to (0.0000", 3.5000").
4. Select ZOOM, then select scale 5 to magnify the display.
5. Select ORIGIN to set the current pointer location to (0.0000", 0.0000").
6. Select % MACRO. The Press Macro Capture Key box at right displays.
Chapter 4: Introducing Edit Layout

7. Press <Alt><S>. Alt S displays in the Press Macro Capture Key prompt box, as shown at right. The macro internal code `\x011F` also displays.

\[NOTE: If you press an invalid key combination, it does not display in the Press Macro Capture Key prompt box. Select a valid key or combination.\]

8. Press <Enter>. The highlighted prompt “Macro Capture” displays in the lower right corner of the screen, reminding you that you are defining a macro. Any commands you execute while “Macro Capture” displays, including pointer movements and selections, are added to the list of commands stored in the macro.

9. Type P O B for PLACE Outline Begin.
   Use the arrow keys to move the pointer (not the arrow keys on the numeric keypad). The pointer moves one grid dot at a time, drawing the outline segment as it moves. You could use the mouse to draw the outline, but using the arrow keys makes it easier to follow the coordinates.

10. Use the right arrow key to move the pointer to (1.5000", 0.0000"). Refer to the coordinates in the lower left corner of the screen. The outline segment appears as a dotted line.

11. Type B (for Begin). A new outline segment begins at that point. The previous outline segment changes to a solid line.

12. Use the down arrow key to move the pointer to (1.5000", 1.0000").

13. Type B. Use the left arrow key to move the pointer to (0.0000", 1.0000").

14. Type B. Use the up arrow key to move the pointer to (0.0000", 0.0000"), completing the outline.
15. Type E (for End). The outline is complete, and all segments display as solid lines.

16. Type P T (for PLACE Text). The Text entry box displays (figure 4-16).

![Figure 4-16. The Text entry box.](image)

17. Enter MACRO. The text displays in the Text entry box.

18. Use the right and down arrow keys to move the text to (0.7000", 0.5000"), then type S (for Set). The Edit Text dialog box displays (figure 4-17).

![Figure 4-17. The Edit Text dialog box.](image)

19. Place the pointer in the Character Height entry box and click the left mouse button. The text cursor displays.

20. Press the <Alt><Backspace> keys to delete the entry, then enter 0.1500.

21. Press <Home> to move the pointer to the OK button, then press <Enter> to close the Edit Text dialog box. The text is now 0.1500 inches high.
22. Type P (for Place) to place the text. The Text entry box displays. Press <Esc> to dismiss the Text entry box.

23. Type % (for % MACRO) to end the macro recording session. The outline and text should look like the example at right. The message "xxx events captured" displays. The value "xxx" reflects the number of mouse movements, keystrokes, and mouse button clicks you performed while recording the macro.

Creating the second macro

1. Use the cursor keys to move the pointer to (0.7000", 1.2000").

2. Select % MACRO again. The Press Macro Capture Key box displays.

3. Press <F1>. F1 displays in the Press Macro Capture Key box, and the macro internal code also displays.

4. Press <Enter>. The message "Macro Capture" displays in the lower right corner of the screen, reminding you that you are defining a macro.

5. Type P T (for PLACE Text). The Text entry box displays.

6. Enter P1 KEY.

7. Type P (Place) to place the text at (0.7000", 1.2000"). Press <Esc> to dismiss the Text entry box.

8. Type % to end this second macro recording session. The message "xxx events captured" displays.

9. Before proceeding with the next sections, reset the origin to the far upper left corner of the work area by moving the pointer to (0.0000", -3.5000") and selecting ORIGIN.
Running the macros

The completed macros are now stored in memory.

1. Move the pointer to the right of the outline and text you just placed and press <Alt><S> to run the first macro.

2. Move the pointer again and press <Fl> to run the second macro.

△ NOTE: If you need to terminate a macro while it is running, press <Ctrl><Break>.

The outline segments and text draw at the new pointer position, rather than the original macro starting point of (0.0000", 0.0000"). Remember that macros capture relative events. This means that objects placed on the board relative to the pointer’s starting position while recording the macro are placed on the board the same relative distance from the new pointer position when the macro is played back.

Saving all macros to a file

In many cases, you will want to use your macros every time you run Edit Layout. To do so, you must first save them to a file.

Edit Layout gives you the ability to save all macros defined in one editing session in a single file, or export a single macro to a file. Use the following steps to save the two macros you just created in a single file:

1. Select GO TO FUNCTION. The menu shown at right displays. Some of the menu items may display as gray on your screen. A menu item displayed in gray is unavailable and cannot be selected.
2. Select Macro Maintenance. The Macro Maintenance dialog box displays (figure 4-18). The two macros you created display in the Defined Macros list box. These macros are now stored only in memory.

Note that some of the buttons at the top of the dialog box display as gray. A gray button is presently inactive and cannot be selected; however, selecting another item in the dialog box may change an inactive button to active.

3. Select Save. The Save ALL Macros to File dialog box displays (figure 4-19).
4. Place the pointer in the entry box below the Files list box and enter the filename **2MACROS.MAC**.

You can use the Directory and Drive list boxes to change the directory where 2MACROS.MAC will be saved. The destination directory is shown in the Current Working Directory entry box. For this example, leave the current working directory at its present setting.

5. Select **OK** to save 2MACROS.MAC in the current working directory. The Save ALL Macros to File dialog box closes and the Macro Maintenance dialog box displays.
To save a single macro from a group, follow these steps:

1. With the Macro Maintenance dialog box displayed, select \x011F Alt S from the Defined Macros list box.

2. Select Export. The Export \x011F Alt S' Macro to File dialog box displays (figure 4-20).

3. Enter ALT - S.MAC in the entry box below the Files list box.

   You can use the Directory and Drive list boxes to change the directory where ALT-S.MAC is saved. The destination directory is shown in the Current Working Directory entry box. For this example, leave the current working directory at its present setting.

4. Select OK to export the macro to ALT-S.MAC. The Save \x011F Alt S' Macro to File dialog box closes and the Macro Maintenance dialog box displays.

\[\text{NOTE: If you save or export a macro to a file that already exists, Edit Layout asks if you want to overwrite the existing file. If you select OK, the new file is written and the old file receives a .BAK extension.}\]
Deleting a macro from the disk

1. With the Macro Maintenance dialog box displayed, select Load. The Load ALL Macros from File dialog box displays and ALT-S.MAC is selected (figure 4-21).

![Load ALL Macros from File dialog box]

Figure 4-21. The Load ALL Macros From File dialog box.

2. Select Delete. The file is deleted from the disk and its filename does not display in the list box. The Delete button becomes inactive.

3. Select Close to exit the dialog box and return to the Macro Maintenance dialog box.

Deleting a macro from Edit Layout

Follow these steps to delete individual macros that are in Edit Layout memory:

1. With the Macro Maintenance dialog box displayed, select \x013B F1 in the Defined Macros list box. The selection highlights and displays in the entry box.

2. Select Delete. The macro is deleted from memory and its name disappears from the list box.

Deleting all macros from Edit Layout

With the Macro Maintenance dialog box displayed, select Delete ALL. All macros in Edit Layout memory are deleted and no macros display in the list box.
Chapter 4: Introducing Edit Layout

Loading a macro from disk

Follow these steps to load a macro file that is saved on the disk into Edit Layout:

1. With the Macro Maintenance dialog box displayed, select Load. The Load ALL Macros From File dialog box displays.

2. Select 2MACROS.MAC from the Files list box. The filename displays in the entry box.

3. Select OK. The Macro Maintenance dialog box displays and the two macros saved in 2MACROS.MAC display in the Defined Macros entry box. The listed macros are now loaded in Edit Layout memory.

Running a defined macro

To run a macro listed in the Macro Maintenance dialog box, follow these steps:

1. Select a macro in the Defined Macros list box. The selection highlights and displays in the entry box.

2. Select Run. The Macro Maintenance dialog box closes and the macro executes, starting at the current pointer location.

Summary

In this chapter you learned how to run Edit Layout and examine and modify work preferences. You also learned how to create and save macros.

The next chapter gives you detailed instructions for creating circuit board modules using the library editor in Edit Layout.
About modules

A module can be defined as a representation of the physical shape, size, and required pad layout of a component that is mounted to a circuit board.

Although PC Board Layout Tools 386+ provides extensive libraries containing over 1000 modules, you may occasionally need a module not found in any library. You can modify an existing module or create an entirely new module from within Edit Layout using the library editor.

In this chapter you will:

- Learn library editor commands
- Edit an existing module
- Create a module library
- Import and export module files
- Create a module
- Save the new module in a library
About the library editor

The library editor in Edit Layout gives you the ability to create complex PC board modules, even while a board file is loaded in Edit Layout. Modules can be routed while they are in the library editor, and an entire routed board file can be imported, edited, and then exported as a single module that can be placed in another board file.

Selecting the library editor

1. Select **GO TO FUNCTION**. The menu at right displays.

2. Select **Library Editor**. The **Initialize to Library File** dialog box displays (figure 5-1).

![Initialize to Library File dialog box](image)

Figure 5-1. The Initialize to Library File dialog box.
Working with module libraries

Note that *.MLB displays in the Filter entry box, and the files that display in the Files list box all have a .MLB extension. These are libraries that contain the modules you place on a board in Edit Layout.

Each library contains module files organized by component type and function. The libraries are named according to the types of modules stored in them. You can rename a library, and you can import and export module files. These functions are described later in this chapter.

In this chapter you work with the DEMO library. To acquaint yourself with library editor commands, you make a copy of the DEMO library, then make a copy of a module in the DEMO library and edit it. Next, you export a copy of the edited module from the DEMO library and import it into the copied library.

Copying the DEMO library

1. Select DEMO.MLB from the Files list box. The library name highlights.
2. Select Copy. The Copy File dialog box displays.
3. Enter DEMOCOPY.MLB in the Copy To: entry box, then select OK. The new library is created and the Initialize to Library File dialog box displays with DEMOCOPY.MLB listed in the Files list box.

Copying and getting a module

It is a good idea to make a copy of a module you want to edit and edit the copy. Also, when you are creating modules that are very similar, it is easier and faster to modify a copy than to build a new module from scratch.

Follow these steps to copy 10HH100 in DEMO.MLB to another name and load the copy into the library editor:

1. Check that DEMO.MLB is the selected library in the Files list box.
2. Select OK. The Get Module dialog box displays (figure 5-2).
3. Select 10HH100 in the Module Name list box.

4. Select Copy. The Copy Module dialog box displays (figure 5-3) and 10HH100 is highlighted.

5. Enter HEADER in the Copy To: entry box, then select OK. The name displays in the Module Name list box.
Chapter 5: Creating board modules

6. Select HEADER in the Module Name list box, then select OK. The module displays in the library editor, as shown in figure 5-4.

Note the three labels, "reference," "value," and "module," that are placed by the module. These are placeholders for values that are associated with the module when it is loaded from a netlist and placed in Edit Layout.

The "reference" placeholder receives the reference designator that is assigned to the 10-pin header schematic part in Draft, such as "JP1." The "value" placeholder receives the schematic part value, such as "10K" for a resistor. The "module" placeholder receives the module name for this part, which is "HEADER."

Updating the library file

Making a copy of a module and loading it into the library editor loads it in memory only. The copy is not actually stored on the disk until you save the file. Follow these steps:

1. Select QUIT. The menu at right displays.

2. Select Update Library File. This saves the module HEADER to DEMO.MLB.
Renaming a module

1. Select GO TO FUNCTION. The menu shown at right displays.


3. Select 10HH100 in the Module Name list box, then select Rename. The Rename Module dialog box displays (figure 5-5).

4. Enter 10PINCON in the Rename To: entry box, then select OK. Module 10HH100 is renamed to 10PINCON.

\[\text{\textbf{Note: Renaming a module renames it in memory only. The new name is not saved in the library until you select QUIT Update Library File.}}\]

5. Select 10PINCON, then select OK. The module displays in the library editor.

6. Select QUIT Update Library File. The renamed file is saved in the library.
Displaying module information

Modules are constructed with many graphic objects, such as outline segments, pads, text, holes, and zones. You display information about these objects using the INQUIRE, VERBOSE INQUIRE, and EDIT commands.

INQUIRE

Follow these steps to familiarize yourself with INQUIRE. You will use 10PINCON, the module you just updated in the library editor, as an example.

1. Press <Enter> or click the left mouse button to display the main menu. The menu shown at right displays. The library editor main menu is very similar to the Edit Layout main menu.

2. Select * LAYER. This sets All Layers as the current working layer, so you can select any object on any layer. The layer color and name display in the lower part of the screen.

3. Place the pointer in the center of the square module pad. The object under the tip of the pointer is the one selected for inquiry.

4. Select INQUIRE. A description of the pad displays in the lower right part of the screen, as shown in the example below.

```
1 Pad: 0.0550" x 0.0550" []
```

The description tells you that it is pad number 1, and that the pad size is 0.0550" square.
5. Place the pointer on each of the pads and select INQUIRE. Note the pad number sequence from pad 1 to pad 10.

6. Place the pointer on one of the outline segments and select INQUIRE. The following description displays:

```
Outline Segment: 0.0080" wide
```

**VERBOSE INQUIRE**

VERBOSE INQUIRE provides information about the entire module, as well as information about the selected module object. Follow these steps to familiarize yourself with VERBOSE INQUIRE:

1. Place the pointer on the square pad and select VERBOSE INQUIRE. The same pad information provided with the INQUIRE command displays in the lower right part of the screen, and the Verbose Inquire - Module dialog box displays (figure 5-6).

```
Verbose Inquire - Module

Reference : 10PINCON
Value : 10PINCON
Module : 10PINCON
Pads : 10
Nets : 0 (Incomplete)
Holes : 0
Interior Segments : 2
Interior Vias : 0

OK
```

*Figure 5-6. The Verbose Inquire - Module dialog box.*

This dialog box contains additional information about the module, such as the module name, number of pads, and internal routing information.

2. Select OK to close the dialog box.
The EDIT command is one of the most powerful tools in the library editor. You use EDIT to change properties, such as line width, size, and layer, of any module object.

EDIT is also context-sensitive. The editing options that display are determined by the type of object you select.

Procedures for editing properties are described later in this chapter. In this example, you use EDIT to acquire information about a module object. Follow these steps to determine what layer an outline segment is on:

1. Make sure that All Layers is the current editing layer.
2. Place the tip of the pointer on one of the outline segments that defines the rectangular shape of the module and select EDIT. The Edit Outline Segment dialog box displays (figure 5-7).

3. Note that Silkscreen Component displays in the Layer droplist box. This is the current layer for the selected outline segment. Silkscreen Component is a graphic layer for the top, or component, side of the board.
4. Select Cancel to close the dialog box without making any changes.
The following procedures describe how to edit a module, using 10PINCON as an example:

**Moving a module**

1. Check that All Layers is the current layer. You can move a module by selecting any of its objects, such as the outline or labels, if All Layers is selected. If the Component Copper or Solder Copper layer is selected, you move a module by selecting one of its pads.

2. Select SET. The **Global Options** dialog box displays.

3. Disable Allow Edits Of Module Objects. With this option disabled, module objects (such as outlines, labels, and pads) cannot be moved individually.

   Leave Stay On Grid enabled. This constrains object movement to a snap grid.

4. Select OK to close the **Global Options** dialog box.

5. Place the pointer in the center of a module label, as shown at right.

6. Select MOVE. The entire module moves as you move the pointer around the screen.

7. Select **PLACE** to place the module in a new location, or press the <Esc> key or the right mouse button to cancel the move.
Rotating a module to a specific angle

1. Place the pointer on a module object and select MOVE.
2. Select Set. The Set Block Parameters dialog box displays (figure 5-8).

3. Enter 90.00 in the Angle entry box, as shown at right.
4. Select OK. The module displays rotated 90° counterclockwise, as shown in figure 5-9. The module rotates using the pointer position as its pivot point.
5. Select Place to place the module.

You can rotate a module in one-hundredth of a degree increments (0.01°), and you can rotate it either counterclockwise or clockwise.

Enter a minus sign before the rotation value to rotate the module clockwise.
If you enter an invalid rotation value in the Angle entry box, the value is not accepted and the following error message displays in the upper left part of the screen:

Allowed Range: -359.99 to 359.99

6. Rotate the module back to its original orientation by repeating steps 1 through 5, entering -90.00 in the Angle entry box.

Rotating a module in preset steps

You can set a rotation step angle so a module rotates by a specified number of degrees clockwise or counterclockwise. Follow these steps to set a rotation step angle of 15°:

1. Place the pointer on a module object and select MOVE.

2. Select Set. The Set Block Parameters dialog box displays.

3. Enter 15 in the Rotation Step Angle entry box, as shown below, then select OK.

   Rotation Step Angle 15.00


5. Select < Rotate Counter Clockwise. The module rotates 15° counterclockwise.

6. Rotate the module back to its original orientation, then select Place.
Mirroring a module along the X axis

Mirroring a module makes it easy to change the pad orientation. To mirror a module along the X axis, follow these steps:

1. Place the pointer above and to the left of the module, as shown in figure 5-10.

2. Select BLOCK.

3. Move the pointer down and to the right. A dotted rectangle stretches to follow it. Place the pointer below and to the right of all module objects, as shown in figure 5-11.

4. Select Block End. The menu shown at right displays.

5. Select Move Block.

6. Select Set. The Set Block Parameters dialog box displays.

7. Enable Mirror X, then select OK.

8. Select Place. The module and its pads mirror along the X axis, as shown at right. Note the new position of pad 1. Use INQUIRE to check the new pad layout.

9. Repeat steps 1 through 8 to return the module to its original orientation.
Mirroring a module along the Y axis

1. Use the BLOCK command to select the module, as described in the previous procedure.

2. Select Move Block, then select Set. The Set Block Parameters dialog box displays.

3. Enable Mirror Y, then select OK. The module mirrors along the Y axis, as shown at right.

4. Return the module to its original orientation.

Mirroring a module along both the X and Y axis

1. Use the BLOCK command to select the module.

2. Select Move Block, then select Set. The Set Block Parameters dialog box displays.

3. Enable both Mirror X and Mirror Y, then select OK. The module mirrors along both the X and Y axis, as shown at right.

4. Return the module to its original orientation.
You can create or modify a module in the library editor so it represents a component mounted on the bottom, or solder, side of the board. Follow these steps to change 10PINCON to a bottom-mounted module.

1. Make sure Allow Edits Of Module Objects in Global Options is not enabled and Stay On Grid is enabled.

2. Make sure that All Layers is the current layer.

3. Place the pointer on any module object and select MOVE.

4. Select Set. The Set Block Parameters dialog box displays.

5. Enable Flip to other side of board, then select OK.

6. Select Place. The module displays as if it is mirrored on the X axis.

7. Move the pointer to an outline segment and select EDIT. Note that the Layer droplist box displays Silkscreen Solder as the current layer. Silkscreen Solder is a graphic layer on the solder, or bottom, side of the board.

△ **NOTE:** The label placeholders flip to the other side of the board, but display normally. When you print or plot the module you specify if you want the text printed or plotted as flipped text.

8. Repeat the previous procedure to flip the module back to the Silkscreen Component side of the board.
Moving a single module object

1. Select SET. The Global Options dialog box displays.
2. Enable Allow Edits Of Module Objects, then select OK. This allows you to edit individual parts of a module.
3. Check that All Layers is the current layer.
4. Place the pointer near the center of one of the module labels, as shown at right.
5. Select MOVE. The label moves with the pointer, but the rest of the module does not move.
6. Select Place to place the label in a new location, or press <Esc> to cancel the move and leave the label in its original position.
7. Place the pointer on a pad and select MOVE. The selected pad moves with the pointer.
8. Move the pad to a new location and select Place, or press <Esc> to cancel the move and leave the pad in its original position.
Moving selected objects within a group

Follow these steps to move the module mounting holes and all outline segments, but not move the pads or text:

1. Place the pointer above and to the left of all the module objects and select BLOCK.

2. Move the pointer down and to the right until all module objects are surrounded by the stretching box.

3. Select Set. The Set Block Parameters dialog box displays. Note the list of check boxes beneath Objects Affected, and that all check boxes are enabled.

   Enabled objects within the block boundaries are selected if the layer the object is on is active. Disabled objects, and enabled objects on inactive layers, are not selected.

4. Disable Module/Pad and Text in the Objects Affected menu, then select OK.

   You do not need to disable any of the other objects in the list because the area defined by the block does not contain any of those objects.

5. Select Block End. The BLOCK menu at right displays.

6. Select Move Block. Move the pointer and note that the two holes and the outline segments move, but the disabled objects in the block (the module outline, pads, and text) do not move. Press <Esc> or click the right mouse button to cancel the Move Block command.
Moving an off-grid object

Some parts of a module, such as an outline segment, may be placed off-grid because the outline dimensions do not coincide with the grid setting. To move an object that is off grid, follow these steps:

1. Select SET to display the Global Options dialog box.
2. Disable Stay On Grid, then select OK.

△ NOTE: When Stay On Grid is enabled, you may not be able to select an object that is off-grid if no part of the object is on an on-grid point.

3. Make sure All Layers is the current layer.
4. Place the pointer on one of the module outline segments and select MOVE. The segment moves with the pointer.
5. Select Place to place the segment off-grid.

Moving an off-grid object back on grid

1. Place the pointer on the off-grid segment.
2. Select MOVE, then select SET. The Set Block Parameters dialog box displays.
4. Enable Stay On Grid, then select OK.
5. Select OK or Cancel to close the Set Block Parameters dialog box. Notice that the outline segment movement is constrained to the snap grid.
6. Select Place to place the segment on the grid.

△ NOTE: You cannot place an off-grid pad back on grid using these procedures.
Deleting and undeleting module objects

These procedures acquaint you with the various methods used to delete and undelete module objects.

△ NOTE: The module placeholders "reference," "module," and "value" cannot be deleted. These items are fields that receive values from the netlist when the module is placed in Edit Layout.

Deleting objects on any layer

Follow these steps to set up the library editor so you can delete any module object on any layer.

1. Make sure All Layers is the current layer.
2. Select SET to display the Global Options dialog box.
3. Enable Allow Edits Of Module Objects, then select OK.
4. Place the pointer on the square pad and select DELETE. The square pad is deleted.
5. Place the pointer on the top outline segment and select DELETE, then place the pointer on the left mounting hole and select DELETE.
6. Select ZOOM Refresh to redraw the display.

Deleting objects on a specific layer

If a module has overlapping objects on different layers, you specify only the layer containing the object you want to delete so you do not delete the wrong object.

1. Select / OTHER to specify the Component Copper layer as the current layer. The /OTHER command toggles between the copper layer pairs that are currently set, which are the Component Copper and Solder Copper layers.
2. Place the pointer on one of the round pads and select DELETE. The pad is deleted.
3. Place the pointer on an outline segment and select DELETE. The message "Nothing to delete" displays in the lower right corner of the display.
The outline segment is not deleted because it is on the Silkscreen Component layer, which is not selected.

4. Select LAYER. The Layer dialog box displays.

5. Select Silkscreen Component from the Current Layer menu, then select OK. The Silkscreen Component layer becomes the current layer.

6. Place the pointer on the same outline segment and select DELETE. The segment is deleted.

**UNDELETE**

After you delete an object, it is stored in an undelete buffer. You can restore the object from the undelete buffer by selecting the **UNDELETE** command. Follow these steps:

1. Select **UNDELETE**. The last deleted object reappears.

2. Continue selecting **UNDELETE**. All deleted objects are recovered in the reverse order they were deleted. When all objects are undeleted, the message “Nothing to Undelete” appears in the lower right part of the screen.

   Up to 254 levels of undelete can be performed. This means that all objects deleted during the last 254 delete commands in the same editing session can be undeleted.

   \[\text{\textbf{\textcopyright}}\] \textit{\textbf{NOTE: The contents of the undelete buffer are not saved when you save the board file.}}

**SELECTIVE**

You use **SELECTIVE** to select which objects you want to undelete from the undelete buffer.

1. Select * LAYER to set All Layers as the current layer.

2. Delete the square pad, the top outline segment, and the left mounting hole.

3. Select **SELECTIVE**. Notice that the deleted objects reappear in their original display colors, but the other module objects now display in dark gray. This provides visual contrast between deleted objects and the other objects on the screen, which cannot be selected while in this mode.
4. Place the pointer on the square pad and select Undelete. The color of the square pad changes to dark gray, indicating that it is removed from the undelete buffer.

5. Place the pointer on the outline segment and select Undelete.

6. Select Quit Selective Undelete. The module objects display with their normal layer colors and the previously deleted objects are restored. Note that the left mounting hole does not display because it is still in the undelete buffer.

As you build and revise a module, you may want to permanently delete old objects from the undelete buffer. Follow these steps to permanently delete the mounting hole:

1. Choose SELECTIVE. The left mounting hole in the undelete buffer displays in its normal color, while the rest of the module displays in dark gray.

2. Place the pointer on the mounting hole and select Permanently Delete. This deletes the mounting hole from the undelete buffer and the display. The mounting hole cannot be restored.

3. Select Quit Selective Undelete to return to the library editor.

△ NOTE: These changes are stored in memory only, until you save the file.

4. Select QUIT Update Library File. This saves the revised 10PINCON module to the DEMO library.
Exporting and importing modules

As you create, revise, and organize modules in a library you may need to transfer them from one library to another. You transfer a module to another library by exporting it to a file, then importing the file into the destination library.

Follow these steps to export 10PINCON from the DEMO library to a file, and import the file into the DEMOCOPY library you created earlier in this chapter.

1. Select GO TO FUNCTION Module Selection. The Get Module dialog box displays and 10PINCON is selected in the Module Name list box.

2. Select Export. The Export Module to File dialog box displays (figure 5-12).

3. Enter 10PINCON.EXP in the entry box below the Files list box. The .EXP extension is not required, but it is helpful when you want to identify or find the file.

4. Select OK. Edit Layout writes the file in the directory specified in the Current Working Directory entry box. The Get Module dialog box displays and 10PINCON is selected.

5. Select OK. The 10PINCON module displays in the library editor.
Chapter 5: Creating board modules

Importing

1. Select QUIT Initialize to Library. The Initialize to Library File dialog box displays.

2. Select DEMOCOPY.MLB from the Files list box and select OK. The Get Module dialog box displays, and the modules in DEMOCOPY.MLB display in the Module Name list box.

3. Enter NEWITEM in the entry box below the Module Name list box.

4. Select Import. The Import Module from File dialog box displays (figure 5-13).

5. Enter * . EXP in the Filter entry box. Only files with a .EXP extension display in the Files list box.

6. Select 10PINCON.EXP in the Files list box, then select OK. The exported module is imported into DEMOCOPY.MLB and receives the name NEWITEM. The Get Module dialog box displays and NEWITEM is selected.
NOTE: NEWITEM is stored in memory only. You must select QUIT Update Library File to save NEWITEM in DEMOCOPY.MLB.

7. Select OK. NEWITEM displays in the library editor.

8. Select QUIT Update Library File to save the module.

9. Verify that NEWITEM is in DEMOCOPY.MLB. Select GO TO FUNCTION Module Selection. The Get Module dialog box displays and NEWITEM is selected in the Module Name list box.

10. Select OK. The module displays in the library editor.

Deleting exported modules

After you import a module into a library and save it, you can delete the exported module file. Follow these steps to delete 10PINCON.EXP:

1. Select QUIT Initialize to Library. The Initialize to Library File dialog box displays.

2. Enter *.EXP in the Filter entry box. Only files with a .EXP extension display in the Files list box.

3. Select 10PINCON.EXP in the Files list box and select Delete. The file is deleted from the disk and removed from the list box.

4. Select Close to return to the library editor.
Creating a module

As your design needs increase, you may require a module for your board layout that is not available in any of the supplied OrCAD module libraries. You use the library editor in Edit Layout to automatically create complex pad arrays and edit all objects in a module.

In this section you learn how to design a module by creating 7PINDEMO, a seven pin connector with two mounting holes. When you are done with the exercises, the finished module looks like figure 5-14.

![Figure 5-14. The finished 7PINDEMO module.](image)

Starting a new module

1. Select QUIT Initialize to Library. The Initialize to Library File dialog box displays.
2. Enter *.MLB in the Filter entry box. A list of module libraries displays in the Files list box.
3. Select DEMO.MLB from the list and select OK. The Get Module dialog box displays and the modules in DEMO.MLB display in the Module Name list box.
4. Enter 7PINDEMO in the entry box below the Module Name list box and select OK.

This assigns the filename 7PINDEMO to the current editing session. The library editor displays, containing the three text strings shown in figure 5-15.

![Figure 5-15. Module property placeholders.](image)

These three text strings are module properties associated with every module, and they automatically display in the library editor when you create a new module. These properties are placeholders for values that are associated with the module when it is loaded from a netlist and placed in Edit Layout.
The "reference" placeholder receives the reference designator that is assigned to the schematic symbol for the module in Draft. The "value" placeholder receives the schematic part value, such as "10K" for a resistor. The "module" placeholder receives the module filename, which is 7PINDEMO.

**Setting preferences**

Before you begin laying out your new module, note some preset library editor preferences. Preferences are saved with the module when you save the file.

1. Select SET. The Global Options dialog box displays. The following options are automatically enabled whenever you start a new module:
   - **Outline tracks.** This displays segments as outlined objects, which makes them easier to edit.
   - **Stay On Grid.** You should always use a grid to lay out a module. Manually routing and autorouting tracks between module pads in Edit Layout is more efficient when module pads are placed on a grid.
   - **Crosshair Cursor.** This displays a pointer with a full-screen crosshair, which makes it easier to align objects.
   - **Allow Edits of Module Objects.** When this is enabled you can select individual module objects.
   - **Grid Size** is set at 0.025000" (25 mils).

2. Select a different grid dot color in Dots if you want to make the grid dots more visible. Select black to make the grid dots invisible.

3. Select Layer. The Layer dialog box displays. Note that SilkScreen Component is selected as the current layer.

4. To display the Silkscreen Component layer in another color, select Other Colors/Enables/... to display the Other Colors/Enables/... dialog box.

5. Select another display color in Silkscreen Component, then select OK to return to the Layer dialog box.

6. Select OK to close the Layer dialog box, then select OK to close Global Options.
Changing the view of the display

1. Move the pointer to (0.2250", 0.0250") and select WINDOW ZOOM.

2. Move the pointer and stretch the zoom window to (1.1500", 0.6500"), then select Window Zoom End. The display zooms to the selected area.

Drawing methods

This tutorial describes two methods for drawing the module outline. You can perform either one of these procedures, or both if you wish:

- Drawing the outline with 90 degree corners, then placing four arcs and modifying the rectangular outline.
- Drawing the outline with orthogonal arc corners.

Use the arrow keys to move the pointer for these procedures—it makes it easier to follow the coordinates.

Drawing the outline with 90 degree corners

1. Move the pointer to (0.4000", 0.4000") and select ORIGIN. Setting the origin where you begin the module outline makes it easier to see that it is drawn to scale.

2. From (0.0000", 0.0000"), select PLACE Outline Begin.

3. Move the pointer to the right to (0.7000", 0.0000"). The outline segment stretches with the pointer.

4. Select Begin, then draw the segment down to (0.7000", 0.2000").

5. Select Begin, then draw the segment to the left to (0.0000", 0.2000").

6. Select Begin, then draw the segment up to (0.0000", 0.0000").

7. Select End to complete the rectangular outline. Press <Esc> to dismiss the PLACE menu. The outline looks like figure 5-16.

You place non-copper module graphic objects, like its outline, on the SilkScreen Component layer when you want these graphic objects silkscreened to the Component Copper side of the board.
Adding arcs

The four corners of this module outline need to be changed to a 0.050" radius. Follow these steps to add a circle constructed of four separate arcs:

1. Move the pointer to (0.3000", 0.1000").

2. Select PLACE Circle.
   An X-shaped cursor displays in the center of the pointer (figure 5-17). The ‘X’ represents the center of the circle.

3. Select Set. The Edit Circle dialog box displays (figure 5-18).
4. Enable Place Circle As Four Arcs, then select OK. Circles created with this option enabled are built with four 90° arc segments, which can be individually moved.

5. Select Begin. Move the pointer to the right to (0.3500", 0.1000"). The circle enlarges as the pointer moves.

6. Select End to complete the circle. Notice that another 'X' cursor appears in the center of the pointer, indicating you can place another circle.

7. Press <Esc> twice to dismiss the PLACE Circle menu. The circle looks like figure 5-19.

When objects display with outlined segments it is easier to see the end points of each segment. The four small circles inside the outlined circle are the overlapping end points for the four arcs.

Figure 5-18. The Edit Circle dialog box.

Figure 5-19. Finished circle, showing arc endpoints.
Positioning the arcs

1. Place the pointer at (0.3000", 0.0500") and select MOVE. The upper left arc moves with the pointer (figure 5-20).

2. Move the arc to (0.0500", 0.0000"). The endpoints of the arc align with the top and left outline segments (figure 5-21).

3. Select Place to position the arc.

4. Use the same procedures to move the three remaining arcs to their respective outline corners, aligning the endpoints with the outline segments. The module looks like figure 5-22.

Figure 5-20. Moving the upper left arc.

Figure 5-21. Magnified view of arc placement.

Figure 5-22. The module outline with an arc in each corner.
Deleting the 90° corners

Now that the arcs are in each corner, you need to delete the 90° corners from the outline. Perform these steps:

1. Move the pointer to (0.0000", 0.0500"). This places the pointer on the lower endpoint of the upper left arc.

2. Select CUT. This cuts the left straight outline segment into two segments, and “Cut” displays at the bottom of the screen.

3. Move the pointer to (0.0500", 0.0000"), the other endpoint for the arc, and select CUT. The top outline segment is cut into two segments at the pointer location.

4. Move the pointer to (0.0000", 0.0000"). The pointer is on the intersection of the left and top cut outline segments (figure 5-23).

5. Select DELETE. One of the cut outline segments is deleted.

6. Select DELETE again to delete the other cut outline segment. You delete a segment by placing the pointer anywhere along its length and selecting DELETE.

△ NOTE: If you have objects intersecting at one point but you do not want to delete all of them, do not try to delete an object at the intersection—you will probably delete the wrong one. If you delete the wrong object, select UNDELETE to recover it.
Now the upper left corner of the outline looks like figure 5-24.

7. Cut and delete the 90° outline segments from the three remaining corners, using the procedures previously described.

After making these revisions, the completed module outline looks like figure 5-25.

Figure 5-24. Enlarged view of corner after deleting 90° segments.

Figure 5-25. The completed module outline.
**Chapter 5: Creating board modules**

### Drawing the outline with arc corners

1. If you already drew the outline using the previous procedures, delete the outline and start with step 3 (you do not need to set the origin again). If you skipped to this section to draw the outline, start with step 2.

2. Move the pointer to (0.4000", 0.4000") and select **ORIGIN**. Setting the origin where you begin the module outline makes it easier to see that it is drawn to scale.

3. Move the pointer to (0.0500", 0.0000") and select **PLACE Outline**.

4. Select **Set**. The **Edit Outline Segment** dialog box displays.

5. Select the **Drawing Method** droplist button to display the selections in the droplist box.

6. Select **Draw Orthogonal Arc Corners**, then select **OK**.

7. Select **Begin** and draw the segment to the right, then down to (0.7000", 0.0500"). The segment automatically draws an arc as you move the pointer down.

8. Select **Begin**, then draw the segment down to (0.7000", 0.1500"). The outline looks like figure 5-26.

![module](module.png)

*Figure 5-26. Drawing the outline with arc corners.*

9. Select **Begin** and move the pointer down, then to the left to (0.6500", 0.2000") to draw the second arc segment.
10. Select **Begin** and move the pointer to the right to 
(0.0500", 0.2000"").

11. Select **Begin** and move the pointer left, then up to 
(0.0000", 0.1500") to complete the third arc segment.

12. Select **Begin** and move the pointer up to 
(0.0000", 0.0500").

13. Select **Begin** and move the pointer up, then to the right to 
(0.0500", 0.0000").

14. Select **End** to complete the module outline. Press <Esc> to 
dismiss the PLACE menu. The outline looks like 
figure 5-25.
Adding holes

Follow these steps to add the two mounting holes:

1. Select PLACE Hole to display a hole.
2. Select Set to display the Edit Hole dialog box (figure 5-27).

![Edit Hole dialog box]

Figure 5-27. The Edit Hole dialog box.

3. Select the Drill Diameter droplist button to display its droplist box, as shown at right.
4. Select 0.0800. The droplist box closes and 0.0800 displays as the new drill diameter.
5. Select OK. The hole is now larger.
6. Move the pointer to (0.0500", 0.1000") and select Place. Another hole displays.
7. Move the second hole to (0.650", 0.1000") and select Place.

8. Press <Esc> twice to dismiss the PLACE menu. The module looks like figure 5-28.

![Figure 5-28. The module, with mounting holes added.](image)

Placing the pad array

The library editor has a powerful pad array generator that makes the task of laying out complex sets of module pads an easy task.

The module needs seven pads placed in two rows, with the three pads in the bottom row offset so they are centered beneath the pads in the top row. See the illustration at right. Follow these steps to create this seven pin pad array:

1. Select PLACE Pad to display a pad.

2. Select Set. The Edit Pad dialog box displays (figure 5-29).
Figure 5-29. The Edit Pad dialog box.

Note the values in the Pad Symbol dropdown box. The values describe the size, shape, and drill hole size of the selected pad. Refer to figure 5-30 for more information on these pad symbol values.

Figure 5-30. Description of pad symbol values.
Selecting a new pad symbol

The pads for 7PINDEMO require a 0.0450" drill hole, but the current pad symbol has a 0.0350" drill hole. Follow these steps to select a new pad symbol:

1. Select the Pad Symbol drop list button to display its drop list box.

2. Select 0.0550" x 0.0550" () Pad 0.0450" drill #. The new pad displays in the Pad Symbol entry box.
Designing the pad array

1. Select Pad Array Settings. The Edit Pad Array Settings dialog box displays (figure 5-31).

![Edit Pad Array Settings dialog box]

Figure 5-31. The Edit Pad Array Settings dialog box.

You choose the type of pad array you want from the selections in the Style menu. Note that Single Pad is selected because you have only one pad placed in the library editor.

Style Sample displays an example of the pad array chosen in Style, and provides visual aids for configuring entry boxes in the menus. A single pad displays in Style Sample because Single Pad is selected in the Style menu.

Select the other options in the Style menu and see how the sample layout changes in Style Sample.

2. Select Connector Stagger X. The layout in Style Sample displays what you want—you just need to define the number of rows, the number of pads in each row, and the pad spacing.
3. The number of pads in the X direction is 4, so enter 4 in the Number (p) entry box in X Direction.

4. The number of pads in the Y direction is 2, so enter 2 in the Number (q) entry box in Y Direction.

5. The spacing between the pads should be one tenth of an inch (0.1000 inch) in both the X and Y direction, so leave Spacing (x) in X Direction and Spacing (y) in Y Direction at their default values.

6. The first pad in the array should be pad number 1, and the other pad numbers should increment by 1. Leave Start Value and Increment in X Direction set at their default values.

7. Leave Numeric as the default selection in X Direction. This assigns a numeric value to the pads. Alphabetic assigns alphabetic values to the pads, which increment according to the value in Increment.

8. Leave Center Array enabled in Options. This places the pointer in the center of the pad array, which makes it easier to align the pad array with other objects.

9. Enter 0.0500 in Stagger (w). This staggers the bottom row of pads so they are offset 50 mils from the pads in the top row. Refer to the sample displayed in Style Sample.

10. The pad array needs seven pads, but there are eight pads currently defined (two rows of four pads). Enter -1 in Row Delta. This deletes the last pad (pad 8) from the bottom row of the pad array.

11. Select OK to close the Edit Pad Array Settings dialog box, then select OK to close the Edit Pad dialog box.

12. The pad array displays in the library editor. Move the pad array to (0.3500", 0.1000"), the center of the module.

13. Select Place. Another pad array displays. Press <Esc> or click the right mouse button twice to dismiss the additional pad array and the PLACE menu.
Now the module looks like figure 5-32.

You are almost finished designing 7PINDEMO. All that is left to do is position the three module value placeholders.

Positioning the placeholders

It is recommended that you position the module value placeholders close to the module outline. If you place modules close together on a board, it is easier to select the correct module value if you keep the placeholders close to the module outline.

Follow these steps to position the module placeholders:

1. Make sure Silkscreen Component is the current layer.
2. Place the pointer in the center of “value” and select MOVE.
3. Move the placeholder to the position shown in figure 5-32 and select Place.
4. Perform the same steps for “module” and “reference,” placing them as shown in figure 5-33.

Figure 5-32. The pad array.

Figure 5-33. The positioned module placeholders.
Saving the module

You are now ready to save the finished module. Select QUIT Update Library File to save 7PINDEMO in the DEMO library.

Leaving the library editor

Select QUIT Leave Library Editor to return to the board editor in Edit Layout. The module you just designed remains in the library editor while you work in Edit Layout, even if you load another board file.

Δ NOTE: You can also return to the Edit Layout board editor from the library editor by selecting GO TO FUNCTION, then selecting Board Editor from the menu.

Summary

In this chapter you learned how to load, edit, and create a module using the library editor.

The next chapter describes how to begin a board layout by loading a netlist and placing modules on the TUTOR board.
About layout placement

The most critical part of the PC board design process is placing the modules on the board. During module placement, some of the important considerations are:

- Placing modules so tracks between pads on a net are as short as possible
- Placing modules so their physical size and location on the board does not interfere with other PC boards or components in the assembled instrument
- Isolating modules that create electrical interference or generate excessive heat

This chapter introduces you to the processes used to create the TUTOR board layout. In this chapter you learn how to:

- Load the Edit Layout template file
- Draw a board outline
- Load a netlist
- Display Edit Layout placement aids
- Place modules
- Edit module placement
- Place other board objects

If you want to do this part of the tutorial at another time and proceed to Chapter 7: Routing the TUTOR board, select QUIT Initialize Board File, then select PLACED.BD1 and proceed to chapter 8.

PLACED.BD1 represents how the TUTOR board appears when you complete this chapter.
Loading the Edit Layout template

To begin this part of the tutorial you load the Edit Layout template, which is an empty file with predefined configurations.

Follow these steps to load the Edit Layout template file:

1. From Edit Layout, select QUIT Initialize Board File. The Initialize Board File dialog box displays.
2. Delete the filename from the entry box beneath the Files list box, then press <Enter>.
3. Select OK. The template loads into Edit Layout.

Setting options

Follow these steps to set some options for Edit Layout:

1. Select SET to display the Global Options dialog box.
2. Enable Outline Pads.
3. Enter 0.025000 in the Grid Size entry box to set a 25 mil grid.
4. Select OK.
5. Select ZOOM Set Scale. The Set Zoom Scale dialog box displays.
6. Enter 14 in the Scale entry box, then select OK. The new zoom scale displays at the bottom of the screen, next to the pointer coordinates.
Chapter 6: Placing the TUTOR board

Drawing the board outline

You define the shape and size of the board by drawing an outline. Follow these steps to draw the TUTOR board outline:

△ NOTE: Use a combination of the <Ctrl> key and the arrow keys to draw the board outline. The <Ctrl> key causes the pointer to move five grid spaces at a time. The arrow keys make it easier to follow the coordinates. You must release the <Ctrl> key before you select Begin.

1. Select * LAYER to set All Layers as the current layer. This places the outline on all board layers.
2. Use the arrow keys to move the pointer to (0.7500", 0.2500").
4. Hold down the <Ctrl> and right arrow keys to draw the first outline segment to (7.1250", 0.2500").
5. Select Begin, then use the down arrow key to draw a segment to (7.1250", 3.3750").
6. Select Begin, then use the left arrow key to draw a segment to (0.7500", 3.3750").
7. Select Begin, then use the up arrow key to draw a segment to (0.7500", 0.2500").
8. Select End to complete the board outline.

Tip

You can draw two sides of a rectangular board outline at the same time. Select PLACE Outline Begin where you want the upper left corner of the outline, then move the pointer down and to the right—the outline segment automatically bends to follow the pointer.

Select New where you want the lower right corner of the board outline, then select Begin to start a new segment. Move the pointer up and to the left to the beginning of the board outline—the outline segment automatically bends to follow the pointer.

Select End to complete the board outline.
Loading the netlist

Now that you completed the TUTOR board outline, you are ready to load the netlist.

\[\text{NOTE: You must have TUTOR386.MLB as the configured module library. If you did not configure TUTOR386.MLB in the Configuring PC Board Layout Tools section of chapter 5, save this file as TUTOR.BD1, quit Edit Layout, and configure TUTOR.MLB using the instructions in chapter 5.}\]

When a netlist is loaded, the modules assigned to the parts on the schematic are loaded into the board layout and nets are assigned to the appropriate pads on the modules. A net is a common signal name shared by two or more module pads on a board.

Before you load the netlist, you define a block to load the modules into and specify the netlist filename.

### Defining a netlist block and selecting TUTOR386.NET

1. Select **GO TO FUNCTION**. The menu at right displays.

2. Select **Netlist Loader**.

3. Move the pointer to \((0.7500", 3.3750")\), the lower left corner of the board outline.

4. Select **Block Begin**, then move the pointer to \((8.5000", 6.2500")\). A bounding box stretches to follow the pointer. The modules loaded from the netlist are placed inside the block area.

5. Select **Block End**. The **Load Netlist File** dialog box displays (figure 6-1).
6. Select TUTOR386.NET from the Files list box, then select OK. The Netlist Load Options dialog box (figure 6-2) displays.

7. Select OK to accept the default configurations in the Netlist Load Options dialog box. The netlist loads and modules display in the area defined by the block.
The netlist loader spaces the modules evenly inside the block, and uses the size of the block to calculate the amount of space between the modules. If the defined block is small, some of the modules may overlap. Making a large netlist block minimizes the number of overlapping modules.

8. Pan the display, if necessary, until the board outline and all modules display.

Figure 6-3 shows how your board layout appears after loading TUTOR386.NET.

Figure 6-3. The TUTOR board, after loading TUTOR386.NET.
There are thirty modules to place on the TUTOR board: eight capacitors, eight resistors, six display chips, four integrated circuits, two switches, one transistor, and one battery.

When you place modules on a board, you must consider the electrical relationships among the modules. Ideally, all modules sharing common nets are placed together so tracks between the modules are as short as possible. Complex circuits make this difficult to accomplish, so you usually achieve optimum placement by using visual placement aids. Placement aids display circuit connectivity, which describes the electrical paths that the nets make through the modules on the board.

Edit Layout provides two visual aids to help you determine optimum module placement:

- **Ratsnest**
- **Force vector**

**Ratsnest**
A ratsnest is a straight line visual connection between two or more pads in a layout that are electrically, but not physically, connected.

**Force vector**
A force vector is a single vector representing the mathematical sum of all the ratsnest vectors for a module. The length of the vector indicates the length of the routes and how close to optimum a module's position is on the board. The goal is to place the module so the vector is as short as possible.
Displaying the ratsnest

1. Place the pointer above and to the left of the modules, then select BLOCK.

2. Enclose all the modules in a block by moving the pointer below and to the right of the group and selecting Block End. The menu at right displays.

3. Select RatsNest Block. All modules in the block display their ratsnest connections (figure 6-4).

Turning off the ratsnest

1. Position the pointer so it is not on any of the modules.

2. Select X SHOW RATSNEST. The ratsnest is removed from the display and the message “Show RatsNest Cleared” displays in the lower right corner.
Chapter 6: Placing the TUTOR board

Displaying force vectors

1. Select SET to display the Global Options dialog box.
2. Enable Show Force Vectors, then select OK. All modules display their force vectors (figure 6-5).

![Force vectors for the TUTOR layout.](image)

Turning off force vectors

1. Select SET to display the Global Options dialog box.
2. Disable Show Force Vectors, then select OK. The force vectors do not display.
Placing the modules

This tutorial describes two methods for placing the modules on the TUTOR board. The method you select is a matter of personal preference. Refer to the section describing the method you wish to use. The two methods are described below.

Coordinate placement—You select the module reference designator from the Place Module dialog box, then enter its Block X, Block Y, and Angle values shown in table 6-1 into the Set Block Parameters dialog box.

Dynamic placement—You select the module reference designator from the Place Module dialog box, then rotate the module, if specified, and use the mouse and arrow keys to move it to the coordinates shown in table 6-1.

Coordinate placement

Follow these steps to place the modules by entering the values from table 6-1:

1. Select PLACE Module. The Place Module dialog box displays (figure 6-6).

2. Select BT1, the first item in the Module Name list box, then select OK. The pointer jumps to the module and the module is ready to move.

Figure 6-6. The Place Module dialog box.
Chapter 6: Placing the TUTOR board

3. Select Set. The Set Block Parameters dialog box displays.

4. Enter the value in the Block X column of table 6-1 into the Block X entry box.

5. Enter the value in the Block Y column of table 6-1 into the Block Y entry box.

6. Enter the value in the Angle column of table 6-1 into the Angle entry box if the value in table 6-1 is other than zero.

7. Select OK. The module moves to the specified coordinates. Check that the coordinates at the bottom of the screen match the entered coordinates.

8. Select Place. The module is placed and you return to the Place Module dialog box.

9. Repeat steps 2 through 8 for the other modules in the list box.

10. Select Cancel after you place U4, the last module.

After you place all modules, the layout looks like figure 6-7.

---

Figure 6-7. The placed TUTOR board.
<table>
<thead>
<tr>
<th>Reference designator</th>
<th>Block X value</th>
<th>Block Y value</th>
<th>Angle</th>
</tr>
</thead>
<tbody>
<tr>
<td>BT1</td>
<td>3.450</td>
<td>2.450</td>
<td>0.00</td>
</tr>
<tr>
<td>C1</td>
<td>2.200</td>
<td>2.025</td>
<td>0.00</td>
</tr>
<tr>
<td>C2</td>
<td>5.750</td>
<td>2.100</td>
<td>0.00</td>
</tr>
<tr>
<td>C3</td>
<td>3.500</td>
<td>2.025</td>
<td>0.00</td>
</tr>
<tr>
<td>C4</td>
<td>2.100</td>
<td>2.275</td>
<td>0.00</td>
</tr>
<tr>
<td>C5</td>
<td>5.750</td>
<td>1.900</td>
<td>0.00</td>
</tr>
<tr>
<td>C6</td>
<td>5.375</td>
<td>1.825</td>
<td>180.0</td>
</tr>
<tr>
<td>C7</td>
<td>5.750</td>
<td>2.300</td>
<td>0.00</td>
</tr>
<tr>
<td>C8</td>
<td>5.875</td>
<td>1.600</td>
<td>0.00</td>
</tr>
<tr>
<td>D1</td>
<td>1.000</td>
<td>0.525</td>
<td>0.00</td>
</tr>
<tr>
<td>D2</td>
<td>1.650</td>
<td>0.525</td>
<td>0.00</td>
</tr>
<tr>
<td>D3</td>
<td>2.825</td>
<td>0.525</td>
<td>0.00</td>
</tr>
<tr>
<td>D4</td>
<td>3.450</td>
<td>0.525</td>
<td>0.00</td>
</tr>
<tr>
<td>D5</td>
<td>5.925</td>
<td>0.525</td>
<td>0.00</td>
</tr>
<tr>
<td>D6</td>
<td>6.575</td>
<td>0.525</td>
<td>0.00</td>
</tr>
<tr>
<td>Q1</td>
<td>2.850</td>
<td>2.950</td>
<td>0.00</td>
</tr>
<tr>
<td>R1</td>
<td>3.400</td>
<td>1.800</td>
<td>0.00</td>
</tr>
<tr>
<td>R2</td>
<td>2.100</td>
<td>1.800</td>
<td>0.00</td>
</tr>
<tr>
<td>R3</td>
<td>2.750</td>
<td>2.025</td>
<td>0.00</td>
</tr>
<tr>
<td>R4</td>
<td>2.750</td>
<td>1.800</td>
<td>0.00</td>
</tr>
<tr>
<td>R5</td>
<td>4.600</td>
<td>1.825</td>
<td>180.0</td>
</tr>
<tr>
<td>R6</td>
<td>2.750</td>
<td>2.500</td>
<td>0.00</td>
</tr>
<tr>
<td>R7</td>
<td>6.050</td>
<td>2.500</td>
<td>180.0</td>
</tr>
<tr>
<td>R8</td>
<td>6.525</td>
<td>1.600</td>
<td>0.00</td>
</tr>
<tr>
<td>S1</td>
<td>5.700</td>
<td>2.800</td>
<td>0.00</td>
</tr>
<tr>
<td>S2</td>
<td>3.050</td>
<td>2.275</td>
<td>180.0</td>
</tr>
<tr>
<td>U1</td>
<td>2.250</td>
<td>0.525</td>
<td>0.00</td>
</tr>
<tr>
<td>U2</td>
<td>4.100</td>
<td>0.525</td>
<td>0.00</td>
</tr>
<tr>
<td>U3</td>
<td>4.975</td>
<td>0.525</td>
<td>0.00</td>
</tr>
<tr>
<td>U4</td>
<td>6.225</td>
<td>2.000</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Table 6-1. Placement coordinates and rotation angles for TUTOR board modules.
Dynamic placement  Follow these steps to move the modules to their locations using the mouse and arrow keys:

1. Select PLACE Module. The Place Module dialog box displays (figure 6-6).

2. Select the first module in the list, then select OK. The pointer jumps to the module and the module is ready to move.

3. Select > Rotate Clockwise or < Rotate Counter Clockwise to rotate the module to the angle shown in table 6-1. The module does not need to be rotated if the listed angle is 0.00. As you rotate the module, the rotation angle displays in the lower right corner of the screen.

4. Using both the mouse and arrow keys, move the module to the coordinates listed in table 6-1. Use the coordinates that display in the lower left part of the screen as a reference.

5. Select Place. The module is placed and you return to the Place Module dialog box.

6. Repeat steps 2 through 4 for the other modules in the list box.

7. Select Cancel after you place the last module in the list.

After you place all modules, the layout looks like figure 6-7.
You placed the modules in their proper locations, but there is additional work to be done before the module layout is complete.

The modules display text that is not required for the finished layout. Follow these steps to hide specific module text for all modules on the board:

1. Enlarge the view of the layout. Move the pointer to (0.7000", 0.2000") and select WINDOW ZOOM.
2. Enclose the board in a zoom window by moving the pointer to (7.2000", 3.6000").
3. Select Window Zoom End to enlarge the layout view.
4. Select SET. The Global Options dialog box displays.
5. Enable Hide Module Value Text, then select OK. Note that the module values do not display.
6. Select SET to display Global Options again.
7. Enable Hide Module Type Text, then select OK. No module type text displays. Refer to figure 6-8.

Figure 6-8. The TUTOR board, with hidden module text.
Rotating module text  When you placed the modules, these four reference
designators were rotated when you rotated the modules:

- C6
- R5
- R7
- S2

You need to rotate these reference designators to the same
orientation as the rest of the reference designators on the
board. Follow this procedure:

1. Place the pointer at (2.7000", 1.7000") and select
   WINDOW ZOOM.

2. Move the pointer to (6.1000", 3.2000") and select
   Window Zoom End to zoom in to the area of the board
   that has the rotated reference designators.

3. Select SET to display the Global Options dialog box.

4. Enable Allow Edits of Module Objects, then select OK.

5. Place the pointer in the center of rotated reference
designator S2 and select MOVE.

6. Select > Rotate Clockwise to rotate the text 90 degrees
clockwise.

7. Select > Rotate Clockwise to rotate the text another
   90 degrees clockwise. This rotates S2 so it is oriented
   properly.

8 Select Place.

9. Move the pointer to reference designator R5 and repeat
   steps 5 through 8 for R5.

10. Repeat steps 5 through 8 for reference designators C6
    and R7. Move R7 up slightly so it is not overlapping S1.
Flipping a module to the other side of the board

Design requirements or changes may require you to place modules on the other side of the board. Follow these steps to flip the transistor, Q1, to the other side of the board:

1. Select SET to display the Global Options dialog box.
2. Disable Allow Edits Of Module Objects, then select OK. The entire module is selected when this option is disabled.
3. Place the pointer at (2.9500", 2.9500"), the center of the middle pad for Q1, and select MOVE.
4. Select Set to display the Set Block Parameters dialog box.
5. Enable Flip to other side of board, then select OK.
6. Select Place to place the module on the other side of the board.

The module reference designator and silkscreen outline change color to reflect their new layer—the SilkScreen Solder layer.

△ NOTE: When a module flips to the other side of the board, the module text continues to display in its normal orientation, rather than as mirrored text. However, the text prints as mirrored text when you enable Mirror Text in the Printing and Plotting dialog box. See Chapter 9: Printing and Plotting the TUTOR board for more information.
Placing other board objects

Now that the modules are properly placed on the board, you can identify an area of the board to place these additional objects:

- Layer marker
- Board identification
- Fill zone
- Alignment targets

Layer marker

A layer marker indicates the number of enabled board layers. A rectangular outline containing a layer number is automatically placed on each of the enabled layers.

1. Select PLACE Layer Marker. A layer marker displays, as shown at right.

2. Move the layer marker to the left and pan the display until the pointer is at (0.9500", 3.2500"), then select Place to position the layer marker. Another layer marker is attached to the pointer.

3. Press <Esc> or click the right mouse button to dismiss the additional layer marker.
Most board layouts have some kind of identification that is either copper etched or silkscreened on the outer layers. The identification can be a board name or a part number.

You need to draw a rectangular outline, then place the board name inside the rectangle. Follow these steps:

1. Select LAYER to display the Layer dialog box.
2. Select SilkScreen Component, then select OK.
3. Move the pointer to (1.5500", 3.0500") and select PLACE Outline Begin.
4. Draw the outline segment to the right to (2.5500", 3.0500") and select Begin.
5. Draw the segment down to (2.5500", 3.3250") and select Begin.
6. Draw the segment to the left to (1.5500", 3.3250") and select Begin.
7. Draw the segment up to (1.5500", 3.0500") and select End to complete the rectangular outline.
**Chapter 6: Placing the TUTOR board**

**Placing the board name**

1. Select PLACE Text. The Text entry box displays.
2. Enter TUTOR BOARD. The text string displays.
3. Select Set. The Edit Text dialog box displays (figure 6-9).

![Figure 6-9. The Edit Text dialog box.](image)

4. Enter 0.0750 in the Character Height entry box, then select OK. The text displays at its new size.
5. Move the text to (2.0500", 3.2000") and select Place. The Text entry box displays.
6. Press <Esc> or click the right mouse button to dismiss the Text entry box. The completed board identification looks like the illustration at right.
**Placing a fill zone**

A fill zone is a copper-filled area of the board that can be assigned a net name. Fill zones are commonly used to provide additional circuit grounding on the board.

You need to place a fill zone on the Solder Copper layer and assign it to GND, the ground net. Follow these procedures:

**Creating the fill zone**

1. Select / OTHER until Solder Copper displays as the current layer.

2. Select PLACE Fill Zone, then move the pointer to (2.6000", 3.0500") and select Begin.

3. Draw the zone segment to the left to (0.8000", 3.0500") and select Begin.

4. Draw up to (0.8000", 1.5000") and select Begin.

5. Draw right to (1.4250", 1.5000") and select Begin.

6. Draw down to (1.4250", 2.4000") and select Begin.

7. Draw right to (2.2250", 2.4000") and select Begin.

8. Draw up to (2.2250", 2.1750") and select Begin.

9. Draw right to (2.6000", 2.1750") and select Begin.

10. Draw down to (2.6000", 3.0500") and select End to complete the fill zone outline.

11. Select SET to display the Global Options dialog box.

12. Enable Show Copper Pour, then select OK. The fill zone displays as a solid, filled polygon, as shown in figure 6-10.

13. Place the pointer at (1.9000", 2.0000") and select WINDOW ZOOM.

*Figure 6-10. The completed fill zone.*
14. Move the pointer to (2.7000", 2.5000) and select Window Zoom End to zoom in to the area around the fill zone and module C4.

Even though the zone is a filled area, Edit Layout automatically isolates the C4 pad from the zone, as shown in the illustration at right. Edit Layout maintains isolation between these copper objects because they are on the same layer and they are not assigned to the same net. The amount of isolation is specified in Copper To Copper Spacing in the Global Options dialog box. The default value is 0.0150.

Follow these steps to determine the assigned net for the pad surrounded by the fill zone:

1. Move the pointer to (2.5000", 2.2750"), the center of the pad.
2. Select INQUIRE. The following message displays at the bottom of the screen:

   C4 2 gnd Pad: 0.0940" x 0.0940"

The pad is part of the GND net.
Assigning a net to the fill zone

1. Place the pointer at (2.6000", 2.1750"), the upper right corner of the fill zone, and select EDIT. The Edit Zone Segment dialog box displays (figure 6-11).

![Edit Zone Segment dialog box](image)

Figure 6-11. The Edit Zone Segment dialog box.

2. Select Zone Properties. The Edit Zone Properties dialog box displays (figure 6-12).

![Edit Zone Properties dialog box](image)

Figure 6-12. The Edit Zone Properties dialog box.
3. Select GND from the Net Names list box and select OK. This assigns the GND net to the fill zone and closes the Edit Zone Properties dialog box.

4. Select OK to close the Edit Zone Segment dialog box.

The pad and fill zone are now assigned to the same net and are automatically connected, with thermal relief created between the pad and the zone. See the illustration at right.

**Viewing thermal relief**

Follow these steps to change how thermal relief displays between the pad and zone:

1. Select SET. The Global Options dialog box displays.

2. Disable Outline Pads.

3. Enable Show Drill Hole, then select OK. The thermal relief area looks like the illustration at right.

4. Select SET to display the Global Options dialog box.

5. Enable Outline Pads.

6. Disable Show Drill Hole, then select OK to restore the previous display settings.

Refer to the PC Board Layout Tools 386+ Reference Guide for a description of thermal relief.
Placing alignment targets

You use alignment targets as visual aids for aligning separately plotted board layers when you check the board design or when the board is fabricated.

Use this procedure to change the zoom scale so you can view the entire board, then place two alignment targets outside the board outline:

1. Select ZOOM Set Scale.
2. Enter 13 in the Scale entry box and select OK. The entire board displays.
3. Select * LAYER to set All Layers as the current layer.
4. Select PLACE Alignment Target. A default alignment target displays, as shown at right.
5. Select Set. The Edit Alignment Target dialog box displays (figure 6-13).

6. Enter 0.1500 in the Radius entry box.
7. Select the Alignment Target Style drop list button to display its drop list box.
8. Select Crosshair Double from the droplist box, then select OK. The alignment target is smaller, with two concentric circles, as shown at right.

9. Move the pointer to (0.5000", 0.2500") and select Place. Another alignment target is attached to the pointer.

10. Move the pointer to (7.3750", 3.5750") and select Place to position the second alignment target.

11. Press <Esc> or click the right mouse button to dismiss the additional alignment target.

Saving your work

You are finished placing the TUTOR board, but before you proceed to the next chapter you need to save your work. Follow these steps:

1. Select QUIT Update Board File. The Write Board File dialog box displays. This dialog box displays because you loaded the Edit Layout template file at the beginning of this chapter, and Edit Layout requires that you save your work to a new filename.

2. Enter TUTOR.BD1 in the entry box below the Files list box, then select OK. TUTOR.BD1 is saved in the current working directory.

Summary

In this chapter you learned how to create a board outline, load a netlist into Edit Layout, and display ratsnest and force vectors as module placement aids. You also learned how to place and edit modules and other objects, and how to assign a net to a fill zone.

The next chapter gives you detailed instructions for manually routing part of the TUTOR board.
CHAPTER 7

Routing the TUTOR board

About manual routing

Some boards may be designed so that connections between pads on a net must have tracks of a specific length or shape. You must place these tracks using manual routing techniques.

In this chapter you use the processes you have already learned, and also learn how to:

- Highlight a net to locate routing targets
- Manually route part of the TUTOR board
- Edit routed tracks

Getting started

In these exercises you manually route only a portion of the TUTOR board. Follow these steps to zoom in on the area where you will route the board.

Zooming in on the routing area

1. Place the pointer at (1.7500", 1.3000") and select WINDOW ZOOM.

2. Move the pointer to (4.5000", 3.1000") and select Window Zoom End. The enclosed area magnifies to fill the screen.
Highlighting a net  Before you start routing the TUTOR board, you need to identify module pads that share the same net. Follow these steps to highlight net N00034, which has two pads:

1. Select SET to display the Global Options dialog box.
2. Enable Find Highlights, then click OK. Find Highlights highlights an object that is located with the FIND command. If the object is a pad attached to a net, then the entire net is highlighted.
3. Select FIND. The Find entry box displays (figure 7-1).

![Figure 7-1. The Find entry box.](image1)

4. Enter ? in the entry box. The Find dialog box displays (figure 7-2).

![Figure 7-2 The Find dialog box.](image2)

5. Select N00034 from the Net Names list box, then select OK. Two pads highlight and the pointer jumps to one of the pads.
You can enhance the display of highlighted objects by following this procedure:

1. Select SET to display Global Options.

2. Note Copper To Copper Spacing to see if it is set at 0.0150, the default setting. This determines the allowable spacing between tracks, pads, vias, and testpoints.

3. Enable Show Highlight Guard. This displays the guard zone established with Copper To Copper Spacing.

4. Select Layer, then select Other Colors/Enables/... from the Layer dialog box.

5. Change the color for SolderMask Solder to a bright color, such as yellow, then select OK.

6. Select OK to close the Layer dialog box, then select OK to close the Global Options dialog box.

The highlighted pads now display an additional yellow band, which is the soldermask guard.

You can also locate routing targets by displaying a single ratsnest between a selected pad and its nearest pad on the same net. Follow these steps:

1. Place the pointer on the highlighted pad for R4.

2. Select X SHOW RATSNEST. A single ratsnest vector displays between the two highlighted pads.

3. Move the pointer away from any module pad and select X SHOW RATSNEST to clear the ratsnest from the display.
Creating a new copper tool

You need to create a new copper tool for one of the nets that you route later in this chapter. Follow these steps to create a copper tool that produces 30 mil wide tracks:

1. Select GO TO FUNCTION. The menu shown at right displays.

2. Select Copper Tool Editor. The Edit Copper Tool dialog box displays (figure 7-3).

3. Enter 0.0300 in the Width entry box.

4. Select Build Name. The entry box below the Copper Tool list box displays 0.0300" width.

5. Select Add. The new copper tool is added to the Copper Tool list box, as shown in the example at right.

6. Select OK.
Routing the board

In these exercises you manually route three nets, using a variety of routing techniques.

Setting conditions

1. Select SET to display Global Options.
2. Enable Show Copper And Guard While Routing. This displays the copper to copper guard zone while you are routing the track.
3. Select the Drawing Method droplist button to display the droplist box.
4. Select Draw Orthogonal 45 Degree Corners, then select OK to close the Global Options dialog box.
5. Select / OTHER until Solder Copper displays as the current routing layer.

Routing the first track

1. Place the pointer in the center of the highlighted pad for R4.
2. Select ROUTE Begin. A ratsnest vector displays, indicating the nearest target pad.
3. Move the pointer to the left to (3.0500", 1.8000"). A track segment, displaying its copper to copper spacing guard, follows the pointer.
4. Move the pointer straight down to (3.0500", 2.2750"), the center of the other highlighted pad (module 52). Notice that the segment automatically bends at a 45° angle.
5. Select End. The highlight guard disappears and the message “N00034: Complete” displays at the bottom of the screen, indicating that the net is completely routed. You just routed a track.
6. Move the pointer to an empty part of the display and select HIGHLIGHT. This removes the highlight from the pads.
Routing with vias

This time you will route net N00037 and place a via.

1. Select FIND. The previously routed net name, N00034, displays in the entry box.

2. Enter N00037. The two net pads are highlighted and the pointer jumps to one of the pads.

3. With the pointer in the center of the highlighted pad for C1, select ROUTE Begin. The ratsnest for the net displays.

4. Move the pointer straight up from the pad, then move it to the right to (2.7000", 1.7500").

5. Select / OTHER. A via is placed at the pointer location and the current routing layer switches from Solder Copper to Component Copper.

   Another ratsnest vector displays, connecting the highlighted pad for C1 and the via. This indicates that the connection up to the via is a subnet of the entire net.

6. Draw the new segment straight to the right, then down to (3.8000", 1.8000"), the other highlighted pad, and select End. The message “N00037: Complete” displays and the track is completed.
You can route with arc segments in Edit Layout. Arc segments can be used to tightly route around and through pads or other objects. Follow these steps to begin a track, then change to an arc segment and route around a pad:

1. Move the pointer to an empty part of the display and select HIGHLIGHT to remove the highlight from the previously routed pads.

2. Select FIND, then erase the net name in the entry box and enter ? to display the Find? dialog box.

3. Select N00038 from the Net Names list box and select OK. Three pads highlight.

4. Move the pointer to (3.0500", 2.9500"), the center of the highlighted pad for Q1.

5. Select ROUTE Begin and draw the segment to the right to (3.4500", 2.9500"), the highlighted pad for BT1.

6. Select Begin, then draw a segment straight up to (3.4500", 2.5500").

7. Select Begin, then select Set. The Edit Net Segment dialog box displays (figure 7-4). Note that the current selection for Drawing Method is Draw Orthogonal 45 Degree Corners.

Figure 7-4. The Edit Net Segment dialog box.
8. Select the **Drawing Method** droplist button to display the options in the droplist box.

9. Select **Draw Orthogonal Arc Corners**. The droplist box closes and the new selection displays.

10. Select **OK** to close the **Edit Net Segment** dialog box.

11. Move the pointer to the left, then up to (3.3500", 2.4500"). Notice that the segment following the pointer is now an arc. See the illustration at right.

12. Select **Begin**. Move the pointer up, then to the right to (3.4500", 2.3500"). The two arc segments follow the contour of the large pad. See the illustration at right.

13. Select **Begin**, then select **Set** to display the **Edit Net Segment** dialog box.

14. Select the **Drawing Method** droplist button and select **Draw Orthogonal 45 Degree Corners** from the droplist box. Select **OK**.

15. Move the pointer up, then right to (3.5000", 2.0250"), the center of the third highlighted pad. Notice that the segment now draws at a 45 degree angle.

16. Select **End** to complete this complex track.

17. Move the pointer away from the pad and select **HIGHLIGHT**. The three finished tracks look like figure 7-5.
Figure 7-5. The three completed tracks.
Performing a DRC check

One of the tracks you routed is too close to several pads. The allowed clearance between tracks, pads, and vias is specified in Copper To Copper Spacing in Global Options. The default value is 0.0150 inches (15 mils).

You locate routing and clearance violations by performing a DRC check. DRC stand for Design Rule Check.

You perform a DRC check two ways in Edit Layout: a whole board check and a block check, which checks a selected area. Follow these steps to perform a block DRC check:

Running a block DRC check

1. Move the pointer to (1.9000", 1.6000") and select GO TO FUNCTION Autorouter.
2. Select Block, then move the pointer and stretch the bounding box to (3.9000", 3.0000"), enclosing the routed area inside the block.
3. Select Block End. The menu shown at right displays.
4. Select Spacing/DRC Check Block. The dialog box shown at right displays.
5. Select OK to ALL. The display zooms in to the block area. In a moment the dialog box shown at right displays, showing the number of spacing/DRC errors found within the block.
6. Select OK. DRC markers display, pointing to the violated objects. See figure 7-6.
7. Press <Esc> to dismiss the autorouter menu.
Chapter 7: Routing the TUTOR board

Identifying DRC violations

A DRC marker looks like the example shown at right. The converging lines point to the center of the violated object.

You identify DRC violations using two commands: INQUIRE and JUMP. Both methods are described in the following procedures.

Using INQUIRE

1. Select LAYER to display the Layer dialog box.
2. Select Comment Layer, then select OK. When Comment Layer is the current layer you can easily select DRC markers because all other objects assigned to layers are not selectable.
3. Move the pointer to (2.5500", 1.8500") and select INQUIRE. The dialog box shown below displays.

   ![Segment Spacing Error Dialog Box]

The dialog box identifies the type of violation, and the coordinates for the violated object.
4. Select OK.
5. Move the pointer to (2.7000", 1.7500"), the center of the via, and select INQUIRE. The dialog box shown at right displays. This dialog box tells that the copper to copper spacing zone of the via is violated by another object (a via spacing error).

6. Select OK.

Using JUMP

1. Select JUMP to display the Jump To dialog box.

2. Select DRCs to display all design errors in the DRCs list box, as shown in figure 7-7.

3. Select Segment Spacing Error 3, then select OK. The pointer jumps to that DRC marker. Select INQUIRE to verify that it is Segment Spacing Error 3, then select OK.

4. Select JUMP and select the other items in the DRCs list box to see how you can locate a specific violation.

Viewing the violated areas

You can change the way objects on copper layers display so you can view the areas where the design violations occurred. Follow these steps:

1. Zoom in on the area containing the violations by placing the pointer at (2.4000", 1.6500") and selecting WINDOW ZOOM.

2. Move the pointer to (3.5000", 1.9500") and select Window Zoom End. The display magnifies to the selected area.

3. Select SET to display the Global Options dialog box.
Chapter 7: Routing the TUTOR board

4. Enable Show Copper And Guard While Drawing, then select OK. All copper objects display their copper to copper spacing boundaries, as shown in figure 7-8.

![Figure 7-8. Copper to copper spacing of violated objects.](image)

The guard band for the track extends inside the guard bands for the four marked pads. The guard band for the marked via extends inside the adjacent module pad.

5. Select SET to display Global Options.

6. Disable Show Copper And Guard While Drawing, then select OK. The copper to copper spacing does not display.

The next section describes how to correct DRC errors.
After manually routing a board, you may need to edit your work to correct DRC errors and make design modifications.

Follow these procedures to correct the DRC errors described earlier in Performing a DRC check:

1. Select ZOOM Set Scale, then enter 3.25 in the Scale entry box and select OK. The display zooms out so you can view the entire track that is violating the pads.
2. Select / OTHER until Solder Copper is the current layer.
3. Move the pointer to (2.2000", 2.0250"), a routed pad.
4. Select ROUTE Begin. A ratsnest vector displays, between the pad and the via, indicating the nearest subnet routing connection. See the PC Board Layout Tools 386+ Reference Guide for a description of subnets.
5. Draw the segment straight up, then to the right to (2.8000", 1.9000") and select / OTHER to place a via and switch the routing layer to Component Copper.
6. Draw the segment to the right to (3.9000", 1.9000"), then select New to complete the segment.
7. Move the pointer to (3.8000", 1.8000"), the routed pad for R1, and select Begin to start another segment.
8. Draw the segment to (3.7000", 1.9000") and select End. The message "N00037: Complete" displays and the new track is complete, as shown in figure 7-9.

Figure 7-9. The old and new tracks for net N00037.
Chapter 7: Routing the TUTOR board

Deleting the offending track

1. Place the pointer at (3.7500", 1.7500"), the junction where the horizontal segment for the old track angles into the pad for R1.

2. Select DELETE to delete one of the segments that intersects at the pointer location.

3. Select DELETE to delete the other intersecting segment.

4. Select / OTHER to select Solder Copper as the current layer.

5. Move the pointer to (2.4750", 1.7500"), the junction where the horizontal segment for the old track angles into the pad for C1.

6. Select DELETE twice to delete the two segments.

7. Place the pointer at (2.7000", 1.7500"), the center of the via with a DRC marker, and select DELETE.

8. Select ZOOM Refresh to redraw the display. The old track is deleted, but the DRC markers remain.

Running another DRC check

Now you can check your work to see if the new track has enough clearance from the pads. Follow these steps:

1. Select GO TO FUNCTION Autorouter.

2. Select Whole Board. The menu shown at right displays.

3. Select Spacing/DRC Check Whole Board. The entire board displays, and in a moment the dialog box shown at right displays. The number of DRC errors is zero.

\[ \text{NOTE: You can also define a block around the modified area and select Spacing/DRC Check Block.} \]
4. Select OK to close the dialog box. Pan the display so you can view the routed area. The DRC markers are removed.

5. Press <Esc> to dismiss the autorouter menu.

6. Place the pointer at (1.9500", 1.5000") and select WINDOW ZOOM.

7. Move the pointer to (4.0000", 3.1000") and select Window Zoom End so the display looks similar to figure 7-10.

![Diagram of edited routing area](image.png)

Figure 7-10. The edited routing area.
Chapter 7: Routing the TUTOR board

Deleting a stub

The track you just completed has an unneeded extra segment, called a "stub" (see figure 7-10). Delete the stub using one of the two methods described below:

1. Select / OTHER to select Component Copper as the current layer.
2. Place the pointer at (3.9000", 1.9000"), the end of the stub, and select DELETE. Remember that deleted objects are retained in the undelete buffer.

Or, use this method:

1. Select QUIT Cleanup Stubs. The dialog box shown below displays.

```
Are You Sure?

Cleanup Stubs?
Cancel OK

NOTE: Deleted stubs will be saved in Undelete buffer
```
2. Select OK. The stub is deleted from the board, but it is retained in the undelete buffer.

Deleting and undeleting a track

It is easy to delete and undelete an entire track, including vias, that connects two pads. Follow this procedure:

1. Place the pointer at (3.3000", 1.9000"), which positions it on a Component Copper segment of the track connecting C1 and R1.
2. Select TRACK DELETE. The entire track, including the via, is deleted.
3. Select UNDELETE. The entire track is recovered.

\[ NOTE: \text{You can also use TRACK DELETE to delete sections of a track that have been replaced by rerouted segments. See Changing a track path for an example.} \]
Changing a track path

As you route your board, you usually have tracks that need to be rerouted to improve the board layout.

1. Place the pointer at (3.8000", 1.8000"), the center of the routed pad for R1, and select ROUTE Begin.

2. Draw the segment up, then left to (3.4500", 1.6500") and select Begin.

3. Draw the segment to (3.2000", 1.9000") and select End. The track looks like the illustration below.

4. Place the pointer at (3.7250", 1.8750"), the first segment of the original track connected to R1. See the illustration at right.

5. Select TRACK DELETE. The track segments delete from the pad to the point where the new track intersects the original segments.

6. Select ZOOM Refresh to redraw the display.
Changing track width

Some of the nets in the design require wide tracks because of the types of signals they carry. Wide power and ground nets are common because those signals usually require larger copper areas.

Follow these steps to change the track width for a routed net to 0.0300" (30 mils):

1. Move the pointer to \( (3.0500", 2.1000") \), which positions it on a track. Note that the track color indicates it is on the Solder Copper layer.

2. Select `/ OTHER` to change the current layer to Solder Copper.

3. Select `EDIT`. The Edit Net Segment dialog box displays (figure 7-11).

4. Select the Copper Tool droplist button to display the droplist box. `Standard Route` is highlighted.

5. Select 0.0300" width. The droplist box closes and the new copper tool displays in the dialog box.

6. Enable `Apply Copper Tool to All Net Segments & Arcs`, then select `OK`. All segments for the routed net are redrawn using the selected 30 mil copper tool.
Saving your work

It is time to save your work again before you go to the next chapter. Select QUIT Update Board File to save the file as TUTOR.BD1.

Summary

In this chapter you learned how to identify routing targets and manually route part of the board. You also learned how to perform a DRC check and edit tracks.

The next chapter tells you how to automatically route the TUTOR board using the Edit Layout autorouter.
About autorouting

Autorouting a board can save you many hours of manual routing work by automatically routing connections.

The autorouter in Edit Layout has many routing options that you can select from to successfully route your board and enhance the routing results.

In this chapter you learn how to:
- Place an autoroute zone
- Lock an existing route
- Set routing conditions for a net
- Specify autoroute options
- Autoroute a board
- Optimize an autorouted board

Preparing for autorouting

Before autorouting the board, you need to do these tasks:
- Place an autoroute zone to specify the area of the board to autoroute
- Lock a manually routed net so it will not be rerouted
- Specify a different copper tool for autorouting the ground net
- Set autorouting options
Placing an autoroute zone

The autorouter requires an autoroute zone to define the area of the board where the autorouter places tracks on the enabled layers. If you do not define a zone, a zone is created by the autorouter before routing begins.

In this tutorial you create an autoroute zone that leaves a 50 mil border around the edge of the board. The area outside the autoroute zone will not contain any copper tracks because all tracks stay inside the autoroute zone.

Follow these steps to create an autoroute zone:

1. Select ZOOM Set Scale, then enter 13 in the Scale entry box and select OK. The entire board displays.

2. Select * LAYER to set All Layers as the current layer. The autoroute zone is placed on all board layers when All Layers is selected.

3. Select PLACE Autoroute Zone.

4. Select Set. The Edit Zone Segment dialog box displays.

5. Select the Drawing Method droplist button to display the droplist box. Draw Orthogonal 45 Degree Corners is the current selection.

6. Select Draw Orthogonal 90 Degree Corners, then select OK.

7. Move the pointer to (0.8000", 0.3000"), then select Begin.

8. Draw the zone segment to the right to (7.0750", 0.3000") and select Begin.

9. Draw the segment down to (7.0750", 3.3250") and select Begin.

10. Draw the segment left to (0.8000", 3.3250") and select Begin.

11. Draw the segment up to (0.8000", 0.3000") and select End to complete the autoroute zone.
Locking an existing route

To prevent the autorouter from changing a critical route that you routed manually, you lock the route using the following procedure:

1. Move the pointer to (0.7000", 0.2000"), the upper left corner of the board, and select WINDOW ZOOM.

2. Move the pointer to (3.9000", 3.4000") and select Window Zoom End, magnifying the left half of the board.

3. With All Layers selected, place the pointer at (3.3000", 2.9500"). This places the pointer on the segment connecting module Q1 and the battery, BT1.

4. Select EDIT. The Edit Net Segment dialog box displays.

5. Select Net Properties. The Edit Net Properties dialog box displays (figure 8-1). N00038, the net name for the selected track, is selected in the Net Names list box.

6. Enable Lock Existing Routes, then select OK. The autorouter will not reroute the selected net.

7. Select OK to close the Edit Net Segment dialog box.
Setting routing conditions for a net

You can give the autorouter specific instructions on how to route a net. The following procedures set routing conditions for the GND net.

Specifying a copper tool

The GND net should have wide copper tracks so proper grounding is achieved. Follow these steps to assign a 30 mil copper tool to the net:

1. Select GO TO FUNCTION. The menu shown at right displays.
2. Select Net Property Editor. The Edit Net Properties dialog box displays.
3. Select GND from the Net Names list box. This is the net name for the ground net.
4. Select the Nominal Copper Tool droplist button to display the droplist box.
5. Select 0.0300" width from the droplist box. The selection displays in the Nominal Copper Tool list box.

Excluding vias

Follow these steps to specify that the GND net should be routed without using vias:

1. With the Edit Net Properties dialog box still displayed, enable Do Not Allow Vias.
2. Select OK to accept the selections for the GND net and close the Edit Net Properties dialog box.
Setting autorouter options

You select autorouter options in Edit Layout to specify the type of routing pattern and the direction that the routing sweep window moves. The options you select depend on the way the board is laid out, and the types of modules placed on the board.

Procedures are described in the following sections for setting autorouting options for the TUTOR board.

Setting an autoroute method

1. Select GO TO FUNCTION Autorouter, then select Set. The Autoroute Options dialog box displays (figure 8-2).

2. Leave Standard selected in Autoroute Method.

Setting a sweep routing direction

You specify the direction of travel for the sweep window based on the density of the surrounding circuitry. Ideally, the sweep window should work its way from the most dense area of the board to the least dense.

1. Select Down, Right in Sweep Routing Direction.

2. Select OK to accept the autoroute options and close the dialog box.
Authorouting the board

You can autoroute an entire board, or you can autoroute a selected area and manually route the rest of the board. The methods you use depend on the routing requirements of the design.

Authorouting a section of the board

1. Place the pointer at (0.8000", 0.3000"), the upper left corner of the autoroute zone, and select Block.

2. Move the pointer to (2.7000", 1.4000") and select Block End. The menu shown at right displays.

3. Select Area Autoroute.

The selected autoroute area magnifies to fill the screen and the autorouter begins routing tracks between pads. Tracks that connect to pads outside the block route to the edge of the block.

During autorouting, routing statistics display at the bottom of the screen, as shown below.

Standard-Window Completed 15 Failed 0 Remaining 11

These statistics list the current autorouting options, and the number of completed, failed, and remaining connections. The statistics update as routing connections are made.

\[\text{NOTE: You can terminate an autorouting session at any time by pressing \(<\text{Esc}\)>.}\]

When the block autoroute is finished, the Finished dialog box displays (figure 8-3).

![Finished dialog box](image)

Figure 8-3. The Finished dialog box.
Chapter 8: Autorouting the TUTOR board

The Finished dialog box displays total routing time and connection statistics.

4. Select OK to close the dialog box. The ratsnest for the whole board displays, and the ratsnest for the incomplete routed tracks displays at their unconnected endpoints. See figure 8-4.

![Figure 8-4. The finished autorouted area.](image)

Note the wider tracks. These are the 30 mil GND net tracks that you specified earlier in this chapter in Setting routing conditions for a net.

5. Press <Esc> to exit the Autorouter menu.

6. Select ? CONDITIONS to display the Conditions dialog box (figure 8-5). Conditions displays additional routing information, such as the number of incomplete nets, the number of vias placed during autorouting, and the total length of all routed tracks.

![Figure 8-5. The Conditions dialog box.](image)

7. Select OK to close the dialog box.
Autorouting the whole board

Before you autoroute a complex board you should review the layout to determine where you want the autorouter to start. The autorouter may complete all connections in one routing pass if you direct its movement properly.

If there are incomplete connections after the first rigorous routing pass, you select a more liberal routing option for the second pass to complete the remaining connections.

For example, you could select Standard option for the first pass, then select the more liberal Maximal for the second routing pass. After all connections are complete, you could select Via Reduction to reduce the number of vias.

The TUTOR board is not a complex layout, but you will use routing methods that are used on complex boards so you can become familiar with the options.

Setting a sweep window

You set a sweep window to define the routing origin and window size. You usually set a sweep window to initially route in the densest area of the board, then follow a sweep pattern to the least dense area. Follow these steps to set a sweep window:

1. Select ZOOM Set Scale, then enter 13 in the Scale entry box and select OK. The entire board displays, showing the ratsnest for all pads.

2. Select GO TO FUNCTION Autorouter.

3. Select Whole Board. The menu shown at right displays.

   Autoroute Whole Board  
   Spacing/DRC Check Whole Board  
   Set Sweep Window

4. Select Set Sweep Window.

5. Place the pointer at (7.0750", 0.3000"), the upper right corner of the autoroute zone, then select Sweep Window Begin.

6. Move the pointer left, then down to (5.0500", 1.9000") and select Sweep Window End. The menu shown above displays again.
Begin autorouting

1. With the menu still displayed, select **Autoroute Whole Board** to begin autorouting.

The display zooms in to the initial sweep window and routes connections. The display pans as the autorouter moves to the next sweep window and routes connections.

Statistics describing the routing status of the current sweep window display at the bottom of the screen, as shown in the example below.

```
Standard-Maze: Completed 15 Failed 0 Remaining 8
```

When the autorouter is finished, the **Finished** dialog box displays. Note that there are no incomplete connections or partial routes.

2. Select **OK** to close the dialog box.

3. Select **? CONDITIONS** to display the **Conditions** dialog box. Note that the number of incomplete nets is zero. Also note the number of vias used on the board and the total route length.

4. Select **OK** to close the dialog box.

5. Verify that the manually routed track that you specified as a locked track remains unchanged after the autorouting process.

Move the pointer and pan the display until the pointer is at (3.4500", 2.9500"), the lower large pad for battery BT1. The track including the arc segments around the other BT1 pad is unchanged. Refer to figure 8-6.

![Locked track](image)
Via reduction

After the autorouter completes all connections you may want to reroute the board to eliminate some of the vias. Follow these steps to automatically remove vias from the board:

1. Select Set to display the Autoroute Options dialog box.
2. Select Via Reduction in Autoroute Method. Up, Left is automatically selected in Sweep Routing Direction.
3. Select OK.
4. Select Whole Board, then select Autoroute Whole Board to begin the via reduction process.

The display pans when the via reduction process in the initial sweep window is complete, and the sweep window moves to the next section of the board.

Statistics describing the routing status of the current sweep window display at the bottom of the screen.

When the via reduction process is complete the Finished dialog box displays.

5. Select OK to close the dialog box.
6. Select ZOOM Set Scale, then enter 13 in the Scale entry box and select OK. Pan the display to view the entire board, as shown in figure 8-7.

Figure 8-7. The autorouted TUTOR board, after via reduction.
Additional processing

Performing another via reduction may remove more vias, but the additional rerouting of tracks could increase the total track length on some boards.

Follow these steps to perform another via reduction:

1. Select Whole Board, then select Autoroute Whole Board from the menu. The autorouter reroutes tracks and removes additional vias. When the process is complete, the Finished dialog box displays.

2. Press <Esc> to exit the Autorouter menu.

3. Select ? CONDITIONS to display the Conditions dialog box. Note the number of vias.

4. Select OK to close the dialog box.

Erasing all routes

You can completely erase all tracks and vias if you want to autoroute the board using different options. This part of the tutorial is optional. Follow these steps:

1. Select QUIT Erase All Routes. The dialog box shown at right displays.

2. Select OK. All tracks and vias are deleted, but saved in the undelete buffer.

If you do not want to retain the objects in the undelete buffer, you can flush the undelete buffer by selecting QUIT Flush Undelete Buffer and selecting OK in the displayed dialog box.

3. Repeat the steps in this chapter, starting with Autorouting the whole board. Select different autorouting and sweep window options.
Finishing the layout

Now that the board is completely routed, you need to move the silkscreened reference designators so they are not positioned over any tracks on the Component Copper layer.

You also need to place an assembly outline around all objects in the board file and add dimensions. The assembly outline defines the printing and plotting area when you print and plot the board in Chapter 9: Printing and plotting the TUTOR board.

Moving reference designators

1. Select SET to display the Global Options dialog box.
2. Enable Allow Edits Of Module Objects so you can move the reference designators, but not move the rest of the module objects.
3. Select Layer to display the Layer dialog box.
4. Select SilkScreen Component, then select OK to set SilkScreen Component as the current layer.
5. Select OK to accept the changes and close the Global Options dialog box.
6. Place the pointer on a reference designator and select MOVE.
7. Move the reference designator so it is close to the module, but not positioned over a pad, via, or track on the Component Copper layer.

If you need to change the grid size to achieve better placement, select Set to display the Set Block Parameters dialog box. Select Global to display the Global Options dialog box, then enter the new grid size in Grid Size. Select OK to close Global Options, then select OK to close Set Block Parameters. The grid size is changed.

8. Select Place.
9. Pan to adjacent parts of the board and move all reference designators that are on the SilkScreen Component layer.
10. To move the reference designator for Q1, which is on the SilkScreen Solder layer, select LAYER to display the Layer dialog box. Select SilkScreen Solder, then select OK and move the reference designator.

11. Select ZOOM Set Scale, then enter 1.3 in the Scale entry box and select OK. Pan the display to view the entire TUTOR board.

**Placing an assembly outline**

1. Select LAYER. The Layer dialog box displays.

2. Select Assembly Drawing, then select OK to set the Assembly Drawing layer as the current layer.

3. Move the pointer to (0.0000", 0.0000") and select PLACE Outline Begin.

4. Draw the outline segment to the right to (7.7000", 0.0000") and select Begin.

5. Draw another segment down to (7.7000", 5.0000") and select Begin.

6. Draw another segment left to (0.0000", 5.0000") and select Begin.

7. Draw another segment up to (0.0000", 0.0000") and select End to complete the outline.

You can include additional objects on the Assembly Drawing layer, such as documentation, a title block, and even a company logo if the logo is drawn in the library editor and placed as a module.
Placing dimensions

Edit Layout has many options for placing dimensions. Follow these steps to place dimensions on the board:

1. Make sure Assembly Drawing displays as the current layer.
2. Place the pointer at $(0.7500", 3.5000")$. The pointer is below the bottom left corner of the board outline.
3. Select PLACE Dimension. The object shown at right displays at the pointer location.
4. Select Begin and move the pointer to the right. The dimension value changes as you move the pointer, indicating the distance from where you selected Begin. Two end bars and dimension arrows display.
5. Move the pointer down. The dimension pivots on its beginning point but the text remains horizontal, as shown in the example at right.

![Figure 8-8. The Edit Dimension Text dialog box.](image)
7. Enter 0.2500 in the End Bar Height entry box and select OK. The end bars are shortened from one inch high to a quarter of an inch, as shown at right.

8. Move the pointer up, then to the right. As you move the pointer the current dimension length, its rotation angle, and an alignment aid display in the lower right part of the screen, as shown in the example below.

\[2.980 \ 351.81^\circ \ X\]

An “X” displays when the dimension is at an angle other than 0°, 90°, 180°, 270°, or 360°. A “+” displays when the dimension is rotated to one of those five angles.

9. Place the pointer at (7.1250", 3.5000") and select End to complete the dimension.

10. Press <Esc> to dismiss the additional dimension text.

11. With the pointer still at (7.1250", 3.5000"), select MOVE.

12. Move the dimension text down to (7.1250", 3.5500") and select Place. This moves the end bars away from the board outline. The first dimension is complete.
Placing the second dimension

1. Move the pointer to (7.1250", 3.3750") and select PLACE Dimension Begin.

2. Select Set to display the Edit Dimension Text dialog box.

3. Enable Rotate text with dimension, then select OK. As you move the pointer, the text rotates to follow the angle of the dimension lines. See the example at right.

4. Move the pointer to (7.1250", 0.2500") and select END.

5. Press <Esc> to dismiss the additional dimension text.

6. Move the pointer to (7.2500", 0.2500") and select MOVE.

7. Move the dimension to the right to (7.4250", 0.2500") and select Place.

The completed TUTOR board looks like figure 8-9.

Figure 8-9. The completed TUTOR board.
Saving your work

It is time to save your work again before you go to the next chapter. Select QUIT Update Board File to save the file as TUTOR.BDI.

△ NOTE: After you save your work you can compare your board with DEMO.BD1, which was prepared using the procedures in this tutorial.

Summary

In this chapter you learned how to set routing conditions for individual nets. You learned how to define a sweep window, then applied these settings when you autorouted the TUTOR board. You also learned how to optimize your autorouted design.

The next chapter gives you instructions for printing and plotting the TUTOR board.
There are two basic types of output devices you can use with Edit Layout: printers and plotters. These devices are categorized by the type of input they require.

If a device accepts raster commands, it is a printer. A raster is an array of dots. When you draw a line to a raster device, you must specify each and every dot.

If a device accepts vector commands, it is considered a plotter. A vector is a series of points with a specific function defined. For example, a line has a beginning point and an ending point. A circle has a center and a radius.

The device needs to know what the vector information is, but does not need every point along the vector.

You can print working copies of your designs on dot matrix printers and laser printers. The output quality depends on the maximum printer resolution.

You can plot final artwork for your designs on pen plotters, laser photoplotters, and Gerber photoplotters.

All plotting functions are contained in Edit Layout. You configure a printer driver when you configure Edit Layout in the ESP design environment. You must have a printer driver configured before you can print in Edit Layout.

See Chapter 4: Introducing Edit Layout for information on configuring your printer driver in the ESP design environment.
You produce a print or plot in **Edit Layout** by selecting layers and objects, then assigning those layers and objects to a “page”. You can specify how objects will print on each page, and you can print one page at a time or print all pages.

You print the currently loaded board file with these settings. If you want to print another board file using the same settings, load the file into **Edit Layout**. The print settings remain intact for the new file.

**Printing**

Follow these steps to display the **Printing and Plotting** dialog box:

1. Select **GO TO FUNCTION**. The menu at right displays.

2. Select **Printing and Plotting**. The **Printing and Plotting** dialog box displays (figure 9-1).

![Figure 9-1. The Printing and Plotting dialog box.](image-url)
Before you can print your board file you need to configure printer options so Edit Layout can find the device. You also need to configure the page settings. Follow these steps:

1. Select Driver. The Driver Configuration dialog box displays (figure 9-2).

![Driver Configuration dialog box]

Figure 9-2. The Driver Configuration dialog box.

This tutorial assumes you have a laser printer that is assigned to printer port LPT1, printing in portrait mode, and the configured printer driver is HPLASER4.DRV.

The following settings are already configured in the Driver Configuration dialog box:

- HPLASER4.DRV displays in the Raster Device list box, and is the default selected printing device.
- Narrow is selected as the print orientation in Destination Device.
- File is selected in Destination, and the contents of the current working directory display in the Files list box.
- Append is selected in Destination. Multiple print jobs are appended to a single printer file when this option is selected.
2. Select LPT1 in Destination to specify you are printing directly to the printer through LPT1. The File options do not display.

3. Leave Magnification and Overlap at their default settings.

4. Select OK. The Printing and Plotting dialog box displays.

Configuring pages

When you print a board file in Edit Layout you need to consider these factors:

♦ The board layers that you want to print
♦ The layers and objects that you want to include on each printed page
♦ How you want the objects to appear on the page

You define three pages in this chapter: a TOP COPPER LAYER page, a BOTTOM COPPER LAYER page, and an ASSEMBLY DRAWING page.

Building the TOP COPPER LAYER page

Follow these steps to select the layers and objects needed to create the TOP COPPER LAYER page:

1. Select the Layer droplist button to display the droplist box.

2. Select Component Copper.

3. Select the Component Copper objects that will appear on the page by enabling these options:

♦ Pads
♦ Nets
♦ Vias
♦ Layer Markers
♦ Alignment Targets
♦ Outlines
4. Select the way the layer marker text appears on the print by enabling these options:

- Outline Text
- Stick Text

These options print the layer marker text as single line segments, rather than as wide, filled lines.

5. Select Insert for Page Contents. The selected options display in the Page Contents list box.

6. Place the pointer in the entry box below the Pages list box and click the left mouse button. Enter TOP COPPER LAYER.

7. Select Insert for Pages. TOP COPPER LAYER displays in the Pages list box. The page name TOP COPPER LAYER is associated with the items listed in Page Contents. The dialog box looks like figure 9-3.

![Figure 9-3. The configured TOP COPPER LAYER page.](image-url)
The TOP COPPER LAYER page is built, now follow these steps to build the BOTTOM COPPER LAYER page:

1. Select Delete for Page Contents to clear the entry in the Page Contents list box. The enabled options and selected layer remain unchanged.

2. Select the Layer droplist button to display the droplist box.

3. Select Solder Copper from the droplist box.

4. Do not disable any of the enabled options. The selections also apply to objects on the Solder Copper layer.

5. Enable Zones to print the fill zone that you placed on the Solder Copper layer.

6. Enable Copper Pour to print the fill zone as a solid, filled area.

7. Select Insert for Page Contents. The selected options display in the Page Contents list box.

8. Place the pointer in the entry box below the Pages list box and click the left mouse button. TOP COPPER LAYER displays in the entry box.

9. Press <Alt><Backspace> to clear the entry box, then enter BOTTOM COPPER LAYER.

10. Select Append for Pages to add BOTTOM COPPER LAYER below TOP COPPER LAYER in the Pages list box. The items in the Page Contents list box are associated with BOTTOM COPPER LAYER.
Building the ASSEMBLY DRAWING page

The BOTTOM COPPER LAYER page is complete, now you need to build the ASSEMBLY DRAWING page, which contains the board outline, module outlines, pads, reference designators, and the outline on the Assembly Layer.

Follow these steps to build the ASSEMBLY DRAWING page:

1. Select Delete for Page Contents to delete the entry in the Page Contents list box.
2. Select All Layers from the Layer droplist box.
3. Disable these options:
   - Copper Pour
   - Nets
   - Vias
   - Layer Markers
   - Zones
4. Enable these options:
   - Outline Pads
   - Text
5. Select Insert for Page Contents to add the selections to the Page Contents list box.
6. Enter ASSEMBLY DRAWING in the entry box below the Pages list box.
7. Select Append for Pages to add ASSEMBLY DRAWING to the bottom of the list of items in the Pages list box.

The three pages are configured, as shown in figure 9-4.
Figure 9-4. The configured printer pages.

**Printing pages**  Now that you have configured the three pages, you are ready to print them. You can print all pages, or you can print selected pages. Follow the procedures listed below.

**Printing all pages**  Select Begin All to print all pages listed in the Pages list box. The pages print in the sequence they display in the list box.

The Printing and Plotting dialog box is replaced by a screen displaying the page being printed and the size of the print window. When the first page is done, the next page displays. When all pages are sent to the printer, the Printing and Plotting dialog box displays.
Chapter 9: Printing and Plotting the TUTOR board

Printing selected pages

1. Select **TOP COPPER LAYER** in the **Pages** list box.

The item in the **Page Contents** list box is the page contents for the **ASSEMBLY DRAWING** layer, which was the last page added to **Pages**. You need to replace the item in **Page Contents** with the page contents of **TOP COPPER LAYER**.

2. Select **Edit for Pages**. The item in **Page Contents** changes to the page contents of the **TOP COPPER LAYER** page. The selected layer and enabled options change to reflect the settings of the new item in **Page Contents**.

3. Select **Begin**. Only the page contents of **TOP COPPER LAYER** print.

The three printed pages look like figures 9-5 through 9-7.

![Diagram](image)

*Figure 9-5. The **TOP COPPER LAYER** page.*
Figure 9-6. The BOTTOM COPPER LAYER page

Figure 9-7. The ASSEMBLY DRAWING page.
Saving a printer setup

You can save your page configurations to a file, then load the configurations when you want to print the pages. Follow these steps:

1. With the three pages listed in the Pages list box, select Save. The Save Print/Plot Setup to File dialog box displays (figure 9-8).

2. Enter TUTOR386.SET in the entry box below the Files list box.

3. Select OK. The page configurations are saved to the file and the Printing and Plotting dialog box displays.

Figure 9-8. The Save Print/Plot Setup to File dialog box.
Loading a printer setup

Follow these steps to load the previously saved setup file into the Printing and Plotting dialog box:

1. With TOP COPPER LAYER already highlighted in Pages, select Delete for Pages. The entry is deleted.

2. Select BOTTOM COPPER LAYER and select Delete for Pages, then select ASSEMBLY DRAWING and select Delete for Pages. All entries are deleted from the Pages list box.

3. Select Delete for Page Contents to delete the highlighted entry. The Pages and Page Contents list boxes are now empty.

4. Select Load. The Load Print/Plot Setup from File dialog box displays, and TUTOR386.SET is selected (figure 9-9).

5. Select OK. The configured pages load and display in the Pages list box.

Figure 9-9. The Load Print/Plot Setup from File dialog box.
Plotting

You create high resolution images of your board file pages by sending them to a plotting device, or plotter.

Edit Layout supports the following vector plotting devices:

- Fire 9xxx
- Gerber (274-D)
- Gerber (274-X)
- HP-GL/2
- Postscript

Refer to the procedures below to plot the three pages to a specific vector device.

Gerber (274-X)

1. Select Driver. The Driver Configuration dialog box displays.

2. Select Vector Device, then select the droplist button to display the droplist box.

   The options shown at right display below the Vector Device droplist box.

4. Select a format for the photoplotter.

5. Select the configured printer port in Destination, then select OK to display the Printing and Plotting dialog box.

6. Select Begin All to photoplot the three pages.

All aperture information required by the Gerber (274-X) photoplotter is embedded in the board file.
HP-GL/2

1. Select Driver. The Driver Configuration dialog box displays.

2. Select Vector Device, then select HPGL2 from the droplist box. The Pen entry box shown at right displays below the Vector Device droplist box.

3. Enter a number that corresponds to the pen slot in the carrousel containing the pen you will use to plot the layout.

4. Select the configured printer port in Destination, then select OK to display the Printing and Plotting dialog box.

5. Select Begin All to plot the three pages.

Postscript

1. Select Driver to display the Driver Configuration dialog box. A postscript printer should be connected to your computer.

2. Select Vector Device, then select PostScript from the droplist box.

3. Select the configured printer port in Destination, then select OK to display the Printing and Plotting dialog box.

4. Select Begin All to print the three pages.
This appendix provides detailed information about the Phar Lap memory extender.

About the CFIG386 utility

All Phar Lap programs support the use of command line switches to override the default operation of the program. The CFIG386 utility allows you to customize a Phar Lap program by specifying command line switches to be automatically processed every time the program is run. Judicious use of the CFIG386 utility will allow you to avoid having to type commonly used switches every time you run a program.

When a Phar Lap program is run, the program defaults are first set up. Then, any switches configured into the program are processed from left to right. If any conflicting switches are given, the last switch processed takes precedence. Thus, switches configured into a program can be overridden with command line switches.

How to run CFIG386

The command line format for CFIG386 is the following:

```
CFIG386 PCB386.EXE [switches]
```

The first command operand is the name of the .EXE file that you want to configure, in this case, PCB386.EXE. The file must be a Phar Lap program. The operand must follow the standard DOS filename conventions. For example:

```
\MYLI99B\386ASM
Run386
A:386LINK.EXE
```
If a filename extension is not specified, then "*.EXE" is assumed.

Following the filename, you list one or more command switches. The switches are given in the same format they are given on the command line for the program being configured. CFIG386 determines which Phar Lap program is being configured from a program signature in the program's configuration block. The specified switches are added to the configuration block of the program, after any switches that are already there from any previous configurations. For example:

```
CFIG386 386ASM -NOLIST -8086
CFIG386 RUN386 -MINREAL 100h -MAXREAL 400h
CFIG386 386DEBUG -BO
```

CFIG386 recognizes one switch that alters its own processing. This is the -CLEAR switch. The -CLEAR switch causes CFIG386 to erase the current contents of the program's configuration block. Any switches specified after the clear switch are added to the just-cleared configuration block. For example:

```
CFIG386 386LINK -CLEAR
CFIG386 MINIBUG -CLEAR -CEMM
```

If CFIG386 is run with no switches on the command line, it will display the current contents of the program's configuration block. For example:

```
CFIG386 386ASM
CFIG386: 1.ls--Copyright (c) 1986,1987 PharLap Software, Inc.
Phar Lap program type: 386|ASM Version 1.1s
Configured switch values:
   -INCLUDE \INCLUDES\
   -TWOCASE
   -386p
```
NOTE: CFIG386 does NOT check the values of any of the switches or switch parameters which it stores in the program's configuration block. Thus, it is possible to configure invalid switch values into a Phar Lap program. You should ALWAYS run the program after configuring it to make sure that the configured switch values have the desired effect.

**Error messages**

Several errors can be reported by the CFIG386 utility. The error messages and their causes are listed below.

- **Filename too long: filename**
  The filename specified is too long for an internal CFIG386 buffer.

- **System Error**
  An internal error in the CFIG386 utility has been detected. Save a copy of the program which causes the error and contact Par Lap.

- **Unable to open: filename: reason**
  An error of some sort prevented filename from being opened. The reason gives more detail as to why the error occurred.

- **Unable to read configuration for: filename**
  The .EXE file being configured has a valid Par Lap signature but an invalid switch block of some sort. This error should never occur with a Phar Lap program. Save a copy of the program and contact Par Lap.

- **Out of configured space on switch: switchname**
  The internal switch buffer in the program being configured has overflowed. Switch switchname and all following switches will not be added to the default switch table.

- **Unable to save new configured switch values in executable file**
  The program file is either write-locked or has been corrupted between the time the switch block was read from the file and when it was written back.
Unable to read config block from: filename

The .EXE file being configured has a valid Phar Lap signature but an invalid switch block of some sort. This error should never occur with a Phar Lap program. Save a copy of the program and contact Par Lap.

Not a Phar Lap program: filename

The specified .EXE file does not have one of the Phar Lap program signatures that the CFIG386 utility expects to find in the program's configuration block.
Command line switches are used to change the default operation of 386|DOS-Extender. By default, 386|DOS-Extender:

- Uses all memory below 640K for your application and leaves no free real mode memory for other applications.
- Allocates four kilobytes for the data buffer used on DOS and BIOS function calls.
- Allocates four buffers of one kilobyte each to be used for stack memory when switching from protected mode to real mode.

Command line switches begin with a minus sign ("-" ) character, followed by the name of the switch. There are two forms of each switch name: a long form and a short form. Any argument to the switch must immediately follow the switch name, with a space as a separator. If conflicting switches are given on a command line, the last (right-most) switch takes precedence.

Some of the command line switches take a number as an argument. By default, the number is considered to be a decimal (base 10) number.

Hexadecimal (base 16) numbers may be specified by appending the character "h" or "H" to the number. The following two examples both give the same number as an argument to the switch -MAXREAL:

```plaintext
run386 -maxreal 200h hello
run386 -maxreal 512 hello
```

386|DOS-Extender switches may be specified in three different ways: (1) some switches may be given when the program is linked, (2) switches may be configured into the 386|DOS-Extender task image (RUN386.EXE) using the CFIG386 utility, and (3) switches may be entered on the command line, when the program is actually run. The link time switch settings are processed first, then configured in switches, and, last, the command line switches. If conflicting switch settings are given, the last switch processed takes precedence.
The -MINREAL and -MAXREAL switches are used to control how much conventional memory (memory below 640K) is left free by 386\*DOS-Extender. By default, 386\*DOS-Extender allocates all the available conventional memory for use by the application program.

If the application program you are executing ever makes a system call to execute another program, you must make sure that 386\*DOS-Extender leave sufficient conventional memory free for the second program's needs. The -MINREAL switch specifies the minimum amount of conventional memory to leave free; 386\*DOS-Extender refuses to run the program if it cannot leave at least this amount of memory free. The -MAXREAL switch specifies the maximum amount of conventional memory to leave free. 386\*DOS-Extender guarantees that at least MINREAL memory is left free and that as much as possible, up to MAXREAL memory, is left free.

The -MINREAL and -MAXREAL switches both take a number as an argument. The number specifies memory size in units of 16-byte paragraphs (the standard unit of memory allocation under MS-DOS).

The number must be less than or equal to 65535 (FFFFh). By default, 386\*DOS-Extender sets both -MINREAL and -MAXREAL to zero. These switches may also be specified at program link time.

### Syntax

- **MINREAL** nparagraphs
- **MAXREAL** nparagraphs

### Short form

- **MINR** nparagraphs
- **MAXR** nparagraphs

### Examples

- run386 -minreal 100h hello
- run386 -minr 128 -maxr 512 hello
Systems Call Data Buffer Switches
The -MINIBUF and -MAXIBUF switches are used to control how much memory is allocated to the data buffer used for DOS and BIOS function calls. This buffer is most important for file I/O; if your program reads or writes large amounts of data at a time, you should allocate a large buffer for efficiency.

The -MINIBUF and -MAXIBUF switches both take a number as an argument. The number specifies the buffer size in units of one kilobyte and must be between one and 64, inclusive. By default, 386|DOS-Extender sets MINIBUF to one kilobyte, and MAXIBUF to four. If 386|DOS-Extender cannot allocate at least MINIBUF kilobytes for the interrupt buffer, it refuses to run the program. If possible, MAXIBUF kilobytes are allocated. If there is not enough memory available to satisfy both the MAXREAL and MAXIBUF parameters, MAXIBUF takes precedence. These switches may also be specified at program link time.

Syntax
-MINIBUF nkilobytes
-MAXIBUF nkilobytes

Short form
-MINI nkilobytes
-MAXI nkilobytes

Examples
run386 -maxibuf 2 hello
run386 -mini 64 filecopy
Mixed mode program switches

The -REALBREAK and -CALLBUFS switches are used to control program loading of programs that contain both real mode and protected mode code.

The -REALBREAK switch controls how much of the program must be loaded into conventional memory, so that it can be accessed and/or executed in real mode. It takes an argument specifying the number of bytes at the beginning of the program, which must be loaded in conventional memory. This switch may also be specified at link time. It is usually more convenient to specify this switch at link time, when the argument can be the name of a public symbol appearing at the end of the real mode code and data. If this switch is used at run time, the argument must be an absolute number which has been calculated from the information in the link map.

The -CALLBUFS switch controls the size of the intermode call buffer, which is allocated in conventional memory for use by the application program as a data buffer on intermode procedure calls. The buffer address is obtained at run time with a 386|DOS-Extender system call. The argument is the size of the buffer in kilobytes and must be less than or equal to 64. The default buffer size is zero. This switch may also be specified at link time.

Syntax

-REALBREAK nbytes
-CALLBUFS nkilobytes

Short form

-REALB nbytes
-CALLB nkilobytes

Examples

run386 -realbreak 200h -callb 2 switch
386link switch -realb END_REAL -callbufs 2
Appendix A: Phar Lap technical information

Stack allocation switches

The -NISTACK and -ISTKSIZE switches are used to control how much memory is allocated to the buffers used to provide stack space when switching the 80386 from protected mode to real mode. For the vast majority of application programs, the default settings of these parameters are sufficient.

Both switches take a number as an argument. The -NISTACK switch specifies the number of stack buffers to allocate and must be four or greater. The -ISTKSIZE switch specifies the size of each stack buffer in kilobytes and must be between one and 64, inclusive. By default, 386 DOS-Extender allocates four stack buffers of one kilobyte each. These switches may also be specified at program link time.

**Syntax**

-NISTACK nbuffers
-ISTKSIZE nkilobytes

**Short form**

-NI nbuffers
-ISTK nkilobytes

**Example**

run386 -ni 6 -istk 2 switch.exp
Extended memory switches

The -EXTLOW and -EXTHIGH switches are used to limit the amount of extended memory (memory above one megabyte) that 386/DOS-Extender allows the application program to use. By default, all extended memory that is not allocated to other programs is available for use by the application. Other programs which may have allocated extended memory include RAM disk programs, disk cache programs, and EMS simulators.

Both the -EXTLOW and -EXTHIGH switches take a number as an argument. The number specifies a physical memory address in extended memory. By default, 386/DOS-Extender sets EXTLOW to 100000h (one megabyte) and EXTHIGH to FFFFFFFFh (four gigabytes).

386/DOS-Extender uses only extended memory above the address specified with the -EXTLOW switch, or memory used by other programs, whichever is higher. Similarly, it uses only extended memory below the address specified with the -EXTHIGH switch, or memory used by other programs, whichever is lower.

Normally, it is not necessary to use the -EXTLOW or -EXTHIGH switches. However, if your system has a program installed that uses extended memory and does not use either (1) the VDISK or RAMDRIVE standards for allocating memory from one megabyte up, or (2) the INT 15th function 88h BIOS call for allocating extended memory from the top of memory down, it may be necessary to use one or both of these switches to prevent 386/DOS-Extender from allocating extended memory used by the installed program.

**Syntax**

- -EXTLOW address
- -EXTHIGH address

**Short form**

- -EXTL address
- -EXTH address

**Examples**

run386 -extlow 200000h hello
run386 -extl 180000h -exth 400000h hello
**Weitek 1167 switch**

The -1167 switch is used to select how detection of the Weitek 1167 floating point coprocessor is performed. If 386\|DOS-Extender detects the presence of the 1167, segment selector 003Ch is initialized to map the memory space used by the 1167 and segment register FS is initialized to contain selector 003Ch. A program can, therefore, test for the presence of the 1167 at run time by examining the contents of the FS register.

The -1167 switch has three settings. The "-1167 AUTO" setting instructs 386\|DOS-Extender to use the Weitek-approved BIOS presence detection call. This may not work correctly on all machines, if the appropriate BIOS is not installed. The setting "-1167 ON" instructs 386\|DOS-Extender to assume the 1167 is present, and setting "-1167 OFF" assumes the 1167 is not present. The default setting is "-1167 AUTO."

**Syntax**

-1167 AUTO
-1167 ON
-1167 OFF

**Short form**

-1167 AUTO
-1167 ON
-1167 OFF

**Example**

run386 -1167 on float.exp
Interrupt relocation switches

The -HWIVEC and -PRIVEC switches are used to select the interrupt vectors to be used for interrupts, which 386/DOS-Extender must relocate due to compatibility problems between the 80386 processor exceptions and PC/AT-compatible interrupts.

The -HWIVEC switch selects a block of eight interrupt vectors to use for hardware interrupts IRQ0 through IRQ7. These interrupts must be relocated, because they are normally vectored through interrupts 08h-0Fh, which are also used for processor exceptions. The switch argument is the interrupt vector number to use for hardware interrupt IRQ0. The default, if no command line switch is used, is interrupt vector 78h. This switch is not available when executing under the DESQview 386 environment, since DESQview also relocates hardware interrupts.

The -PRIVEC switch selects the interrupt vector to use for the BIOS print screen function call. This interrupt must be relocated, because it normally uses vector 05h, which is also used for the processor bounds exception. The default setting for this switch is interrupt vector 80h.

**Syntax**

- HWIVEC vector
- PRIVEC vector

**Short form**

- HWI vector
- PRI vector

**Example**

run386 -hwivec 50h -pri 78h hello
The -INTMAP and -PRIMAP switches prevent 386\textsuperscript{1}DOS-Extender from relocating any interrupt vectors, and to specify where the relevant interrupt vectors are already mapped. The primary purpose of these switches is to allow protected mode memory-resident (TSR) programs to be installed. A secondary purpose is compatibility with other programs which relocate interrupts.

The -INTMAP switch disables 386\textsuperscript{1}DOS-Extenders's remapping of hardware interrupts, and specifies the block of eight interrupt vectors to which hardware interrupts IRQ0 through IRQ7 are already mapped. When running under DOS, these interrupts are normally mapped to vectors 08h - 0Fh, but it is possible for other programs to relocate them. If the -INTMAP 8 switch is used, 386\textsuperscript{1}DOS-Extender (and a debugger, if one is being used) does not take over processor exceptions 08h - 0Fh, because it assumes these interrupt vectors are being used for hardware interrupts. The -INTMAP 8 switch should ONLY be used with TSR (or other) programs that have already been debugged, since a processor exception in a buggy program run with this switch would be interpreted as a hardware interrupt, causing the machine to crash.

The -PRIMAP switch disables 386\textsuperscript{1}DOS-Extender's remapping of the BIOS print screen function call, and specifies the interrupt vector to which this call is already mapped. The print screen function is normally invoked through INT 5. If the -PRIMAP 5 switch is used, 386\textsuperscript{1}DOS-Extender (and a debugger, if one is being used) does not take over processor exception 5, the BOUND exception. If a buggy program which causes a BOUND exception is run with the -PRIMAP 5 switch, a print screen will occur instead of an abnormal program termination.

### Syntax

- -INTMAP vector
- -PRIMAP vector

### Short form

- -INTM vector
- -PRIM vector

### Examples

run386 -intm 8 -primap 5 kbdtrap
run386 -intmap 78h -prim 80h program2
Paging disable switch

The -NOPAGE switch is used to prevent 386|DOS-Extender from using the 80386's hardware paging functionality to perform memory management. The ONLY reason to use this switch is to avoid a bug in early versions of the 80386 chip. Chip steps B1 and earlier have the bug. Chip step D0, which does not have the problem, was released by Intel in the second quarter of 1988. The problem only appears in programs which use the 80387 numeric coprocessor, and manifests itself as the machine halting, with not even the DOS system reboot command, CTRL-ALT-DEL, available. This problem does not occur on all 80386 machines (e.g., the Compaq DESKPRO 386/20), because it can be affected by the design of the system motherboard.

When this problem occurs in a program which uses floating point arithmetic, the only workarounds available are: (1) to simulate floating point operations in software to avoid using the 80387 coprocessor, (2) to install a step D0 or later 80386 chip in the machine, or (3) to disable paging, which removes one of the hardware conditions necessary for the problem to occur. The -NOPAGE switch disables paging.

There are disadvantages to using the -NOPAGE switch. Programs run with this switch cannot be linked with the -OFFSET switch. Programs run with -NOPAGE are loaded entirely in extended memory, with conventional DOS memory (below 640 K) not available. In addition, the dynamic memory allocation system services provided by 386|DOS-Extender are disabled when this switch is used. This means that the program must specify at link time (using the -MINDATA and -MAXDATA linker switches) how much memory it needs at run time. Heap memory allocation performed by compiler run-time libraries is normally done out of memory allocated under the control of the linker -MAXDATA switch, and is not affected by the use of -NOPAGE. The system memory allocation calls which are disabled by the use of this switch are INT 21h functions 48h, 49h, 4Ah, and 250Ah.

Syntax

-NOPAGE

Short form

-NOP

Example

run386 -nopage numcrunch
Compaq built-in memory switch

The -NOBIM switch is used to disable the automatic use of Compaq built-in memory. By default, 386|DOS-Extender attempts to use built-in memory mapped above 14 megabytes on Compaq 386 machines, if the memory is not allocated to another program. This switch instructs 386|DOS-Extender not to check for Compaq built-in memory. Normally, it is not necessary to use this switch.

Syntax
-NOBIM

Short form
-NOB

Example
run386 -nobim hello

VDISK compatibility switch

The -VDISK switch is a workaround for compatibility problems with other programs which do not correctly follow the VDISK standard for allocating extended memory.

If 386|DOS-Extender refuses to run an application program because of inconsistent VDISK allocation signatures, this switch can be used to force 386|DOS-Extender to run the program. The larger of the two allocation marks present will be used. Before using this switch, you should check the allocation sizes printed out with the error message when 386|DOS-Extender refuses to run the program. If the larger of the two numbers printed out does not seem reasonable, it will be necessary to calculate how much extended memory is in use by other programs and to use the -EXTLOW switch to inform 386|DOS-Extender of the correct value.

Syntax
-VDISK

Short form
-VDISK

Example
run386 -vdisk hello
### 80386 step B0 switch

The `-B0` switch is used to enable operation of 386|DOS-Extender on a system that has a step B0 80386 chip. 386|DOS-Extender is able to run only on 80386 chips that are step B0 or later because of a bug in earlier chips that did not permit the processor to be switched from protected mode back to real mode.

386|DOS-Extender can only check at run time whether the 80386 chip is step B1 or later. By default, it will refuse to run the application program if the chip is earlier than step B1. The `-B0` switch can be used to force 386|DOS-Extender to run. Note that if the `-B0` switch is used with a chip that is earlier than step B0, the system will crash.

#### Syntax

```
-B0
```

#### Short form

```
-B0
```

#### Example

```
run386 -B0 hello.exp
```
Appendix A: Phar Lap technical information

EMS simulator switch  The -CEMM switch is used to turn off the COMPAQ CEMM or compatible EMS (Lotus/Intel/Microsoft Expanded Memory Specification) simulator programs. These EMS simulators operate in the 80386's Virtual 8086 Mode.

386DOS-Extender normally cannot run in Virtual 8086 Mode and, by default, will refuse to run the application program if the 80386 is in virtual mode and if the VCPI interface provided by the Quarterdeck QEMM and some other programs is not present. The -CEMM switch can be used to have 386DOS-Extender automatically disable the EMS simulator program and switch the 8086 back to real mode. Note that the same thing can be accomplished by manually disabling the EMS simulator before running 386DOS-Extender. If the -CEMM switch is used when some program other than a Compaq-compatible EMS simulator has switched the 80386 into virtual mode, 386DOS-Extender prints an error message and refuses to run the program.

Syntax       -CEMM

Short form   -CEMM

Example      run386 -CEMM hello
Address line 20 switch

The -A20 switch is used to control how address line 20 is enabled or disabled. 80386 systems that conform to the IBM PC/AT standard have hardware either to allow full 32-bit addressing ("enable A20") or to truncate addresses to 20 bits ("disable A20"). When executing in real mode, A20 is normally disabled for compatibility with programs that take advantage of the address space wrap-around occurring at one megabyte on 8088/8086 systems. Very few programs rely on this behavior; the most common example is copy protection programs.

By default, 386|DOS-Extender enables A20 before starting the application running, and restores the original A20 setting when the program terminates. The -A20 switch can be used to force 386|DOS-Extender to disable A20 each time the 80386 is switched to real mode, and to re-enable A20 each time the 80386 is switched to protected mode. This can be important if, for example, a software diver, which can gain control at any time via a hardware interrupt, and which relies on one-megabyte addressing wrap-around, is installed on your machine.

There is a penalty associated with the -A20 switch. Depending on the hardware in your system, it can take several milliseconds to enable or disable A20. Thus, using the -A20 switch slows down the switch to 80386 real mode, then back to protected mode, that occurs whenever there is a hardware interrupt or DOS or BIOS function call.

Syntax

-A20

Short form

-A20

Example

run386 -A20 hello
Appendix A: Phar Lap technical information

PC and PC/XT detection switch

The -XT switch is used to inform 386\textdagger DOS-Extender that it is executing on a IBM-Compatible PC or PC/XT with a 386 board, such as the Intel Inboard/PC, installed. 386\textdagger DOS-Extender normally detects such configurations automatically, but it may not be able to detect systems which do not have the IBM standard system ID byte in the BIOS. If 386\textdagger DOS Extender does not correctly detect a PC environment, this switch can be used to allow the program to execute successfully.

Syntax

\texttt{-XT}

Short form

\texttt{-XT}

Example

\texttt{run386 \textdagger xt hello}
386 | VMM command line switches

Command line switches are used to change the default operation of 386 | DOS-Extender, and of 386 | VMM. This section documents the 386 | DOS-Extender command line switches that apply specifically to operation with virtual memory. These switches can be used with 386 | DOS-Extender even if 386 | VMM is not used. Except where noted, the 386 | VMM-specific switches are ignored by 386 | DOS-Extender when used without virtual memory. By default, 386 | VMM:

- Places the swap file used for paging in the root directory of the device from which the application program was loaded.
- Always increases the swap file size when the program allocates additional virtual memory.
- Uses a least-frequently-used algorithm for selecting the page to be replaced (written to the swap file), when bringing another page into memory.
- Updates the virtual page aging information (used by the page replacement algorithm) every four seconds.

Command line switches begin with a minus sign ("-"), character, followed by the name of the switch (e.g., -LFU). There are two forms of each switch name: a long form and a short form. Any argument to the switch must immediately follow the switch name, with a space as a separator (e.g., -VSCAN 4000). If conflicting switches are given on a command line, the last (right-most) switch takes precedence.

Some of the command line switches take a number as an argument. By default, the number is considered to be a decimal (base 10) number. Hexadecimal (base 16) numbers may be specified by appending the character "h" or "H" to the number. The following two examples both give the same number as an argument to the switch "-VSCAN":

```
run386 -vscan 2048 -vmfile vmmdrv bigsort
run386 -vscan 800h -vmfile vmmdrv bigsort
```
Virtual memory driver switches

The -VMFILE switch is used to specify the name and location of the development version of 386\|VMM to be loaded by 386\|DOS-Extender during initialization. Using this switch causes the application program to run in a virtual memory environment.

The -VMFILE switch is normally not used with bound applications (programs which have the redistribution versions of 386\|DOS-Extender and 386\|VMM bound to them in a single .EXE file). If the -VMFILE switch is used with a bound application, the virtual memory driver specified with the switch will be loaded, instead of the virtual memory driver that is bound to the application. This can be useful for testing a bound application with a later release of 386\|VMM.

The -NOVM switch instructs 386\|DOS-Extender not to load a virtual memory driver, regardless of whether the -VMFILE switch was used, or whether 386\|VMM is bound to the application program. It causes the program to run in a non-virtual environment.

Syntax

-VMFILE filename

-NOVM

Short form

-VM filename

-NOVM

Examples

run386 -vmfile vmm.mdrv hello
386debug -vm \pharlap\vmm.mdrv.exp hello
cfig386 hello.exe -novm
Swap file location switch

The -SWAPDIR switch specifies the device and directory in which to place the page swap file. The default location for the swap file is the root directory of the device from which the application program was loaded. This switch is useful for placing the swap file on a device which has sufficient free space to allow the swap file to grow as needed.

The directory name specified with this switch must not end with a "\" character, as 386\VMM appends a "\" before adding the name of the swap file. The swap file is created with a unique filename (using the DOS Create Temporary File system call).

Syntax

-SWAPDIR filename
-SWD dirname

Short form

-VM filename
-NOVM

Examples

run386 -swapdir d: -vm vmmdrv hello
minibug -swd e:\tmp -vm vmmdrv hello

Page replacement policy switches

386\VMM supports two switch-selectable page replacement policies. The page replacement policy defines the algorithm used to select a page to be swapped to disk when a page already on disk needs to be brought into memory. The performance of a program in a virtual memory environment depends to some extent on whether the system usually replaces pages that are not needed for a long time; ideally, the page selected for replacement is the page not referenced by the program for the longest time into the future. Depending on the memory referencing patterns of an application, one of the page replacement algorithms supported by 386\VMM may yield better performance than the other.
Appendix A: Phar Lap technical information

The -LFU switch selects the Least-Frequently-Used replacement policy. A reference frequency count is kept with each page. Periodically, the page tables are scanned, and the count is either incremented or decremented, depending whether the page was referenced since the last scan. The page with the lowest count (the least-frequently-used page) is the page selected for replacement. This is the default page replacement policy if no switches are used.

The -NUR switch selects the Not-Used-Recently replacement policy. This algorithm chooses a page for replacement based on whether the page has been accessed by the program, and whether it is dirty (its contents have been modified). Periodically, the page tables are scanned to mark all pages not accessed. The page accessed information thus identifies pages which have been referenced recently (since the last page table scan).

The -VSCAN switch selects how frequently the page tables are scanned in order to update the page aging information used by the page replacement policy. Changing the scan period affects which pages are selected for replacement, and therefore, affects program performance. The -VSCAN switch takes as an argument a time expressed in milliseconds (ms). The minimum value which may be given is 1000 ms (1 second). The default scan period is 4000 ms. (Note that 386/VMM assumes the timer tick interrupt occurs 18.2 times per second; application programs which change this standard timer operation must adjust the value specified with the -VSCAN switch appropriately).

**Syntax**

```
-LFU
-NUR
-VSCAN nMilliseconds
```

**Short form**

```
-LFU
-NUR
-VS nMilliseconds
```

**Examples**

```
run386 -lfu -vm vmmdrv hello
minibug -nur -vm vmmdrv hello
386debug -nur -vscan 2000 -vm vmmdrv hello
```
The page swap file can potentially grow very large, if the virtual address space required by the program is large. Under these circumstances, it is possible to run out of disk space. The tradeoff is using up more disk space than is actually needed versus taking the risk of running out of swap space during a page fault, in which case 386\ VMM is forced to abort the program.

The -SWAPCHK switch is used to select when the size of the swap file is increased by 386\ VMM. The -SWAPCHK MAX setting causes the swap file to grow whenever the virtual address space of the program is increased. The size of the swap file is always set to the size of the program's virtual address space, which is the largest size that could possibly be needed. If the swap file cannot be grown when a memory allocate system call is made, the memory allocate call returns failure, so the program can deal with the condition gracefully. This is the safest setting, because it guarantees that 386\ VMM will always have swap space available when a page fault occurs. It does, however, result in the largest swap file. (Remember that disk space problems can sometimes be solved by placing the swap file on a different disk drive with the -SWAPDIR switch.)

The default setting is -SWAPCHK FORCE, which still causes the swap file to grow whenever additional virtual memory is allocated. However, the size to which it is increased is smaller than the virtual address space for the program, while still large enough to guarantee that sufficient swap space is available when a page fault occurs. This setting is a good compromise. It results in a smaller swap file, but ensures that no unexpected program aborts will occur. However, if this setting is used and too large a value is specified by the -CODESIZE switch, it can result in an out of swap space condition during a page fault.
The `-SWAPCHK ON` setting does not grow the swap file when virtual memory is allocated; instead, the swap file size is increased by the page fault handler as it needs new swap space. When additional virtual memory is allocated, the amount of free space on the disk is checked to make sure there is sufficient free space, using the same swap space requirements as those imposed by the `-SWAPCHK FORCE` setting. However, the swap file is not actually grown until the space is needed. This setting minimizes the size of the swap file; but if the program uses up disk space for another purpose between the time the memory allocate is performed and the swap file needs to be grown, a fatal out-of-swap-space error may occur in the page fault handler. In addition, there will be some performance degradation, because it is more expensive to grow the swap file one page at a time than to grow it in large chunks when additional virtual memory is allocated. For these reasons, it is normally better to use the `-SWAPCHK FORCE` setting.

The `-SWAPCHK OFF` setting disables all swap space checking when virtual memory is allocated. The swap file is grown as needed when a page fault occurs. As with `-SWAPCHK ON`, this minimizes swap file size, but leaves the program vulnerable to out-of-swap-space fatal errors when a page fault occurs. If this setting is used, the program should install an out-of-swap-space handler that attempts either to create more swap space, or clean up and exit, when this condition occurs.

The `-CODESIZE` switch specifies the number of bytes of code which can be paged to disk without seriously affecting program performance. It is equal to the total size of the program's code, in bytes, minus the program's code "working set," that is, the amount of code that needs to be in memory at any given time to avoid excessive paging. This information is used with the `-SWAPCHK FORCE` and `-SWAPCHK ON` settings to calculate the minimum swap space required. Increasing the value reduces the swap file size. Specifying too large a value with this switch may result in unacceptable program performance, or even in fatal out-of-swap-space errors.
The -SWFGROW1ST and -NOSWFGROW1ST switches specify what the page fault handler should do when it needs a page in the swap file and one is not available. -SWFGROW1ST is the default, and causes the page fault handler, first, to attempt to grow the swap file, and then if that fails, to attempt to take a swap file page away from a virtual page currently in memory (this can be done because a page in memory does not need space in the swap file). The -NOSWFGROW1ST setting reverses the order; it causes a swap page to be taken away from an in-memory page first, and the swap file to be grown only if no in-memory page owns a page in the swap file. The tradeoff is performance versus disk space. Disk space requirements are reduced if the -NOSWFGROW1ST switch is used, but program performance suffers.

The -MAXSWFIZE switch is used to limit the maximum disk space that is allocated to the swap file. It specifies a size, in bytes, beyond which the swap file is never increased. If this switch is not used, the only upper bound on swap file size is the amount of free space available on the disk.

**Syntax**

- SWAPCHK OFF
- SWAPCHK ON
- SWAPCHK FORCE
- SWAPCHK MAX
- SWFGROW1ST
- NOSWFGROW1ST
- CODESIZE nbytes
- MAXSWFSIZE nbytes
### Short form

- SWC OFF
- SWC ON
- SWC FORCE
- SWC MAX
- SWFG
- NOSWFG
- CODES nbytes
- MAXS nbytes

### Examples

- run386 -swapchk force -vm vmmdrv hello
- run386 -swc off -noswfg -vm vmmdrv hello
- minibug -swc on -codes 900 -vm vmmdrv hello
- run386 -maxs A00000h -vm vmmdrv hello
Special characters

<Ctrl> key 16
= BOOKMARK command 65, 168
? CONDITIONS command 180, 183, 185
- LAYER command 61
% MACRO command 76, 79
* LAYER command 61, 93
+ LAYER command 61
/ OTHER command 61
1167 switch
weitek 217
386 | DOS-Extender
command line switches 211
386 | VMM
command line switches 226
80386
step B0 switch 222
90-degree corners
drawing 113, 117

A

about
autorouting 175
CFIG386 utility 207
IFORM 37
ILINK 36
INET 35
layout placement 129
library editor 88
manual routing 155
module placement 135
modules 87
printing and plotting 193
adding
arcs 114
holes 121
additional processing 185
address
line 20 switch 224
aids, placement 135
alignment targets, placing 152
all macros, deleting 84
all pages, printing 200
allocation, stack, switches 215
Annotate Schematic 24
any layer, deleting objects 105
arc segments, routing 161
arcs
adding 114
positioning 116
ASSEMBLY DRAWING page, building 199
assembly outline, placing 187
assigning a net 150
autoroute method, setting 179
Autoroute Options dialog box 179
autoroute zone, placing 176
autorouter options, setting 179
autorouting
about 175
begin 183
board 180
preparing 175
section of board 180
whole board 182
Available Display Drivers 43
Available Printer Drivers 43
PC Board Layout Tools 386+ User's Guide

B
basics, mouse 16
BLOCK command 49, 99, 100
board
  autorouting 180
  file prefix 45
  identification 146
  modules, creating 9
  name, placing 147
  objects, placing 145
  outline, drawing 131
  routing 159
  saving and backing up 71
bookmark
  creating 65
  deleting 67
  jumping 66
  using 65
BOOKMARK command 67
Bookmark dialog box 65
BOTTOM COPPER LAYER page, building 198
boxes
  check 52
  dialog 50
  droplist 52
  entry 53
  list 51
building, page
  ASSEMBLY DRAWING page 199
  BOTTOM COPPER LAYER 198
  TOP COPPER LAYER 196
built-in memory switch, compag 221
buttons 50

C
CFI386 utility, about 207
changing
  grid color 70
  origin 68
  startup design 20
  track path 172
  track width 173
  TUTOR design 19
  view of display 113
  view of the layout 62
check box 52
Check Design Integrity, running 34
Check Electrical Rules 31
configuring 33
checking design integrity 31
Cleanup Schematic 31
configuring 31
command line switches
  386 VMM 226
command prompt, operating system 17
commands
  % MACRO 76, 79
  * LAYER 61, 93
  + LAYER 61
  / OTHER 61
  = BOOKMARK 65, 168
  ? CONDITIONS 180, 183, 185
  BLOCK 49, 99, 100
  BOOKMARK 67
  CUT 117
  DELETE 105, 117, 169, 171
  EDIT 95, 150, 173
  EXIT 75
  FIND 156, 160, 161
  GO TO FUNCTION 80, 88, 92, 179
  HIGHLIGHT 159, 161
  INQUIRE 93, 99, 165
  JUMP 65, 66, 68, 165
  LAYER 60, 165
  MOVE 96, 97, 98
Index

ORIGIN 68, 76, 79, 113, 119
PLACE 77, 78, 79, 96
QUIT 24, 31, 71, 72, 73, 74, 75, 91
ROUTE 155, 159, 160, 161, 172
SELECTIVE 106, 107
SET 47, 56, 69, 70, 76, 96, 130, 148
TRACK DELETE 171, 172
UNDELETE 106, 171
VERBOSE INQUIRE 94
WINDOW ZOOM 63, 148
X SHOW RATSNEST 136, 157
ZOOM 47, 62, 63, 64, 76
- LAYER 61

copag built-in memory switch 221
Compare Netlist 8
compatibility switch, vdisk 221
conditions, setting 159
configuration
  minimum 3
  saving 71
Configured Display Drivers 43
Configured Printer Drivers 43
configuring
  DRAFT 22
  Edit Layout 46
  IFORM 37
  ILINK 36
  INET 35
  key fields 24
  pages 196
  PC Board Layout Tools 42
  printer options 195
  Schematic Design Tools 22, 23
  virtual memory 3
conventional memory switches 212
decordate placement 138
Copper Colors/Enables/... dialog box 58
copper tool
  creating 158
  specifying 178
Copy File dialog box 73
copying
  DEMO library 89
file 73
  module 89
  correcting DRC errors 168
Create NC Drill File 7
creating
  board module 9
  bookmark 65
  copper tool 158
  fill zone 148
  first macro 76
  module 111
  netlist 35
  second macro 79
  stuff file 28
Cross Reference Parts 31
  configuring 32
Current Object Settings dialog box 59
current settings 59
CUT command 117

237
D

data buffer switches, systems calls 213
defined macro, running 85
defining a netlist block 132
DELETE command 105, 117, 169, 171
deleted objects, permanently deleting 107
deleting
  90° corner 117
  bookmark 67
  exported modules 110
  file 74
  macro file 84
  module objects 105
  objects 105
  offending track 169
  stub 171
  track 171
DEMO library, copy 89
design
  environment 18
  integrity, checking 31
  size 2
  designing a pad array 125
dialog boxes
  Autoroute Options 179
  Bookmark 65
  Copper Colors/Enables/... 58
  Copy File 73
  Current Object Settings 59
  Driver Configuration 195
  Edit Alignment Target 152
  Edit Circle 114
  Edit Copper Tool 158
  Edit Hole 121
  Edit Net Properties 177
  Edit Net Segment 161, 173
  Edit Outline Segment 95
  Edit Pad 122
  Edit Pad Array Settings 125
  Edit Text 147
  Edit Zone Properties 150
  Edit Zone Segment 150
  Export Module to File 108
  Find 156
  Finished 183
  Get Module 89
  Global Options 56
  Import Module from File 109
  Initialize to Library File 88
  Jump To 66, 166
  Layer 57
  Load ALL Macros from File 84
  Load Print/Plot Setup from File 204
  Macro Maintenance 81
  Netlist Load Options 133
  Place Module 141
  Printing and Plotting 194
  Rename File 73
  Rename Module 92
  Save "\x011F Alt S" Macro to File 83
  Save ALL Macros to File 82
  Save Print/Plot Setup to File 203
  Set Block Parameters 97, 139
  Verbose Inquire 94
  Write Board File 72
dimensions, placing 188
disable switch, paging 220
disabling snap grid 70
displaying
  force vectors 137
  main menu 48
  module information 93
  ratsnest 136, 157
  schematic 24
DRAFT, configuring 22
drawing
  board outline 131
  methods 113
  new track 168
  outline 113, 119, 146
DRC
  check 164
DRC errors, correcting 168
DRC violations, identifying 165
Driver Configuration dialog box 195
driver switches, virtual memory 227
drivers 11
droplist box 52
dynamic, placement 141

E
Edit Alignment Target dialog box 152
Edit Circle dialog box 114
EDIT command 95, 150, 173
Edit Copper Tool dialog box 158
Edit File editors 7
Edit Hole dialog box 121
Edit Layout
  command basics 48
  configuring 46
  deleting macro 84
  editors 7
  introducing 9, 41
  running 47
  template, loading 130
Edit Net Properties dialog box 177
Edit Net Segment dialog box 161, 173
Edit Outline Segment dialog box 95
Edit Pad Array Settings dialog box 125
Edit Pad dialog box 122
Edit Text dialog box 147
Edit Zone Properties dialog box 150
Edit Zone Segment dialog box 150
editing
  module 96, 142
  routed board 168
editors
  Edit File 7
  Edit Layout 7
  View Reference 7
ems, simulator switch 223
entry box 53
erasing
  routes 185
error messages 209
ESP design environment
  changing designs 19
  running 18
  startup designs 20
excluding vias 178
existing route, locking 177
EXIT command 75
Export Module to File dialog box 108
exported modules, deleting 110
exporting
  macro file 83
  modules 108
extended memory 2, 3
  switches 216
F
field contents, updating 30
file
copying 73
deleting 74
names 17
renaming 73
updating 71
writing 72
fill zone
creating 148
placing 148
FIND command 156, 160, 161
Find dialog box 156
finding information 4
Finished dialog box 183
finishing, layout 186
first dimension, placing 188
first macro, creating 76
first track, routing 159
Fix Time Stamps 7
flipping 101, 144
force vector 135

G
Gerber 205
Get Module dialog box 89
getting started 155, 193
Global Options 130
Find Highlights 130
Grid Size 130
Outline Pads 130
Global Options dialog box 56
GO TO FUNCTION command 80, 88, 92, 179
grid
color, changing 70
divisor, setting 70
options, setting 69
size, setting 69

H
hiding module text 142
HIGHLIGHT command 159, 161
highlighting a net 156
holes, adding 121
HP-GL/2 206
I
identification, board 146
identifying DRC violations 165
IFORM
    about 37
    configuring 37
ILINK
    about 36
    configuring 36
Import Module from File dialog box 109
INET
    about 35
    configuring 35
information, finding 4
Initialize to Library File dialog box 88
input, keyboard 17
INQUIRE
    command 93, 99, 165
    using 165
installing PC Board Layout Tools 386+ 9, 12
interface, command 49
interrupt
    mapping switches 219
    relocation switches 218
introducing
    Edit Layout 9, 41
isolating, modules 129

J
JUMP command 65, 66, 68, 165
Jump To dialog box 66, 166
jumping to bookmark 66

K
key fields, viewing 27
keyboard input 16, 17

L
layer 57
    marker 145
    selecting 60
LAYER command 60, 165
Layer dialog box 57
layout
    finishing 186
    placement 129
learning PC Board Layout Tools 386+ 9
leaving the library editor 128
librarians
    Make Board Template 7
    Make Library 7
library
    editor
        leaving 128
        selecting 88
    file, updating 91
        prefix 44
line 20, address, switch 224
list, boxes 51
Load ALL Macros from File dialog box 84
Load Netlist File dialog box 133
Load Print/Plot Setup from File dialog box 204
loading
    macro 85
    netlist 132
    printer setup 204
    template file 130
location switch, swap file 228
locking existing route 177
M

macro file
deleing 84
exporting 83
loading from 85
saving 80
Macro Maintenance dialog box 81
macros 76, 80
main menu
displaying 48
returning 55
Make Board Template 7
Make Library 7
manual routing 155
mapping, interrupt, switches 219
marker, layer 145
memory
extended 2, 3
switches
conventional 212
extended 216
menus 49
messages, error 209
minimum configuration 3
mirroring
modules 99, 100
mixed mode program switches 214
Modify Modules 7
Module Report 8
modules 12
copying 89
creating 111
flipping 144
importing 108
information, displaying 93
isolating 129
libraries 89
moving 96
placing 129, 135, 138
renaming 92
saving 128
size 2
text
hiding 142
rotating 143
mouse basics 16
MOVE command 96, 97, 98
movement, pointer, resolution 64
moving
around screen 47
back onto grid 104
module 96
off-grid object 104
reference designators 186
selected objects within a group 103
single module object 102
Index

N

net
  assigning 150
  highlighting 156
netlist
  creating 35
  loading 132
  prefix 45
  viewing 38
netlist block, defining 132
Netlist Load Options dialog box 133
Netlist Loader 132
new module, starting 111
new pad symbol, selecting 124
new track, drawing 168

O

d-off-grid object, moving 104
offending track, deleting 169
operating system command prompt 17
options, setting 130
OrCAD/PCB II, upgrading 11
ORIGIN command 68, 76, 79, 113, 119
origin, changing 68
outlines, drawing 119, 146

P

pad array
  designing 125
  placing 122
page replacement policy switches 228
pages
  configuring 196
  printing 200
paging disable switch 220
pc and pc/xt detection switch 225
PC Board Layout Tools
  configuring 42
  installation 12
  introduction 2
  learning 9
performing DRC check 164
Phar Lap, technical information 10
PLACE command 77, 78, 79, 96
Place Module dialog box 141
placeholders, positioning 127
placement
  aids 135
  coordinate 138
  dynamic 141
placing
  assembly outline 187
  autoroute zone 176
  board name 147
  board objects 145
  dimensions 188
  fill zone 148
  first dimension 188
  modules 129, 138
  pad array 122
  second dimension 190
  TUTOR board 9
plotting 205
pointer movement resolution 64
policy switches
   page replacement 228
   swap file 230
positioning
   arcs 116
   placeholders 127
PostScript 206
preferences, setting 112
prefix
   board file 45
   library 44
   netlist 45
   temp file 45
preset steps, rotating in 98
printer
   configuring 195
   setup 203
printing and plotting
   all pages 200
   pages 200
   selected pages 201
   TUTOR board 10
Printing and Plotting dialog box 194
processors
   Create NC Drill File 7
   Fix Time Stamps 7
   Modify Modules 7
   Reannotate Board File 7
program switches, mixed mode 214
project, organizing by 4

Q
QUIT command 24, 31, 71, 72, 73, 74, 75, 91

R
radio button 53
ratsnest 135, 136
Reannotate Board File 7
recommendations 3
reduction, via 184
reference designators, moving 186
relocation, interrupt, switches 218
Rename File dialog box 73
Rename Module dialog box 92
renaming
   file 73
   module 92
reporters
   Compare Netlist 8
   Module Report 8
returning to main menu level 55
rotating
   module text 143
   preset steps 98
   specific module angle 97
ROUTE command 155, 159, 160, 161, 172
routed board, editing 168
routes, erasing 185
routing
   arc segments 161
   area, zooming 155
   board 159
   conditions net, setting 178
   first track 159
   manually 155
   TUTOR board 10
   vias 160
running
   CFIG386 207
   Check Design Integrity 34
defined macro 85
   DRC check 164, 169
   Edit Layout 47
   ESP design environment 18
   macros 80
   To Layout 38
Index

S

Save Alt S’ Macro to File dialog box 83
Save ALL Macros to File dialog box 82
Save Print/Plot Setup to File dialog box 203

saving
  board file 71
  configuration 71
  macro 80
  module 128
  printer setup 203
  work 153, 174, 191

schematic
  annotating 25
  displaying 24
  transferring to layout 9

Schematic Design Tools, configuring 22, 23

scroll buttons 53

second dimension, placing 190

second macro, creating 79

section of board, autorouting 180

selected
  objects within a group, moving 103

selecting
  layer 60
  library editor 88
  new pad symbol 124
  TUTOR386.NET 132
  zoom window 63

SELECTIVE command 106, 107

Set Block Parameters dialog box 97, 139

SET command 47, 56, 69, 70, 76, 96, 130, 148

setting
  autoroute method 179
  autorouter options 179
  conditions 159
  grid divisor 70
  grid options 69
  grid size 69
  preferences 112
  routing conditions net 178
  sweep routing direction 179
  sweep window 182
  zoom scale 63

work conditions in Edit Layout 56

settings, current 59

simulator switch, ems 223

snap grid, disabling 70

specific layer, deleting objects on 105

specifying a copper tool 178

stack allocation switches 215

starting
  new module 111

startup design, changing 20

step B0 switch, 80386 222

stub, deleting 171

stuff file, creating 28

summary 39, 85, 128, 153, 174, 191

suspend to system 75

swap file
  location switch 228
  policy switches 230

sweep
  routing direction 179
  window 182

system
  call data buffer switches 213
  requirements 2, 3
  suspend to 75
T

targets, alignment, placing 152
technical information, Phar Lap 10
temp file prefix 45
thermal relief, viewing 151
To Digital Simulation 8
To Layout 38
To Main 8
To PLD 8
To Schematic 8
tools
editors 6
librarians 6
processors 6
reporters 6
transfers 6
TOP COPPER LAYER page, building 196
track, deleting and undeleting 171
track
path 172
width 173
TRACK DELETE command 171, 172
transferring from schematic to layout 9
transfers
   To Digital Simulation 8
   To Main 8
   To PLD 8
   To Schematic 8
turning off
   force vectors 137
   ratsnest 136
TUTOR board
   autorouting 10
   placing 9
   printing and plotting 10
   routing 10
TUTOR design, changing 19
TUTOR386.NET, selecting 132

U

UNDELETE command 106, 171
undeleting a track 171
Update Field Contents 24
updating
   field contents 30
   file 91, 71
upgrading from OrCAD/PCB II 11, 12
using
   bookmarks 65
   INQUIRE 165
   JUMP 166

V

vdisk compatibility switch 221
vector, force 135
VERBOSE INQUIRE command 94
Verbose Inquire dialog box 94
via reduction 184
vias
   excluding 178
   routing 160
view of display, changing 113
view of layout, changing 62
View Reference 7
viewing
   key fields 27
   netlist 38
   thermal relief 151
   violated areas 166
violated areas, viewing 166
virtual memory 2
   configuring 3
   disk space 2, 3
   driver switches 227
   recommendations 3
Index

W
weitek 1167 switch 217
whole board autorouting 182
WINDOW ZOOM command 63, 148
work, saving 153, 174, 191
working
  environment 5
  libraries 89
Write Board File dialog box 72

X
X SHOW RATSNEST command 136, 157

Z
zone/pad isolation 149
ZOOM command 47, 62, 63, 64, 76
zoom
  scale, setting 63
  window
    selecting 63
    routing area 155