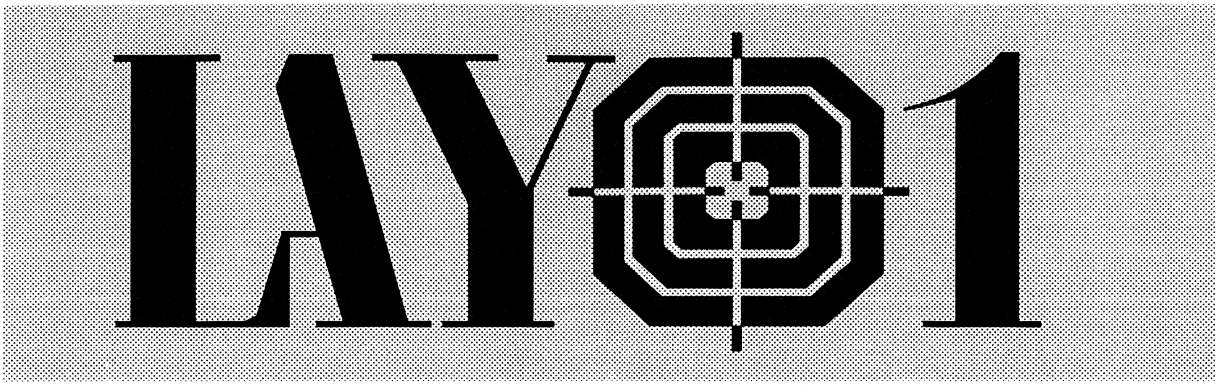


Reference Guide

Layo1 PCB CAD system

© 1986 ... 1992 Peter Baas
All rights reserved

Layo1 Level 4

Registratie Nummer:

I403115322

The Layo1 programs, utilities, component banks and all associated documentation is copyright by Peter Baas, Baas Electronics BV, Netherlands. No part of it may be copied, multiplied or disclosed in any way, except with the explicit written consent of Baas Electronics BV or as noted. You are licensed to use this program on a single computer. Copies of the program diskettes may be made only for personal backup use. Violation of this copyright is a breach of applicable law.

This program and its documentation are sold 'as is'. Baas Electronics BV does not accept any liability for damages, either directly or indirectly, that arises from the use of Layo1. No guarantees are given for its fitness for a particular use, if any. Baas Electronics BV's liability will be limited to the replacement only of defective diskettes or documentation unless the defects are caused by misuse.

Layo1 is a trademark of Peter Baas, Numansdorp, the Netherlands;
OrCAD is a registered trademark of OrCAD L.P.;
PC-DOS and IBM are a registered trademarks of International Business Machines Corp.;
MS-DOS is a registered trademark of Microsoft corporation;
HP/GL is a trademark of Hewlett-Packard Company;
DM/PL is a trademark of Houston Instruments;
Postscript is a registered trademark of Adobe Systems Incorporated;
All other brand and product names mentioned herein are used for identification purposes only, and are trademarks or registered trademarks of their respective holders.

Manual layout and design: JamaSys Electronics, Po Box 223, NL-4760 AE Zevenbergen, The Netherlands.

manual version 4 of jan. 92

C O N T E N T S

List of figures v

Overview 1-1
Contents overview 1-1
Conventions 1-3
Mouse keys 1-3

Layo1 Structure 2-1
Vectors 2-1
Blocknumbers 2-2
Layo1 components 2-2
Layo1 subdraws 2-3
Design control 2-3
Net and component list 2-4
Schematic capture 2-4
Directory and program file 2-7
Graphics editor command line . 2-7
File system 2-9
.LMC 2-9
.PLY 2-10
.CNF 2-10
.LML 2-10
.SMB 2-12
USRXX.CFG 2-13
.LPU 2-13
.LIB 2-13

Graphics Editor Reference 3-1
Format 3-1
Menu functions 3-1
Secondary functions 3-2
External functions 3-2
Selecting from a menu 3-2
Mouse keys conventions 3-3
Function list 3-3
Attributes display 3-3
Auto settings 3-4
Block 3-5
Box 3-6
Center 3-6
Check connections 3-7
Circle 3-7
Color mode 3-8
Color settings 3-8
Configure language 3-10
Configure video driver 3-11
Configure directories 3-12
Configure libraries 3-13
Configure printer 3-15
Coordinates 3-15
Copy repeat 3-16
Copy pad 3-17
Copy line 3-18
Copy object 3-18
Corner lines 3-18
Cursor select 3-19
Delete file 3-19
Delete layer 3-20
Delete object 3-20
Design rule check 3-22
Diagonal lines 3-23

Directories 3-23
Display options 3-24
Display CAM 3-24
Draw line 3-26
Draw menu 3-29
Drill group 3-31
Edit Design CNF 3-33
Edit net 3-34
Edit 3-36
Ellipse 3-36
File/directory select 3-37
File menu 3-39
Fill 3-41
Grid dots 3-42
Grid lines 3-42
Grid units 3-43
Grid set 3-43
Grid define 3-44
Layer display 3-44
Layer change 3-46
Layer select 3-47
Load component 3-48
Load file 3-49
Macro create 3-49
Main menu 3-50
Manipulate block 3-51
Manipulate window 3-52
Manipulate line 3-52
Manipulate group 3-53
Mirror 3-54
Mouse sensit. 3-54
MultiFunc menu 3-55
New design 3-56
New component 3-56
Offset 3-57
Optimize traces 3-58
Options menu 3-59
Origin 3-60
Pad menu 3-60
Pad define 3-61
Pad place 3-64
Pad replace 3-65
Pad select 3-66
Pen 3-66
Penwidth 3-67
Pin numbers 3-68
Printer checkplot 3-68
Quit 3-70
Ratsnest 3-71
Redraw screen 3-72
Reference PCB 3-72
Rename 3-73
Reroute all 3-74
Reroute net 3-74
Reroute track 3-75
Reset program 3-75
Rotate object 3-76
Rotate design 3-76
Route window 3-77
Route all 3-77
Route net 3-78
Route pad 3-78
Route manual 3-78
Router setup 3-79
Router menu 3-82
Rubberband 3-84
Ruler origin 3-84
Save user configuration 3-85

Save design 3-85
Save file 3-86
Save as .LMC 3-86
Scale design 3-87
Shape display 3-87
Snap design 3-87
Snap cursor 3-88
Snap component 3-88
Step nodes 3-89
Swap block 3-89
Text draw 3-90
Text options 3-92
Text display 3-95
Toolbox menu 3-95
Tool select 3-96
User xx menu 3-97
Via 3-98
Window define 3-99
Window delete 3-99
Zoom 3-100

Text Editor Reference 4-1
Editing screen 4-1
Status line 4-2
Command line 4-3
Editing commands 4-4
Cursor positioning 4-4
Insert and delete 4-5
Search and replace 4-5
Search/replace options 4-6

Project Manager Reference 5-1
Project Manager file structure . . 5-2
PM.BAT 5-3
PM.CNF 5-4
PM.DAT 5-4
Menu command repertoire 5-4

Output Driver Reference 6-1
Start-up 6-1
Output menu 6-2
Output main window 6-3
Output Input/output window . . 6-6
General window 6-8
Device configuration menu 6-9
Device configuration main window 6-9
Device basics window 6-10
Example window 6-11

Font Editor Reference 7-1
Character set 7-1
Features 7-2
Layer 7-2
Pen 7-2
Grid 7-3
Mode 7-3
Commands 7-3
Browse font 7-4
Character end 7-5
Character select 7-5
Cross, diagonal 7-5
Cross, rectangular 7-6
Draw mode 7-6
Edit mode 7-6
Font load 7-6
Font save 7-7
Grid select 7-7

Grid toggle 7-7
Help 7-7
Node insert 7-7
Node delete 7-8
Pen up/down 7-8
Redisplay, overlap 7-8
Redisplay, mix 7-8
Origin 7-8
Ruler select 7-9
X input 7-9
Y input 7-9
Zoom 7-9

Layo1 Component Banks A-1

Index i-1

List of figures

Layo1 Structure

Fig. 2-1 Layo1 default directory structure 2-8
Fig. 2-2 Example netlist fragment 2-12
Fig. 2-3 Example component list fragment 2-12
Fig. 2-4 Layo1 library fragment 2-14

Graphics Editor Reference

Fig. 3-1 Typical color setting screen 3-10
Fig. 3-2 Display adapter parameters 3-11
Fig. 3-3 Typical configured directories list 3-13
Fig. 3-4 Configured user libraries list 3-14
Fig. 3-5 Configure printer settings display 3-16
Fig. 3-6 Delete layer CAM data list 3-21
Fig. 3-7 Delete multifunc/ group mouse functions 3-22
Fig. 3-8 Typical File/ directories browse display 3-24
Fig. 3-9 Display options, component name options 3-25
Fig. 3-10 Display CAM screen 3-26
Fig. 3-11 Full scale default penwidths 3-27
Fig. 3-12 Draw mouse functions 3-28
Fig. 3-13 Draw menu functions 3-30
Fig. 3-14 Drill group matrix 3-32
Fig. 3-15 Edit Design CNF parameter screen 3-34
Fig. 3-16 Directory/file select window 3-38
Fig. 3-17 File/directory select directory window 3-38
Fig. 3-18 File/directory select recent files 3-39
Fig. 3-19 File/directory select Layo1p directory 3-39
Fig. 3-20 File menu functions 3-40
Fig. 3-21 Layer display configuration menu 3-46
Fig. 3-22 Layer change table 3-47
Fig. 3-23 Layer select menu 3-48
Fig. 3-24 File, load submenu 3-49
Fig. 3-25 Main menu selections 3-51

Fig. 3-26 Block shortcut mouse actions 3-52
Fig. 3-27 Multifunc, {P1..P2} mouse functions 3-54
Fig. 3-28 Mouse sensitivity settings 3-55
Fig. 3-29 MultiFunc functions 3-56
Fig. 3-30 Optimize options menu 3-59
Fig. 3-31 Options menu 3-60
Fig. 3-32 Pad menu 3-61
Fig. 3-33 Pad define submenu 3-62
Fig. 3-34 Typical pad resize display 3-64
Fig. 3-35 Pad place mouse functions 3-65
Fig. 3-36 Pen selection menu 3-67
Fig. 3-37 Printer checkplot submenu 3-69
Fig. 3-38 Reference PCB 3-73
Fig. 3-39 Router setup submenu 3-80
Fig. 3-40 Router menu 3-83
Fig. 3-41 Text object submenu 3-90
Fig. 3-42 Text draw mouse functions 3-91
Fig. 3-43 Draw/ text/ options submenu 3-93
Fig. 3-44 Options, layo1 toolbox submenu 3-96
Fig. 3-45 Default tool diameters 3-97
Fig. 3-46 User parameter menu 3-98

Text Editor Reference

Fig. 4-1 Editor opening screen 4-2

Project Manager Reference

Fig. 5-1 Project Manager screen 5-2
Fig. 5-2 Project Manager custom menu page 5-12

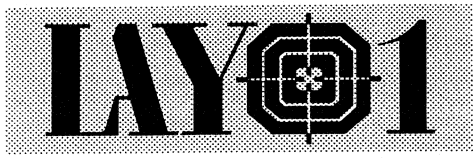
Output Driver Reference

Fig. 6-1 Output Driver Output screen 6-2
Fig. 6-2 Output Driver plot layers specification 6-4
Fig. 6-3 Output Driver plot tools/pens specification 6-4
Fig. 6-4 Output Driver drill program specification 6-5
Fig. 6-5 Output driver selections 6-6
Fig. 6-6 Output device configuration menu 6-10
Fig. 6-7 Output driver formats 6-11

Font Editor Reference

Fig. 7-1 Font editor opening screen 7-4

Notes



CHAPTER 1

Overview

This guide contains all reference information for Layo1 and its support programs, the Layo1 Project Manager with built-in text-editor and the Layo1 Output Driver.

The guide is valid for all Layo1 package levels. The various package levels differ *only* in the size of the PCB, specified in vectors, that can be designed. All packages can handle the same number of layers, to the same complexity. All packages are complete with the Layo1 Project Manager, Layo1 Output Driver, all Component Libraries and the Tools. Designs, subdrawings and user-customiseable options are fully portable across packages.

Contents overview

This chapter will give you an overview of this Reference Guide, and introduce some conventions used in the rest of the guide.

Chapter 2

The second chapter contains an introduction to the structure of Layo1, its files and related topics. This chapter is not required reading for using Layo1, but it will enhance your understanding of the various functions and relationships. It is especially usefull if you go into manipulating parts of a design or if you interface Layo1 with a schematic design package and want to control the PCB configuration from the schematic diagram.

Chapter 3 This chapter contains descriptions and specifications for all Layo1-functions and commands. Functions are listed in alphabetic order. Each entry will note all methods to activate the function, both through the menu's, via the mouse or directly with a keyboard shortcut. A short description is given, along with any specific features and limitations.

Chapter 4 This is a reference for the text editor contained in the Layo1 Project Manager. It lists all the editor's functions and commands. Because the editor is largely compatible with Wordstar and Sidekick, this chapter is not required reading.

Chapter 5 Chapter 5 introduces the Layo1 Project Manager, its functions and programming commands. The Project Manager is a programmable menu system, acting as a command shell for the Layo1 Graphics Editor and its support utilities. This chapter discusses the structure of the programmable Project Manager, its internal commands and how to customize it to your personal needs. This is of interest is you want to customize the Project Manager. The use of the Project Manager and the delivered menu structure is discussed in the User Guide.

Chapter 6 This chapter contains the reference information for the Output Driver program. The stand-alone Output Driver program is used to generate the high-quality layouts to produce the actual PCB. It accepts various Layo1 file formats, and in turn generates outputs to drive a wide variety of output devices. You will want to read this chapter before producing production-quality output plots or layouts.

Chapter 7 This last chapter contains the reference information for the Layo1 Font Editor. This editor is a separate graphics editor within the Layo1 system. It is primarily meant for experienced Layo1 users to design or edit character sets, logos or symbols.

You should get some experience with Layo1's regular Graphics Editor before you use the Font Edit.

Appendix Layo1 contains an extensive library of component shapes. These are found in the component banks (directories) under the .LMC directory. The Appendix contains a printout of all shapes and their names. You should browse this appendix before starting to use Layo1.

Index As usual, an index is found at the end of this guide.

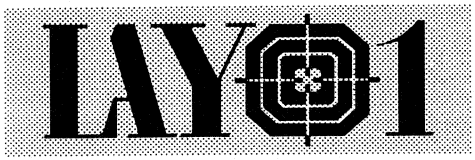
Conventions The following conventions are used in the text:

Keypresses Single keypresses are represented by **[keyname]**. Keyname may be one or more keys. If **[↵]** has to be added, this will be explicitly shown. Shift-key combinations are shown as in **[↑f]**, which is identical to **[F]**. Control-key combinations are shown as **[^F3]**, which means to hold the control-key down while pressing function key F3.

Mouse keys Mouse key presses are indicated as 'click ■□□', which means to press and release the left mouse key. If a pressing of two or more mouse buttons is indicated, as in □■■, the action is taken only after both keys are released. This means that it is not necessary to press or release the buttons exactly simultaneously. It is enough if they have been depressed together at some time before release.

Strings A series of key presses, like a component name or a filename are indicated as **'enter [filename]'**. Unless noted, the string *must* be followed by **[↵]**. Quoted strings like **'Net edit'** refer to a menu selection, a message or other string appearing on the screen.

▸**item** This references an entry under the name **item** which contains additional information related to the current subject. If no further reference is given, the entry is in this chapter. The entry may be shown in the contents; if not, you can look it up in the index. You may want to review this entry.



CHAPTER 2

Layo1 Structure

Layo1 is a user-friendly program, that has many features for the new and advanced user alike. You will find that when you start using it regularly, you will ask yourself: 'what if I could do this'. Most of the time, you will find that you actually can do what you wish. To accelerate this learning curve, the next chapter will give you an insight in the internal structure and organization of Layo1. Although this chapter is not required reading for using Layo1, it will enhance your understanding of the various functions and relationships. It is especially usefull if you go into manipulating parts of a design or if you interface Layo1 with a schematic design package and want to control the PCB configuration from the schematic diagram. This important issue is discussed in the User Guide under ▸Quality Assurance.

Vectors

A Layo1 design is basically nothing more than a collection of vectors, also called 'datalines'. A vector can be a line section, a pad, a track section etc. The level-version of Layo1 you purchased specifies a maximum number of vectors any one design can hold. This number applies therefor to the number

of datalines that can be handled in a design as described below. A dataline is a basic data-unit which contains among others the X- and Y-position(s) of the vector, the type of vector, and the block number.

Blocknumbers

The blocknumber is important because it determines whether several vectors are grouped together as a block. A Layo1 component (or part) or subdrawing in fact consists of one or more blocks, each of which consists of individual datalines with the same blocknumber. As an example, a 14 pin DIP shape will consist of a single block, so that Layo1 will manipulate as a single entity. (You would not want to rotate one row of pads and leave the other stationary, for example). Now suppose you built a little subdrawing for a power supply consisting of a TO-220 shape (for the regulator), a bridge and a few capacitors. If you save this subdrawing, it will consist of several blocks, being the individual components. Each block in turn will be a collection of vectors or datalines specifying the type (pads, lines, text etc.), position etc. of the individual elements. It is important to realize that Layo1 manipulates blocks, as will be explained below (▸Subdraws).

Layo1 components

After the above it will come as no surprise that a Layo1 component consists of the following elements:

- one or more datalines with the same blocknumber;
- a type label (eg '100nF');
- a reference label (eg 'C21');
- a shape label (eg 'CKER-03).

The blocknumber will correspond to the line number of the particular component in the componentlist (to be discussed).

Generally, a datalines of a component specifies a pad and the lines that make up the shape outline.

Layo1 subdraws

A subdrawing consists of a number of blocks, together with a collection of tracks. When you load a subdraw into a design, Layo1 will ask whether it should separate the blocks. This is important, because if you answer no, all elements of the subdraw will have the same blocknumber. This means that you can manipulate the subdraw as a single entity. Moving, rotating etc. is much easier this way (▸Multifunc, block, rotate/move, Chapter 3).

Design control

The Layo1 Graphics Editor contains several features that help you to control the integrity of your design. The program can use information in two text files, the component list and the netlist. If these exist, Layo1 will check all actions you request against the information in these files (▸.NET, ▸.CMP). The chance of a routing error is very slim, unless you do it manually, on purpose. Layo1 will provide an overview of the required connections with the ratsnest (▸Ratsnest, Chapter 3). The ▸Router (Chapter 3) can use the connectivity information in the netlist, along with the setup parameters you provide, to route the connections. When you activate the Draw or Edit functions (▸Draw, ▸Edit, Chapter 3) Layo1 will highlight the pads and connections belonging to a particular net. Also, the color of a pad will indicate its status as follows:

- color a: routed;
- color b: yet to be routed;
- color c: not part of any net.

where color a, b and c can be set to your preference (▸Color settings, Chapter 3).

Net and component list

The net and component lists will normally be provided by a schematic capture program. But note that even if you do not use a schematic editor to provide Layo1 with a component and netlist, you can generate these manually with a text editor (▸.NET, ▸.CMP). Alternatively, you can use Layo1's internal netlist editor (▸Edit net, Chapter 3) to built nets 'on the fly' while developing your design. See also ▸Standalone design and ▸Integrated design in Chapter 4, User Guide.

Schematic capture

Layo1 can basically use the net- and component lists of any schematic capture program. The only condition is that the schematic capture program must be able to output these two lists in a Layo1 compatible format.

Netlist format

Layo1 can process net- and component lists conforming to the **CALAY** format, more specific the **MGIOS** variant. One widely used schematic capture program that can generate such files is **OrCAD/SDT**. The installation procedure for Layo1 gives you the option to install the Project Manager for integration with several versions of OrCAD/SDT. See ▸Installation in the User Guide. The Project Manager will set up its menus and the related parameters to instruct OrCAD/SDT to generate the required CALAY files, and transfer these to the Layo1 PCB directory. Also, the Forward Annotate menu entry will automatically cause an update of OrCAD's net-and component lists, and transfer the changes to Layo1. (See ▸Forward Annotate in the User Guide). If you wish to use another CALAY-compatible schematic capture program, you will either have to generate the net- and

component lists in the schematic capture program, or modify the relevant Project Manager menu entries.

Customizing Project Manager

The detailed command and syntax structure for the Project Manager are discussed in Chapter 5. However, for the following discussion assume that each action for a Project Manager menu entry consists of one or more Dos-command lines. The following fragment shows the commands that Project Manager will pass to Dos to generate the CALAY compatible net- and component list from OrCAD. It is assumed that all files for this project named **prj** are in the \orcad\prj\ directory. More information on the specific OrCAD functions can be found in the OrCAD reference documentation.

Project Manager menu entry: OrCAD/SDT IV NETLIST
(generate net/comp list)

```
CD \ORCAD\prj
INET prj.SCH
ILINK prj.INF /B
IFORM prj CALAY.CF prj.NET prj.CMP /B
```

Project Manager menu entry: Forward annotate
(update net/comp list and transfer to Layo1)

```
CD \ORCAD\prj
INET prj.SCH
INET prj.SCH /B
ILINK prj.INF /B
IFORM prj CALAY.CF prj.NET prj.CMP /B
CD \LAYO1\PMAN
UT UPDATE [arg1] [arg2] [arg3] [arg4]
```

The internal Layo1 utility called **UT** is called with its command **'update'**. This will do the actual updating of Layo1's files. This command needs 4 arguments as shown. These are:

- arg1: the directory where the calay-compatible net- and component files are kept;
- arg2: the directory where the Layo1 files for this design are kept;
- arg3: the directory where the Layo1 executable files are kept (usually ..\layo1p\;
- arg4: the Layo1 user number (see also Chapter 5).

Of course, the above fragments are specifically for use with OrCAD/SDT, but if you want to use another schematic capture program, you should use the same concept:

- assemble the commands to generate the CALAY compatible net- and component list;
- use the **'UT UPDATE'** command to transfer the updated information to Layo1.

Chapter 5 shows you how to set up the menu's for the Project Manager. With the above information and the documentation of your particular Schematic Capture Program, you should be able to do the integration. You may also want to review the contents of the **PM.BAT** file after reviewing the relevant entries in Chapter 5.

Annotate

When your schematic capture program has the possibility to update the net- and component list, and you want to transfer the changes to Layo1, you must be careful about what exactly you want to change. Modifications to the net- and

componentlist can only be accommodated if they concern additions or deletions. *Never renumber components! For instance, do not renumber R6 to R8, this will make the netlist and thus the autorouter unusable!* This is not a real limitation, because you can make any required changes to the schematic diagram just by adding and deleting components. Just do not renumber the existing components in the process.

Directory and program file

To avoid errors, you should keep Layo1's pcb files in a different directory from the schematic files. However, it is necessary that the netlist and componentlist files generated by a schematic capture program have the same name as Layo1's project name. Otherwise you are not able to use the Forward Annotate function to update the PCB design from the schematic diagram. This requirement can be satisfied by using a single project name for all your schematic and PCB related files, with only the extensions differing, and keeping a separate directory for schematic and PCB (layo1) related files. If you use Project Manager, this is automatically set up at install time.

Graphics editor command line

If you do not use Project Manager, but prefer to use a batch file, a Dos command line or some other menu system, you can call the Graphics Editor with the following DOS command line:

```
layo1edi d:\pcb\%1.cnf /SCHd:\sch\%1\
```

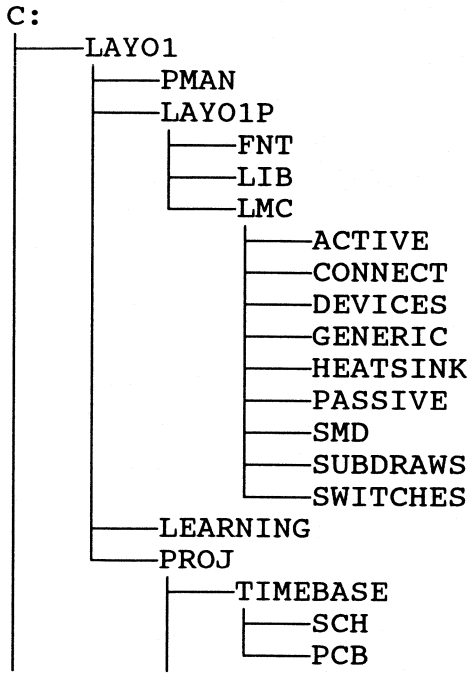


Fig. 2-1 Layo1 default directory structure

This is assuming that %1 is the project name, d:\pcb\ is the directory where you keep the PCB related files and d:\sch\%1\ the directory where you keep the schematic diagram related files (the calay compatible .net and .cmp list).

If the above command line is put in a batch file called `gredit.bat`, and the project name is TIMEBASE, the following command will call the Graphics Editor, load the TIMEBASE design and set up access to the relevant schematic design files:

```
gredit timebase
```

File system

A Layo1 graphical design basically is contained in a .PLY file. However, for the complete PCB design, several other files are used, which all have the same project name, but differ in their extension. Some information on the design is kept in files that are associated with a user rather than a design. This gives different users the possibility to customize colors, padshapes, penwidths etcetera to their own preferences.

.LMC .LMC files are the basic building blocks (components or parts) which you use to build your design. A .LMC file is basically a collection of datalines. Each dataline consists of 5 fields of 2 bytes each. The fields are:

- Blocknumber;
- Attribute word;
- X-coordinate;
- Y-coordinate;
- Netnumber.

The attribute word holds information on the type of dataline represented and related information like layer, pen, pad and drill diameter. An .LMC does not normally contain any tracks. A more detailed description on the data structure can be found in the **TOOLS** directory.

.PLY A .PLY file is a binary file that contains all graphical information about the PCB design. It is built up from a collection of components (.LMC's), traces, text, corner lines etcetera. But basically it still is a collection of vectors or datalines.

.CNF With each project will be associated a configuration file with extension **.CNF**. This file contains the following information:

- The PCB size as indicated by the corner lines;
- The corner lines offset relative to the cursor;
- The path to the project's .LML file if any;
- The path to the project's .PLY file;
- The path to the project's .CMP file;
- The path to the project's .NET file;
- The drill diameter definitions;
- The pad definitions;
- The penwidth definitions.

The .LML file is optional (▸.LML). The last three definitions are initially copied from the active USRXX.CFG file. After that, they are user-independent and may be edited for the current project with ▸Edit Design CNF, ▸Pad define and ▸Penwidth (Chapter 3) respectively.

The modified pad and pen definitions however may be saved in a user-related **.SMB** file. They can then be loaded while working on another design, so that a user-defined pad and pen set can be used in all projects worked on by a particular user. (See ▸Save file and ▸Load file in Chapter 3).

.LML If you edit the design configuration (▸Edit Design CNF, Chapter 3) to include the **.LML**, Layo1 will generate this file in addition to the regular .PLY, every time you save the

design. The .LML contains all design data including the pad and pen definitions from the original design. This means that a .LML needs also the corresponding .CNF file. You can deliver a .LML with its .CNF to a production site, and have the design produced using the Layo1 Output Driver. Logically however a .LML is not different from a ▸.LMC or a subdraw. In fact, you can rename a .LML to a .LMC. The only change is that you lose the custom pad/pen settings contained in the related .CNF. A .LMC uses either default settings, or assumes the settings of a design where it is imported.

.NET The netlist contains all nets and the connections (pins) that are part of it. Each net starts at col 0 with a '/', followed with the net name. Depending on whether you generate the netlist manually or with some schematic capture program, the netname may be a simple increasing number (eg **/N00006**) or a meaningful name (eg **/VCC**). Following the net name are the connections for that net as a component reference and the pin number (see following illustration). Each net is terminated with a ';'.

.CMP The component list consists of maximum 6 columns with the following format:

Column 1: **type** label, starts at col 0, max 20 characters;
Column 2: **ref** label, starts at col 21, max 8 characters;
Column 3: **shape** label, starts at col 41, max 8 characters;
Column 4: **x** coord, starts at col 61, in 1/1280 inch steps;
Column 5: **y** coord, starts at col 71, in 1/1280 inch steps;
Column 6: **rotation**, starts at col 80, in 90 degree steps.

```
/N00013    U2(8) JP1(4) U3(1);
/N00014    JP2(1) U3(3);
/N00015    JP2(2) U3(4);
/N00016    JP2(3) U3(5);
/N00017    JP2(4) U3(6) U3(13);
/N00018    JP3(1) U3(11);
/N00019    JP3(2) U3(10);
/N00020    JP3(3) U3(9);
/N00021    JP3(4) U3(8);
/VCC       C3(1) C2(1) JP4(2) U3(14) U4(14),
           U2(14) U1(14);
/GND       C3(2) C2(2) U2(12) U3(2) U3(12),
           JP4(1) U3(7) U2(7), U4(7) U1(7);
```

Fig. 2-2 Example netlist fragment

74LS08	U4	DIL-14	1216	512	1
74LS393	U3	DIL-14	2752	512	1
2.4576 MHZ	X1	CKER-03	1828	320	1
4 HEADER	JP1	SIL-1X04	3712	320	1
HEADER 2	JP4	SIL-1X02	4864	192	1
1K	R1	SFR25-01	5248	204	1
1 UF	C3	CTANT-01	6404	192	1
10NF	C4	CMKT-02	6784	256	1

Fig. 2-3 Example component list fragment

Both the netlist and the component list may be manually generated or edited with a standard text editor. **However, the files should not contain any control characters!**

.SMB This type of file can be used to save custom pad and pen definitions. Normally, if you change the pad shapes/dimensions and/or penwidths in a design, these are only

valid for that particular design, and will be stored in the .CNF. However, by saving them from the ▸Save file (Chapter 3) menu, you can again load them in another design using the ▸File load (Chapter 3) menu. You must provide a filename, but the extension will be .SMB.

USRXX.CFG This file contains all user defined parameters from the ▸User xx menu (Chapter 3). A particular user environment can be selected from the Project Manager menu, or be specified at the ▸Command line.

.LPU When you leave the Layo1 program, all settings related to the currently selected user and project are saved in a .LPU file. When the program is started up again, the environment, including the selected project name and user number, will be the same as when you exited the program.

.LIB A library file contains a list of component types (the values) and the Layo1 shape related to it (including the path where the shape can be found). Layo1 normally takes the shape for a particular component from the component list. If you work integrated with OrCAD, you will use the **Fieldstuff** function (see ▸Fieldstuffing in the User Guide) to provide the shapes.

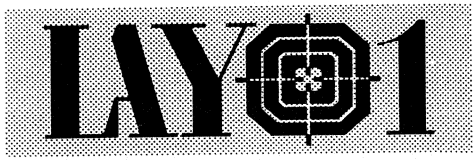
When you use Layo1 standalone, you will built the component list as you load the components into the design. When building the design, Layo1 will search the configured directories (see Chapter 3, ▸Configure libraries and ▸Configure directories) to load the shape. If the shape cannot be found, Layo1 will search the configured libraries.

Layo1 contains many predefined libraries. You can also create your own libraries with any text editor (see below for library file format). This may be necessary if you create custom

shapes. It is recommended not to make changes to an existing library, but rather to create new ones and configuring them in the ▸User xx menu, see Chapter 3. A typical library file fragment is shown below.

```
1 UF
=E:\LAYO1\LAYO1P\LMC\PASSIVE\CTANT-01
22NF
=E:\LAYO1\LAYO1P\LMC\PASSIVE\CMKT-02
1K
=E:\LAYO1\LAYO1P\LMC\PASSIVE\SFR25-01
2.4576 MHZ
=E:\LAYO1\LAYO1P\LMC\PASSIVE\CKER-03
4 HEADER
=E:\LAYO1\LAYO1P\LMC\CONNECT\SIL-1X04
74LS04
74LS393
=E:\LAYO1\LAYO1P\LMC\ACTIVE\DIL-14
HEADER 2
=E:\LAYO1\LAYO1P\LMC\CONNECT\SIL-1X02
```

Fig. 2-4 Layo1 library fragment



CHAPTER 3

Graphics Editor Reference

This chapter contains descriptions and specifications for all Layo1-functions and commands. Functions are listed in alphabetic order. The entry will indicate whether the function is a primary or a secondary function. Primary functions are available in a menu, although often there is a shortcut to directly activate them. A few functions are external to the Graphic Editor and must be run from the Project Manager. Each entry will note all methods to activate the function, both through the menu's, via the mouse or directly with a keyboard shortcut. A short description is given, along with any specific features, limitations etc. See also ▸Conventions in Chapter 1 for the notational conventions used.

Format

All available functions are listed, independent of the type of function or its use. There are, however, three groups of functions. They differ in the way that they are accessed and the situation they can be activated.

Menu functions These are accessed by opening a Layo1 menu and selecting the desired function. This will be explained below. The name of the menu will be indicated between brackets, to the right of the function name. Example: 'Load (file, load)' means that the 'Load' function is an entry in the file menu titled 'load'. Sometimes there is a shortcut to activate the function without

going through the menus. This will be shown as applicable.

Secondary functions

A secondary function is a function that is not used on its own, but instead is used to modify the effect of a primary function. or otherwise influence the display or appearance of the design. As an example, *Draw* is a primary function, while *Pen* is a secondary function defining the penwidth with which the drawing is done. They are shown in the function list as: 'Pen (secondary)'.

External functions

These are functions that are run from outside the Graphics Editor from the Project Manager, like 'Design rule check'.

Selecting from a menu

Activating a menu function is done by opening the submenu and selecting the required function. Opening the submenu is done by first opening the main menu from the Top Level, and then opening the required submenu. The Top Level is reached when you exit any other function and/or pressing **[Esc]** until the Layo1 logo is shown in the top left corner of the screen.

You open the main menu by pressing [↵] or clicking □□■. You select a submenu by placing the highlight over it and pressing [↵] or clicking ■□□. The highlight is moved by moving the mouse or with the [→] and [←] cursor keys. You select the desired function by moving the highlight over it as described above and pressing [↵] or clicking ■□□.

Mouse keys conventions

Although the mouse keys have often different effects in different functions, there are some global conventions. Firstly, clicking ■□□ *always* has the effect of accepting a default choice, opening a menu or selecting a menu or list entry with the highlight. This can be considered as a general confirming action. Clicking □□■ *always* terminates an ongoing function, exits from a menu or declines a default choice. This is a general escape action, and indeed has exactly the same effect as pressing **[Esc]**. When reference is made to 'confirming' or 'escaping', the two action described in this paragraph are meant.

Function list

The next large section contains a complete function list, with all information for their use. The list is ordered alphabetically. Note that each menu function entry will also show the menu selections necessary to activate the function. For all secondary functions the entry shows the key(s) or mousebuttons to be pressed to execute it.

Attributes display

(secondary)

Activated by: Press [’].

Function: Cycle component attribute display.

Description: Every time [’] is pressed, the program cycles the display of the component attributes in the following order:

- display component reference (eg **U4**);
- display component name (eg. **74LS74**);
- display component shape (eg. **DIL14**);
- no attribute display.

Note that each attribute text string has a unique block number and can be manipulated as any other object.

Auto settings

(user xx, auto settings)

Shortcut: None.

Function: To set parameters for auto functions.

Description: This selection opens a further submenu with the following choices.

Auto save time — This selection lets you set the autosave time between 0 and 120 minutes. The program will ask you if you wish to save your work if the preset time after the last save action has been expired. Note however that this request is postponed till you press any key or mouse button, to make sure that there is anybody at the keyboard to react to the request.

Auto move — This selection lets you set **auto move** to **ON** or **OFF**. If you set it to ON, and you perform a move ▸Block operation, the selected block will follow the cursor if you move it to the new location. Otherwise the block will only move to the new location after you press [↵] or click ■□□. Setting auto move to on improves ease of working, but may slow down the operation, particular on a machine that is not very fast.

Auto move delay — When Auto move is **ON** (▸Auto settings), this option will let you select how fast the manipulated block follows the movements of the cursor. The delay can be set between 0 and 500. %00 gives the longest delay. The absolute delay for a given setting depends on your particular PC and should be to your own preference. Automove on with a suitable delay can give a good compromise between speed (with automove off) and ease of working.

Auto pan — Auto pan can be set to on (**Y**) or off (**N**). When ON, the design will be automatically panned if the cursor is moved off-screen. With autopan off, the design will remain at its current position, even if the cursor would be moved off-screen.

Block (secondary)

Shortcut: Press **[b]**.

Function: To select a block.

Description: This function is used to attach a block to the cursor. This block can now be manipulated in various ways. It can be deleted, rotated, moved etc. See also the ▸Delete... , ▸Manipulate... , and ▸Copy... functions. The block is selected by placing the cursor over it and confirming. If your choice is ambiguous, Layo1 will cycle through the possibilities and let you select one.

Box (draw, box)

Shortcut: None.

Function: To draw a rectangle on the active layer with the active penwidth.

Description: This function is usefull as a general drawing aide (for instance for drawing custom component shapes). A box can also be used to confine or exclude ▸Router actions. When the function is activated, it is indicated in the top-left corner of the screen. You can then navigate the cursor where you want one of the corners of the box. After clicking ■□□ box draw is activated, which is indicated by highlighting the status line at the top of the screen. If you now move the mouse, a square will be drawn anchored at the first defined point, with the opposite point at the current cursor position. Clicking ■□□ again fixes the box. Note that Layo1 automatically adjusts the drawing to a perfect square, independent of you mouse movements. You can continue drawing another box by clicking ■□□ again. You can exit the box draw function at any time by clicking □□■, and a box that is not fixed yet will be deleted. In this function several additional mouse actions are enabled, which can be reviewed as usual with **[F1]**.

Center (secondary)

Activated by: Press **[c]**.

Function: To center the design.

Description: The cursor will be placed at the center of the screen. The design will be centered around the cursor

Check connections

(secondary)

Activated by: Press **[^F6]**.

Function: Show connection errors.

Description: This function will show all connection errors that violate the netlist. Each error is indicated by a cross in the defined color (see ▸Color settings) placed over a pad that is connected to a net it does not belong to. The crosses are placed on layer 13, so this layer must be visible for this function to work. The crosses are removed if you remove the error and select the function again.

Circle

(draw, circle)

Shortcut: None.

Function: To draw a circle on the active layer with the active penwidth.

Description: This function is similar to drawing a ▸Box. Initially, a square box will appear with the center at the position where the cursor was when this function was activated. Moving the mouse will increase the dimension of the square, but its center will remain fixed. When the box is fixed by clicking ■□□, it will change into a circle with a diameter equal to the side of the square. You can draw a circle on any layer, but for best

results you should select layer 8 or above. If you start drawing a circle and the current layer is layer 7 or below, Layo1 will display the ▸Layer select menu so you must make a selection. You can select a penwidth different from pen 1, but drawing on layer 8 or above will always be done with pen 1 anyway. If you later copy the circle to layer 7 or below, the penwidth that was originally selected will then be used to draw the circle.

Color mode

(secondary)

Activated by: Press **[↑F5]**.

Function: Toggles color mode between overlap and mix.

Description: Toggling the color mode to mix makes it easier to view the exact position of traces and/or pads at nodes. The normal mode is overlap which gives a better view of the run of longer traces.

Color settings

(user xx, color settings)

Shortcut: None.

Function: To set the colors of the various displayed layout elements to personal preference.

Description: When selected, a new window will be shown, listing three groups of objects: layers, netlist pads, other. The netlist pads refers to pads that are part of a netlist only.

Layers — If you select this choice, a window will open from which you can repeatedly select any layer. Then yet another window will open where you can select the color for that layer.

Netlist pads — If you select this choice, a window will open from which you can repeatedly select several combinations of layers and pad status (not used, used but not routed, used and routed). Another window will open where you can select the color for that layer/pad status combination. When you are done you exit with **[Esc]** or clicking □□■, at which time you return to the User xx menu.

Other — If you select this choice, a window will open from which you can select cursor, grid dot or grid raster. Another window will open where you can select the color for the selected object. '▸Grid dots' refers to the dots used to show the .1 inch grid. '▸Grid lines' refers to the lines used to show the selected raster grid.

When you are done with any subselection, you exit with **[Esc]** or clicking □□■, at which time you return to the User xx menu. A typical screen display for this function is shown below.

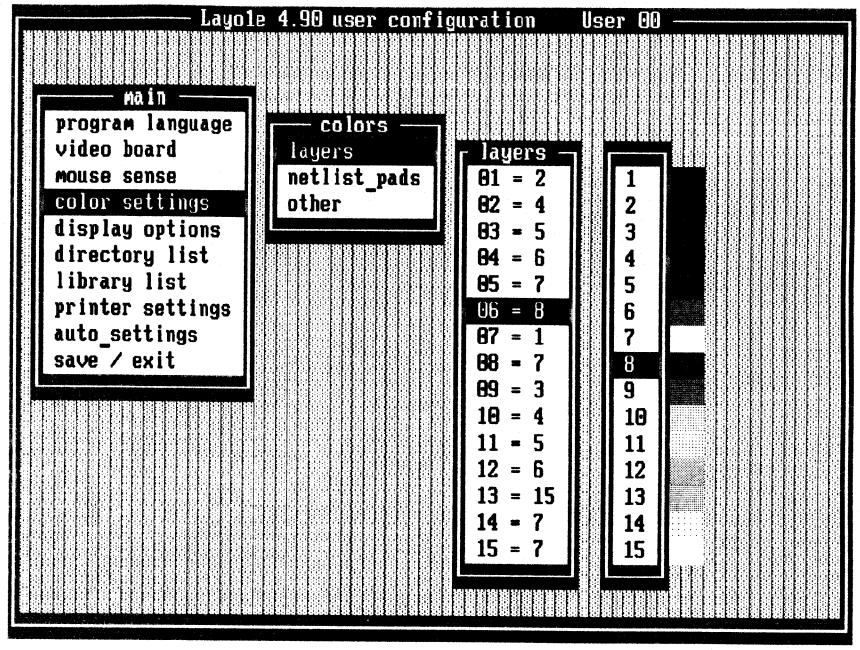


Fig. 3-1 Typical color setting screen

Configure language

(user xx, program language)

Shortcut: None.

Function: Select language for messages and menus.

Description: A window will open allowing you to select a language for the program messages and menus. The selection will only be valid for the current user number. Note that the menus of the Project Manager are not affected.

Configure video driver

(user xx, video board)

Shortcut: None.

Function: Select a display adapter and related parameters.

Description: This function will open a submenu where you can set several display adapter parameters, see the figure below.

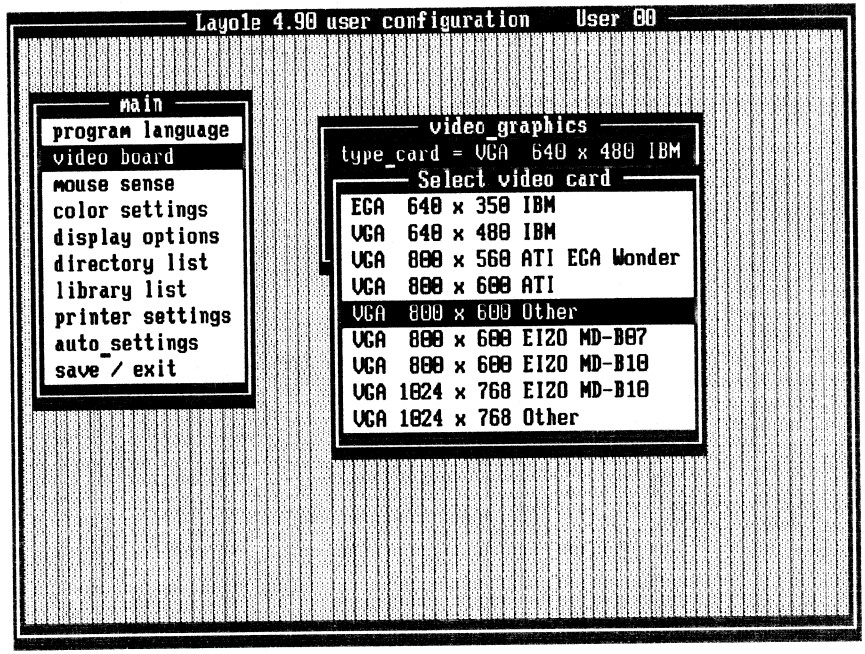


Fig. 3-2 Display adapter parameters

The recommended procedure is first to select a board type. Layo1 will open a window where you can select several pre-defined adapters. If your adapter is not listed, select a standard IBM mode that corresponds to your board, and set the board mode to the selected IBM mode when starting Layo1. When a board is selected, the other parameters will be set to the default that fits the adapter. In case you have a non-standard board, you can however change several parameters. There are a bewildering number of adapters and modes, and Layo1 will support many of them. However, in case of difficulty, especially with extended resolution boards, you may have to go to a standard IBM-compatible mode. See your video adapter manual for more information on supported modes.

Configure directories

(user xx, directories)

Shortcut: None.

Function: To configure directories for Layo1.

Description: This selection will open a window where you can choose two directory groups to configure. One group is the directories where the components are located (the .LMC files), the other group are the directories where the Layo1 designs are located (default under the PROJ directory). These directories are used by Layo1 to search for components or designs. Each group, when selected, opens a window listing the currently configured directories. You can remove or edit an entry by selecting it and then editing the path with the usual editing keys. Selecting an empty slot will enable you to add another directory path. A typical display is shown below.

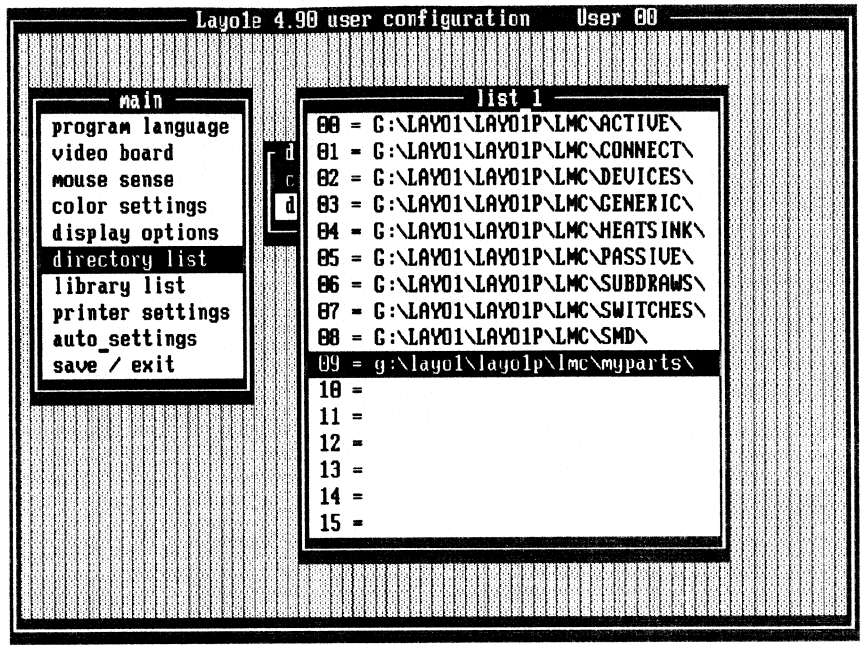


Fig. 3-3 Typical configured directories list

Configure libraries

(user xx, libraries)

Shortcut: None.

Function: Select libraries to be used.

Description: When the Libraries function is selected from the user menu, a list of currently configured libraries is opened as shown below.

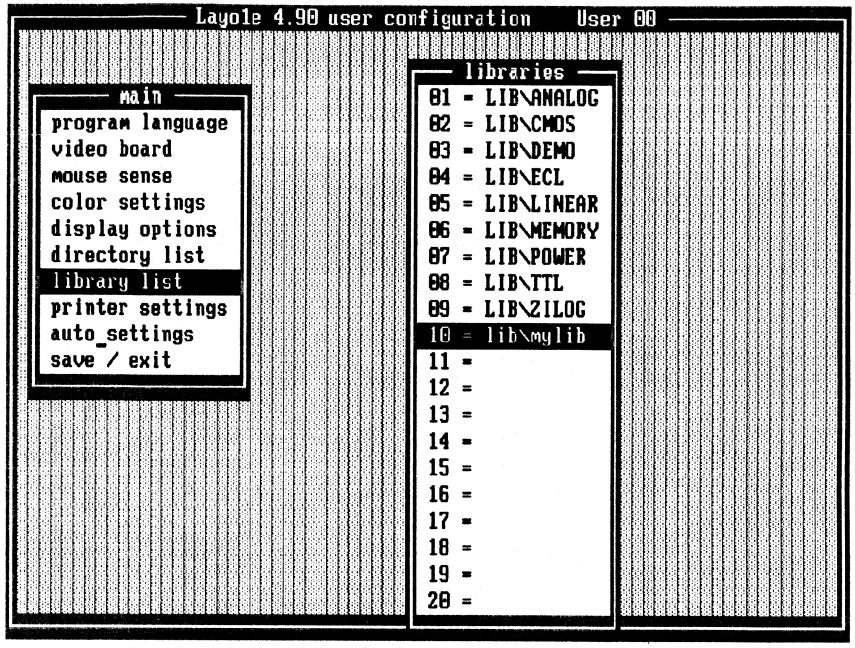


Fig. 3-4 Configured user libraries list

These are the libraries that Layo1 will search when looking for a particular component shape, if the shape cannot be found in the configured directories. See also ▸Configure directories, and ▸.LIB in Chapter 2 for the contents of a library. If you place the highlight cursor on an existing entry, and press [↵] or click ■□□, you can change or delete the entry with the usual editing keys. You can add libraries by selecting an empty entry and filling in the path and filename of the new library you want to configure. Note that this list is only valid for the current user.

Configure printer

(user xx, printer settings)

Shortcut: None.

Function: Configure the printer to be used with ▸Printer checkplot.

Description: Selection of this option from the ▸User xx menu gives you two sub-choices. You can select printer **type** or printer **port**. If you select type, a list of printers is displayed from which you can select the printer to be used for ▸Printer checkplot. Currently, Epson Matrix printers and Hewlett Packard Deskjet and Laserjet printers can be selected. Selecting printer port gives access to further selections from LPT1..4 through COM1..4. If you select a COM port, you are further queried for the port setup parameters. A typical display is shown below.

Coordinates

(secondary)

Activated by: Press **[x]** or **[y]**.

Function: Lets you position the cursor to an x- and/or y-coordinate.

Description: When activated, a window opens where you can enter the required coordinate. The value will be interpreted in the currently active scale type (mm or inch). A decimal value can be entered with a maximum of two fractional digits.

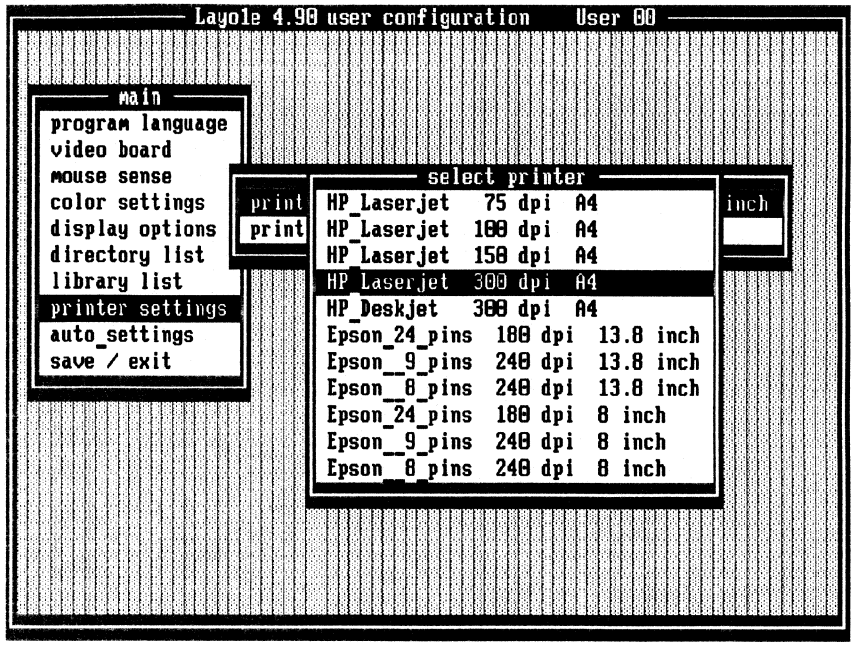


Fig. 3-5 Configure printer settings display

Copy repeat (secondary)

Activated by: Click ■□□. with object attached to cursor.

Function: To copy an object a specified number of times, on a specified distance, in a specified direction.

Description: This is a very powerful function to generate patterns of objects, for instance bus traces. First copy the object to the first position. Do not move the mouse, and activate this function. If necessary press **[p]** or click ■■□ to set the origin at the cursor (▸Origin). Now move the cursor to the next position. Enter a number from 01 to 99 *on the number keys, not on the numeric keypad!* The number will be shown at the screens top-left corner. Now press **[Ins]** and the object will be copied as specified.

Tip: If you wish to autoroute the connections to repeat-copied objects, make sure that you have a grid of 1/20 or coarser active, and the cursor snapped to the grid, before starting the copying. ▸Grid set, ▸Snap cursor.

Note: When you do a repeat copy in a .PLY design, you can only use the autorouter (and its associated error checking feature, ▸Check connections) if you update the netlist. See ▸Edit net and ▸Forward annotate in the User Guide.

Copy pad

(pad, copy)

Shortcut: None.

Function: To copy an existing pad to another location.

Description: The existing pad is 'picked up' by placing the cursor over it and clicking ■□□. The cursor with a copy of the pad can then be moved to another location. The pad is dropped by clicking ■□□. You exit the function with **[Esc]**. See also ▸Copy repeat.

Copy line (draw, copy)

Shortcut: None.

Function: To copy a line attached to the cursor.

Description: You select the line or track section to copy by placing the cursor over it and confirming. As long as it is attached to the cursor, it can be copied to any position, any number of times. The object is placed by clicking ■□□. Exit with **[Esc]**. See also ▸Copy repeat.

Copy object (multifunc, block; line; window; {P1..P2})

Shortcut: None.

Function: To copy an object, line, window or group attached to the cursor.

Description: As long as the selected object is attached to the cursor, it can be copied to any position, any number of times. The object is placed by clicking ■□□. See also ▸Copy repeat.

Corner lines (draw, corners)

Shortcut: None.

Function: To draw the PCB corner lines.

Description: When you draw the corner lines, they define the PCB dimensions in the .CNF file. Alternatively, the corner lines will be automatically adjusted if you set the PCB dimensions in the .CNF (▸Edit Design CNF). Drawing is identical to drawing a ▸Box, except that you can define a rectangle as well as a square.
Also, when the box is fixed, it will change into 4 angled line segments at the PCB corners. Corner lines are automatically drawn on Layer 15.

Cursor select

(secondary)

Activated by: Press **[A]**.

Function: Select a cursor shape.

Description: This function will cycle the cursor shape through arrow, long and short. This corresponds to the choices available from the menu ▸User xx, display options, cursor type.

Delete file

(file, delete)

Shortcut: None.

Function: Delete a selected file from disk.

Description: When the function is activated, you are asked to enter a filename/pathname. If you just press **[↵]**, you will enter the ▸Directories function so you can navigate the disk to select the file for deletion. In that case, the file will be selected by placing the highlight on it and click ■□□ or press **[↵]**.

If the file can be located, the program will ask you if you wish to delete it and do so if affirmed. If you type in the filename/path, all file name editing functions as described for ▸File/directory select are available, *except* the **[F7]** and **[F8]** selection options.

Delete layer

(options, layo1 toolbox, delete layer)

Shortcut: None.

Function: Permanently delete all objects from a layer.

Description: This function enables you to *permanently* delete all objects from a layer in your design. When you select this function from the Toolbox menu, Layo1 will show the CAM data list as shown below.
You can only move the highlight cursor to a layer in the left column that contains objects. Pressing **[↵]** or clicking ■□□ causes the program to request confirmation, and removes all objects from the layer. You cannot remove objects from layer 0.

Delete object

(draw, delete; pad, delete; multifunc, object, delete)

Shortcut: Press **[Del]** for delete line, press **[b], [Del]** for a block.

Function: To delete an object from the currently active layer.

Description: You must select the object to be deleted, either by confirming it, or by going through the multifunc menu. A block object can also be selected directly by pressing **[b]**.

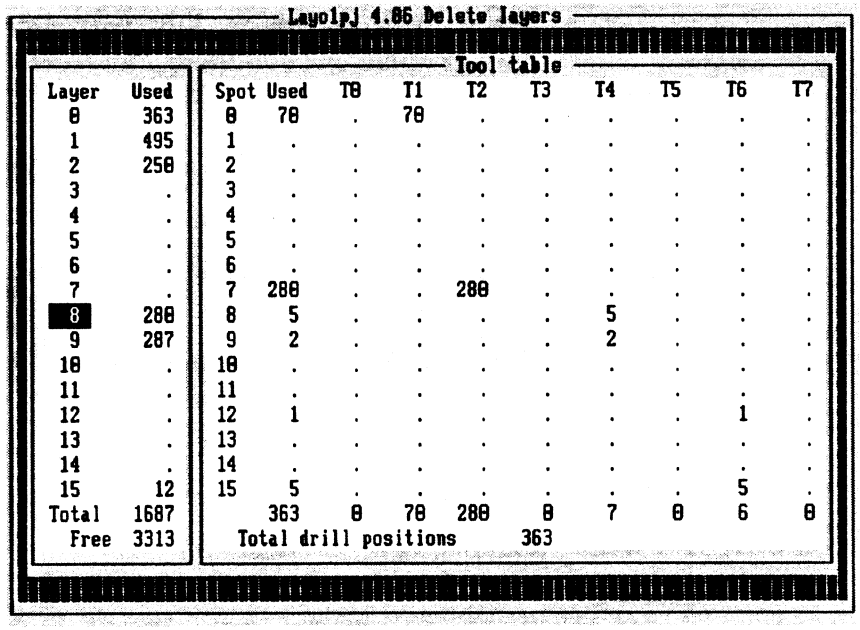


Fig. 3-6 Delete layer CAM data list

The selected object is deleted by clicking □■■, or by pressing **[Del]** in case of a block. When you press **[Del]** from the Top Level, you will enter the delete line mode. You can now delete a line segment by confirming it. You exit this function by pressing **[Esc]** or clicking □□■.

Pad

If you wish to delete a pad, you must place the cursor *exactly* in the hart of the pad. It may be necessary to select a finer grid and/or a larger zoom factor (▸Zoom) to be able to do this. Also, note that if you are working on a design based on a netlist, you *cannot* delete a pad that is part of the netlist.

MultiFunc

If you want to delete an object that is contained in the netlist, Layo1 will indicate this. If you nevertheless want to continue, the object is deleted from the componentlist and the netlist as well as from the design. You make the netlist, ratsnest, componentlist and autorouter functions for the design unusable. Be very careful in these cases!

See also ▸MultiFunc menu, ▸Manipulate window, ▸Block, ▸Manipulate group.

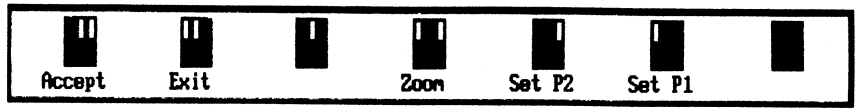


Fig. 3-7 Delete multifunc/ group mouse functions

Design rule check

(external, Project Manager)

Shortcut: None.

Function: Check the design for clearances as specified.

Description: This is a separate utility that can be started from the Project Manager. There are three choices for the clearance limits. This utility 'DRC' report its findings on-screen and also to the file **'projectname.DRC'**. This file can be viewed with the editor. You can also start DRC from a DOS command line with the command:

```
drc [projectname] xx
```

'Projectname' is the path and filename (without extension), and 'xx' is the clearance specification. This value is in 1/1280 of an inch. For instance, if you set xx to 32, DRC will check for clearances less than 32/1280 = 1/20 inch. If you omit xx, the specification will be set to the default 10 (1/128 inch).

Diagonal lines

(secondary)

Activated by: Press **[q]**.

Function: Displays diagonal help lines.

Description: The lines are displayed from the cursor position. When you move the cursor and again press **[q]**, another set of diagonal lines is displayed. All diagonal lines are erased when you redraw the screen (▸Redraw screen).

Directories

(file, directories)

Shortcut: None.

Function: To let you navigate through the directory tree on your harddisk.

Description: This function is usefull if you are looking for a specific file or file in a specific directory for loading or saving components. You do not have to leave Layo1 to go to the DOS level. Using the highlight, you may display the subdirectories on your hard disk, or display the files in a selected directory.

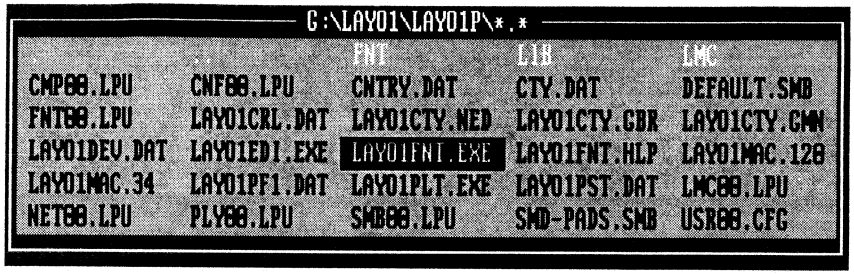


Fig. 3-8 Typical File/ directories browse display

Display options

(user xx, display options)

Shortcut: None.

Function: Set various display options for the active user.

Description: This enables you to set the display options indicated in the figure below. The cursor- and component name settings can be selected from a submenu as shown. When the selections are saved they will be valid for any subsequent session for the currently active user number.

Display CAM

(options, display cam)

Shortcut: None.

Function: Display design data for the current design.

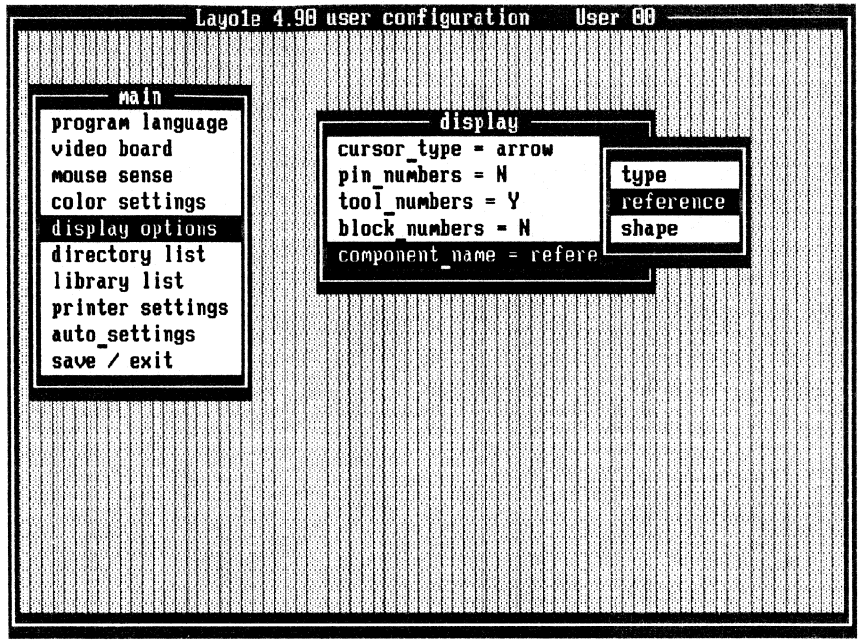
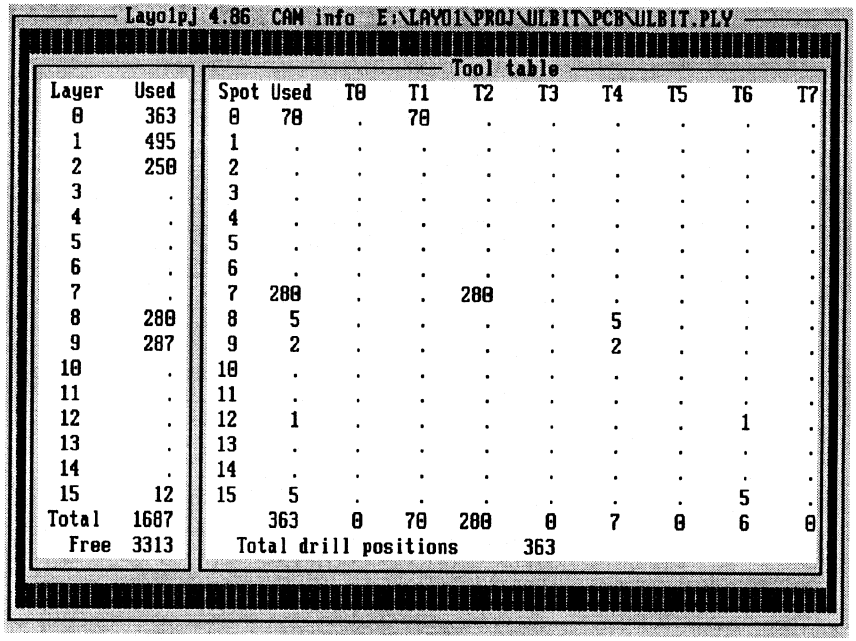


Fig. 3-9 Display options, component name options

Description: The display shows the number of vectors present on each of the layers. Also, the matrix shows all combinations of tools and spots (the 'real' pads from set 0, ▸Pad select) and the number of occurrences. The screen also shows the total used vectors and the remaining free ones, and the total number of drill positions.



Layo1pj 4.86 CAM info E:\LAYO1\PROJ\ULBIT\PCB\ULBIT.PLY

Layer	Used
0	363
1	495
2	250
3	.
4	.
5	.
6	.
7	.
8	200
9	287
10	.
11	.
12	.
13	.
14	.
15	12
Total	1607
Free	3313

Tool table

Spot	Used	T0	T1	T2	T3	T4	T5	T6	T7
0	70	.	70	.	.	.	.	.	.
1	.	.	.	.	.	.	.	.	.
2	.	.	.	.	.	.	.	.	.
3	.	.	.	.	.	.	.	.	.
4	.	.	.	.	.	.	.	.	.
5	.	.	.	.	.	.	.	.	.
6	.	.	.	.	.	.	.	.	.
7	200	.	.	200	.	.	.	.	.
8	5	.	.	.	.	5	.	.	.
9	2	.	.	.	.	2	.	.	.
10	.	.	.	.	.	.	.	.	.
11	.	.	.	.	.	.	.	.	.
12	1	.	.	.	.	.	.	1	.
13	.	.	.	.	.	.	.	.	.
14	.	.	.	.	.	.	.	.	.
15	5	.	.	.	.	.	.	5	.
	363	0	70	200	0	7	0	6	0

Total drill positions 363

Fig. 3-10 Display CAM screen

Draw line (draw, draw)

Shortcut: Click ■□□

Function: Drawing lines/tracks.

Description: This function lets you draw lines in your design at any layer and with any of the 7 penwidths. You can change the penwidth with ▸Pen and ▸Penwidth. The layer on which the drawing takes place can be selected with ▸Layer select. While drawing you can back-trace over the line to modify it, to add extra curves or change the routing.

Remember that on layers 7 and below, Layo1 will only use 90- and 45-degree turns. Any turns and arcs are possible on layers 8 and above. The figure below shows the default penwidths at full scale.

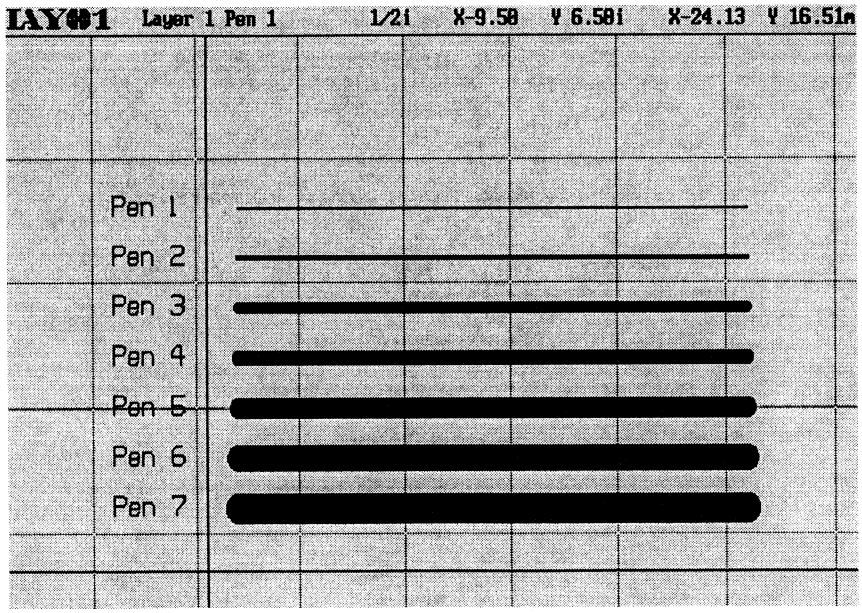


Fig. 3-11 Full scale default penwidths

There is no difference in drawing a line or drawing a trace. This function is therefor used both for manual routing as well as for any general drawing or building new components and shapes. Note however that when you use the ▸Route manual function to draw traces, Layo1 will check the legibility of it. This means, that you can only route (or draw) between pads. The general draw functions from the ▸Draw menu like Draw, Box, Ellipse etc. do not have this limitation. You start a line by placing the cursor at the start position and clicking ■□□.

This places the program in the **'pen down'** position. Move the cursor to the end point of the line or the segment and again click □□■ to end the line, or click ■□□ to place a node and continue to the next point. Note that in the pen down state, the mouse buttons offer the functions shown below.

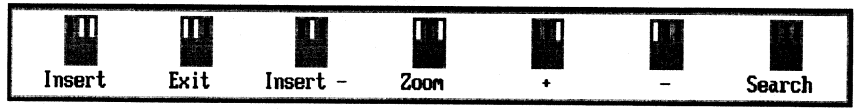


Fig. 3-12 Draw mouse functions

Insert — This action starts the drawing process, and each time you click this, a node is inserted in the line, enabling you to change direction.

Exit — Clicking this key places the final node of a line and terminates the drawing function. Same as **[Esc]**.

Insert- — Clicking this will enable you to go backwards from the last node inserted. As you move the cursor backwards over the line, you will see that the line is attached to the cursor. That is, you can 'pull' parts of the line to another position. This enables you to modify the shape and routing of a line (or a trace). When you do change the line shape or routing, you can fix it in the new position in several ways. To just fix it click ■□□. This is the regular 'insert' to place a node. You can, however also fix and automatically jump to the previous node (click □■■) or to the next node (click ■■□).

+ Fix the current node and jump to the next node (see above).

- Fix the current node and jump to the previous node (see above).

Find All the above functions are available for the line currently being drawn. However, if you want to modify a line previously drawn, you can make that the currently active line by clicking ■■■. This will blink the line (or segment) under the cursor. Layo1 will ask you to confirm (see ▸Edit).
If you do, all above described editing functions are available for the selected line. This function is identical with the ▸Edit function in the draw menu, and can also be accessed by just pressing [=].

Draw menu

(main, draw)

Shortcut: None.

Function: To allow access to various drawing functions in the Graphics Editor.

Description: The Draw Menu gives access to various drawing and editing functions as shown below.

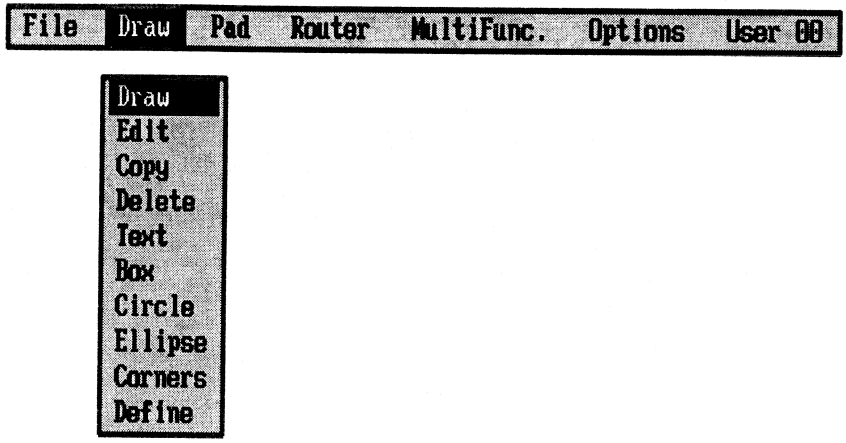


Fig. 3-13 Draw menu functions

Although most of the drawing functions can be executed on any of the 16 layers (numbered 0 through 15) and with any of the 7 penwidths (numbered 1 through 7), there are some defaults and restrictions.

Layers 1 and 2 are used for placing SMD pads, laying down traces and general drawing under 90- and 45 degree angles. These layers are generally used as component and solder side of single or dual layer board. Drawing under any angle or curves is best done on layers 8 and above. The default layer settings are as follows:

- Layer 0: not for user; contains pad/holes which are visible at any layer;
- Layer 1: traces and SMD pads (component side);
- Layer 2: traces and SMD pads (solder side);
- Layer 8: component shapes (component side);
- Layer 9: text (component side);
- Layer 10: component shapes (solder side);
- Layer 11: text (solder side);
- Layer 13: error indications (▸Check connections);
- Layer 15: corner lines.

These are based on the design of single or double sided PCBs. For multilayers, you should determine beforehand the layer usage to minimize the chance for errors. Note that at any time you can copy the elements at any layer to another layer, or ▸Layer change.

Drill group

(options, layo1 toolbox, drill group)

Shortcut: None.

Function: Group identical pads for identical drill diameters.

Description: This function will show a matrix of design information as depicted below.
The left column shows usefull design information: the number of vectors present on each layer, the total number of vectors used and the number remaining. The right column shows how many of each of the pads are used, and the drill diameters associated with them. Often, different components with identical pads can have different drill diameters. For efficient production it is important that as little different drill diameters are used as absolutely necessary.

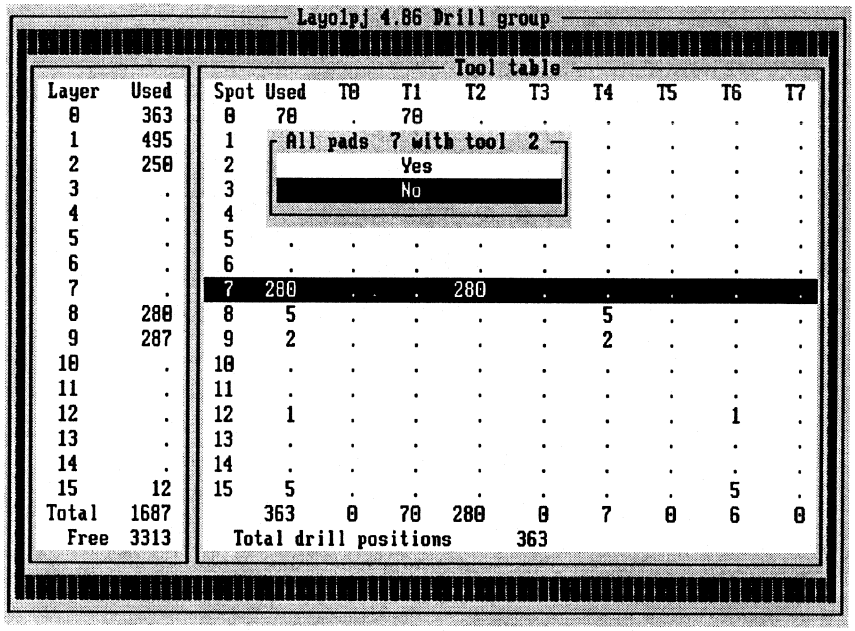


Fig. 3-14 Drill group matrix

With this function you can group drill sizes for a particular pad. Place the cursor highlight on a tool number (this can be a tool that is not used at presently) and click ■□□. Layo1 will ask for confirmation and then set the tool number for all instances of the selected pad to the new tool.

Edit Design CNF

(options)

Shortcut: None.

Function: Set up or modify design parameters.

Description: This function will let you change the most important parameters that define the design, like dimensions, tool diameters, file locations etc. The screen is shown below.

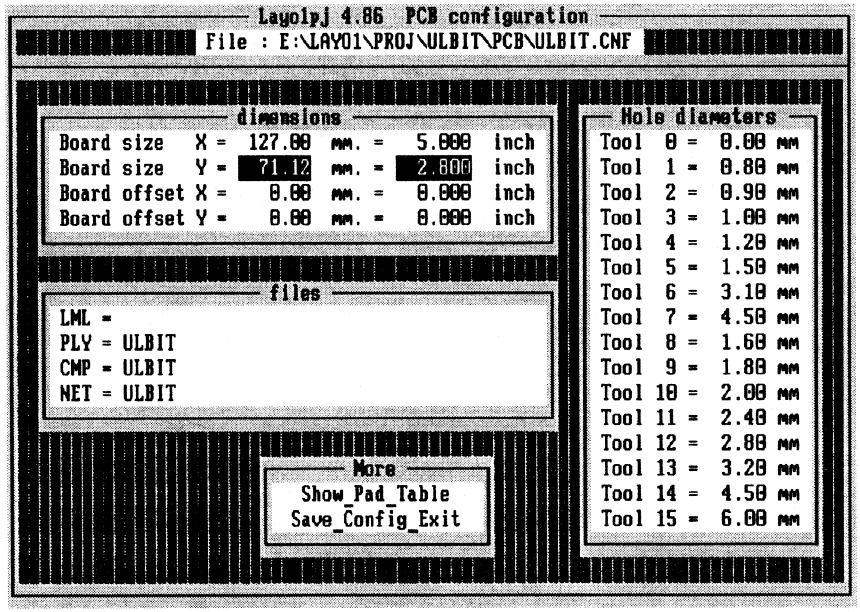


Fig. 3-15 Edit Design CNF parameter screen

All parameters can be modified by placing the highlight over it and pressing [↵] or clicking ■□□. Mostly there will be a window to enter the new setting for the selected parameter. Note that if you enter a filename for **LML**, your design will be saved both as a **.PLY** and as a **.LML** every time you save it. See ▸.LML and ▸.PLY in chapter 2.

Note: The changes that you save are only valid for the current design. If you want to customize pads and penwidths for personal use, you should modify them with ▸Pad define and ▸Penwidth and save the changes with ▸Save file, PadPen Def.

Edit net

(options, edit net)

Shortcut: None.

Function: Modify the contents of a net.

Description: When activated, you will return to the graphics editor screen, with the top left indicating that you are in the neteditor. The neteditor acts on nodes that can be part of a net. A node is a pad of a component that has a pin number associated with it. You can execute three possible action:

- add a node to a net;
- delete a node from a net;
- transfer a node from one net to another.

add a node — You select the net to which you want to add a node by clicking any of its nodes. All other connections for that net will then be highlighted. You now click the node you want to add to the net.

delete a node — Again, select the net as described above. Then click the node you wish to remove, and Layo1 will ask you if you really want to do it. If affirmative, the node is removed.

transfer a node — Select the net to which you want to transfer a node as above. Then click the node from another net to be transferred. Again, Layo1 will ask you to confirm your request.

If you click an uncommitted node before selecting a net, Layo1 asks if you want to start a new net. If you confirm, the clicked node will be the first one in a new net. See also the tutorial in Chapter 4, ▸Stand-alone design, in the User Guide. You exit the neteditor by clicking □□■. You are then asked if you want to save the changes you made, or to leave the design as it was before.

Note: If you are running under PM with a schematic capture program, Layo1 will not let you use the neteditor. Instead, you should use ▸Forward annotate as described in the User Guide.

Edit

(draw, edit)

Shortcut: Press [=] or click ■■■.

Function: To modify lines, tracks or pad positions.

Description: You can only enter this mode when the cursor is on an editable element. If the cursor is on more than one editable element, Layo1 requests conformation on the object you want to edit. If you are editing in a design based on a netlist, Layo1 will highlight all elements that are part of the selected net. See also ▸Draw line

Ellipse

(draw, ellipse)

Shortcut: None.

Function: To draw an ellipse on the active layer with the active pen.

Description: This function is similar to drawing a ▸Box. Initially, a rectangular box will appear with the center at the position where the cursor was when this function was activated. Moving the mouse will increase the dimension of the rectangle, but its center will remain fixed. When the box is fixed by clicking ■□□, it will change into an ellipse. The ellipse will have dimensions equal to the drawn rectangle. You can draw an ellipse on any layer, but for best results you should select layer 8 or above. If you start drawing an ellipse and the current layer is layer 7 or below, Layo1 will display the ▸Layer select menu so you must make a selection. You can select a penwidth different from pen 1, but drawing on layer 8 or above will always be done with the minimum line width anyway. If you later copy the ellipse to layer 7 or below, the penwidth that was originally selected will then be used to draw the ellipse.

File/directory select

(secondary)

Shortcut: None.

Function: Select the file to be used in the current function.

Description: Whenever you activate a function that operates on a file, Layo1 allows you to select the file in different ways. A window will open where you can enter the file and/or the full pathname. If possible, Layo1 will show the pathname with the cursor set at the last character so you just have to complete the filename. An example is shown below.

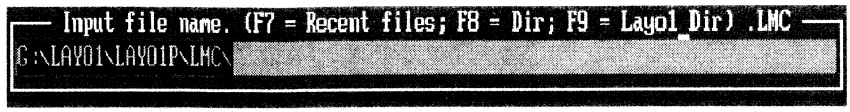


Fig. 3-16 Directory/file select window

Note: If you just start typing, Layo1 assumes that you want to start from the very first character, and erases the preset path. If you want to complete the filename, start by pressing **[→]** before entering the file name. Of course you can also edit the path with the usual edit keys.

Other directory

You can also select another directory by pressing **[F8]**, and then select a directory with the highlight and pressing **[↵]** or clicking ■□□.

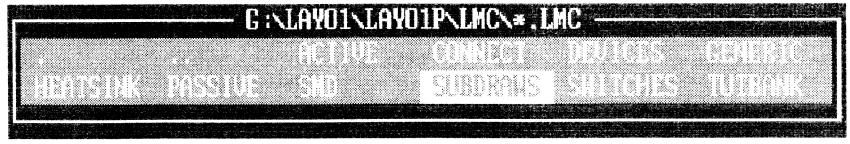


Fig. 3-17 File/directory select directory window

Recent files

Pressing **[F7]** will display a list of recently accessed files. You can select a file as described above.

Layo1 directory

Finally, pressing **[F9]** will open a window showing the directories in the Layo1p directory for selection.

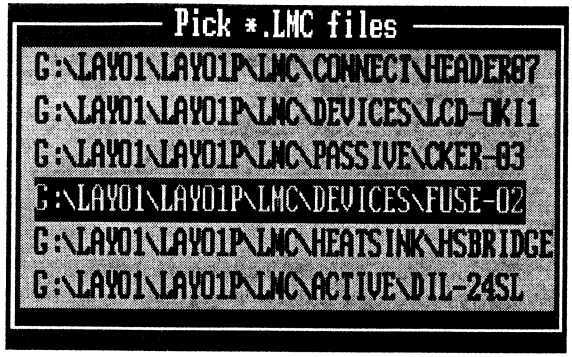


Fig. 3-18 File/directory select recent files

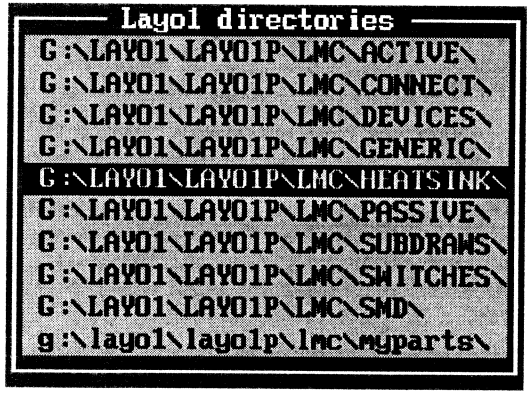


Fig. 3-19 File/directory select Layo1p directory

File menu (main, file)

Shortcut: None.

Function: Saving and loading designs, rename and delete files, reset and quit Layo1 and others.

Description: The File Menu gives access to various file- and DOS related functions. These include functions to load and save designs, subdraws and parts. Limited DOS related rename and delete functions are also available. Furthermore, this menu will let you restart Layo1 with a clean screen and exit the program.

DOS functions The Layo1 file system is organised around a project. All files related to a design (or project) have the same name. See also ▸File system in Chapter 2. Therefor, normally there is no need to access file related functions from the DOS level. As for creating or removing the necessary files and directories, this is best done from the appropriate menu selections from the Project Manager menus. Project Manager also offers functions for backing up a project, and for exporting and importing to /from a floppy. If you work in an integrated environment, the import and export functions will also write and read the appropriate OrCAD/SDT files. However, the file menu does offer some limited functions. You can rename or delete any file by selecting the appropriate function from this menu. There is also a function to browse a disk if you are looking for a particular directory or file (▸Directories).

Save functions One thing to remember is that Layo1 will not automatically save *all* project related files at once. It does save the design files whenever you do a save with ▸[F4], or a timed save (see ▸Auto settings). However, the custom penwidth and pad definitions must be saved by you. This way, you can give this file (with suffix ▸.SMB) a meaningfull name, which is not necessarily related to your current project.

Activating the File Menu opens a drop-down menu with functions as shown below.

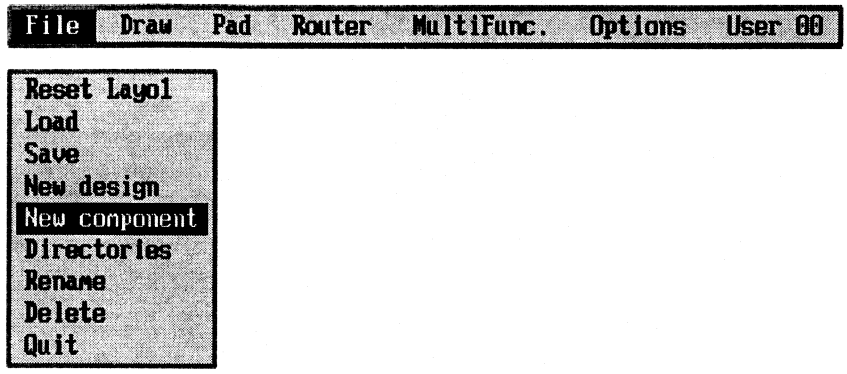


Fig. 3-20 File menu functions

These can be selected by moving the highlight to the desired choice with the cursor keys or the mouse. The selected choice is activated by clicking ■□□ or by pressing [↵]. The following menu choices are available:

- ▸Reset program;
- ▸Load file;
- ▸Save file;
- ▸New design;
- ▸New component;
- ▸Directories;
- ▸Rename;
- ▸Delete file;
- ▸Quit.

Fill (secondary)

Activated by: Press [F].

Function: To fill a bounded area.

Description: The cursor must be on a line or trace fully enclosing an area when **[F]** is pressed. Layo1 will blink the line and ask you to confirm that this is the boundary of the area you wish to fill. If you confirm, the area will be filled with 1/80 inch wide horizontal lines. Due to this process, the area must be bounded by lines that are vertical, horizontal or run under a 45 degree angle. Filling other shapes produces coarse results. The fill pattern may be placed on any layer. You can also fill areas that contain other areas, for instance a box containing another box. If you fill the outer box, the inner box remains clean.

Grid dots (secondary)

Activated by: Press **[.]**.

Function: Toggle the display of the dot help grid.

Description: When toggled on, a dot grid is displayed in a 1/10 inch raster. The grid is aligned with the top left corner line, or if not present, with the cursor position at the time the design (or component) was loaded.

Grid lines (secondary)

Activated by: Press **[↑ >]**.

Function: Toggles the display of the raster help grid.

Description: When toggled on, a raster grid is displayed in the current grid size (▸Grid set). The grid is aligned with the current cursor position.

However, to prevent cluttering the display to much, this function is inhibited at the lower zoom settings. Exactly where the inhibiting starts depends on your design dimensions, because Layo1 will set the zoom scales so that there is one setting at which the design exactly fills the screen.

Note: If the imperial ruler is active, the raster grid will be aligned with the ▸Grid dots. With the metric ruler active, the raster grid will be aligned with the cursor position at the time the raster grid was set, see ▸Grid set.

Grid units (secondary)

Activated by: Press **[m]** or **[↑F9]** to set to mm, or **[i]** to set to inches.

Function: Set the grid units to use mm (metric) or inch (imperial).

Description: These functions set the grid units to millimeters or inches. This has several effects. The grid steps will change to the selected units as shown in the status line. (Note that both mm and inch coordinates will be displayed in each case). It will effect the way **[F9]** and **[F10]** increase and decrease the grid size, see ▸Grid set and ▸Grid define. Any cursor movements with the mouse or the cursor keys will be done in steps as set by **[F9]** or **[F10]**.

Grid set (secondary)

Activated by: Press **[F9]** or **[F10]**.

Function: To increase or decrease the current drawing and placing grid.

Description: Pressing **[F9]** decreases the drawing grid size, while **[F10]** increases it. If the currently active ruler system is inch, the grid size will be halved or doubled each time you press one of these keys. If the current ruler is millimeters, the gridsize will change according to the settings specified with the with the ▸Grid define function.

Grid define (secondary)

Activated by: Press **[^F9]**.

Function: To define metric drawing and placing grid sizes.

Description: You can define up to 10 different grid sizes, that can be set when the metric ruler is active. See ▸Grid set.

Layer display (secondary)

Activated by: Press **[d]**.

Function: Select display of individual or groups of layers.

Description: When activated, a menu as shown below is presented. It shows several items that can be set and/or selected.

ActualGroup

This column shows eight names for groups of layers. The names are user definable, only the default names are shown here. The names are edited by placing the cursor highlight on them and pressing **[F9]**. The name can now be modified using the usual editing keys. You should select a meaningful name, such as AllCopperSide for all layers containing objects on the PCB copper side, or PadsOnly for the display of only pads. The display of a particular group is selected by pressing its function key (**[F1]** through **[F8]**). The numbers above the columns are the layer number. The last four columns represent the following groups:

- P0: the 'pad layer', ie all pads excluding vias;
- S1: All SMD pads on layer 1;
- S2: All SMD pads on layer 2;
- Via: All vias.

Grouping layers

You select or deselect the layers to be displayed in a particular group by placing the highlight on its position in the group and toggling the checkmark on or off by pressing **[F9]**.

Note: The display of the layers will also effect which layers will be printed from the ▸Printer checkplot option. However, when you produce output with the ▸Output Driver (Chapter 6), you can individually select layers for output independent of the display in the Graphics Editor.

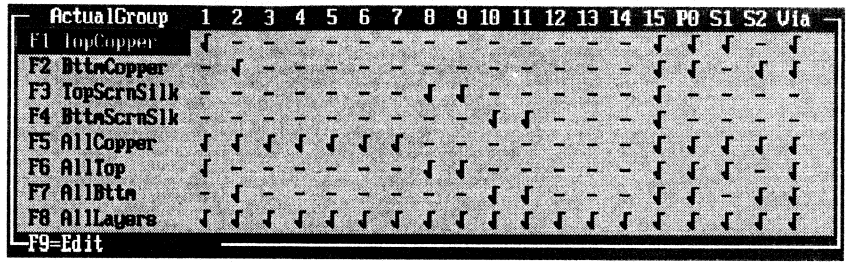


Fig. 3-21 Layer display configuration menu

Layer change

(options, layo1 toolbox, change layer)

Shortcut: None.

Function: Move all objects from a layer to another.

Description: When this function is selected, a layers table is shown, see below. The left column lists the source layer, from which the objects will be moved. Layer 0 (pads) cannot be moved. The right column is the target layer, where the objects of the source layer will be moved to. Move the cursor to the source layer and click ■□□. Next, move the cursor to the target layer and again click ■□□. The selected layers will blink. Move the cursor to **'Start'** and again click ■□□. This will execute the change.

Warning! Combining objects from different layers on one layer can not be undone! For instance, if you move the traces from layer 1 to layer 2, you have turned a double sided design into a single sided one, for ever...

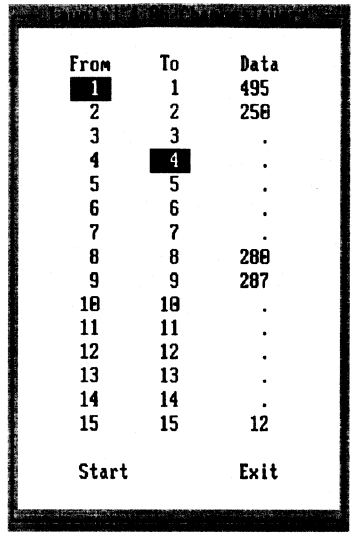


Fig. 3-22 Layer change table

Layer select (secondary)

Activated by: Press **[l]**.

Function: To select the layer on which the current operation (draw, place etc.) will take place.

Description: When activated, a menu line is presented for selecting a layer. Note that some operations are default done on predefined layers, see ▸Draw line.
There are two shortcut keys to let you select the most used layers. The default setting is that **[F7]** selects layer 1 and **[F8]** selects layer 2. Pressing **[^F7]** or **[^F8]** lets you enter a layernumber, that can be selected by pressing **[F7]** and **[F8]**.

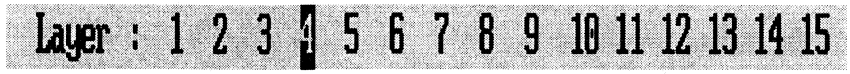


Fig. 3-23 Layer select menu

Load component

(secondary)

Shortcut: Press **[F3]**.

Function: To load a component only.

Description: When this function is activated, another selection window will open where you can either enter the component name, or use the ▸File/directory select options.

.PLY If you are working with a netlist based design (a .PLY file) and you load an .LMC, Layo1 will check if the component contains pads. If not, it will be considered as a block only. If yes, then Layo1 will request you to enter a component reference and type, and append this component to the component list. (You must use ▸Save file to make the addition permanent).

.LMC If you are working in a design not based on a netlist, and you load a .LMC, Layo1 checks if the component contains more than one block. If yes, you are asked if the separate blocks must be kept separate. If you decline, all objects in the component will get the same blocknumber and can the be manipulated as one entity.

Load file (file, load)

Shortcut: **[F3]** can be pressed to *load a component (.LMC) only.*

Function: To load a design, subdraw, component or other data file.

Description: When this function is activated, another selection window will open as shown below. You are now able to highlight and load several entities as shown. After making your selection, you will have to enter a filename, eventually with a full pathname. ▸File/directory select is available for this, if you do not want to type in the name/path.

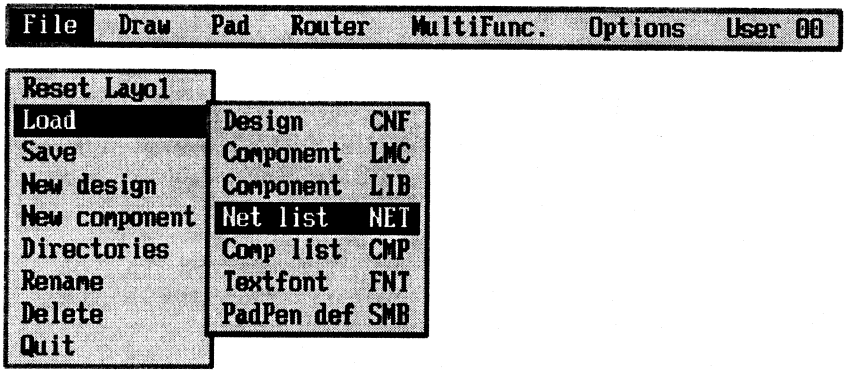


Fig. 3-24 File, load submenu

Macro create (options, create macro)

Shortcut: None.

Function: Store a sequence of keypresses under an **[Alt][key]** combination.

Description: When activated, Layo1 will request the **[Alt][key]** combination to attach the macro to. Next, all key presses you do will be recorded. Note that the mouse actions are disabled during a macro recording. All positioning must be done from the keyboard. To insure that the macro has always the same effect, it must be executed from the same position in the program as when it was defined. To insure this, start the macro with several **[Esc]** presses to go to the Top Level, and then start the intended action. Every time you open a menu, press **[Home]** to be sure you are at the menu top and them go to your selection. When you finish, stop macro recording by pressing **[Alt][F10]**. The macro can now be recalled by pressing the defined **[Alt][key]** combination.

Note: You cannot call a macro from within another macro (no macro nesting).

Main menu (top level)

Shortcut: Click □□■ from Top Level.
Press **[↵]** or **[Esc]** from Top Level.

Function: Selection of function menus.

Description: The Main Menu is activated from the program's Top Level, ie when no functions or commands are active. You are at the Top Level if the Layo1 logo is displayed in the upper left corner of the screen. You can return to Top Level from anywhere in the program by pressing **[Esc]** until you see the Layo1 logo. Activating the main menu opens the main menu line. Subsequent function selection can be done through moving the cursor to the required selection using the cursor keys or the mouse. A selection is activated by clicking □□■ or pressing **[↵]**. The available menus are shown below.

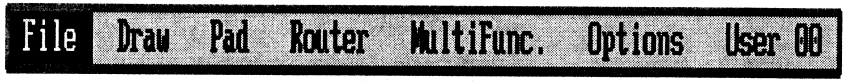


Fig. 3-25 Main menu selections

Manipulate block

(multifunc, block)

Shortcut: Press **[b]**.

Function: Manipulate (delete, copy, move, rotate, save as .LMC) a single block.

Description: You must first select the object (component, subdraw or text-block) to manipulate by putting the cursor on it and clicking ■□□. The selected block will be highlighted. Now use the mouse buttons to perform the operation, or move the mouse to move the object. If you use the shortcut, you cannot use 'Save as .LMC'. If you wish to copy the block, you must click ■□□ once more. See also ▸Copy... , ▸Copy repeat.

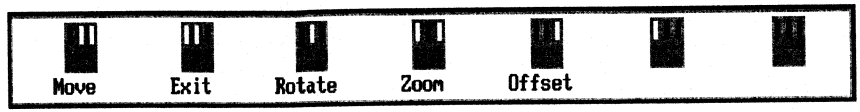


Fig. 3-26 Block shortcut mouse actions

Manipulate window

(multifunc, window)

Shortcut: None.

Function: Manipulate (delete, copy, rotate, move, save as .LMC) a group of objects in an area.

Description: Before you specify the object(s) to manipulate, you must specify the type of manipulation from the submenu that opens when you select **window**. The window is defined by confirming two opposite corners. The selected objects will be highlighted. You can now perform the manipulation with the mouse. You can also use ▸Copy repeat. See also ▸Copy object.

Note: Use this function if you want to manipulate an area containing electrical significant objects like traces and pad. The function ▸Window define is easier to use for general drawing and manipulating text and non-track line objects.

Manipulate line

(multifunc, line)

Shortcut: None.

Function: Manipulate (delete, copy, rotate, move, save as .LMC) a line segment.

Description: Before you specify the line to manipulate, you must specify the type of manipulation from the submenu that opens when you select **line**. The line is selected by confirming it. The selected line will be highlighted. You can now perform the manipulation with the mouse, or use **[F2]** to rotate. See also use ▸Copy repeat. See also ▸Copy line.

Manipulate group

(multifunc, {P1..P2})

Shortcut: None.

Function: Manipulate (delete, copy, rotate, move, save as .LMC) a group of objects.

Description: Before you specify the object(s) to manipulate, you must specify the type of manipulation from the submenu that opens when you select **{P1..P2}**. P1 and P2 specify the first and last datalines or vectors (▸.LMC, Chapter 2) of the group you wish to manipulate. You can step forward or backward through your design with **[+]** or **[-]** until you reach the first dataline (pad, track etc.) of the group. Mark it with **[F7]**. Use the same method to step to the last dataline of the group. Mark this with **[F8]**. In the top left corner you will see which datalines are selected. They are also highlighted on the layout. You can now perform the manipulation with the mouse as shown below. ▸Copy... , ▸Rotate object, ▸Delete object.

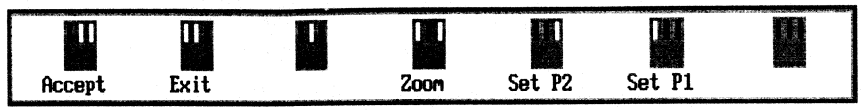


Fig. 3-27 Multifunc, {P1..P2} mouse functions

Mirror

(options, layo1 toolbox, mirror)

Shortcut: None.

Function: Mirror a complete design.

Description: When this is selected, the current design will be mirrored around a horizontal line, through the cursor position. Note that in the interest of preserving design integrity, you cannot mirror a netlist based design. You can mirror components, general drawings and subdrawings. (If you absolutely have to mirror a netlist based design, save it as an .LML, see ▸Edit Design CNF, rename it to an .LMC and load this .LMC).

Mouse sensit.

(user xx, mouse sensit.)

Shortcut: None.

Function: Set mouse sensitivity.

Description: Selecting this function opens a window for selection of mouse sensitivity in text mode, or in graphics mode. Each of the sensitivities can be set between 1 and 9. Setting 1 gives the lowest sensitivity, and setting 9 the highest. The higher the sensitivity, the quicker the display will react to mouse

movements. Sensitivity is default set to 5 for both graphics and text modes as shown below.

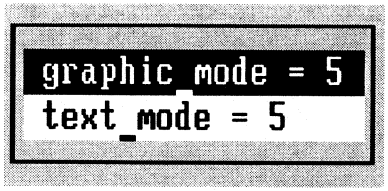


Fig. 3-28 Mouse sensitivity settings

MultiFunc menu

(main, multifunc)

Shortcut: None.

Function: Allow manipulation of objects and parts of a design.

Description: This menu allows you to select a part of the design, either a ▸Block, a set of datalines called a group (▸Manipulate group), a line (▸Manipulate line) or an area inside a window (▸Manipulate window). The type of manipulation is selected from a submenu. The following manipulations are available (moving is inherent to moving the mouse):

▸Copy... ;	▸Rotate;
▸Delete... ;	▸Save as .LMC.

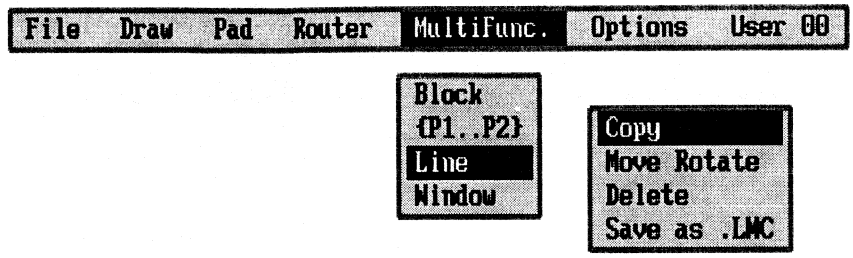


Fig. 3-29 MultiFunc functions

New design

(file, new design)

Shortcut: None.

Function: Load a new design by filename.

Description: Once the function is activated, you will be asked to enter a filename, eventually with a pathname. You may enter the name or use the ▸File/directory select option.

New component

(file, new component)

Shortcut: None.

Function: Create or modify a component or subdraw.

Description: When this function is activated, Layo1 will automatically set itself up to generate or modify a component or subdraw. The following actions are taken:

- Set up: Layo1 will reset itself and request a filename for the new/modified component. You may use ▸File/directory select. If the name exists, the component or subdrawing will be loaded. *The name specified will be kept as the name for the object you are creating or editing!*

- Built the component: you will return to the graphics editor screen with the drawing grid set to 1/20 inch, and the imperial ruler activated. You can now built the object by drawing, inserting other components or subdraws etc.

- Save the component: you can save your work with **[F4]**. If you want to quit the program, Layo1 will ask you if you wish to save the work. It will use the name specified at step 1 above, unless you type in another name. If the name does already exists, Layo1 asks if you want to overwrite it. Be careful, if you start this function by loading an existing component that you want to change or extend, be sure to save the work under another name!

Offset (options, layo1 toolbox, xy offset)

Shortcut: Pres **[x]** or **[y]**.

Function: The design is shifted in X and Y.

Description: When this is selected, the top left of the screen will display **'XY offset'**. When you click ■□□, the design will be shifted so that the new origin (0,0) will be at the current cursor position. *Note that this will also modify the ▸Grid dots alignment with respect to the drawing.* The design can also be shifted in X and Y by entering an XY offset in the ▸Edit Design CNF function or by the shortcuts as indicated above.

Optimize traces

(options, optimize)

Shortcut: None.

Function: Optimize corners of traces.

Description: This function gives access to a submenu with three choices as shown below. These will each have a slightly different effect.

Sq45 opt 1/40 — All trace corners are mitered to 45 degrees, with all corners placed at a 1/40 inch grid.

Sq45 opt 1/80 — All trace corners are mitered to 45 degrees, with all corners placed at a 1/80 inch grid.

Optimize45 — All trace are optimized to a routing of straight and 45 degree sections.

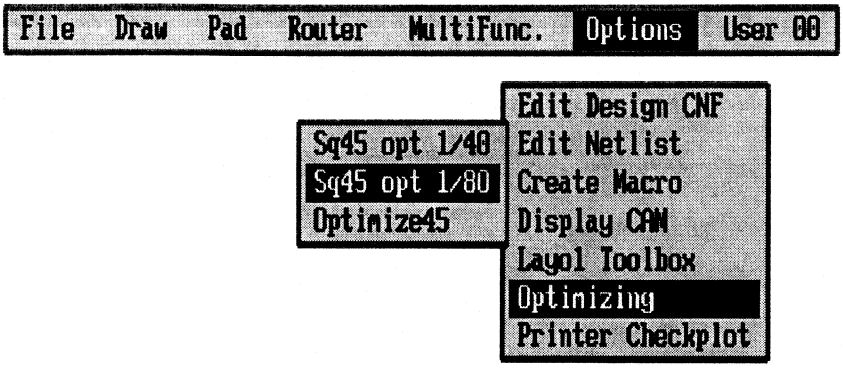


Fig. 3-30 Optimize options menu

Options menu (main, options)

Shortcut: None.

Function: Allow selection from a collection of miscellaneous functions.

Description: The Options menu is shown below. Each of the entry is listed separately.
The options are discussed in the following entries:

- ▸Edit Design CNF;
- ▸Edit net;
- ▸Macro create;
- ▸Display CAM;
- ▸Toolbox menu;
- ▸Optimize traces;
- ▸Printer checkplot.

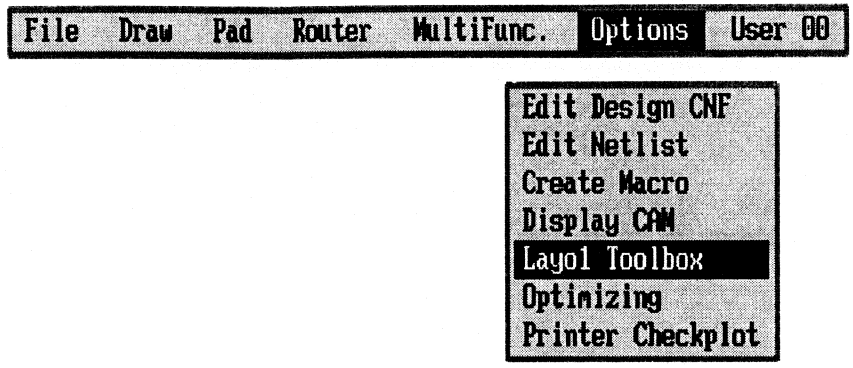


Fig. 3-31 Options menu

Origin (secondary)

Activated by: Press **[F6]**.

Function: Set a temporary origin.

Description: This function will set a temporary origin at the cursor position. The ruler(s) at the top of the display are set to 0,0. This is usefull if precise placement has to be done relative to some point or component of the design. This function is also related to ▸Ruler origin.

Pad menu (main, pad)

Shortcut: None.

Function: Access to pad placing and editing functions, and defining custom pads and drill hole diameters (spots).

Description: This menu gives access to the choices as shown below. All functions are described under the following entries:

- ▸Pad place;
- ▸Copy... ;
- ▸Tool select;
- ▸Pad replace;
- ▸Delete object;
- ▸Pad define.

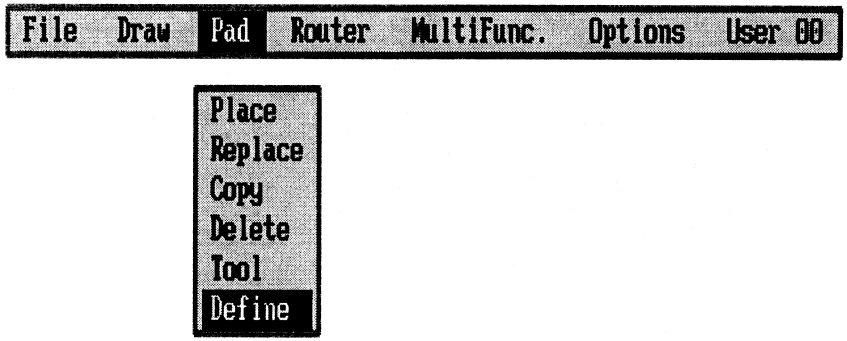


Fig. 3-32 Pad menu

Pad define (pad, define)

Shortcut: None.

Function: Customize pad shape and/or size.

Description: This function lets you select a pad from the current set, and/or customize the shape and/or size. When the function is selected, a submenu is opened as shown below.

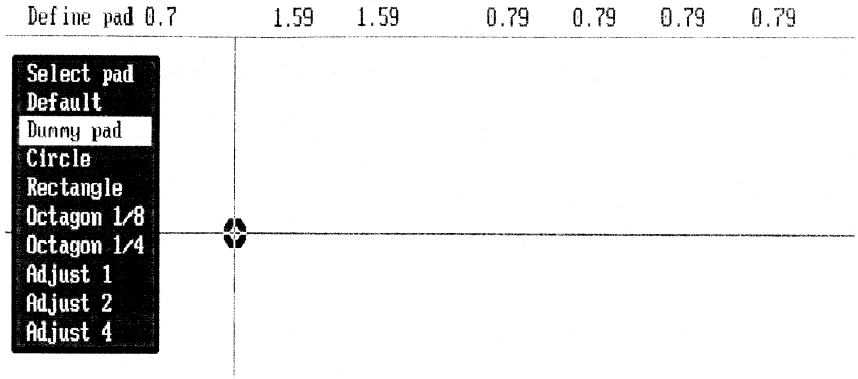


Fig. 3-33 Pad define submenu

The current pad is shown above the menu as **'Pad a.b'**, where **'a'** is the set-number and **'b'** is the pad number within the set. There are 8 sets with each 16 pads. Set 0 is a set of regular pads with a tool included. The other sets are pads to be used for surface mount devices (SMD) and general use. These can be placed at any layer, and do not have an associated tool. In other words, they are just copper areas, while the first set actually are holes through all layers, with a copper area around it on a specific layer.

Select pad

With the choice **'Select pad'** you can step through the pads and sets by moving the mouse. Vertical movement changes the set-number up and down, while horizontal movement changes the pad number within the set. The current selection is shown at the cursor position. You may want to zoom-in before activating this function, so that the pad shapes are easier to distinguish. The selection is confirmed by clicking ■□□.

Select shape — Once you have selected the required pad, you may move the highlight to a shape choice (default, dummy, circle, rectangle, octagon 1/8, octagon 1/4) and click ■□□ for selection. Again, the shape of the pad at the cursor position follows your selections.

default — This will reset the pad shape and size to the default shipping values.

dummy — This is a special case of a pad that has no display and does not appear in the design. Suppose you must design a board from a schematic design program output. The components in Layo1 must have pads numbered corresponding to the numbering in the schematic design software's component. In a Layo1 component, all pads are numbered in the order placed. This is true for the delivered libraries as well as for any custom components. This poses a problem if the numbering in the schematic design component is not consecutive, i.e. if some numbers are missing. Placing a dummy pad for the missing numbers makes Layo1 number the 'real' pads correctly.

circle — Changes the pad shape to a circle.

rectangle — Changes the pad shape to a rectangle.

octagon 1/8, 1/4 — Changes to a rectangle with mitered corners. The miter has a dimension of 1/8 or 1/4 of the side of the rectangle.

Set sizes — After selecting the required shape, you can modify the pad's sizes. When any of the choices Adj. 1, 2 or 4 are selected, the display will show a pad outline with a small circle cursor. This cursor can be moved to any of the sides, to select the side you want to resize. If you click ■□□, the circle cursor changes to a small square.

Moving the mouse will now change the size of that pad dimension. Clicking ■□□ will fix the size. The actual size will be shown at the top line of the display. The figure below shows a typical display when resizing a pad.

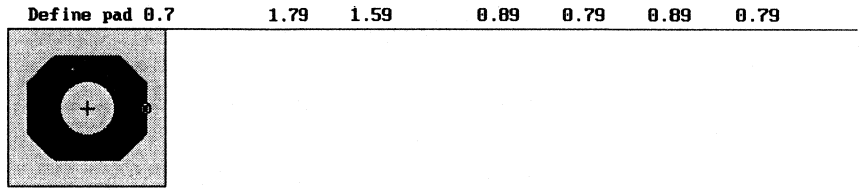


Fig. 3-34 Typical pad resize display

There are three ways to change the shape size (and shape) as described below. Note that for circular pads, the three methods produce identical results.

Adj. 1 With this choice, you can change only one side of the pad. This will produce pads with eccentric holes.

Adj. 2 With this choice, you change two opposite sides of the pad simultaneously. This will let you change the height or the width of the pad, depending whether you place the circle cursor on the top or bottom, or on one of the sides.

Adj. 4 This selection will change all 4 sides of the pad simultaneously, independent of the side where you place the circle cursor.

Pad place (pad, place)

Shortcut: Press **[s]** or click ■■□.

Function: Place a pad at the cursor position.

Description: The currently active pad selection (last selection done) will be attached to the cursor and can now be placed at the required position. The attach point is the point relative to which the pad was defined, see ▸Pad define. For non-SMD pads this will be the drill hole center. The pad is placed by clicking ■□□. There will still be a pad at the cursor, which will be visible when you move the cursor away. This can now be placed at another position, and this can be repeated. See also ▸Copy repeat. The function is ended with **[Esc]** or clicking □□■. You can also select another pad or drill diameter while a pad is attached to the cursor by pressing **[s]**.

While placing pads, including SMDs, you can use **[F7]** and **[F8]** to select layer 1 (component side) or layer 2 (solder side). If you change layers, SMD pads are mirrored, so that a correct placing is obtained even with non-symmetrical SMD-pads. This is separate from the ▸Layer change feature, and is independent of the layers defined with **[^F7]** and **[^F8]** (▸Layer select).

Fig. 3-35 Pad place mouse functions

Pad replace (pad, replace)

Shortcut: None.

Function: Replace a pad with another size or shape pad.

Description: This function works much the same as the ▸Pad place function. When you have a pad attached to the cursor, select the required size and shape with ▸Pad select, place it over the pad to be replaced and click ■□□.

Pad select (secondary)

Activated by: Press **[s]**.

Function: Select a pad shape/size for placing or editing.

Description: You can use this function inside ▸Pad place and ▸Pad replace. When you have a pad attached to the cursor, press **[s]**. The current pad selection (set and number) are displayed in the status line. If you now move the mouse, you will step through the pad palette. The selected pad is shown at the cursor position, and its number at the status line. Moving the mouse horizontal will select another pad from the current set, moving vertical will select another set.

Pen (secondary)

Activated by: Press **[p]**.

Function: To select the pen used for drawing objects.

Description: When activated, a list is presented from which 7 different pens can be selected. The width for each pen is set by the ▸Penwidth function.

Note that when you are drawing text Layo1 will always use a minimum line width. Furthermore, on layers 8 and above, all pens will appear to have the minimum width. However, when output is generated or if the object is moved to layer 7 or below, the penwidth will be correctly used.

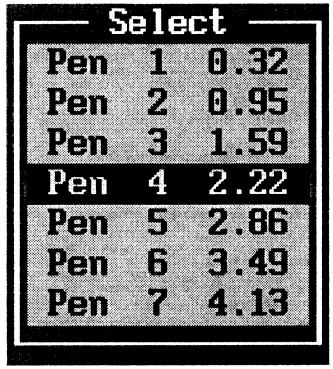


Fig. 3-36 Pen selection menu

Penwidth (draw, define)

Shortcut: None.

Function: To allow defining the widths of the pens selected for drawing.

Description: When this function is selected, a window will open listing the currently defined widths for the available pens. You may select a pen with the highlight. Another windows opens where you can enter the new width for the selected pen. Note that as long you do not save the new setting(s), they will only be valid for the current project (they are saved in the .CNF).

You can save the settings in a user-related ▸.SMB (Chapter 2) by selecting 'Save' from the ▸File menu and select to save PadPen definitions. All objects that were drawn with a particular pen will be updated to the new width for that particular pen.

Pin numbers

(secondary)

Activated by: Press **[n]**.

Function: Toggle pinnumber display.

Description: This function toggles the display of all pinnumbers of the pads that are part of a component or subdrawing. For the sequence of pad numbers see also ▸Pad define, dummy pad. Note that the pinnumbers have a fixed size. You may have to zoom the design to see them clearly.

Printer checkplot

(options, printer checkplot)

Shortcut: None.

Function: Generate hardcopy output from the Graphics Editor.

Description: This function will let you generate prototype-quality output from within Layo1. You do not have to use the Output Driver. Note that only selected layers, that are currently visible, will be printed. However, for best results production output should be made from the Output Driver, preferably on a photoplotter. See also Chapter 6. When this function is selected, a submenu will be opened as shown below.

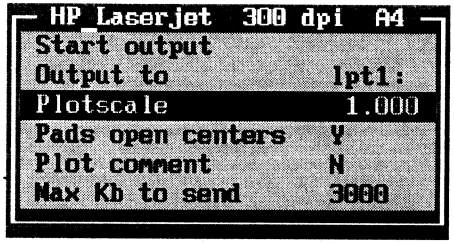


Fig. 3-37 Printer checkplot submenu

Device type — The selected device is shown in the heading of the submenu. This is set in the ▸User xx menu (▸Configure printer).

Start output — Begin sending data to the printer. The amount of data to be send and the progress will be shown in the top-left corner of the display.

Output to: — This selection opens a window where you can select the destination of the checkplot. The first choice is the port configured in the ▸User xx menu with ▸Configure printer. You can also select a file, for which Layo1 will request a name. Instead of a filename you can also enter an LPT or COM port. However, if you specify a COM port, you must have configured this port from DOS or from the ▸User xx menu (▸Configure printer).

Plotscale — You can select this option to change the plotscale from the default 1.000. Scaling can be set between 0.1 and 5.

Pads open center — Your choices or **YES** or **NO**. If you select Yes, all pads are printed with open centers, according to the tool diameter for that pad (▸Tool select). Selection of No will produce full black pads.

Plot comment — If you select **YES** for this option Layo1 will print the date, time and design file name on the top of the output.

Repeat — You can set Repeat to one, which will cause the print head to pass over the print area an extra time. This will give a more fully black print, but is only necessary with old ribbons or order printers. This will have no effect with Laser Printers. Default is 0 (no repeat).

Quit (file, quit)

Shortcut: None.

Function: Leave the Layo1 Graphics Editor.

Description: When this function is activated, Layo1 checks if you are working on a design that was modified since the last save. If not, the program terminates and you are returned to Project Manager (or to DOS if Layo1 was started from the command line). Else, the program asks if you wish to save the current design and will do so if requested.

Warning: If you have customized penwidth or pads, or if you have modified the netlist or component list, these must be save prior to quitting. The **.SMB**, **.CMP** and **.NET** files are not automatically saved!

Ratsnest (secondary)

Activated by: **[^F1,2,3,4,6]** and **[F5]**.

Function: Maintain and show ratsnest.

Description: Layo1 contains several functions to maintain and display the ratsnest while you are working on a design. The ratsnest can only be maintained if there is a netlist for the design. This may be a netlist read in from a schematic capture program, a netlist created by the user with the ▸Edit net function or even a netlist created with a text editor (see ▸.NET, Chapter 2). The following functions are available:

Reset ratsnest — Press **[^F1]** or **[F5]** (▸Redraw screen). The ratsnest (which is drawn on layer 14) will be deleted.

Ratsnest trace — Press **[^F2]**. The ratsnest for the pad at the cursor position is displayed.

Ratsnest component — Press **[^F3]**. The ratsnest for the component attached to the cursor is displayed. See ▸Block on how to attach a component to the cursor.

Ratsnest design — Press **[^F4]**. The ratsnest for the complete design is displayed, *except the connections already routed.*

Ratsnest check — Press **[^F6]**. The routed connections are checked against the ratsnest, and any errors are labeled with a X and displayed on layer 13 at the pad in error. ▸Check connections.

Redraw screen

(secondary)

Activated by: Press **[F5]**.

Function: Clear and redraw the screen.

Description: This function is used to redraw a clean screen after delete, move, rotate or other editing operation. These operations often leave some remains of the manipulated objects on the previous position. Note that redraw also clears the ▸Ratsnest display and any ▸Diagonal lines, if present.

Reference PCB

(secondary)

Activated by: Press **[:]**.

Function: Toggle the reference PCB display.

Description: When toggled on, a reference PCB is displayed in the upper left corner of the graphics screen. This reference PCB is a reduced copy of the design. It will show the position of the pads, and the cursor position. The latter is displayed with a small green '+'. When performing an operation that highlights part of the design (editing or routing for example) the highlight is also visible in the reference PCB.

The advantages of this are twofold: firstly, you have a good idea of the relative density of the design over the board area. Secondly, you can more easily appreciate at which area of the board you are currently working. This saves you repeated zoom-in, especially with large designs. Each time you press **[:]** the display of the reference PCB is toggled. A typical display is shown below.

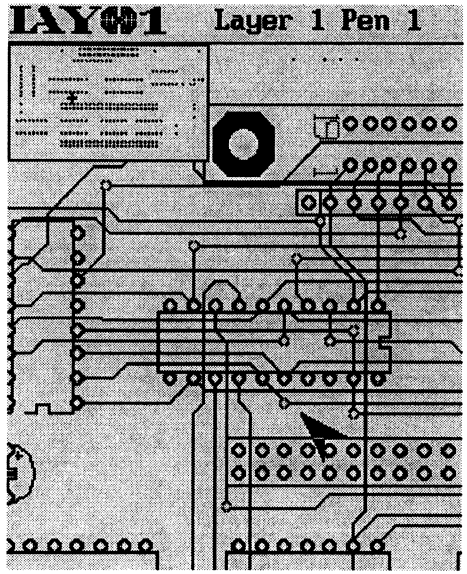


Fig. 3-38 Reference PCB

Rename (file, rename)

Shortcut: None.

Function: Rename a file on a disk.

Description: When the function is activated, you are asked to enter a filename/pathname. If you just press [↵] or confirm, you will enter the ▸Directories function so you can navigate the disk to select the file to rename. In that case, the file will be selected by placing the highlight on it and confirming or pressing [↵]. If the file can be located, you are asked for the new name. If you enter the filename directly, you can use the usual editing keys. If the file cannot be located, Layo1 will return with the name you entered. You can edit the name or **[Esc]** from this function.

Reroute all

(router, reroute all)

Shortcut: None.

Function: Reroute all connections in the design.

Description: When this function is selected, all traces that have been previously routed are erased. The router will start and attempt to route the design to completion. This will typically be used if you are not happy with the routing so far, and after making changes to the ▸Router setup menu.

Reroute net

(router, reroute net)

Shortcut: None.

Function: Reroute all connections in a net.

Description: To use this function, you must first select a net (see ▸Edit net). All traces in the selected net that have been previously routed are erased. The router will start and attempt to route the net to completion. This will typically be used if you are not happy with the routing so far, and after making changes to the ▸Router setup menu.

Reroute track

(router, reroute track)

Shortcut: None.

Function: Reroute a track.

Description: To use this function, you must first select a trace by pressing [=] or by clicking ■■■ (see ▸Edit). If a trace is already present, it will be erased. The router will start and attempt to route the trace. This will typically be used if you are not happy with the routing so far, and after making changes to the ▸Router setup menu.

Reset program

(file, reset layo1)

Shortcut: None.

Function: Start a new design without leaving the Graphics Editor.

Description: The Reset function lets you start a new design session without leaving the Graphics Editor. If a design is active, and you have made any changes to it since the last automatic ▸Save design or manual save, Layo1 will ask if you want to save the current design and do so if requested. The screen will be cleared. You may now start a new design by loading an existing design, subdraw or component. See also ▸New design and ▸New component.

Rotate object

(multifunc, window; line; group; block)

Shortcut: None.

Function: Rotate an object or group of objects.

Description: This function can be used to rotate an object or group of objects in 90-degree steps. The object(s) must first be attached to the cursor with one of the ▸Manipulate... or the ▸Block function. The object(s) is rotated around the point where the cursor is positioned. In most of the situations where an object is attached to the cursor, you can rotate it with a mouse command or with **[F2]**. You can verify that by pressing **[F1]**.

Rotate design

(options, Layo1 toolbox, rotate)

Shortcut: None.

Function: Rotate a complete design.

Description: The full design is rotated in 90-degree steps, counter clockwise. The design is rotated around the point where the cursor is positioned. Note that this is not possible with a net-list controlled .PLY design. You can only rotate designs consisting of a .LMC or a subdrawing. See also ▸File system in Chapter 2.

Route window

(router, route window)

Shortcut: None.

Function: To limit router action to a defined area of the design.

Description: Define a window on the design. You can use a shortcut (press **[w]**), or the mouse (▸Window define). After that, you can start the router with any of the route- or reroute selections. No trace will be laid outside of the window (you must place the cursor inside the window when starting the router).

Route all

(router, route all)

Shortcut: None.

Function: Route complete design.

Description: The router will start and attempt to route the design to completion, within the given constraints. These constraints are set in the ▸Router setup menu, and possibly by a boundary window (see ▸Route window).

Route net

(router, route net)

Shortcut: None.

Function: Route a net.

Description: The router will start and attempt to route the selected net to completion, within the given constraints. These constraints are set in the ▸Router setup menu, and possibly by a boundary window (see ▸Route window). See ▸Edit net on how to select a net.

Route pad

(router, route pad)

Shortcut: None.

Function: Route a net from a pad.

Description: This function is usefull to route a 'star' net configuration. Place the cursor over the pad that is to be the center of the star and click ■□□ or press [↵]. All pads in the net are highlighted and one trace is drawn to another pad. Repeat this until all traces have been drawn.

Route manual

(router, route manual)

Shortcut: None.

Function: Manually use the router trace by trace.

Description: Place the cursor on a pad you want to route, and click ■□□ or press [↵]. The indicated pad and all other pads that are part of the same net are highlighted. Move the cursor to the pad you want to route to and click ■□□ or press **[Ins]**. The router will lay a trace for you. If not, the reason may be:

- The trace is not possible because of the design geometry and the presence of other traces;
- The trace is limited by router boundaries in space (see ▸Route window or ▸Router setup, via router);
- The trace length would exceed the length limits set in ▸Router setup, max xy length, max len error or max len test.

You can continue routing in the same net by placing the cursor on another pad and again clicking ■□□ or pressing [↵]. If you want to go to another net, press **[Esc]** or click □□■ to leave the current net, and start with the new net.

Router setup

(router, router setup)

Shortcut: None.

Function: Set various parameters for the router.

Description: Router setup will display a submenu as shown below. It should be noted that all settings will be kept when the program is terminated. When you restart it, the settings will be as you last defined them.

Note: All choices in the router setup menu are changed by pressing [+] or [-].

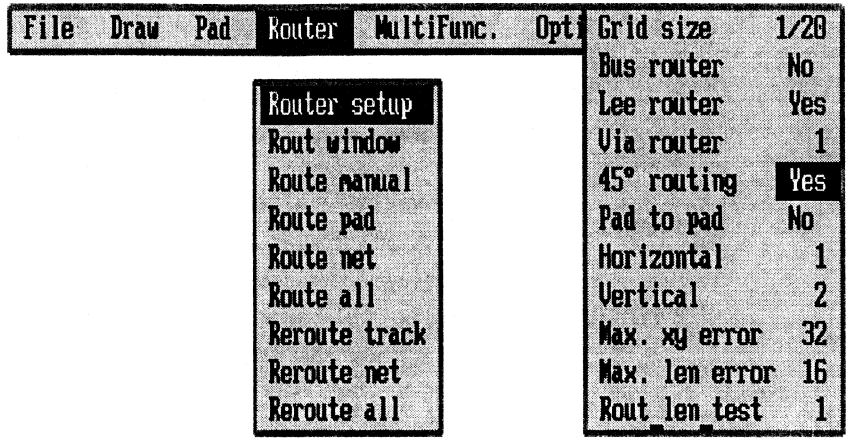


Fig. 3-39 Router setup submenu

Grid size This parameter defines the grid that the router will use to place connections, vias and turns. You can choose a 1/20 or a 1/40 inch grid. This grid will be aligned with the 1/10 inch ▸Grid dots. If you plan to use the autorouter, it is important that you place all components on the grid from the beginning (or use ▸Snap design before routing). Otherwise the router will terminate the traces away from pad centers and generally will not route optimally. With the 1/40 inch grid, an area of about 20 square inch can be route at the time. The 1/20 inch grid setting covers about 80 square inch. Larger areas will have to be routed in parts, using ▸Route window. Using a 1/20 inch grid is fastest, while a 1/40 inch grid does not normally improve the result significantly.

Bus router — The bus router uses an algorithm that does not generate large loops in the tracks. It is best in making short bus-type connections. The choices are **'Yes'** and **'No'** to enable/disable the Bus-algorithm.

Lee router — The Lee router uses an algorithm that restricts it much less than the Bus router, and leads to a much higher completion ratio. However, it can generate very long trace loops, which may be undesirable. The maximum length ratio can be limited with the settings in the ▸Router setup menu. As with the Bus router, your choices are **'Yes'** and **'No'**.

Via router — You can set this parameter to 0, 1 or 2. This number determines the number of via's the router is allowed to insert in trying to complete a route. A via is a small pad used to connect a trace on one layer with a continuing trace on another layer. This parameter only makes sense if routing on more than one layer is enabled. This is the case if different layers are specified for horizontal and vertical routing.

45° routing — If you set this parameter to **'No'**, you force the Lee router (when enabled) to route the traces with 90° corners only. Otherwise, 45° corners will be used.

Pad to pad — Setting this to **'Yes'** will force the router to route the traces from pad to pad, as specified in the netlist (Chapter 2, ▸.Net). No shortcuts will be made to other parts of the same net. If set to **'No'**, the router will use existing traces of a net to connect the other parts.

Horizontal — This parameter specifies the layer on which the router will try to route tracks horizontal. This parameter is connected with the setting for Via.

Vertical — This parameter specifies the layer on which the router will try to route tracks vertical. This parameter is connected with the setting for Via.

Max. xy error — This parameter specifies what extra length is allowed for the x- and y-lengths of a trace, in relation to a straight line connection as showed by the ▸Ratsnest. The extra length (the **'error'**) is specified in grid steps (1/20 or 1/40 as set). This parameter can be set between 0 and 256. It can be used to limit the router to go to great length (pun intended) to complete a route.

Max. Len. error — This parameter works similar to the ▸Max xy error, except that the trace length itself is considered, instead of separate x- and y- lengths.

Route len test — This parameter specifies one of three routing strategies. This entry can be set to 0, 1 or 2. The strategies are as follows:

- 0: route according to ratsnest, sorted on absolute distance;
- 1: start as with strategy 0 above, and if any routes cannot be completed, try to route to other traces;
- 2: route in the order of the netlist, which gives the user a method (through editing the net) to determine the routing order.

Router menu

(main, router)

Shortcut: None.

Function: Select router actions and setup menu.

Description: A menu will be opened as shown below. The ▸Router setup choice will open another submenu, to set the various limits and parameters for the router.

The following selections will route parts of the design or the whole design:

- ▸Route window;
- ▸Route all;
- ▸Route net;
- ▸Route pad.

The following selections will reroute existing tracks after clearing them first:

- ▸Reroute all;
- ▸Reroute net;
- ▸Reroute track.

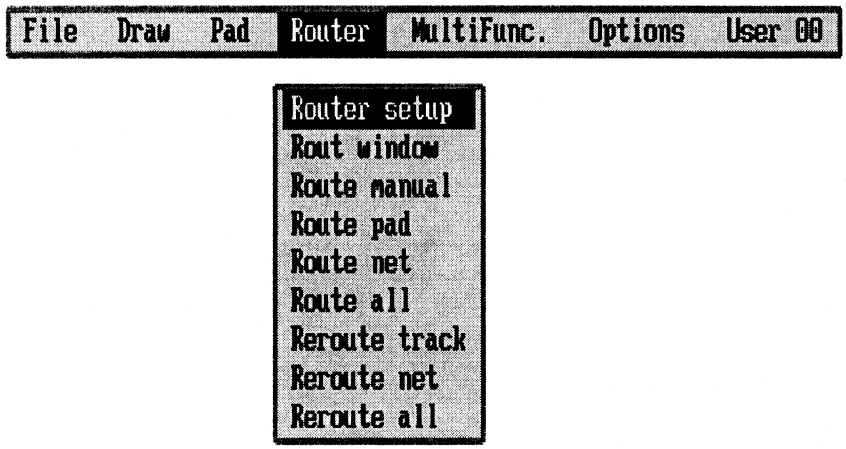


Fig. 3-40 Router menu

Rubberband (secondary)

Activated by: Press **[^F1]**.

Function: Show ratsnest during move.

Description: A window will open asking you if real-time rubberbanding should be set on or off. If on, the ratsnest will be continually updated during a move operation. This feature is very helpful if you try to find an optimum location for a component. It does however slow down screen update, especially if you move large and/or complex objects.

Ruler origin (secondary)

Activated by: Press **[r]**.

Function: Set the ruler origin.

Description: There are three origins which the ruler can use to indicate distances from. Origin 1 (selected with **[r1[1]**) is the top left corner line of the design, e.i. the physical PCB corner. Origin 2 is the top left corner of the display, while origin 3 is the temporary ▸Origin set before with **[F6]**.

Save user configuration

(user xx, save/exit)

Shortcut: None.

Function: Save all user-defined settings and return to the graphics editor.

Description: When selected, a window is opened asking if you wish to save the current settings and exit. If you answer no, the previous settings will be restored. If you save the settings, they will become effective every time the current user number is selected.

Save design

(secondary)

Shortcut: Press **[F4]**.

Function: To save a design only.

Description: Activating this function will save the design and its related file: the .PLY and the .CNF. Layo1 will ask you to confirm the save operation. If you have specified a .LML in the ▸Edit Design CNF menu, your project will also be saved in .LML form. See ▸File structure in Chapter 2 for more information on the difference of an .LML and a .PLY. This is the same function that will be called when you specify a timed backup through the ▸Auto settings menu. Note that custom pads, pens, the netlist and the component list are *not* saved. Use ▸Save file for that.

Save file

(file, save)

Shortcut: Press **[F4]** to save a design only.

Function: To save a design, subdraw, component or other data file.

Description: Once the function is activated, a further window will open. This will let you select the entity you wish to save. You will be asked to enter a filename, eventually with a pathname. ▸File/directory select is available if you do not want to enter the filename/path. The action may be slightly different depending on what you wish to save, see also ▸New component, ▸Edit net and ▸Quit.

Save as .LMC

(multifunc, line; window; group; block)

Shortcut: None.

Function: To save an object or group of objects as a component.

Description: This function will only work if an object has been previously selected with the block, group, window or line selection. Once the function is activated, a window will open and the program will request a new offset. This is usefull if you have selected the object(s) to be saved, and the cursor is at some arbitrary position. You van now place the cursor at a suitable point (for instance pin 1 of an ic). When you later load it, it will be attached to the cursor at the same point. Next you must enter a filename for the object to be saved. The usual ▸File/directory select options are available.

Scale design (options, layo1 toolbox, scale)

Shortcut: None.

Function: To scale a complete design.

Description: This function can be used to scale a complete design. It is not meant to generate a normal PCB layout, but can have several uses. It can be used when drawing equipment fronts, or to scale a design for documentation purposes. The scale can be set from .1 to 10. All dimensions will be scaled, *but pad dimensions and penwidths will not be changed!* You can also use this for components that were drawn to another scale than you design. Scale your design to the scale of the component, load the component and rescale the design.

Shape display (secondary)

Activated by: Press [,].

Function: Toggles the component shape display.

Description: This function alternately displays and hides the component shapes. It has no effect on the design itself, and does not change the number of vectors in the design.

Snap design (options, layo1 toolbox, set grid)

Shortcut: None.

Function: To align all objects in a design to a 1/40 inch grid.

Description: This function will snap all objects in a design to the nearest 1/40 inch grid point. This can be usefull if you want to use the router. In any case, this should be done as early as possible in the design, because it can distort relative positioning in the design. It can also distort already routed connections. If you select a suitable grid such as 1/20 inch, and ▸Snap cursor to it, before loading and placing components, this function will not be necessary.

Snap cursor

(secondary)

Activated by: Press **[*]** or **[**]**.

Function: Snap the cursor to a grid point.

Description: Pressing **[*]** snaps the cursor to the nearest point of the current drawing grid (▸Grid set). Pressing **[**]** however snaps the cursor to the nearest 1/10 inch gridpoint.

Snap component

(secondary)

Activated by: Press **[u]**.

Function: Select a component for manipulation.

Description: When activated, the program opens a window where the component reference (eg. **R12**) must be entered. The component will 'jump' to the cursor. It can now be manipulated as if it was hooked with ▸Block.

Step nodes (secondary)

Activated by: Press **[-]** or **[+]**.

Function: To select the next or the previous dataline (pad, trace, text segment etc) in the design.

Description: This function lets you select items for editing that are grouped together. For instance, if you are editing a trace or other object that consists or many connected lines, you can step from one segment to the other using this function. Pressing **[+]** will move the cursor to the next segment, **[-]** to the previous. Of course, this will only work if the related segments were draw in order, that is, they are entered in the design consecutively. See also the tutorial ▸Create a component in the User Guide.

Swap block (secondary)

Activated by: Press **[↑F2]** while a ▸Block is active.

Function: Move the objects in the block from one layer to another.

Description: This function can only be used in ▸Block mode, when an object is attached to the cursor. The object is moved to another layer according to the following rule:

layer 1	<--->	layer 2
layer 8	<--->	layer 10
layer 9	<--->	layer 11

This feature is especially usefull when working with SMD components. In that case, the SMD pads will be mirrored to insure proper placement of asymmetrical SMD pads.

Text draw (draw, text)

Shortcut: None.

Function: To place text objects in the design.

Description: This command will open a secondary menu, as shown below.

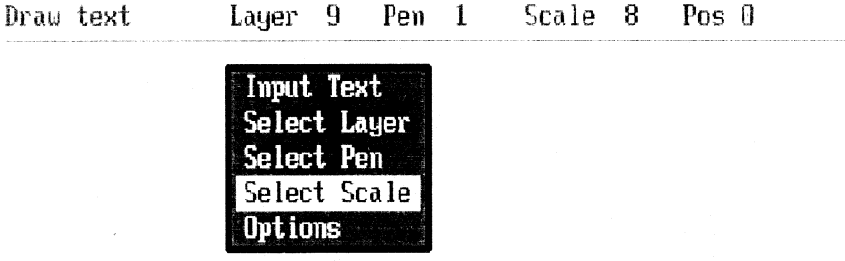


Fig. 3-41 Text object submenu

Text There are four selections to draw user text strings. Besides the text, you can also specify the layer, pen and scale of the text object. The ▸Text options are described elsewhere. If this is the first time during this session that the text function is called, Layo1 will request the font to be used. As delivered, a file **'FONT.FNT'** contains the default font.

A separate ▸Font editor, described in Chapter 7, allows the generation of custom fonts, symbols and logos. ▸File/directory select is available for selection of a font.

Input text

This subselection will open a window where you can enter plain text strings to be placed anywhere on the design. Pressing [↵] puts the text on the screen, attached to the cursor. At this point, several mouse functions are available to rotate, mirror and size the text as shown below.

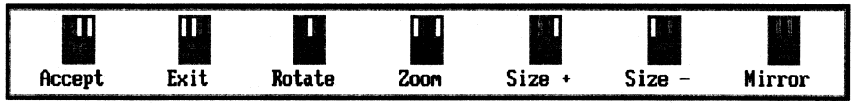


Fig. 3-42 Text draw mouse functions

However, you can also execute these functions from the keyboard as follows:

- Mirror: press **[↑F2]**.
- Rotate: press **[F2]**.
- Enlarge: press **[+]**.
- Shrink: press **[-]**

The text is placed by clicking ■□□ or pressing **[Ins]**.

Select layer

Shortcut (after entering text): press **[l]**. This selection lets you set the layer on which the text must be drawn. Default is layer 9. Note that on layers 8 and above, the text is drawn with the minimum penwidth. See also ▸Pen and ▸Penwidth.

Select pen

Shortcut (after entering text): press **[p]**. Text is default drawn with the default pen width for pen 1, at layer 9. Selecting another pen (and thus penwidth) will only have an effect if you draw the text on layer 7 or below. See also ▸Layer select.

Layo1 will place text default on layer 9. You can select another layer, but you should use layer 8 or above. You may also select a different pen from the default pen 1. On layer 8 or above, this will have no apparent effect. However, on layer 7 or below the text will de drawn in the actually selected pen width. Also, if you move the text from layer 8 or above to layer 7 or below, it will be drawn in the penwidth that was selected at the time the text was entered.

When you exit the text function and reenter it at a later date, all attributes (pen, layer, size, rotation and mirror) will be as selected previously.

Select scale

Selecting this function will let you change the size of the text characters. You can use this before actually entering the text , and the new size will be valid for all subsequent text draw actions. However, once the text is attached to the cursor, it can still be sized it with the mouse buttons or by pressing **[+]** or **[-]**.

Text options

(draw, text, options)

Shortcut: Press **[']** (limited).

Function: To place component attribute text in the design.

Description: This command will open a secondary menu, as shown below. It gives you the possibility to automatically place component type, reference or shape names in your design. With this options you can fully document your design electrically as well as mechanically. Each attribute is placed with a default pen 1 width, and on a default layer: if a component is placed on the component side (default), the attribute text is placed on layer 9. If the component is placed on the solder side (for instance a mirrored SMD device) the attribute text is placed on layer 11. Note that this function is also available as a secondary function ▸Attributes display, but without the scaling option. Note that the selected attributes will be placed at all component locations at the same time. They will not be attached to the cursor, and therefor cannot be manipulated prior to being placed.

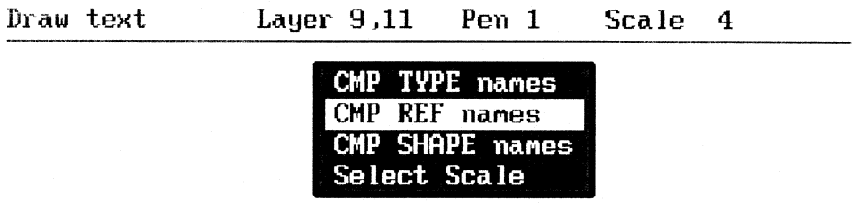


Fig. 3-43 Draw/ text/ options submenu

Type name — This selection places the component type, as contained in the component list, at the center of the component shape outline (e.g. **'4.7 k'**).

Reference name — This selection places the component reference, as contained in the component list, at the center of the component shape outline (e.g. **'R16'**).

Shape name — This selection places the component shape name, as contained in the component list, at the center of the component shape outline (e.g. **'R-05'**).

Select scale — This function can be used to set the size of the component attributes that will be placed. You can select the size from the keyboard or use **[+]** or **[-]** to decrement or increment the scale. Even the mouse buttons can be used.

Tips

- Text objects are made of line segments and thus use up vectors as any other object. Do not place more text in your design as absolutely necessary.

- To keep your design as clear as possible, place text attributes when you are done with the design, to produce the documentation.

- If you want to move attribute text, use the ▸Window define function with only the layer of interest displayed. If you would use the ▸MultiFunc menu, it would be difficult to move just the attribute and not to distort other parts of the design.

- If you want to place more than one attribute at the time, for instance both type and reference, they would overwrite each other. You can avoid this by first placing one attribute, and then using ▸Layer change, to move it to another layer. The next attribute will then be placed at its default layer. In this way you can selectively print or plot the layers for your documentation or silk screen layer.

Text display

(secondary)

Activated by: Press [;].

Function: Hides layer 9, the default text layer.

Description: All objects present at layer 9 are hidden from display. If text is present on other layers, it is not affected.

Toolbox menu

(options, layo1 toolbox)

Shortcut: None.

Function: Access a menu with several general purpose utilities.

Description: Selecting this function opens a submenu listing several utilities for manipulation an entire design, as shown below.
These are described by the following functions:

- ▸Mirror;
- ▸Rotate;
- ▸Offset;
- ▸Delete layer;
- ▸Layer change;
- ▸Drill group;
- ▸Scale design;
- ▸Snap design.

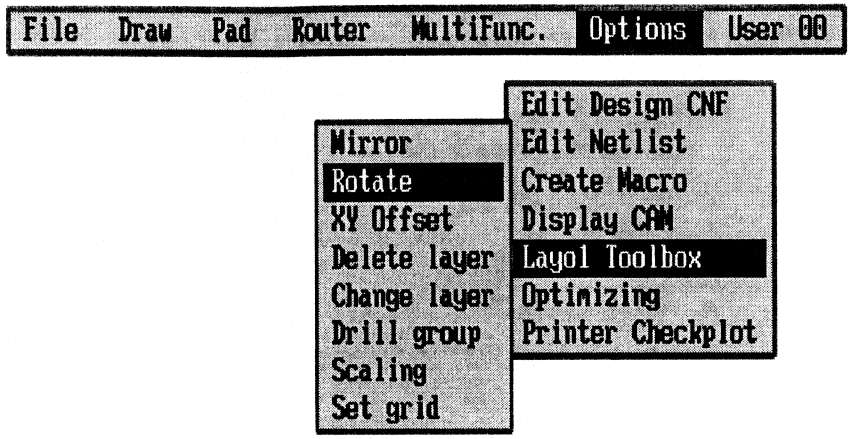


Fig. 3-44 Options, layo1 toolbox submenu

Tool select (secondary)

Activated by: Press **[t]**.

Function: Selection of a tool number and its associated drill diameter for a currently selected pad.

Description: This function can only be activated when a pad is attached to the cursor (see ▸Pad place). The tool selection will be valid for all pads that are the same type (number and set) as the currently attached. Layo1 will open a window showing the tool and the associated drill diameter. The default values are shown in the figure below. You can select and modify an entry in the table.

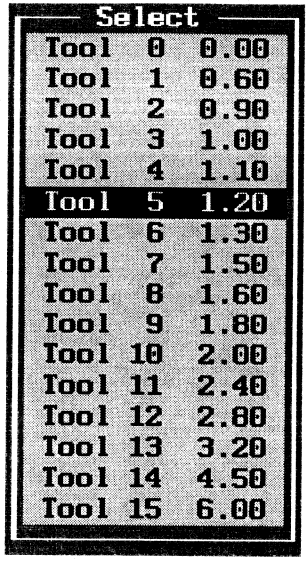


Fig. 3-45 Default tool diameters

User xx menu (main, user xx)

Shortcut: None.

Function: To set all user definable parameters.

Description: This menu gives access to all user definable parameters. When you save them after you changed them, they will remain in effect for the current user number only. The opening menu screen is shown below. The various functions are described in the following entries:

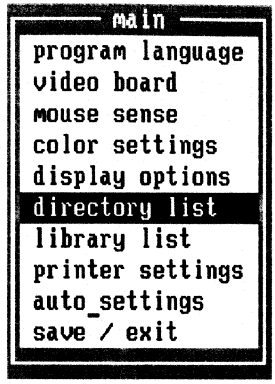


Fig. 3-46 User parameter menu

- ▸Configure language;
- ▸Mouse sensit.;
- ▸Display options;
- ▸Configure libraries;
- ▸Auto settings;
- ▸Configure video driver;
- ▸Color settings;
- ▸Configure directories;
- ▸Configure printer;
- ▸Save user configuration.

Via (secondary)

Activated by: Press **[F2]**.

Function: Place or remove a via.

Description: Pressing **[F2]** places a via at the cursor. If the cursor is already over a via, it is removed. This can also be used in combination with ▸Layer select when drawing traces, to place trace sections on alternate layers. You draw part of a trace, insert a via, select another layer and continue the trace.

Window define

(secondary)

Shortcut: Click ■□■.

Function: Select objects within a rectangular window for manipulation.

Description: This function can be activated from the Top Level. When the window is fixed with ■□□, the program switches to the **move/rotate** mode as shown in the top left of the screen. The selected objects are highlighted. You can then use the mouse to manipulate the objects in the enclosed area. You can also rotate the selected objects with**[F2]**.

Note 1: Window define will only select objects that have no electrical significance. That means that for instance component attributes can be manipulated, but not tracks or pads. If you want to manipulate electrical significant objects, you must use ▸Manipulate window from the Multifunc menu.

Note 2: You cannot define a window when it does not enclose any objects!

Window delete

(multifunc, window)

Shortcut: None.

Function: Delete all objects in an area.

Description: This function will automatically activate the ▸Window define function.

The window start and end corners are fixed with clicking ■□□. See also ▸Manipulate window. When you are working on a netlist based design (.PLY), you cannot delete components and corner lines with window delete.

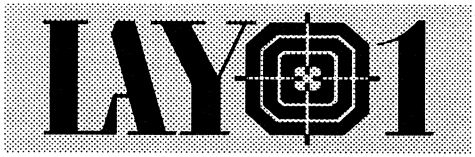
Zoom (secondary)

Activated by: Press **[z]** or click □■□.

Function: Change the display scale factor.

Description: When activated, a menu is shown to select a zoomscale between 1 and 9. Scale 1 will show little detail, but allows maximum overview of a design. Scale 9 shows maximum detail and can be used for precision editing. A special case occurs when you press **[zz]**; this will zoom the design to just fill the display. This will however only work if the design dimensions have been defined, i.e. the ▸Corner lines have been placed. See also ▸Edit Design CNF.

Notes



CHAPTER 4

Text Editor Reference

This chapter lists all commands and functions for the Project Manager's build-in Text Editor. The editor is largely compatible with Wordstar and Sidekick. If you are familiar with any of these programs, you can use the PM editor right away. Nevertheless, this reference section is included for completeness.

When you are in the Project Manager, there are two ways to activate the editor. The first way is to place the cursor bar on a file in the middle or rightmost window you wish to edit, and press [↵]. The other way is using a command-line within the Project Manager menu structure. This is described in detail in chapter 5, Project manager reference.

Editing screen

When the editor starts up, the screen is divided into three sections as shown below. The middle and largest part is taken up by the actual edit screen. Depending on the start-up mode (see also chapter 5), this middle window is empty or contains (part of) the file to be edited. There is a status-line at the top of the screen and a command-line at the bottom.

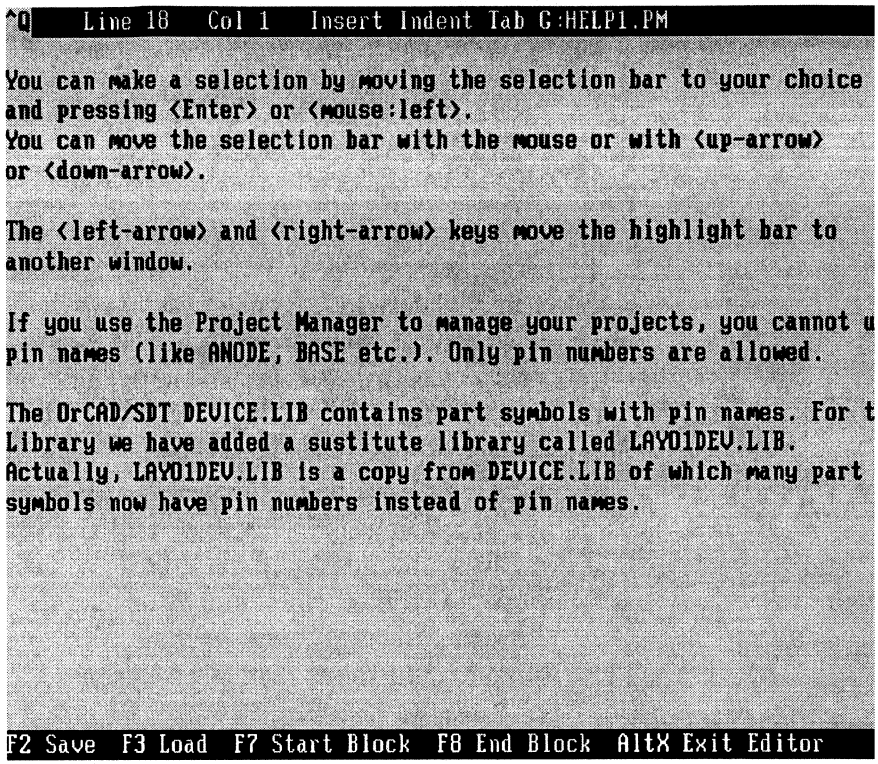


Fig. 4-1 Editor opening screen

Status line The status line contains the following items:

Cmd This field, at the far left, will display the command you are entering. If you enter a multi-argument command like **[^Q][^F]**, ^Q will be shown to indicate that the command has to be completed.

Line n This field shows the line number within the file you are editing where the cursor is positioned.

Col n — Shows the column within the line where the cursor is positioned.

Insert — Indicates the Insert status. When this field is empty, insert is off and any new characters will overtype existing text. If Insert is shown, new text will be inserted. 'Insert' is toggled with **[Ins]**.

Indent — Indicates the Auto Indent status. When Indent is displayed, and **[↵]** is pressed, the editor will automatically position the cursor on the next line in the column of the first character of the previous line. If Auto Indent is off, a new line will start at column 0. The Auto Indent status may be toggled by pressing **[^Q][I]**.

Tab — Indicates the Tab status. When Tab is displayed on the status line, the **[Tab]** key reacts as you would normally expect. That is, it shifts the cursor 8 spaces to the right on the current line. However, you can switch to 'smart tabs' by pressing **[^O][T]**. The word 'TAB' is removed from the status line. If you now press **[Tab]**, the cursor is moved to the right to the position of the next word on the previous line. This gives you the capability to automatically align tables to the first line of the table. You go back to normal tabs by pressing **[^O][T]** again.

filename.ext — Indicates the name of the currently edited file.

Command line — The command line contains the following items:

F2 = Save — The file being edited is saved when you press this key. The original file is saved as **filename.bak**.

F3 = Load — Loads another file from disk. The name will be requested. If you were already editing a file, which was changed after the last save, the program will ask if it should be saved. In any case, the new file will replace the previous one in the editing screen.

F7 = BlockBegin — Define the current cursor position as start of a block of text. The end of the block will only be visible if it was defined previously.

F8 = BlockEnd — Defines the current cursor position as the end of a block of text. The beginning of the block is visible only if it was defined before.

AltX = Exit — Exit the editor and return to the Project Manager. If the currently edited file was changed since the last save, the program will ask if it should be saved.

Editing commands

The Text Editor supports the commands listed in the next section. Note that the two block-define commands (Ctrl-Q B and Ctrl-Q K) have a shortcut as shown in the command line (F7 and F8).

Cursor positioning

Char left	Cursor left
Char right	Cursor right
Word left	Ctrl-cursor left
Word right	Ctrl-cursor right
Line up	Cursor up
Line down	Cursor down
Scroll up	Ctrl-W

Scroll down	Ctrl-Z
Page up	PgUp
Page down	PgDn
Beginning of file	Ctrl-PgUp
End of file	Ctrl-PgDn
Beginning of line	Home
End of line	End
Top of screen	Ctrl-Home
Bottom of screen	Ctrl-End
Beginning of block	Ctrl-Q B
End of block	Ctrl-Q K
To mark 0..3	Ctrl-Q 0..3
Previous position	Ctrl-Q P

Insert and delete

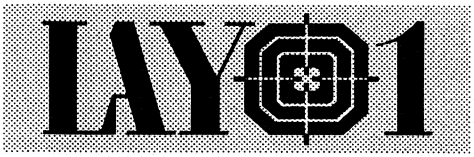
New line	Enter
Insert line	Ctrl-N
Insert Ctrl character	Ctrl-P
Insert Tab	Tab
Insert mark 0..3	Ctrl-K 0..3
Delete character	Del
Delete character left	Backspace
Delete word	Ctrl-T
Delete to end of line	Ctrl-Q Y
Delete line	Ctrl-Y

Search and replace

Search	Ctrl-Q F
Search and replace	Ctrl-Q A
Repeat last search	Ctrl-L

Search/replace options

B	search backwards from cursor
L	search within block only
Nx	searches forward x times
U	case-insensitive search
W	search for identical string only
G	search global in file
N	replace without confirm
Ctrl-Z	continue search



CHAPTER 5

Project Manager Reference

The Project Manager is a programmable menu system, acting as a command shell for the Layo1 Graphics Editor and its support utilities. When a menu entry is selected, Project Manager builds a .BAT-file to execute the requested function. Project Manager removes itself from memory, making room for the selected function, and calls the .BAT file. The last command in the batch file will normally be a call to the Project Manager, so that this will be started up once again. Thus, when the selected function has been executed, you will return to the Project Manager screen.

The displayed menu's and the actions that they will initiate are fully programmable. You do not need any programming experience, but you should be familiar with MS-DOS commands and batch files and commands. The Project Manager can be customized using any text editor, but the use of Project Manager's build-in editor is a natural choice.

This chapter discusses the structure of the programmable Project Manager, its internal commands and how to customize it to your personal needs. The use of the Project Manager and the delivered menu structure is discussed in the User Guide.

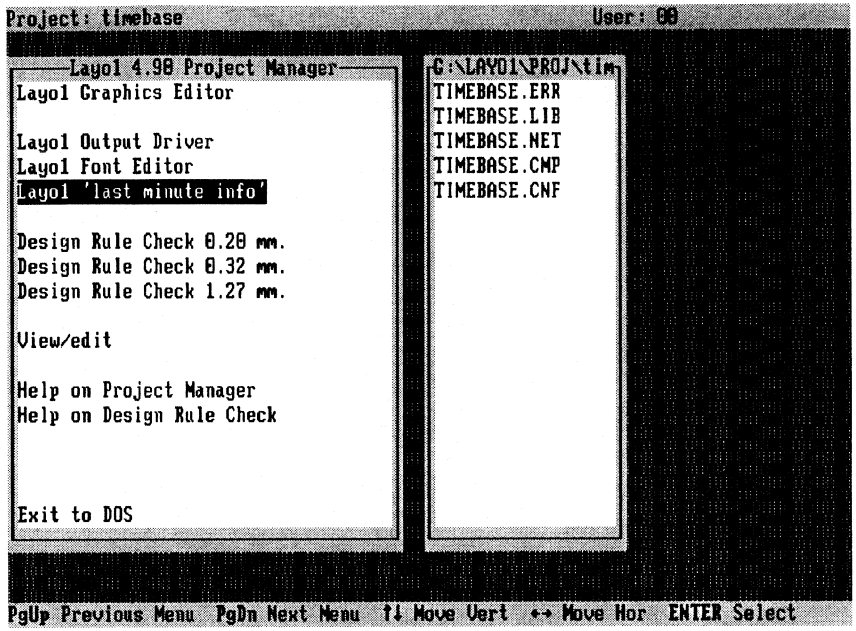


Fig. 5-1 Project Manager screen

Project Manager file structure

The Project Manager uses several files which define the menu structure and associated actions. The menu structure and function definitions file has the extension **.DAT**. A **.CNF** file holds the last selected menu page, project etc. This will insure that Project Manager starts up in the same state and at the same menu page as the last time you ran it. There will also be a **.BAT** file used to start the project manager. All these files will have the name **PM**. You can have more than one Project Manager structure, which would each than have a unique name as described below. Finally, there will be a

PTEMP.BAT file, which holds the DOS commands necessary execute to the selected menu entry.

PM.BAT A typical PM.BAT looks like the following:

```
echo off
c:
cd \PMAN
projmenu pm
ptemp 00 timebase
```

This batch file will change to the indicated drive and directory and then start the Project Manager program **PROJMENU.EXE**. The projmenu program will put up the menu pages as defined in the PM.DAT file. As you see, the name of this definition file is passed to projmenu.exe on the command line. The structure of pm.dat is explained further on. When you make a menu selection, Project Manager will build a batch file named **PTEMP.BAT**. It then terminates itself and pass control to this ptemp.bat. If PTEMP.BAT contains PM.BAT as the last line (this is something you can set up yourself, see below) Project Manager is started up again when ptemp is done. You are now ready for the next menu selection. It sounds complicated, but if you trace execution through the commands, it's really straightforward.

The two arguments to PTEMP represent the currently selected user number and the selected project you are working on. These may be used as replaceable parameters in the selected menu function. These will be explained later.

You can build your own Project Manager, possibly for uses not related to Layo1. If you call this version say **MYPM**, you will need to build a menu structure called **MYPM.DAT** and a

configuration file **MYPM.CNF**. You must then change the argument in line 4 above to MYPM. It is very important to keep the *structure* of PM.BAT **exactly** as above. You may only change the drive, the directory, the projmenu argument and ptemp's user and project arguments. Do not change the number of lines or their order!

PM.CNF The .CNF file contains the last settings of Project Manager (menu's parameters etc.) and is used to start it up in the same state as you exit it. If you want to create a custom **MYPM.CNF** you can copy the existing PM.CNF to MYPM.CNF. When you exit your version of the project manager, MYPM.CNF will have the correct settings.

PM.DAT This is the actual menu structure that Project Manager uses to display the selections and to execute the related functions. Each entry in this file consists of a line beginning with one or more special command characters and the argument(s) to the command. The available commands will be described below.

Menu command repertoire

The Project Manager acts on commands that are contained in the menu structure (.DAT file). Each command consists of one or two characters. Most of the time there will be arguments following the command. A complete command string is terminated by a new-line character. The commands for border styles and colors accept many parameters, which are listed at the end of this section.

Commands The following commands are recognized by the Project Manager:

% This command starts a new Project Manager menu page. The string following % will be displayed as the menu page title at the top of the menu. This title may have up to 36 characters. A menu structure can have a maximum of 10 menu pages. Example:

```
%Layo1 Project Manager
```

& This command uses the string following it as a *label* to display the currently active user number on the top line of the display. Example:

```
&User:
```

This command uses the string following it as a *label* to display the currently selected project on the top line of the display. Example:

```
#Project:
```

}1 This command will open the center directory window. The path following the command (no spaces) determines which files will be shown. The window will show the path on the title line on the top border. DOS wildcard characters may be used. If **%2** is used in the path string, it will be replaced with the arguments specified with the ▸**@A** and ▸**@P** commands (see below). Example:

```
}1c:\layo1\proj\myproj\pcb\myproj.*
```

will display all files starting with myproj in the directory c:\layo1\proj\myproj\pcb\. If you had specified the project as **'myproj'** with @P or @A, you could have used:

```
}1c:\layo1\proj\%2\pcb\%2.*
```

which would automatically select the correct files to display depending on the project selected.

}2 This command will open the right-most directory window. The path may be specified as described for }1 above.

$M The string of numbers following this command will determine the for- and background colors of the various parts of Project Manager's windows. All in all, 11 color attributes can be specified. They are coded as follows (the codes itself will be explained below):

$M aaa bb cc dd ee ff gg hh ii jj kk, where:

aaa = border type;
bb = border background color;
cc = border foreground color;
dd = text background color;
ee = text foreground color;
ff = cursor background color;
gg = cursor foreground color;
hh = highlighted text background color;
ii = highlighted text foreground color;
jj = error/warning message background color;
kk = error/warning message foreground color;

$E The string of numbers following this command will determine the for- and background colors of the build-in editor's screen. The various attributes that are available are coded as follows:

$E aa bb cc dd ee ff gg, where:

aa = top/bottom info/status line background color;
bb = top/bottom info/status line foreground color;
cc = top/bottom info/status line highlight color;
dd = text background color;
ee = text foreground color;
ff = block background color;
gg = block foreground color;

Note: The &, #, $M, $E, }1 and }2 commands should directly follow the %-command which opens the menu page!

* This command will display a menu entry on the current page. Each page can have up to 20 menu choices. Each menu choice name may be up to 36 characters long. The entry can be selected by placing the highlight bar over it and pressing **[↵]** or by entering the first character of the menu entry name. You move the highlight with the cursor keys or the mouse.
The menu entry name must follow the command. Example:

```
*Select PCB
```

will put up a menu choice that can be selected by pressing **[s]**.

@E This command starts the build-in editor. The command may be followed with a filename, which will then be loaded into the editor ready for editing. Example:

```
@Epm.dat
```

This will open the editor with the menu-structure pm.dat loaded, ready to be modified. If you want to edit a file in another than the current directory, you must specify the full pathname. The %2 replaceable argument may be used as described for ▸}1 and ▸}2.

@P This command enables the specification of a parameter. The parameter must be entered as a string of maximum 20 characters. This parameter will be shown at the top left of the screen in the field preceded by the label specified with the ▸#-command. The parameter can also be used in other commands and functions selected by menu choices on the *current* menu page by using the %2 replaceable argument. Example: Suppose you specified **'#Current PCB:'** at the start of this menu page. Suppose you want to have a menu selection to select the name of the PCB you work on, to be displayed on the status line. The following PM.DAT fragment will accomplish it:

```
*Select PCB
@P
```

If you select this function, the cursor will move to the top left of the screen. If you now enter **'newpcb'**, the top left of the screen will show: **'Current PCB: newpcb'**. Furthermore, any reference in the other menu functions to **%2** will be replaced at execution time by **newpcb**. Example:

```
*Show project files
+dir d:\layo1\proj\%2\pcb\%2.*
+pause
+pm
```

will display all files related to the selected project **newpcb** until you 'hit any key to continue' (assuming of course that you have installed Layo1 with the default directory structure on c:).

@A This command works the same as @P above, except that the parameter you enter will be valid for *all menu pages*, rather than just for the current page.

\+ This command makes the Project Manager perform the actual work when you select a menu entry. The line following this command will be treated as a DOS-command. Project Manager will build a PTEMP.BAT file that will contain all +... command lines found in the selected menu entry. Example:

```
+dir /p
+pause
```

will cause the DOS command processor to perform a **'DIR /P'** command, followed by the usual message **'press any key to continue'**.
If you want to return to Project Manager after the selected function is performed, you must include **'+pm'** as the last line. If you omit this or use just the + sign, you are returned to DOS. Example:

```
*Exit Project Manager
+echo Project Manager signing off
+
```

will display a menu choice titled **Exit Project Manager**. If you select this entry, the screen will show the message **'Project Manager signing off'**, and you are returned to the DOS prompt.

Borders The border-attributes for the Project Manager windows (see ▸$M, ▸$E) are coded as follows:

Border:	Bottom	Right	Top	Left
000	S	S	S	S
001	S	S	S	D
002	S	S	D	S
003	S	S	D	D
004	S	D	S	S
005	S	D	S	D
006	S	D	D	S
007	S	D	D	D
008	D	S	S	S
009	D	S	S	D
010	D	S	D	S
011	D	S	D	D
012	D	D	S	S
013	D	D	S	D
014	D	D	D	S
015	D	D	D	D
023	N	N	N	N

S = single border;
D = double border;
N = no border.

Colors The color attributes are coded as follows:

00 = black;	08 = dark gray;
01 = blue;	09 = light blue;
02 = green;	10 = light green;
03 = cyan;	11 = light cyan;
04 = red;	12 = light red;
05 = purple;	13 = light purple;
06 = brown;	14 = yellow;
07 = light gray;	15 = white.

Note: For background attributes, only colors 00 through 07 are valid!

Example As an example, let us construct a custom menu page that lets you perform some DOS services like editing your **AUTOEXEC.BAT** and **CONFIG.SYS** files and format a data disk.

The following Project Manager .DAT file fragment will set up a menu page as shown below:

```
%DOS Services
}1c:\autoexec.*
}2c:\config.*
*Format a disk in A:
+format a:
+pm
*Format a disk in B:
+format b:
+pm
*Exit to DOS
+
```

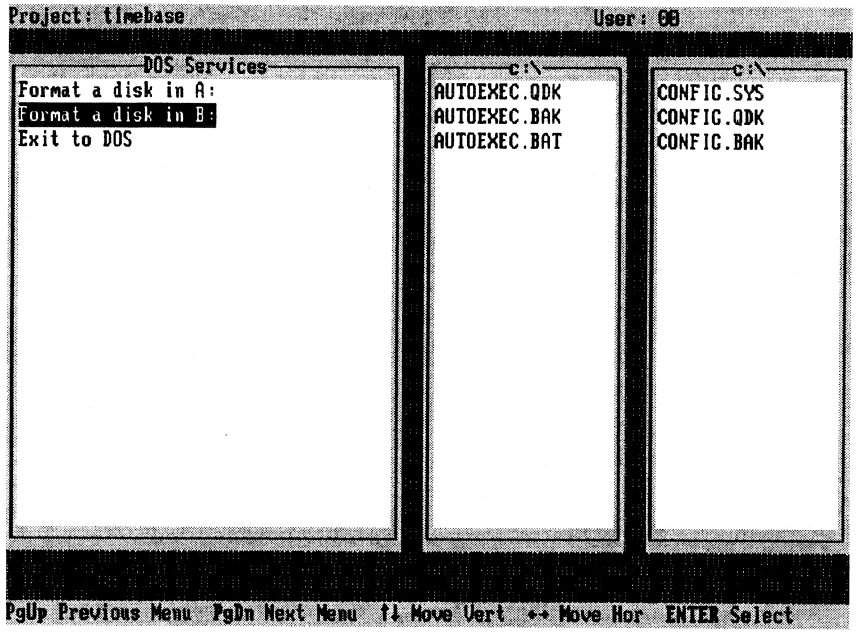


Fig. 5-2 Project Manager custom menu page

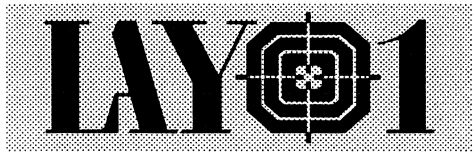
You can now format a disk by selecting the appropriate entry. Because the autoexec.bat and the config.sys files are displayed in one of the directory windows, you can easily edit them. Just place the highlight over the filename and press [↵]. Alternatively, you could have included a separate menu entry for editing autoexec.bat as follows:

```
*Edit autoexec.bat
@Ec:\autoexec.bat
```

If you find this addition usefull, just load your PM.DAT in the editor, and add these lines at the end. (Its a good idea to save

the original first). If you need an example on the use of the border- and color attributes, see how they are used in PM.DAT (look for the **$M** and **$E** described above).

Notes



CHAPTER 6

Output Driver Reference

The stand-alone Output Driver program is used to generate the high-quality layouts to produce the actual PCB. It accepts various Layo1 file formats, and in turn generates outputs to drive a wide variety of output devices. Each output function can be fully configured as to the data to be output and the output device to be driven. Up to 20 output functions or jobs can be saved on a separate configuration page.

Start-up

If you are using Layo1 and Project Manager to design PCB's, you can start the Output Driver from Project Manager's Project Manager menu page. If you are using the Output Driver at a production site to process customer-delivered designs, you can just start the program by with the following command line:

```
layo1plt [filename.ext] [/Uxx]
```

If you do not specify the filename (and if necessary the path) you will have to specify it in the ▸Output menu, input parameters. If you specify a user number, the Output Driver will read the USRXX.CFG file and use the pad, pen and tool definitions from it. If no userfile is specified the default values will be used. If specified but not found, the USRXX.CFG will be created and the parameters set to the default values. If you specify an **.LML** or **.PLY** file as input, the program will

search for a **.CNF** with the same name in the same directory. It this is found, it will use the pen and pad definitions contained in it. If not, the default parameters for pads and penwidths will be used. Note that depending on the type of output device, the terms 'tool' and 'pen' are used interchangeably in this chapter.

Output menu

As soon as the program is started, it will display the **Output Menu** screen as shown below:

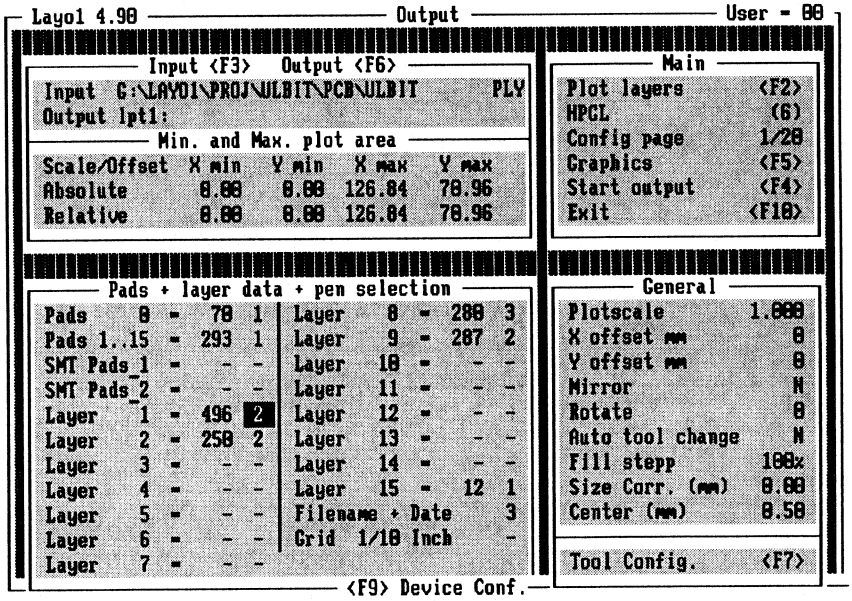


Fig. 6-1 Output Driver Output screen

The Output menu consists of several parameter areas, with default settings that can be (and sometimes must be) changed. Note that basically all selection and parameter entry is initiated by highlighting the appropriate entry and pressing [↵]. Also, many entries have a control-key shortcut as shown in the window. Therefor, in the discussion below, it is assumed that you know how to activate a particular menu entry.

Output main window

This window contains several parameters that can be set, and commands that can be given.

Plot select

This choice opens a window where you can select three different sets of data to be output:

- Plot selected layers;
- Plot pads;
- Plot drill data.

Each of the selections changes the lower left window. This will now show the parameters to be set for the selected output sets.

Plot layers

This will select layers for output. The pad and layer data and the related pen selections are shown in the lower left window. Layers and pads used in the design show a non-zero number, indicating the number of pads per type and number of objects on each layer. For each entry you can select a pen. Selecting pen 0 effectively *deselects* the layer (or pad group) for output. Each pen can be set to a custom width by selecting **Penwidth** or **[F7]** in the lower left corner. A typical screen is shown below.

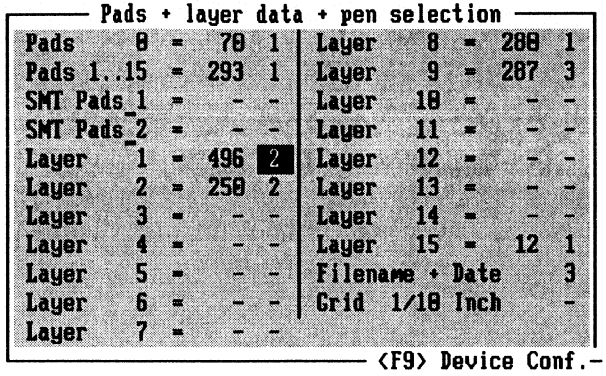


Fig. 6-2 Output Driver plot layers specification

Plot tools

This will select tools/pens to be included in the output. This means that only the objects drawn with the selected tools will be output. Again, the lower left window shows the tool data and related pen selections. The tools used in the design show a non-zero number, indicating the number of objects per tool/pen For each entry you can select a pen. Selecting pen 0 effectively *deselects* the tool data group for output. Each pen can be set to a custom width by selecting **Penwidth** or **[F7]** in the lower left corner. A typical screen is shown below.

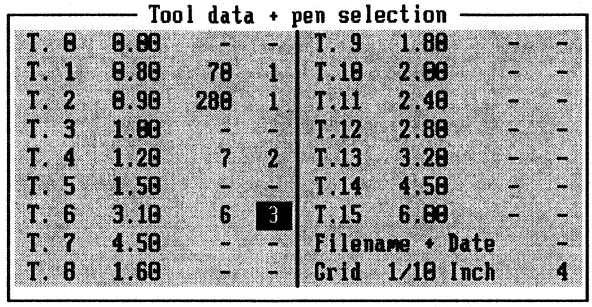


Fig. 6-3 Output Driver plot tools/pens specification

Drill program

This selection is used to prepare the output data for a drilling machine. As with the previous selection, the used tools and the number of objects (really holes in this case) are displayed. However, you *cannot* change the diameter (except of course if you insert a different tool in the machine than requested). You can only select and deselect individual tools. If you use the preview option (**[F5]**) the deselected tool data is shown in a different color (white) from the selected data (red). A typical screen is shown below.

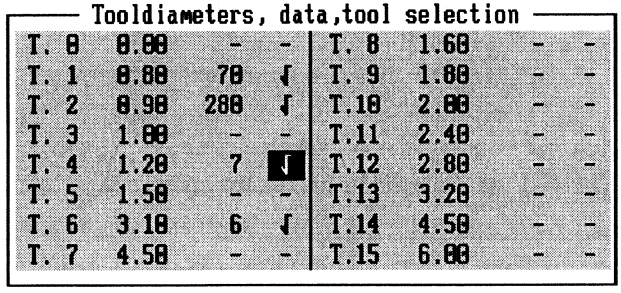


Fig. 6-4 Output Driver drill program specification

Output format The second entry in the main window shows the device name and device number selected. In the example above, **'HPGL (1)'** is shown. When you select this entry, a window will open where you can select up to 10 different devices, each with its own name and format. When you have a device selected, you can edit its parameters by pressing **[F9]** as indicated at the bottom of the screen (▸Device configuration menu).

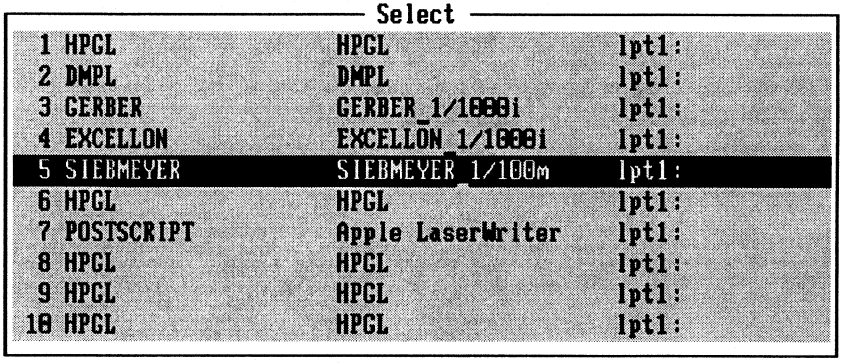


Fig. 6-5 Output driver selections

Configuration page All parameters set in the menu windows can be saved on one of 20 different configuration pages. In this way you can set up 20 different configurations for different output jobs or customers.

Graphics This is strictly a command and will give you a graphic preview of the data (layers, pads etc) selected for output. It is meant as a visual check before committing the output device.

Start output As its name implies, this starts the output process. The output can be aborted by pressing **[Esc]**.

Exit Stop the program and return to the calling process (Project Manager or Dos).

Output Input/output window

This window is used to enter the input file used it generate the output, and also the destination of the output.

Input This entry shows the input file. Selecting it for editing will open a submenu where you must first select the *type* of input file to use. The following selections are available:

Block **.BNK**	Comp **.LMC**
Drawing **.LAY**	Design **.LML**
	Design **.PLY**

The **.BNK** (Block) and **.LAY** (Drawing) formats provide compatibility with previous Layo1 versions. Note that the two **'Design'** formats require the presence of an **.CNF** with the same name to insure that the output will conform to the design with respect of custom penwidths and pads. When you select a format, a window opens where you can enter the filename, or make a selection using the ▸File/directory select as described in Chapter 3. You do not have to supply the file name extension, because the Output Driver will fill that in from your earlier type selection.

Output This parameter can be either a file or a physical device. When you select an output port, you can select an LPT port (LPT1..4) or a COM port (COM1..4). When you select a COM port, additional selection windows are presented for Baud Rate, number of data bits, parity and stop bits.

When you select a filename, a secondary window opens which lets you select a drive to write the output to. The output will be stored in a file with the same name as the input file and extension **.AXX**, in subdirectory **:\BBBB** on the selected drive. The subdirectory will be created if not already present and will have the same name as the selected output format (e.g. **HPGL** for a HPGL printer, **SIEBMEYE** for a SIEBMEYER output device). The first character in the extension (**A**) will be set to the first character of the selected output format (e.g. **H** for HPGL, **P** for Postscript etc). The number **XX** in the file extension will be 01 for the first file output, and increment by one for each following output action to the same directory and with the same format.

You can also select the third (normally blank) line in this window. Here you can enter an arbitrary filename to write the output to.

General window This menu window lists several general parameters that can be used to influence the final result.

Plotscale The final dimension of the output can be set to any reasonable scaling, both over and below 100%.

X and Y offset These two options offer the possibility to precisely position the output on the medium (paper or film). The input is in millimeters, and is limited by the physical size of the output medium.

Mirror, Rotate Mirror will output a mirror image of the design, while Rotate will rotate the design clockwise so that the dimensions of the design can be aligned with the dimensions of the output medium.

Auto tool change This can be set to **Y** or **N**. If set to no, output will pause at each new pen, to enable you to manual change it. If set to Yes, the program will assume that the output device will automatically change the tool (or pen). Auto tool change will of course only occur if the device is capable of it. This parameter must also be set for each configured device in the ▸Device configuration menu.

Fill step This parameter determines the density of filled areas. A fill step of 100% means that each pen stroke in a fill pattern will be drawn exactly one penwidth from the previous one, theoretically exactly covering the area. However, if you set the Fill step to 50%, each new stroke will be only half a penwidth from the previous stroke, thus overlapping and effectively covering the area twice. This might give better results with some devices.

Size correction This option *does not* change the dimensions of the output as a whole. Use ▸Plotscale for that. Rather, it applies a preset correction to the outside diameter of the pads. This can be useful for plotting output to be used for a silkscreen, where you normally would want the blank area around a pad to be slightly larger then the pad itself.

Center For this option you must enter a diameter which will be used as the center hole for pads. You can use this to produce a small hole in the copper of the pads to facilitate 'homing' of the drill tool.

Device configuration menu

When you press **[F9]** from the ▸Output menu, the device configuration menu is displayed. This menu has several parts as shown below.

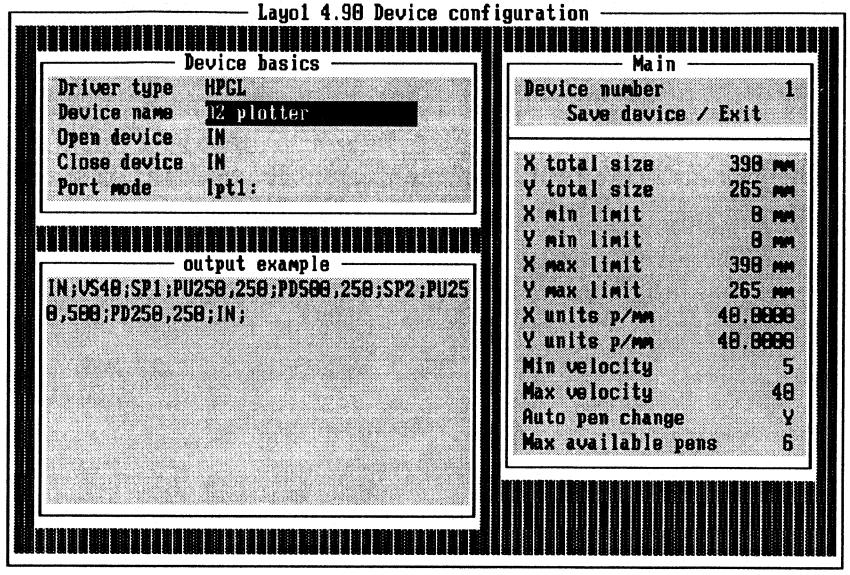


Fig. 6-6 Output device configuration menu

Device configuration main window

The Device configuration main window contains, beside the device number and the Exit/save entry, several device specific parameters. Only the parameters displayed in *red* can be changed; the parameters displayed in *black* are fixed for the particular device. The parameters can be found in the device documentation for optimal output results.

Device basics window

The entries in this window determine the basic device class.

Driver type

The *driver name* will determine the format in which the data will be output. There is a choice in several formats as shown below.

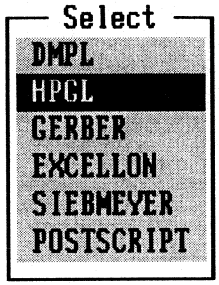


Fig. 6-7 Output driver formats

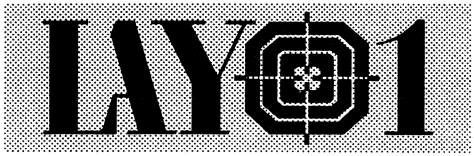
Device name The *device name* you enter is only used for identification purposes, it is the name displayed in the ▸Output menu main window.

Open, close device The commands to open or close the device to accept data are to be entered here. The program will display defaults, and only in rare cases need these to be changed. Check the device documentation for correct commands if the device does not seem to accept data.

Port mode As described for the ▸Output input/output window, you can preset an output port for the device. LPT1..4 or COM1..4 can be selected, with additional selections for the parameters for the COM port.

Example window This window shows an output sample for the currently selected device for an extra check.

Notes



CHAPTER 7

Font Editor Reference

The Layo1 Font Editor is a separate graphics editor within the Layo1 system. It is primarily meant for experienced Layo1 users to design or edit character sets, logos or symbols. It is smaller and faster than the regular editor, and uses a command subset. You should get some experience with Layo1's regular Graphics Editor before you use the Font Edit.

If you want to edit the default character set, be sure to keep a backup of the delivered .FNT file, so that you can always recover.

Character set

A character set is stored in a font file. Each font file will have the extension **.FNT**. A font is organized as a set of characters numbered from 32 to 255 inclusive. This corresponds to the regular IBM character set for printable characters and symbols.

Each character set can contain up to 4000 vectors (see also Chapter 2, ▸Vectors). That means that you can design complex logo's, if you have only one or a few in a set.

To design or edit a character from the set you must first select the font set, and then the character. Selecting the character can be done by pressing the corresponding key (for keyboard characters) or using the Alt key with a three-digit number on the numeric keypad. For instance, to select the character **'A'** you press **[A]** or **[Alt][0][6][5]** (**'A'** corresponds to ASCII code 65).

Features

Although the font editor has many features from the regular graphics editor, it is 'leaner and meaner' in the following aspects.

Layer The font editor uses only one layer. No layer has to be selected. When a character is placed in a design, you can select the layer where it will be placed. See Chapter 3, ▸Text draw.

Pen The font editor cannot uses different pen widths. When a character is placed in a design, you can select the pen width to use. However, at layers 8 or above (default 9) the penwidth will have no effect. If you have a logo you wish to be drawn with a thicker pen, use only 90- and 45 degree angles and place it with the required pen at a layernumber less than 8. See Chapter 3, ▸Text draw.

Grid The font editor uses a relative grid of 255 by 255 steps. Thus a single character or logo has a detail limit of 255 point in x and y direction. When the character or logo is placed in a PCB, the actual size is determined.

Related to this is the help grip. Two different settings are possible, 2 steps or 80 steps per grid. This changes automatically as you zoom in or out. ▸Grid select.

Mode The font editor can operate in two distinct modes, **drawing mode** or **edit mode**. This will be explained in the following section.

Commands

The font editor commands are all entered as single keystrokes from the keyboard. This will enable you to work much quicker than if you had to use menu's instead. As said before, this does mean that you must have some experience with the regular Layo1 Graphics Editor. The limited command set, is in many respects very much like the Layo1 shortcut commands. The optimal setup of the Layo1 Font Editor is also evident in its clean edit screen. When the program is started, the following opening display will be presented:

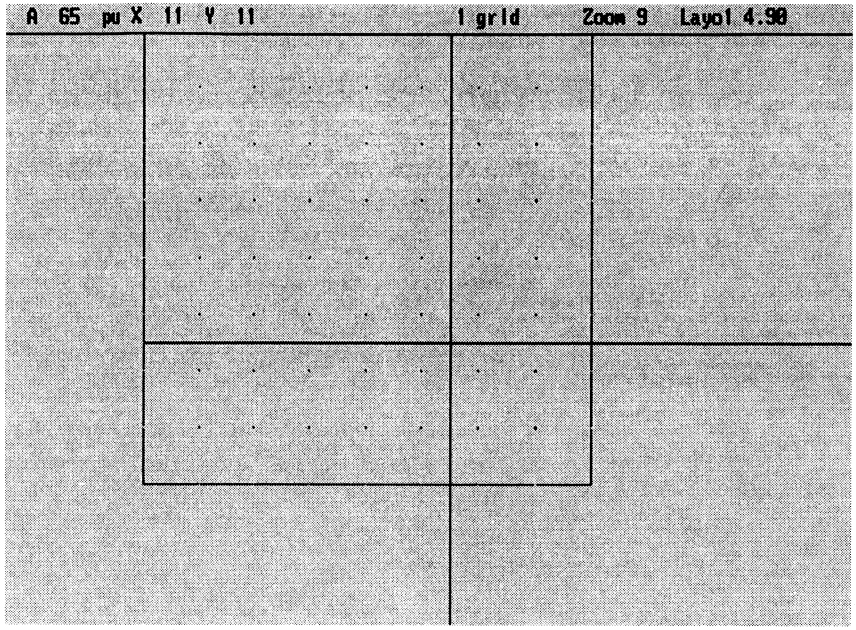


Fig. 7-1 Font editor opening screen

Browse font Press **[F7]**, **[F8]**.

Description: This function enables you to 'page' through the various characters of the font. Pressing **[F7]** will bring up the previous character by its page number. For instance, from 'C' (ASCII 67) you will step to 'B' (ASCII 66). This command can also be called by pressing **[↑-]** or mouse □■■. Pressing **[F8]**, **[↑+]** or mouse ■■□ will bring up the next character in the set.

Character end Press **[F2]**.

Description: This command will define the current cursor position as the top right boundary of the character. The top left boundary is the original origin (0,0) of the character field when loaded. The top left boundary determines the point where the first character in a string will be attached to the cursor when placing text. The top right boundary determines where the top left boundary of the next character in a string will be placed. This lets you set the whitespace between individual characters or symbols.

Character select Press **[c]**.

Description: This function enables you to select a character, either from the keyboard or by number, using Alt plus the numeric keypad (▸Character set). Depending on the situation it might be faster to page to the required character using ▸Browse font.

Cross, diagonal Press **[q]**.

Description: A diagonal cross hair will be placed on the screen at the current cursor position. This is meant as a drawing aide, it will not generate any output. To remove a previously placed cross hair, place the cursor over the cross point and press **[q]** again.

Cross, rectangular Press **[a]**.

Description: A rectangular crosshair will be placed on the screen at the current cursor position. This is meant as a drawing aide, it will not generate any output. To remove a previously placed cross hair, place the cursor over it and select this function again.

Draw mode Press **[t]**.

Description: The font editor will enter the draw mode. A **'T'** will be shown at the top left corner of the screen. Moving the mouse will draw lines. You can insert nodes with ▸Node insert to change directions, or click □□■ to end the current line and move the mouse to a new location without drawing.

Edit mode Press **[e]**.

Description: The font editor will enter the edit mode. An **'E'** will be shown at the top left corner of the screen.

Font load Press **[F3]**.

Description: You are asked to enter the name of the font you want to load. ▸File/directory select (Chapter 3), can be used.

Font save Press **[F4]**.

Description: You are asked to enter the name of the file you want to save the current font to. ▸File/directory select (Chapter 3), can be used.

Grid select Press **[F9]** or **[F10]**.

Description: The cursor step size will be increased if you press **[F10]** and decreased with **[F9]**. The step size can be set from 1 to 50 grid steps in a 1,2,5 sequence.

Grid toggle Press **[.]**.

Description: Toggles the display of the dot grid on or off, every time **[.]** is pressed.

Help Press **[F1]**.

Description: Displays a screen with an abbreviated description for each command.

Node insert Press **[Ins]** or click ■□□.

Description: A node will be inserted at the current cursor position, acting as an anchor for the currently being drawn line. This enables you to change direction while drawing.

Node delete Press **[Del]**.

Description: The node at the current cursor position will be deleted. This will only work in the ▸Edit mode.

Pen up/down Press **[p]**.

Description: The pen up/down status will be toggled. This will only work in the ▸Edit mode.

Redisplay, overlap Press **[F5]**.

Description: This function clears the screen and redisplays an updated picture. It is similar to the Graphics' Editor Refresh screen function. Overlapping lines will be invisible.

Redisplay, mix Press **[↑F5]**.

Description: This function clears the screen and redisplays an updated picture. Overlapping lines will be shown in mixed color.

Origin Press **[F6]**.

Description: This function is similar to the Graphics' Editor Origin function. The current cursor position will be used to set the origin for ruler #3 (0,0).

Ruler select Press **[r]** followed by **[1]** or **[3]**.

Description: This function enables selection of ruler 1, which indicates coordinates from the left top of the character, or ruler 3, which indicates coordinates from the temporary origin set with ▸Origin.

X input Press **[x]**.

Description: A window will open where you can enter an x-coordinate. The cursor will be moved to the entered position.

Y input Press **[y]**.

Description: A window will open where you can enter a y-coordinate. The cursor will be moved to the entered position.

Zoom Press **[z]**.

Description: A selection window will open and you can select a zoomscale between 1 and 9. Zoom scale 9 shows the most detail, while zoom scale 1 will show a full overview.

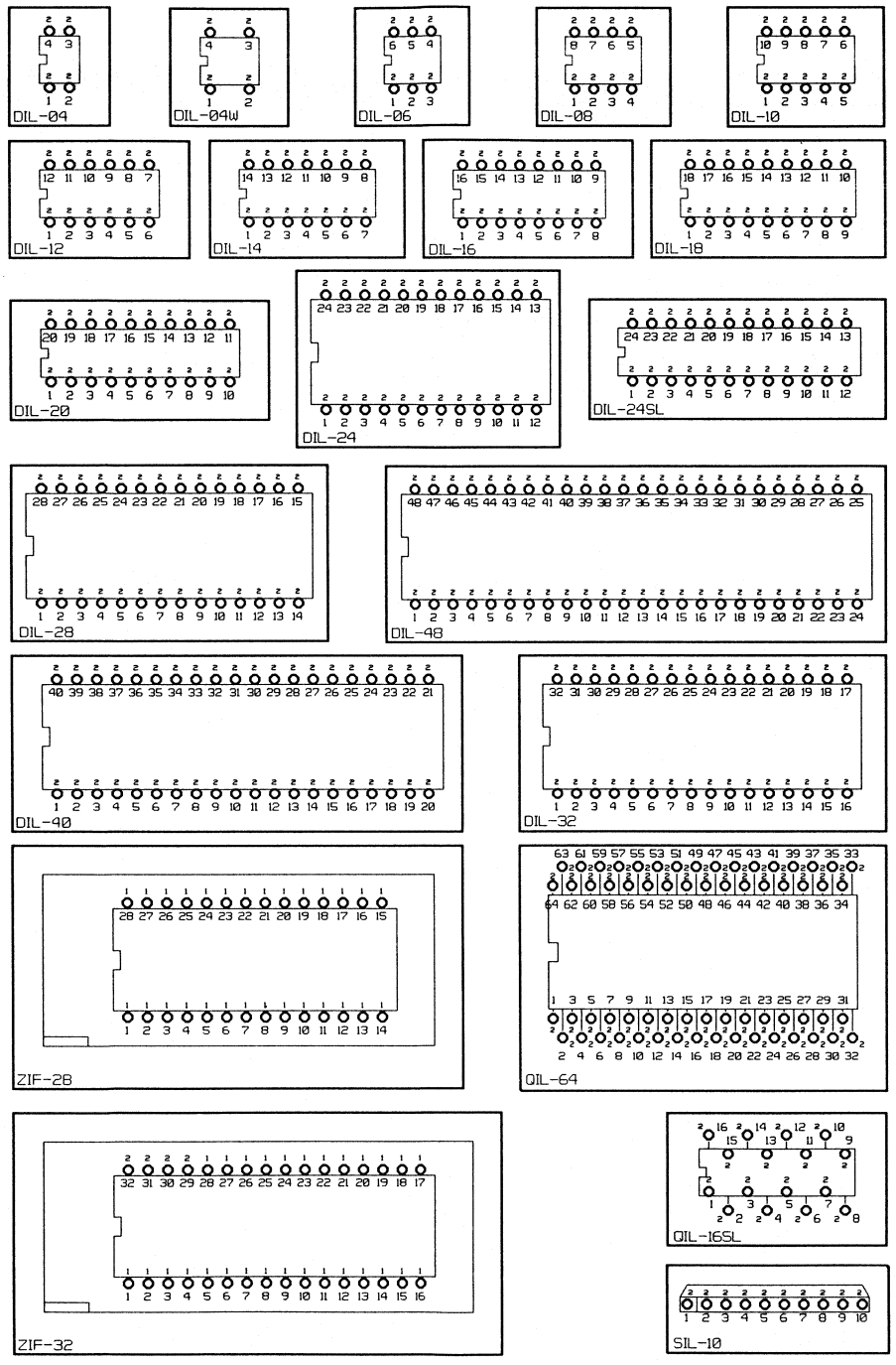
Notes

Appendix

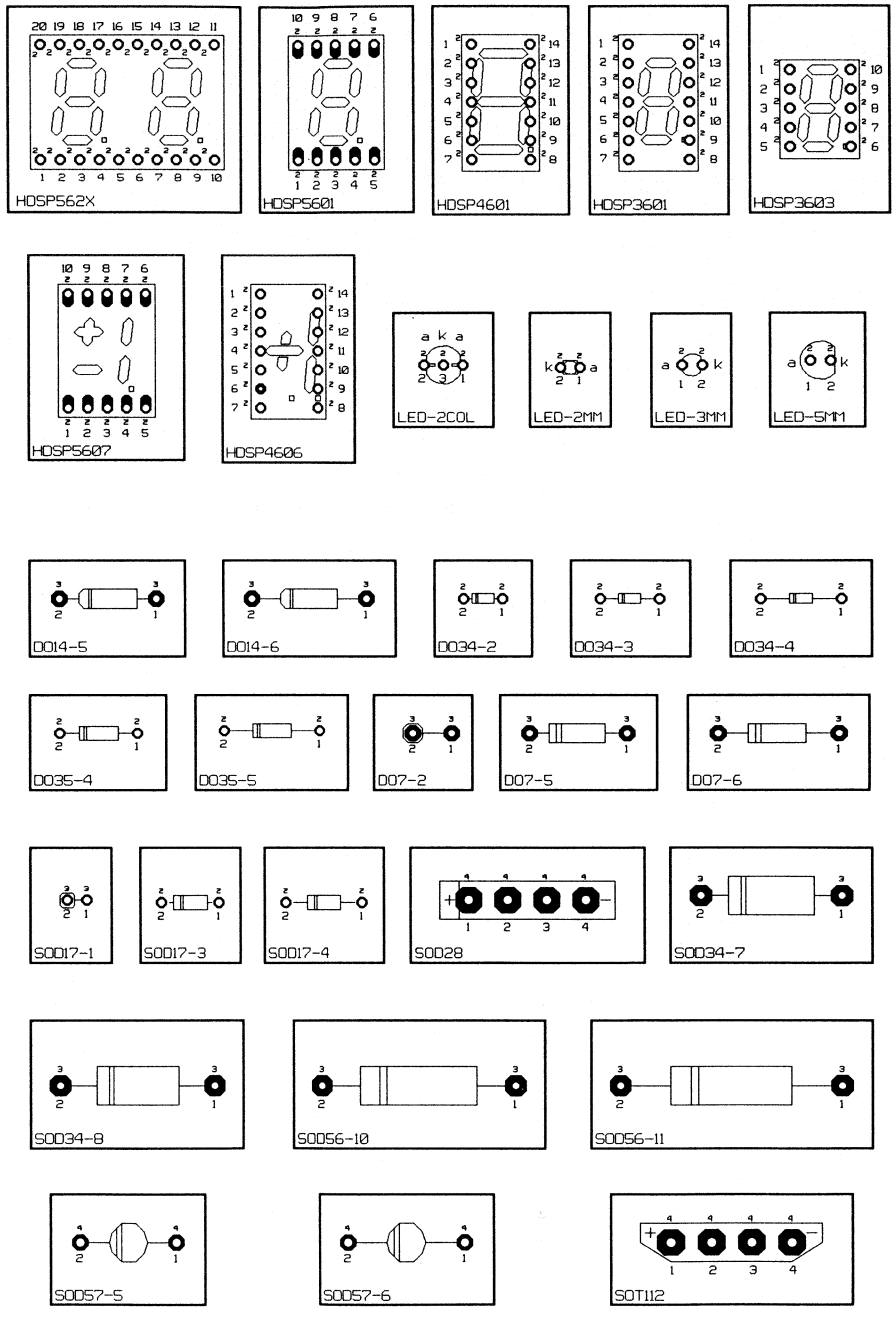
Layo1 Component Banks

The following pages show the Layo1 components that are contained in the component banks or libraries. They are divided over several directories. At the bottom of each page, you will find the name of the directory for the components on that page. You will find an overview of the available banks in Chapter 2 ▸Directory structure. All component banks or directories are found under the master component directory **..\LMC**. For instance, the page **'passive-02'** contains the model for a trimpot named **'R-POT-14'**. This component is found in directory **..\PASSIVE** and is named **R-POT-14.LMC**.

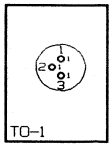
Note that when selecting or loading components, Layo1 will almost always supply the path and .LMC suffix from its configuration data.

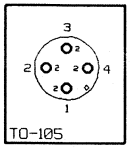

DIL-09
DIL-09W
DIL-06
DIL-08
DIL-10
DIL-12
DIL-14
DIL-16
DIL-18
DIL-20
DIL-24
DIL-24SL
DIL-28
DIL-48
DIL-40
DIL-32
ZIF-28
QIL-64
ZIF-32
QIL-16SL
SIL-10

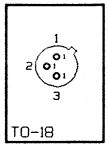
active-01

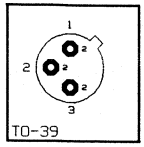

HDSP562X
HDSP5601
HDSP4601
HDSP3601
HDSP3603
HDSP5607
HDSP4606
LED-2COL
LED-2MM
LED-3MM
LED-5MM
D014-5
D014-6
D034-2
D034-3
D034-4
D035-4
D035-5
D07-2
D07-5
D07-6
SOD17-1
SOD17-3
SOD17-4
SOD28
SOD34-7
SOD34-8
SOD56-10
SOD56-11
SOD57-5
SOD57-6
SOT112

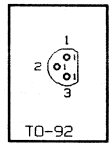
active-02

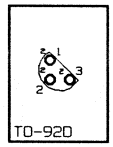

TO-1

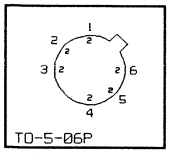

TO-105

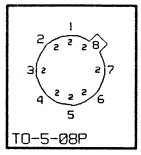

TO-18

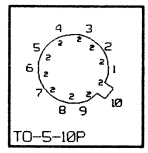

TO-39

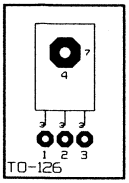

TO-92

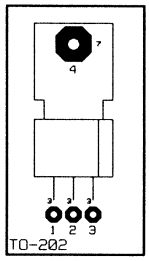

TO-92D

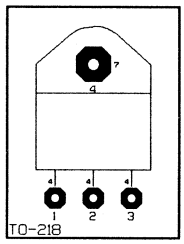

TO-5-06P

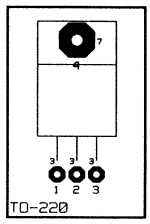

TO-5-08P

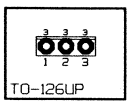

TO-5-10P

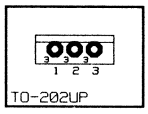

TO-126

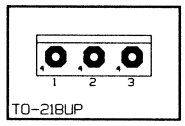

TO-202

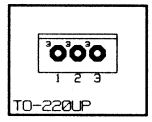

TO-218

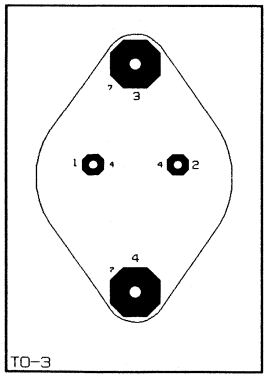

TO-220

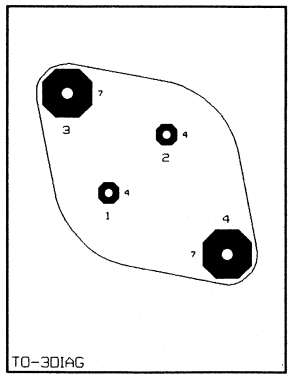

TO-126UP

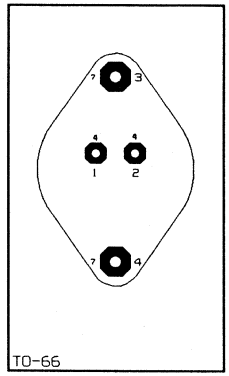

TO-202UP

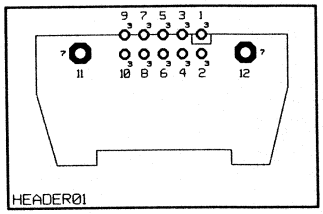

TO-218UP

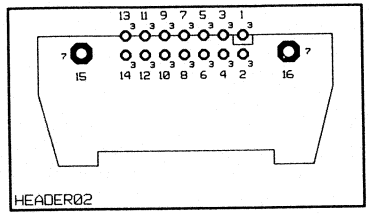

TO-220UP

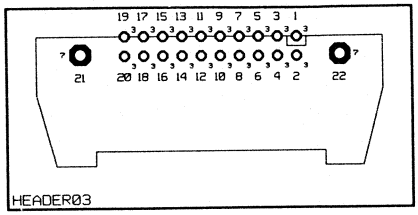

TO-3

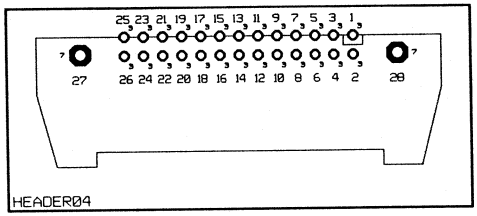

TO-3DIAG

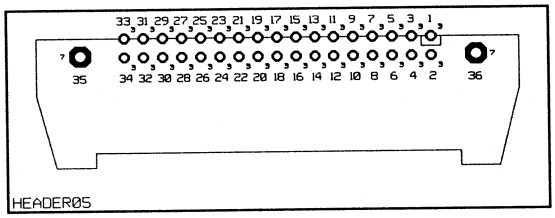

TO-66

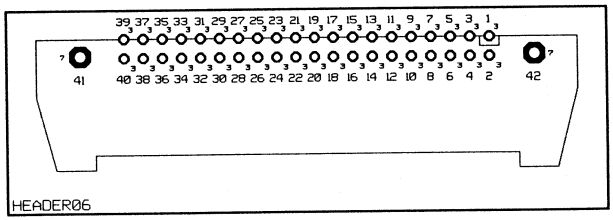

9 7 5 3 1
11
10 8 6 4 2
12
HEADER01

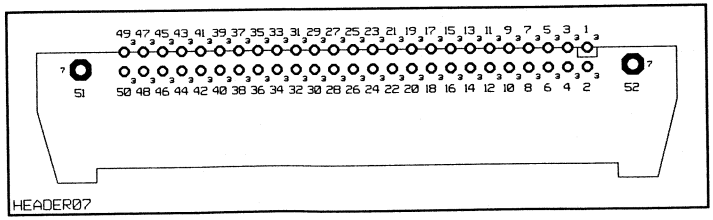

13 11 9 7 5 3 1
15
14 12 10 8 6 4 2
16
HEADER02


19 17 15 13 11 9 7 5 3 1
21
20 18 16 14 12 10 8 6 4 2
22
HEADER03

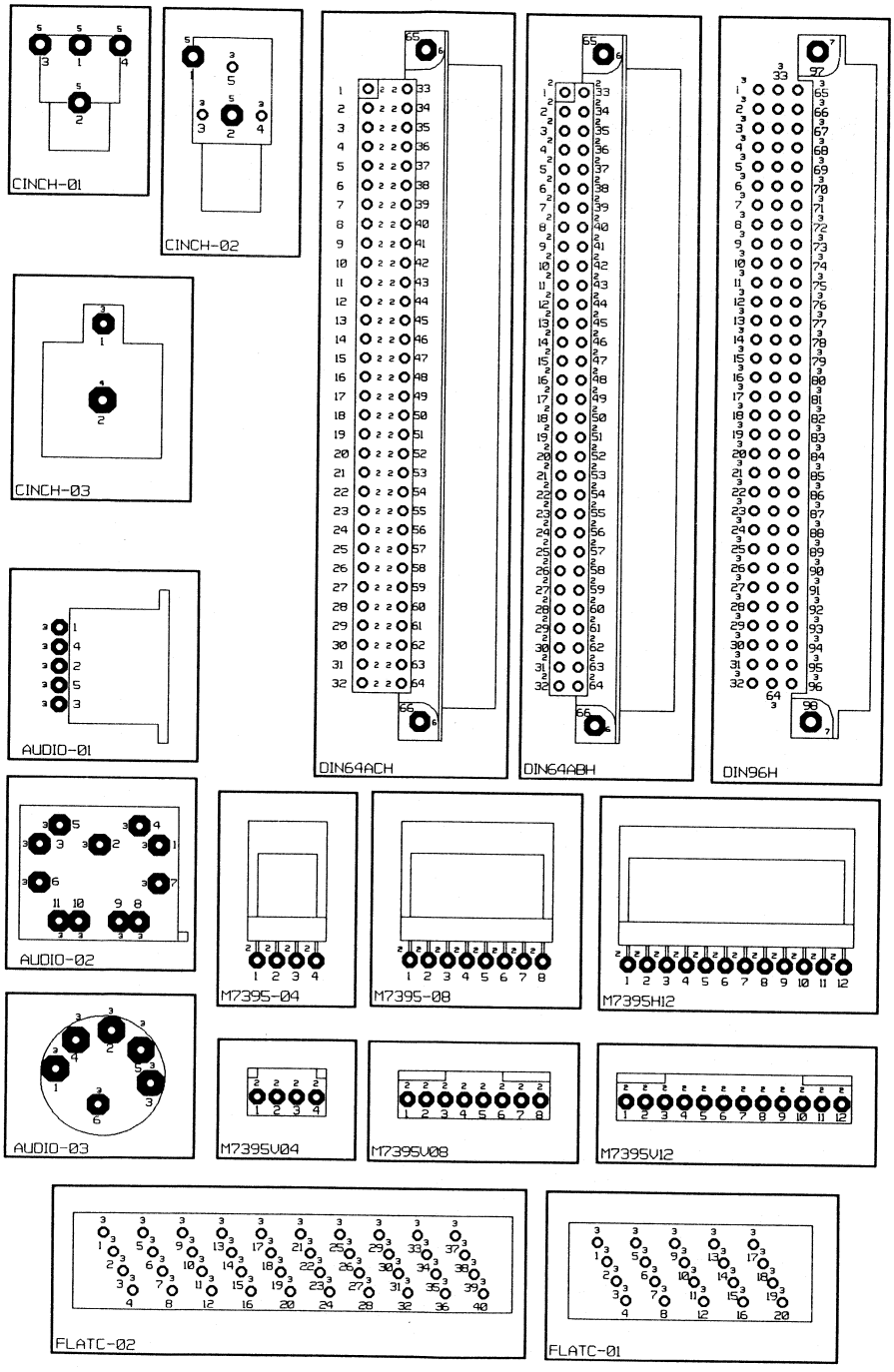

25 23 21 19 17 15 13 11 9 7 5 3 1
27
26 24 22 20 18 16 14 12 10 8 6 4 2
28
HEADER04


33 31 29 27 25 23 21 19 17 15 13 11 9 7 5 3 1
35
34 32 30 28 26 24 22 20 18 16 14 12 10 8 6 4 2
36
HEADER05

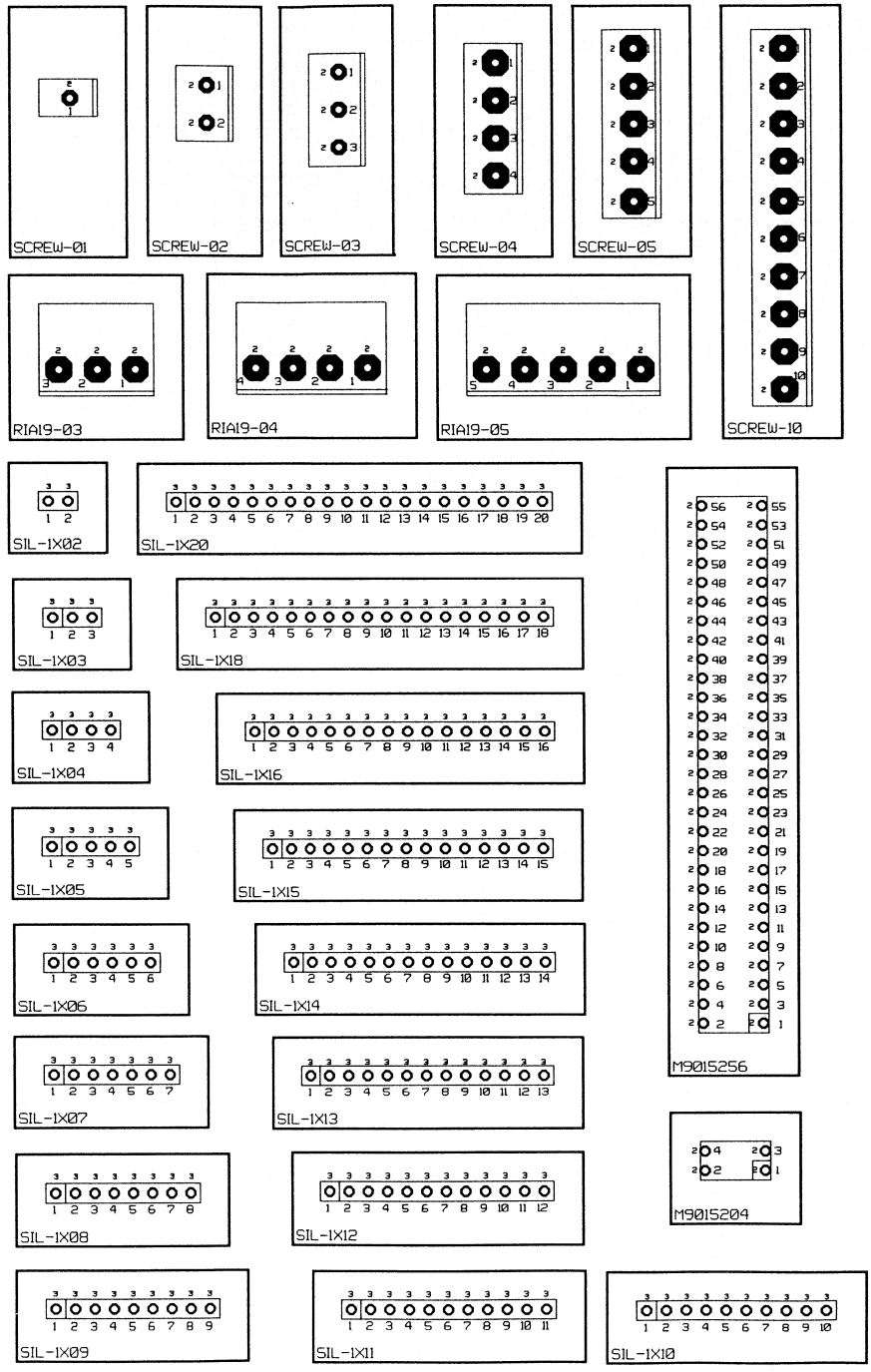

39 37 35 33 31 29 27 25 23 21 19 17 15 13 11 9 7 5 3 1
41
40 38 36 34 32 30 28 26 24 22 20 18 16 14 12 10 8 6 4 2
42
HEADER06


49 47 45 43 41 39 37 35 33 31 29 27 25 23 21 19 17 15 13 11 9 7 5 3 1
51
50 48 46 44 42 40 38 36 34 32 30 28 26 24 22 20 18 16 14 12 10 8 6 4 2
52
HEADER07

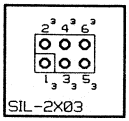
connect-01

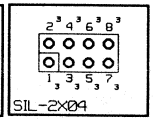

CINCH-01
CINCH-02
CINCH-03
AUDIO-01
AUDIO-02
AUDIO-03
DIN64ACH
DIN64ABH
DIN96H
M7395-04
M7395-08
M7395H12
M7395V04
M7395V08
M7395V12
FLATC-02
FLATC-01

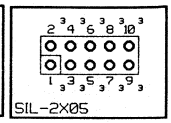
connect-02

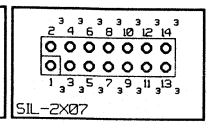

SCREW-01
SCREW-02
SCREW-03
SCREW-04
SCREW-05
SCREW-10
RIA19-03
RIA19-04
RIA19-05
SIL-1X02
SIL-1X20
SIL-1X03
SIL-1X18
SIL-1X04
SIL-1X16
SIL-1X05
SIL-1X15
SIL-1X06
SIL-1X14
SIL-1X07
SIL-1X13
SIL-1X08
SIL-1X12
SIL-1X09
SIL-1X11
SIL-1X10
M9015256
M9015204

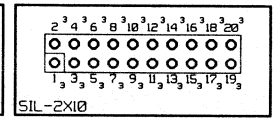
connect-03

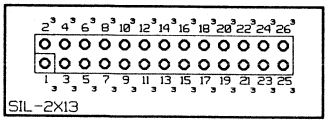

SIL-2X03

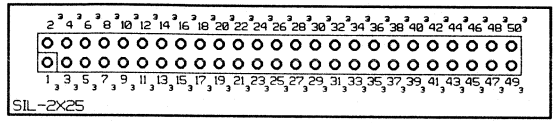

SIL-2X04

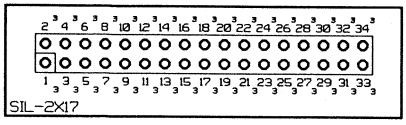

SIL-2X05

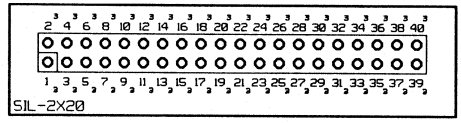

SIL-2X07

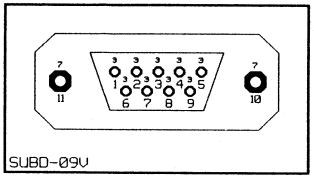

SIL-2X10

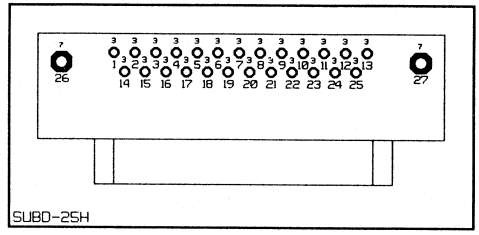

SIL-2X13

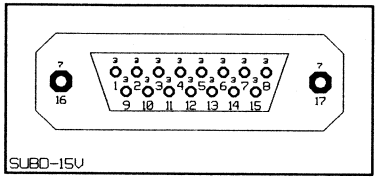

SIL-2X25

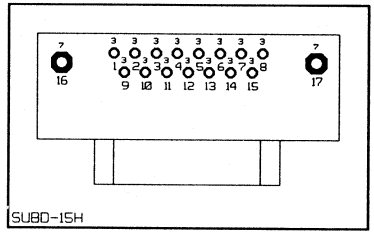

SIL-2X17

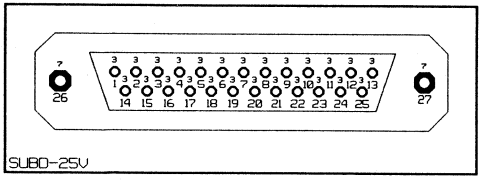

SIL-2X20

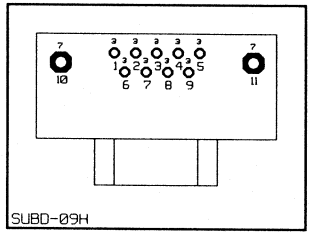

SUBD-09V

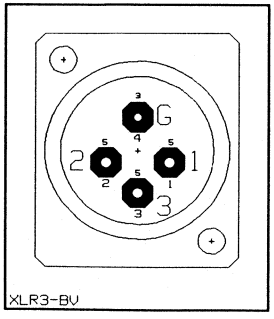

SUBD-25H

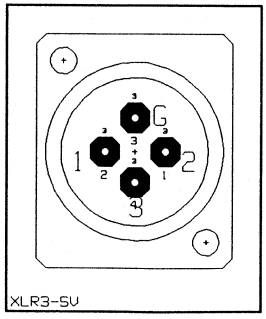

SUBD-15V

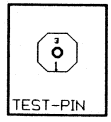

SUBD-15H


SUBD-25V

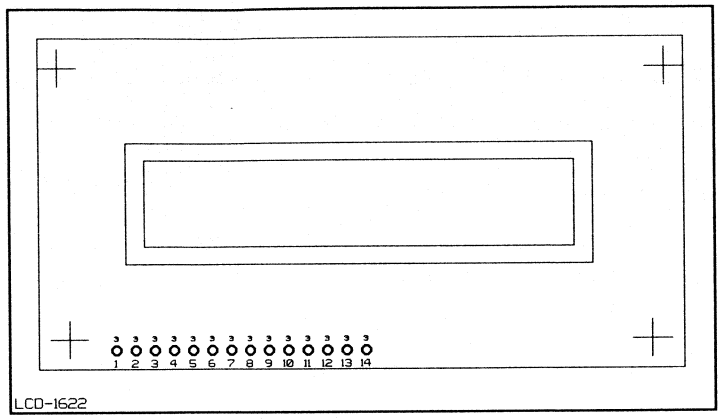

SUBD-09H

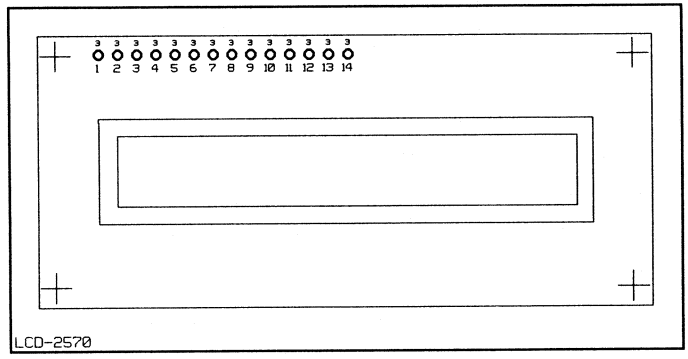

XLR3-8V

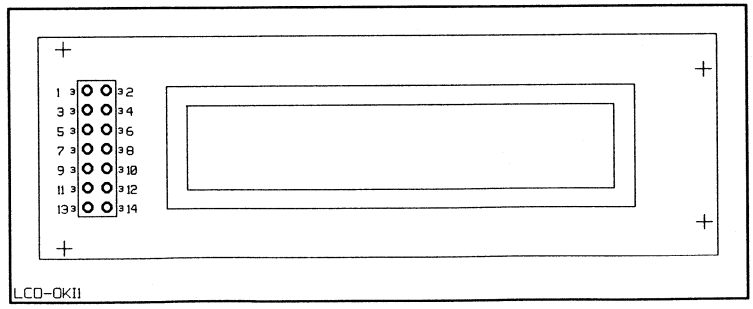

XLR3-5V


TEST-PIN

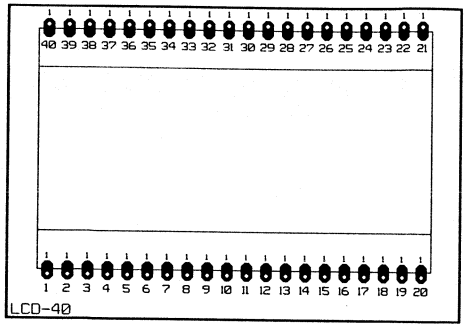
connect-04

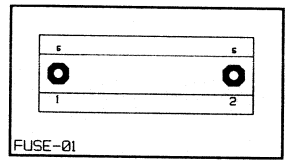

1 2 3 4 5 6 7 8 9 10 11 12 13 14
LCD-1622

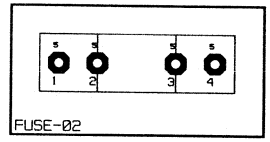

1 2 3 4 5 6 7 8 9 10 11 12 13 14
LCD-2570

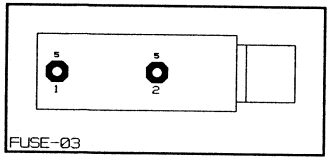

1 2
3 4
5 6
7 8
9 10
11 12
13 14
LCD-OKI1

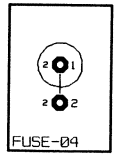
devices-01

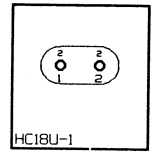

40 39 38 37 36 35 34 33 32 31 30 29 28 27 26 25 24 23 22 21
1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19 20
LCD-40


FUSE-01

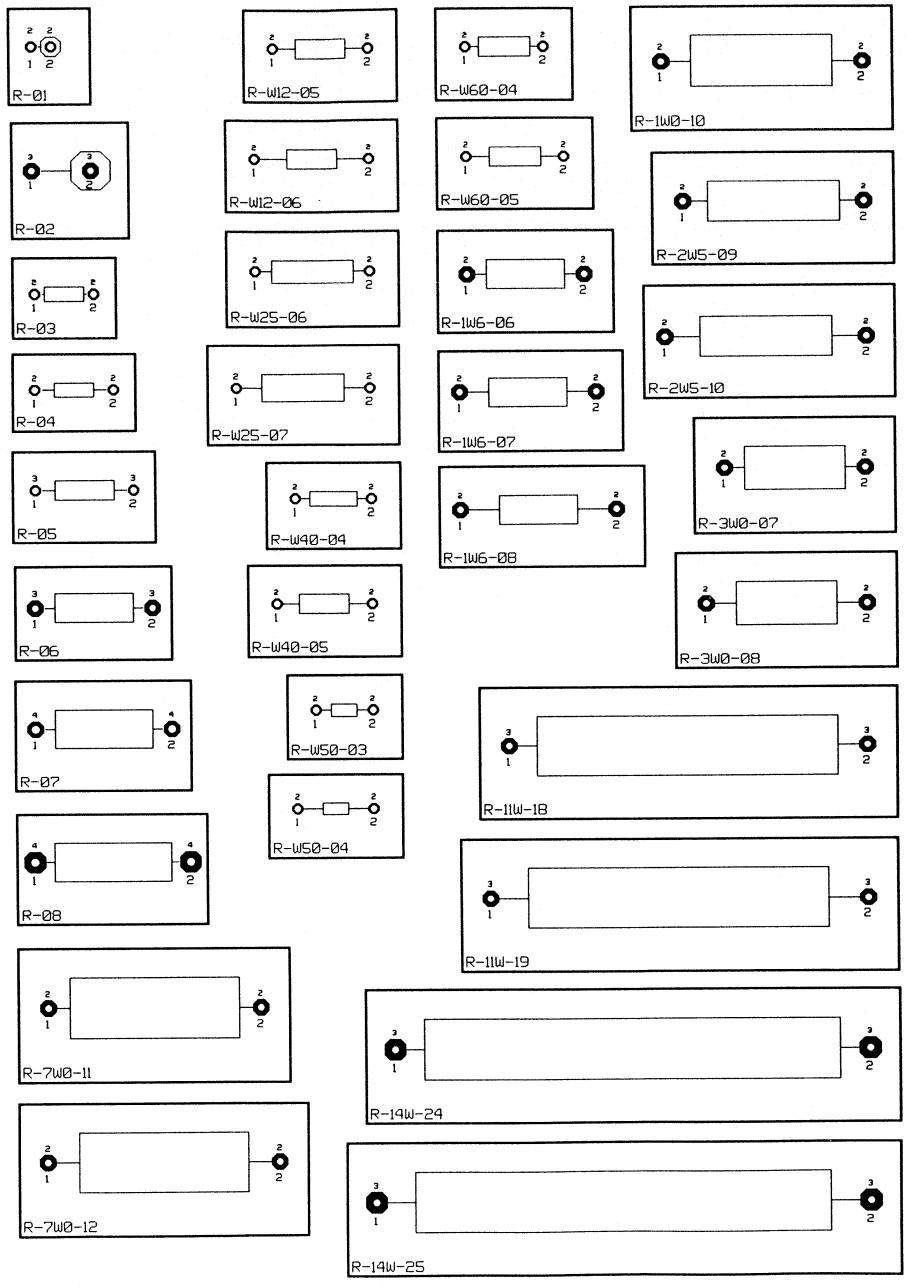

FUSE-02


FUSE-03


FUSE-04


HC18U-1

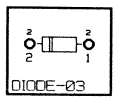
devices-02

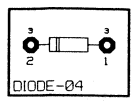

R-01
R-02
R-03
R-04
R-05
R-06
R-07
R-08
R-7W0-11
R-7W0-12
R-W12-05
R-W12-06
R-W25-06
R-W25-07
R-W40-04
R-W40-05
R-W50-03
R-W50-04
R-W60-04
R-W60-05
R-1W6-06
R-1W6-07
R-1W6-08
R-1W0-10
R-2W5-09
R-2W5-10
R-3W0-07
R-3W0-08
R-11W-18
R-11W-19
R-14W-24
R-14W-25

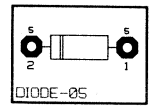
generic-01

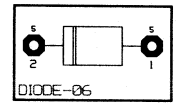
DIODE-01

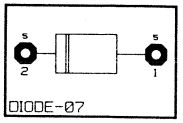
DIODE-02

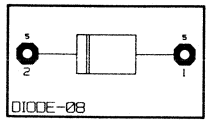

DIODE-03

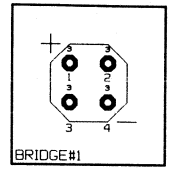

DIODE-04

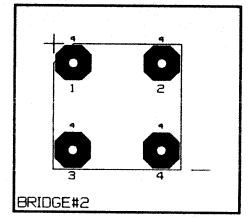

DIODE-05

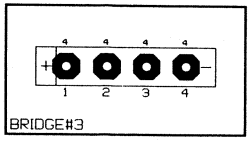

DIODE-06

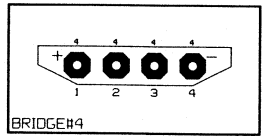

DIODE-07

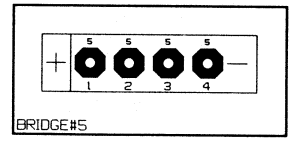

DIODE-08

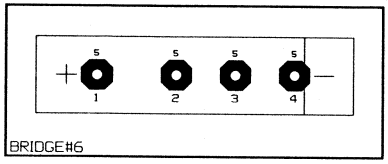

BRIDGE#1

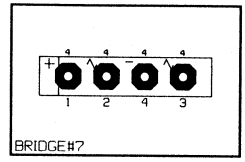

BRIDGE#2

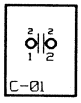

BRIDGE#3

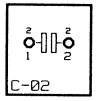

BRIDGE#4

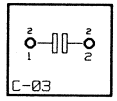

BRIDGE#5

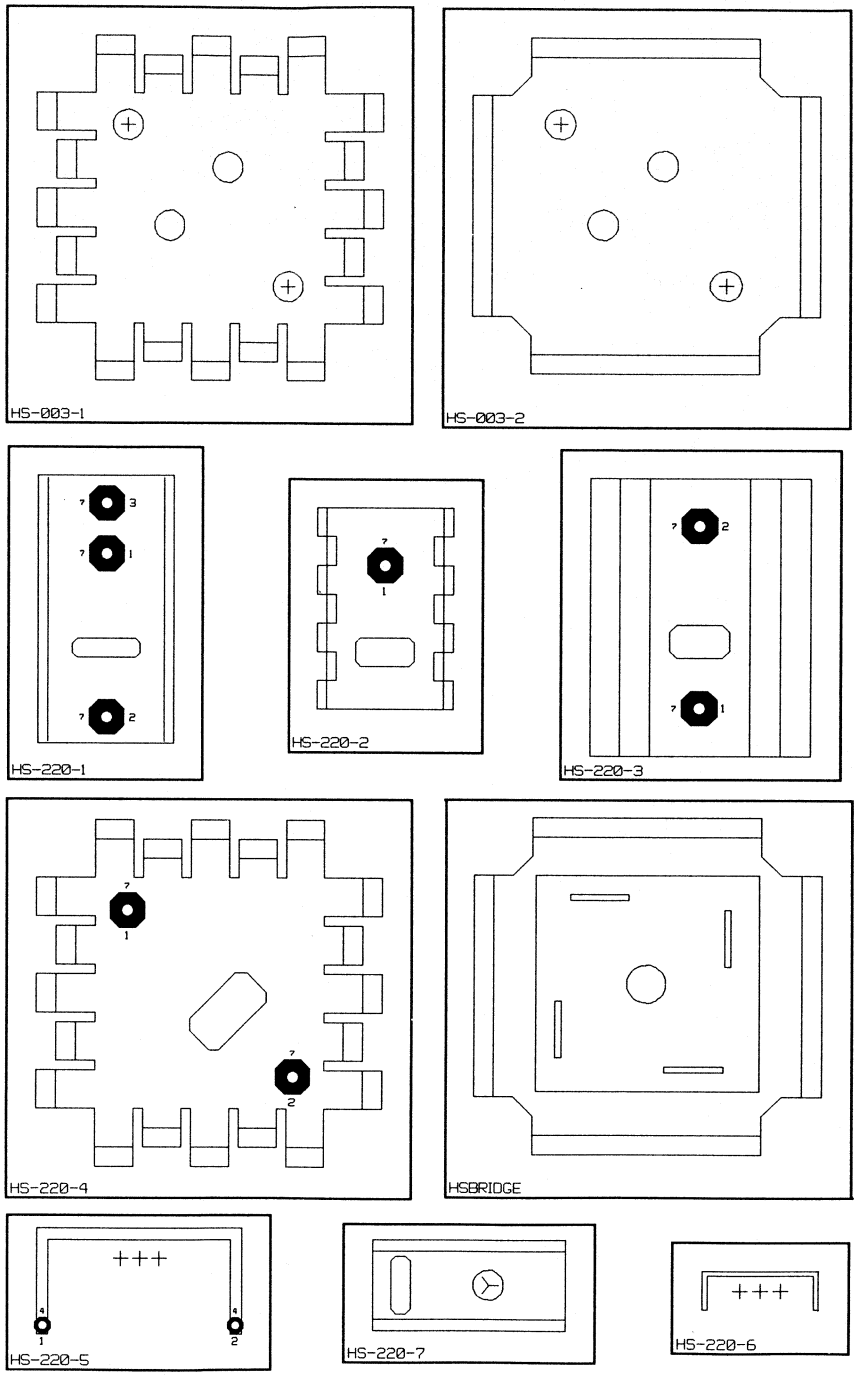

BRIDGE#6


BRIDGE#7

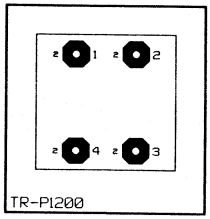

C-01

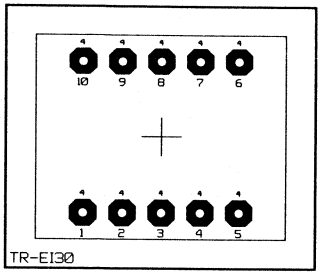

C-02

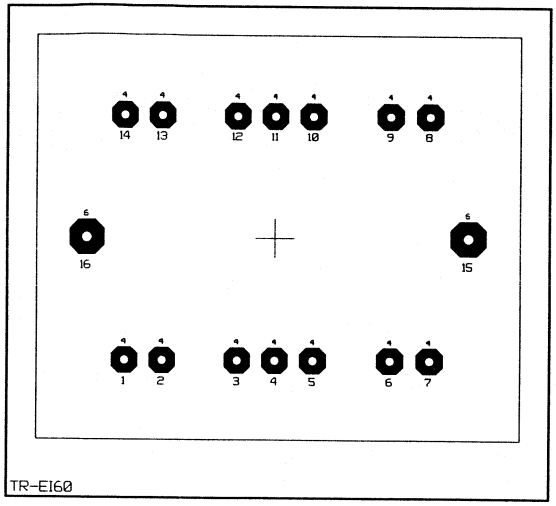

C-03

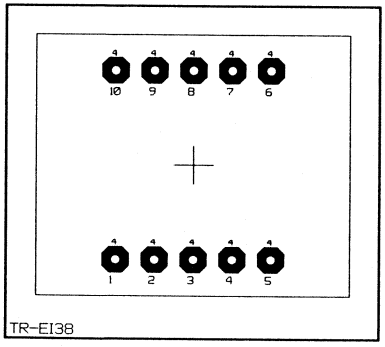

HS-003-1
HS-003-2
HS-220-1
HS-220-2
HS-220-3
HS-220-4
HSBRIDGE
HS-220-5
HS-220-7
HS-220-6

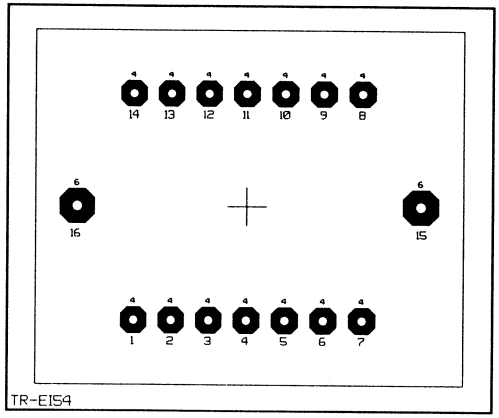
heatsink-01

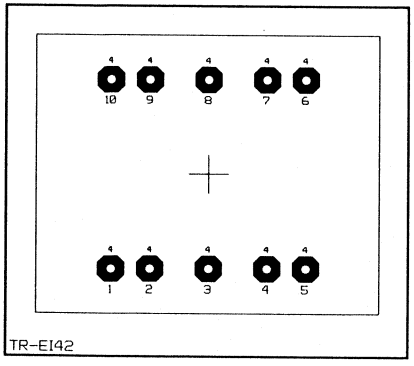


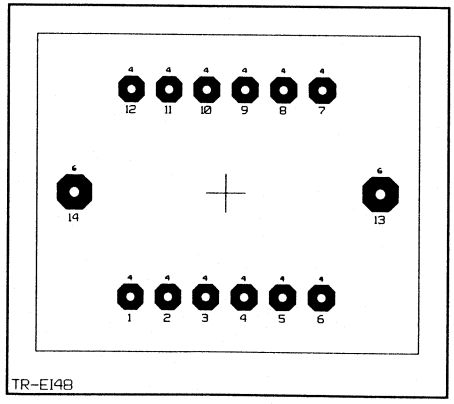




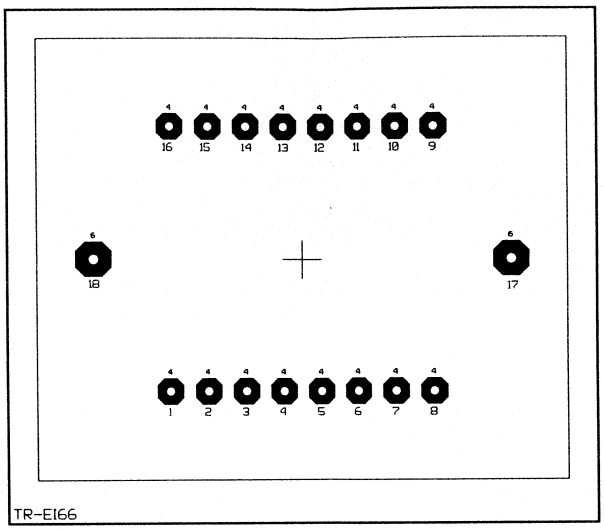


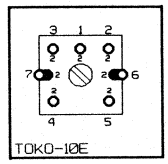


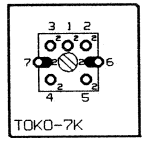


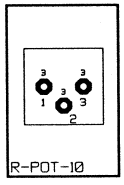


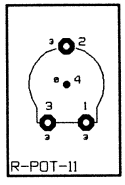
passive-01

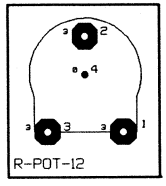

16 15 14 13 12 11 10 9
18
17
1 2 3 4 5 6 7 8
TR-E166

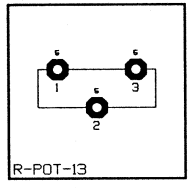

TOKO-10E

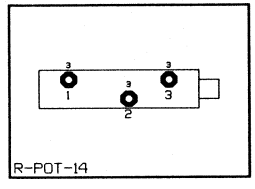

TOKO-7K

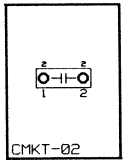

R-POT-10

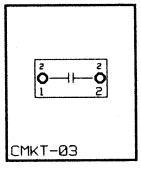

R-POT-11

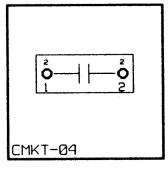

R-POT-12

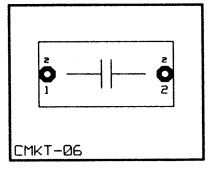

R-POT-13

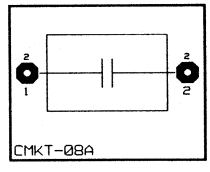

R-POT-14

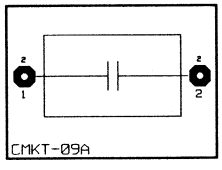

CMKT-02

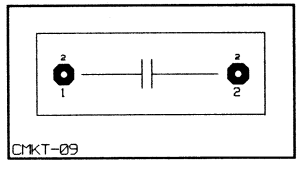

CMKT-03

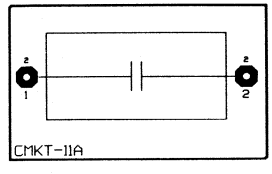

CMKT-04

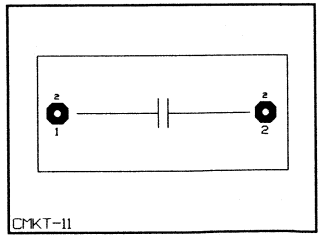

CMKT-06

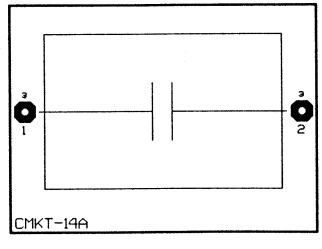

CMKT-08A


CMKT-09A

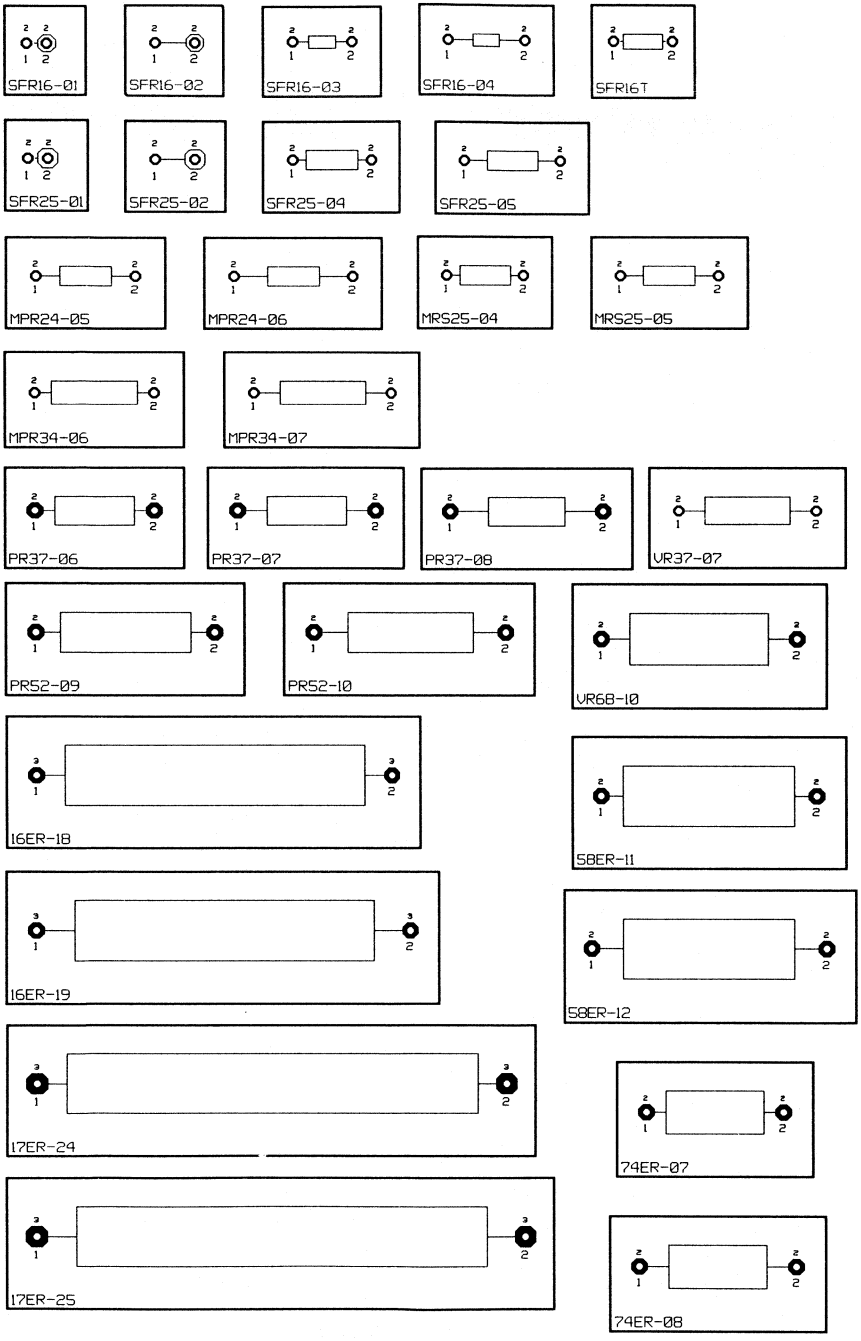

CMKT-09


CMKT-11A

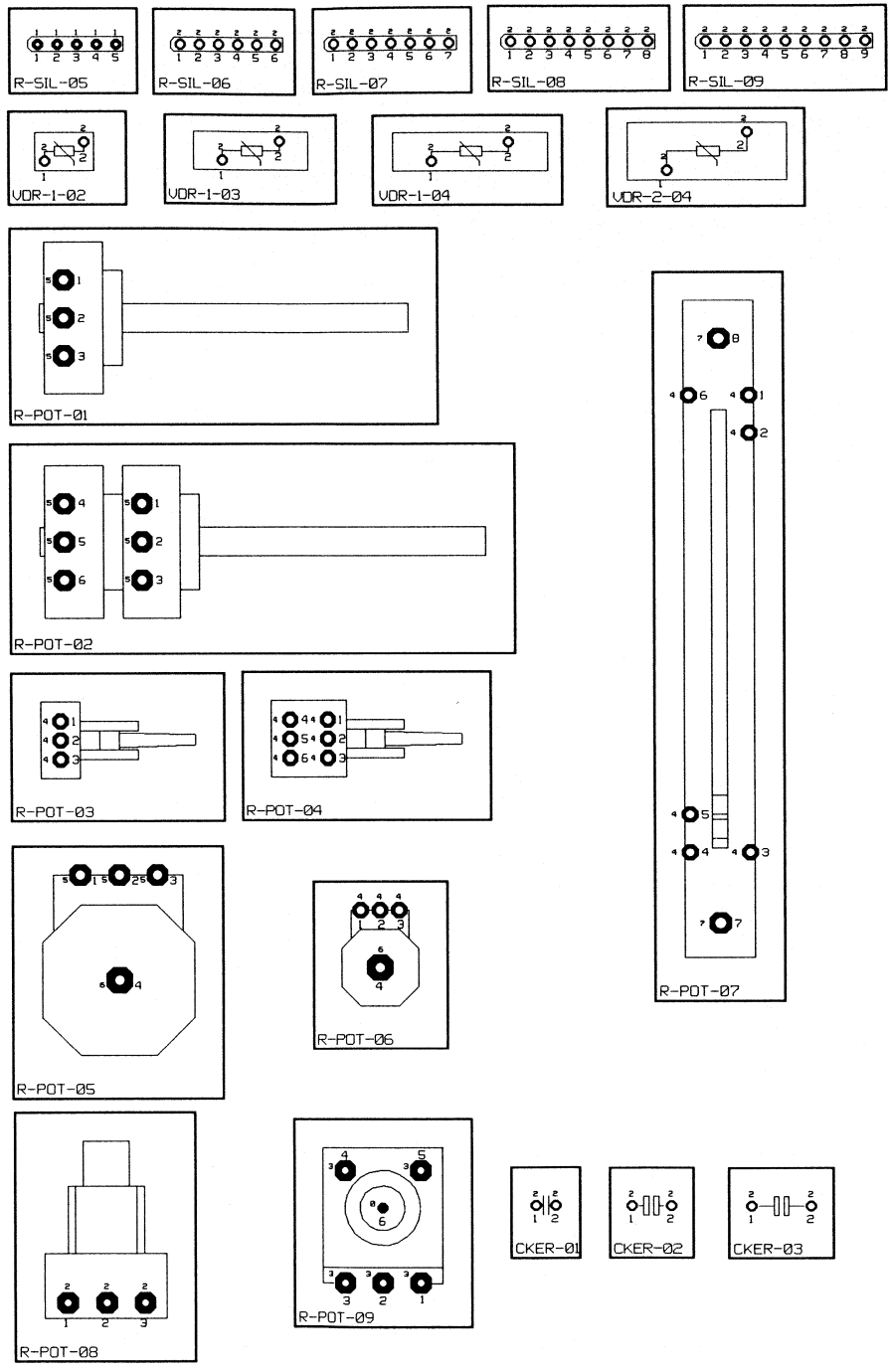

CMKT-11


CMKT-14A

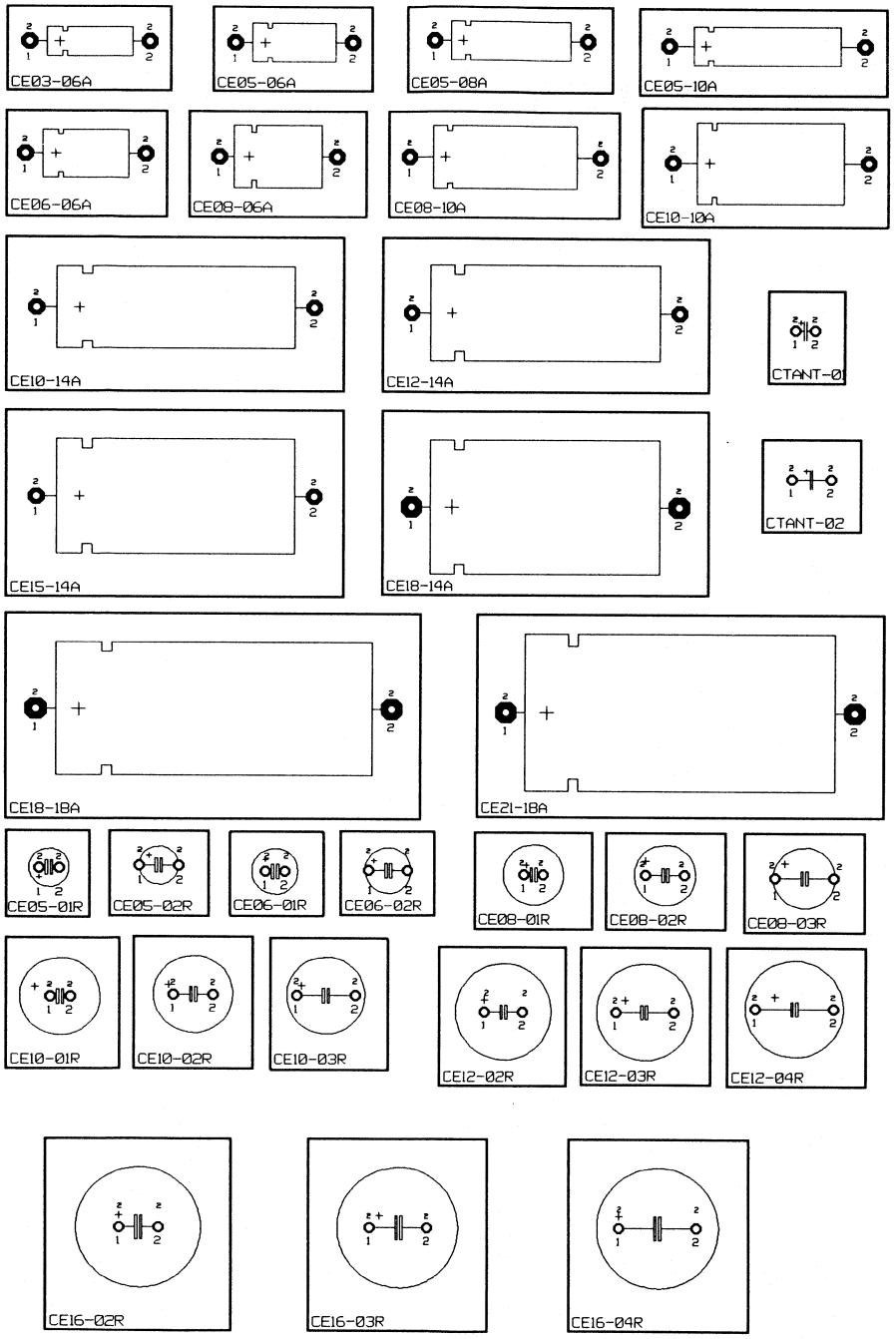
passive-02


SFR16-01
SFR16-02
SFR16-03
SFR16-04
SFR16T
SFR25-01
SFR25-02
SFR25-04
SFR25-05
MPR24-05
MPR24-06
MRS25-04
MRS25-05
MPR34-06
MPR34-07
PR37-06
PR37-07
PR37-08
VR37-07
PR52-09
PR52-10
VR68-10
16ER-18
58ER-11
16ER-19
58ER-12
17ER-24
74ER-07
17ER-25
74ER-08

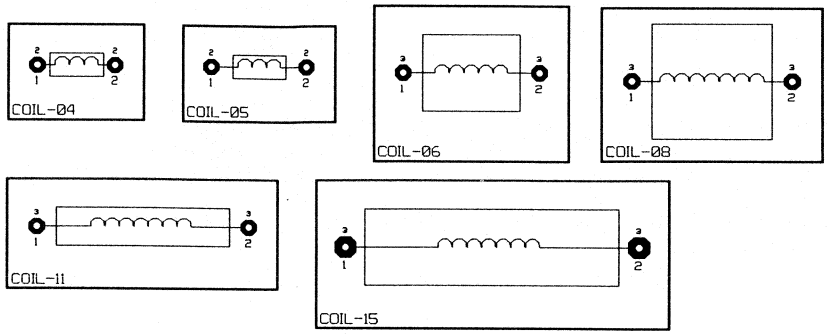
passive-03


R-SIL-05
R-SIL-06
R-SIL-07
R-SIL-08
R-SIL-09
VDR-1-02
VDR-1-03
VDR-1-04
VDR-2-04
R-POT-01
R-POT-02
R-POT-03
R-POT-04
R-POT-07
R-POT-05
R-POT-06
R-POT-08
R-POT-09
CKER-01
CKER-02
CKER-03

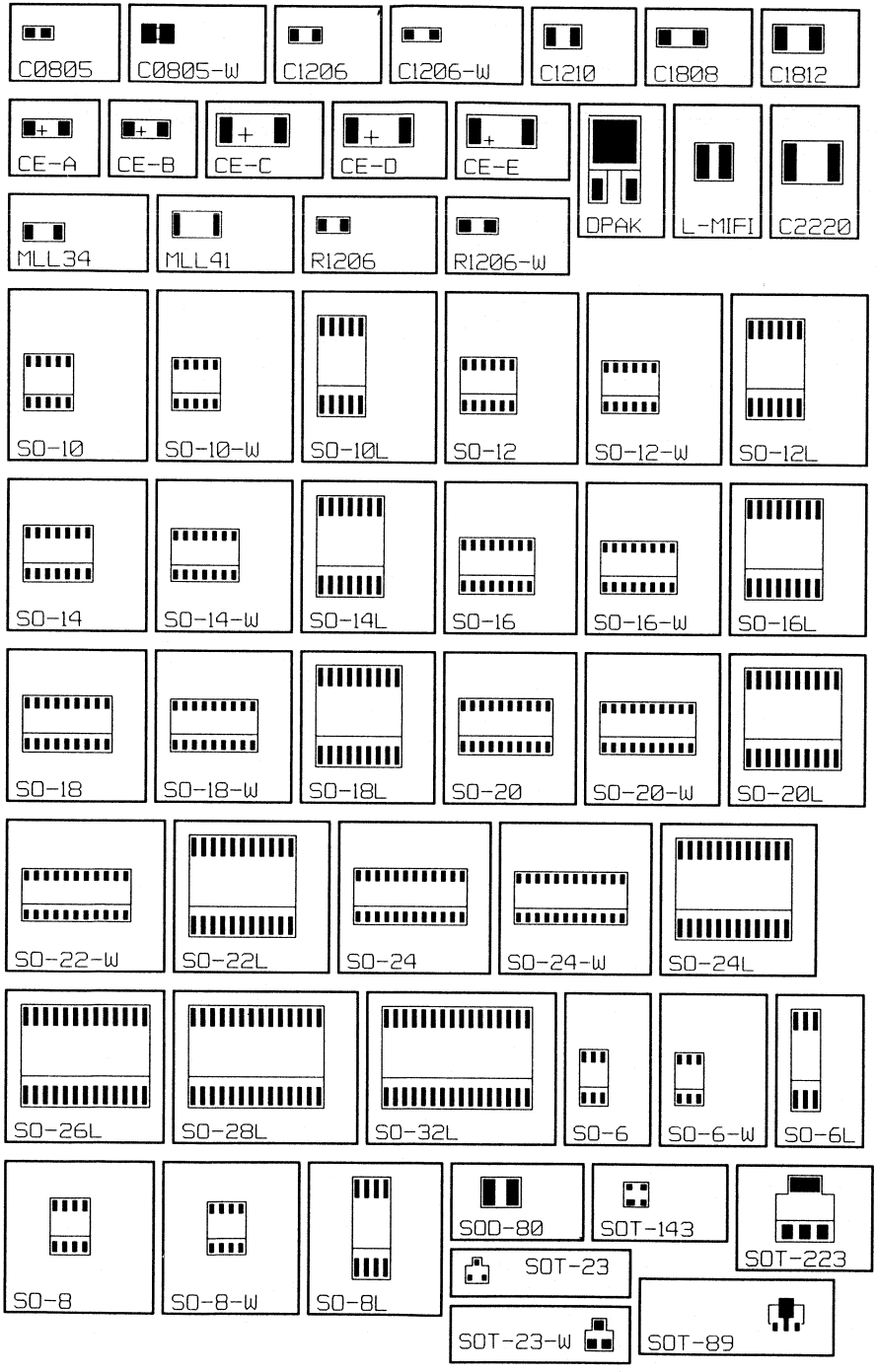
passive-04


CE03-06A
CE05-06A
CE05-08A
CE05-10A
CE06-06A
CE08-06A
CE08-10A
CE10-10A
CE10-14A
CE12-14A
CTANT-01
CE15-14A
CE18-14A
CTANT-02
CE18-18A
CE21-18A
CE05-01R
CE05-02R
CE06-01R
CE06-02R
CE08-01R
CE08-02R
CE08-03R
CE10-01R
CE10-02R
CE10-03R
CE12-02R
CE12-03R
CE12-04R
CE16-02R
CE16-03R
CE16-04R

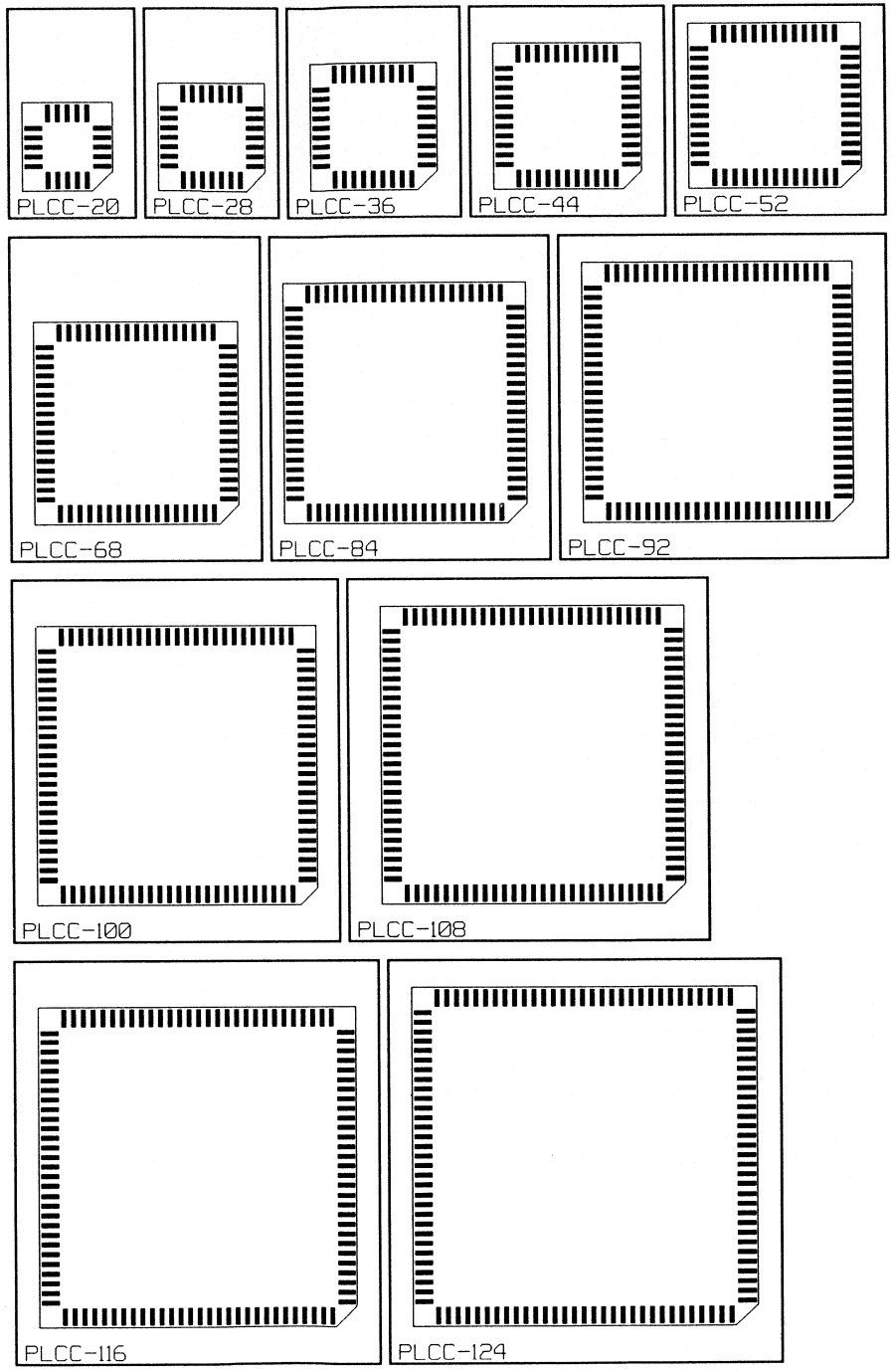
passive-05



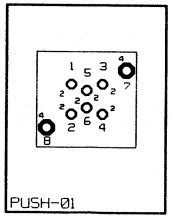
passive-06

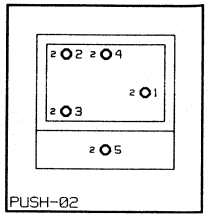

C0805
C0805-W
C1206
C1206-W
C1210
C1808
C1812
CE-A
CE-B
CE-C
CE-D
CE-E
DPAK
L-MIFI
C2220
MLL34
MLL41
R1206
R1206-W
SO-10
SO-10-W
SO-10L
SO-12
SO-12-W
SO-12L
SO-14
SO-14-W
SO-14L
SO-16
SO-16-W
SO-16L
SO-18
SO-18-W
SO-18L
SO-20
SO-20-W
SO-20L
SO-22-W
SO-22L
SO-24
SO-24-W
SO-24L
SO-26L
SO-28L
SO-32L
SO-6
SO-6-W
SO-6L
SO-8
SO-8-W
SO-8L
SOD-80
SOT-143
SOT-223
SOT-23
SOT-23-W
SOT-89

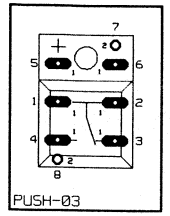
smd-01

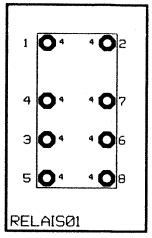

PLCC-20
PLCC-28
PLCC-36
PLCC-44
PLCC-52
PLCC-68
PLCC-84
PLCC-92
PLCC-100
PLCC-108
PLCC-116
PLCC-124

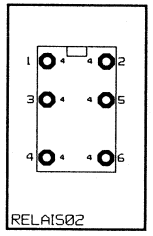
smd-02

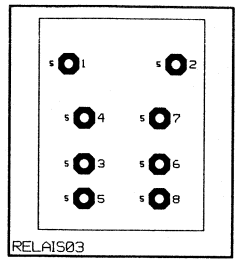

PUSH-01

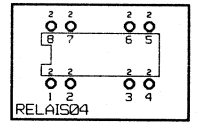

PUSH-02

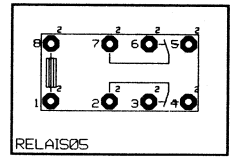

PUSH-03

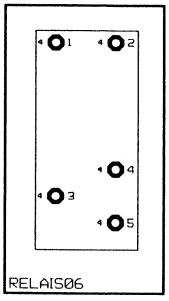

RELAIS01

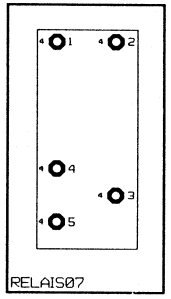

RELAIS02

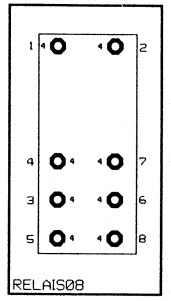

RELAIS03

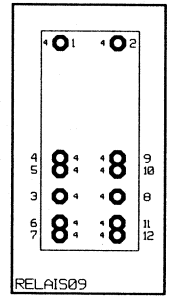

RELAIS04

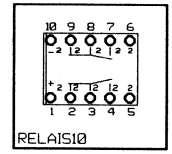

RELAIS05

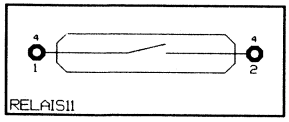

RELAIS06

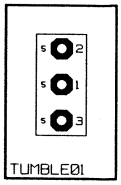

RELAIS07

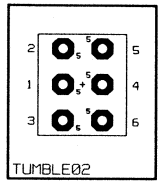

RELAIS08

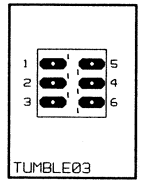

RELAIS09

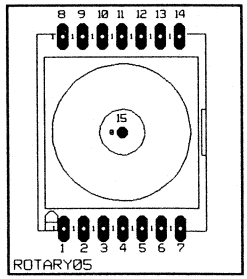

RELAIS10

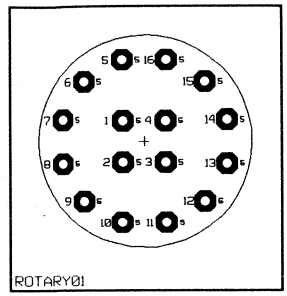

RELAIS11

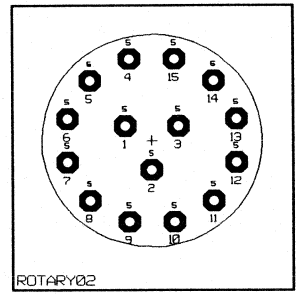

TUMBLE01

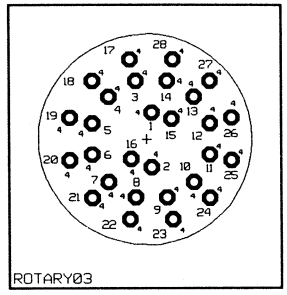

TUMBLE02

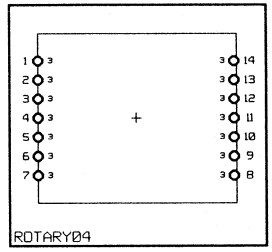

TUMBLE03


8 9 10 11 12 13 14
15
1 2 3 4 5 6 7
ROTARY05


ROTARY01


ROTARY02


ROTARY03


ROTARY04

switches-02

Notes

.CMP 2-11
.CNF 2-10
.LIB 2-13
.LMC 2-9, 3-47
  save as 3-85
.LML 2-10
  create 3-33
.LPU 2-13
.NET 2-11
.PLY 2-10, 3-47
.SMB 2-12
{P1..P2} 3-18
  manipulate 3-52
} 5-5, 5-6
$E 5-7
$M 5-6
+ 5-10
* 5-7
% 5-5
# 5-5
& 5-5
@A 5-9
@E 5-7
@P 5-8
45o routing 3-80

A

ActualGroup 3-44
Align design 3-87
Annotate 2-6
Arguments
  to UT 2-6
attribute
  overwrite 3-93
  Select scale 3-93
Attributes
  Border 5-10
  display 3-3
Auto move 3-4
Auto move delay 3-5
Auto pan 3-5
Auto save time 3-4
Auto tool change 6-9

B

block
  attache point 3-85
  Manipulate 3-50
  Rotate 3-75
  save as .LMC 3-50, 3-85
  swap 3-88
  text editor, text 4-4
Blocknumbers 2-2
Borders
  Project Manager menu 5-10
  text editor 5-10
Bus router 3-80

C

calay
  MGIOS format 2-4
Center 3-6
  pad hole 6-9

C

Pads, open 3-69
Center hole
pads 6-9
Change layer 3-45
Check
connections 3-7
Design rule 3-22
checkplot
comment 3-69
Device 3-68
Output to 3-68
Plotscale 3-69
printer 3-67
prototype-quality 3-68
Repeat 3-69
Color
Layers 3-9
pad 2-3
PM windows 5-6
color mix
font editor 7-8
colors
background 5-12
editor windows 5-7
Netlist pads 3-9
Other 3-9
Project Manager 5-12
text editor 5-12
valid 5-12
Command line
graphics editor 2-7
output driver 6-1
Commands
$E 5-7
font editor 7-3
project manager 5-6
component
attribute text 3-91
banks A-1
datalines 2-2
list 2-4
master directory A-1
New 3-55
place reference 3-92
place shape 3-93
place type 3-92
Reference name 3-92
rotation status 2-11
select attribute scale 3-93
Shape name 3-93
snap 3-87
Type name 3-92
x coord 2-11
y coord 2-11
component elements 2-2
component list
control chars in, 2-12
Configure
output device 6-10
output page 6-5
Conventions 1-3
Keypresses 1-3
Mouse keys 1-3
reference 1-4
Strings 1-3

Copy
{P1..P2} 3-18
line 3-18
pad 3-17
Create
Macro 3-48
Cursor
position, ref PCB 3-71
snap 3-87
Custom pad 3-60
Custom pad design 3-60

D

Dataline
group, first 3-52
group, last 3-52
step next 3-88
step previous 3-88
default tools 3-96
Define
Pad 3-60
Define window 3-98
Delete
editing commands 4-5
file 3-19
layer 3-20
line 3-20
MultiFunc 3-22
object 3-20
Via 3-97
window 3-98
Design
control 2-3
distort 3-87
logo's 7-1
Mirror 3-53
New 3-55
Optimize 3-57
relative density 3-72
rename to .LMC 3-53
Rotate 3-75
save 3-84
scale 3-86
snap 3-86
Device
basics, output 6-11
checkplot 3-68
open, close 6-11
output configuration 6-10
Device basics
output 6-11
Device name 6-11
output 6-11
Directories 3-12
Directory
component A-1
Display
Attributes 3-3
CAM 3-24
Pin numbers 3-67
Ratsnest 3-70
scale 3-99
shape 3-86
text 3-94
Display options 3-24
Draw

D

Box 3-6
Circle 3-7
corners 3-18
line 3-26
text 3-89
DRC 3-22
Drill size
grouping 3-31
Driver type 6-11
output 6-11

E

Edit path 3-37
Editor
Command line 4-3
Exit graphics editor 3-69

F

file
Quit 3-69
Rename 3-72
Reset layo1 3-74
save 3-85
system, Layo1 2-9
File system
.CMP 2-11
.CNF 2-10
.LIB 2-13
.LMC 2-9
.LML 2-10
.LPU 2-13
.NET 2-11
.PLY 2-10
.SMB 2-12
USRXX.CFG 2-13
Files
Recent 3-37
Fill step
output driver 6-9
font 3-89
font editor
Grid 7-3
helplines 7-5, 7-6
kerning 7-5
Mode 7-3
select character 7-2
select set 7-2
set whitespace 7-5
font select 3-89

G

Graphics Editor
Command line 2-7
Grid
define [^F9] 3-43
dots 3-57
font editor 7-3
router 3-79
set [F10] 3-42
set [F9] 3-42
Group
Manipulate 3-52
Rotate 3-75
save as .LMC 3-85

H

Hardcopy 3-67
horizontal router 3-80

I

Input
  output driver 6-6
Input text 3-90
Insert
  editing commands 4-5

L

Layer
  ActualGroup 3-44
  change 3-45
  default 3-30
  display [d] 3-43
  ellipse default 3-36
  font editor 7-2
  Grouping 3-44
  select 3-46, 3-64
  swap block 3-88
  text 3-90
Layo1 directory 3-37
Layo1plt 6-1
Lee router 3-80
Libraries 3-13
Line
  Find 3-29
  Manipulate 3-51
  Rotate 3-75
  save as .LMC 3-52, 3-85
Load
  component 3-47
  file 3-48
  text file 4-4

M

Macro
  nesting 3-49
Macro create 3-48
Main menu 3-49
Manipulate
  {P1..P2} 3-52
  block 3-50
  group 3-52
  line 3-51
  window 3-51
Max len error 3-81
Max xy error 3-81
Menu
  Main 3-49
  MultiFunc 3-54
  Options 3-58
  output, output driver 6-2
  Pad 3-59
  toolbox 3-94
  user xx 3-96
Menu choices
  Draw Menu 3-40
Menu page, example 5-12
Mirror
  design 3-53
  output 6-9
  SMD 3-89

M

SMD pads 3-64
Mode
font editor 7-3
mouse
default sensitivity 3-54
sensitivity 3-53
Mouse keys
conventions 1-3
using 1-3
move
attribute text 3-93
MultiFunc
menu 3-54

N

Netlist 2-4
CALAY 2-4
control chars in, 2-12
New
component 3-55
design 3-55
New component
name 3-56
Node
add 3-34
definition 3-34
delete 3-34
delete, font edit 7-8
Insert 3-28
insert, font edit 7-7
last 3-28
next 3-29
previous 3-28, 3-29
step 3-88
transfer 3-34

O

object
attache point 3-85
Rotate 3-75
Offset
design 3-56
output x,y 6-8
x, y 3-56
Optimize
optimize45 3-57
Sq45 opt 1/40 3-57
Sq45 opt 1/80 3-57
traces 3-57
Options
menu 3-58
text 3-91
text, search/replace 4-6
Origin 3-59, 7-8
ruler 3-83
Other directory 3-37
output
Auto tool change 6-9
Center hole 6-9
Configuration page 6-5
Device basics 6-11
Device name 6-11
Drill program 6-4
Driver type 6-11
Fill step 6-9

O

format 6-5
Graphics 6-6
Input 6-6
input/output window 6-6
output 6-7
output driver 6-7
Plot layers 6-3
Plot select 6-3
Plot tools 6-4
Plotscale 6-8
Port mode 6-11
Size correction 6-9
Start 6-6
Output menu 6-2
overview
Appendix 1-3
manual contents 1-1

P

pad
Adj 3-63
attach point 3-64
circle 3-62
currently active 3-64
custom 3-60
default 3-62
define 3-60
dummy 3-62
menu 3-59
octagon 3-62
open center 3-69
pad number 3-62
Pad to pad router 3-80
palette 3-65
place 3-63
rectangle 3-62
replace 3-64
Route 3-77
select 3-65
Set size 3-62
set-number 3-62
Size correction 6-9
SMD 3-62
PCB
Reference 3-71
Pen
font editor 7-2
minimum width 3-66
select 3-65
text 3-91
up/down, font edit 7-8
width 3-66
Penwidth 3-66
Pin numbers 3-67
place pad 3-63
Plot comment 3-69
Plotscale 3-69
PM.BAT 5-3
PM.DAT 5-4
Port mode
output 6-11
Printer checkplot 3-67
Printer settings 3-15
program
Reset 3-74

Program language 3-10
Project Editor command
  + 5-10
  @A 5-9
  @P 5-8
Project Manager
  customizing 2-5
Project Manager command
  } 5-5, 5-6
  $E 5-7
  $M 5-6
  * 5-7
  % 5-5
  # 5-5
  & 5-5
  @E 5-7
  order 5-7

Q

Quit 3-69

R

Ratsnest 3-70
  check 3-71
  component 3-70
  design 3-70
  Reset 3-70, 3-71
  rubberband 3-83
  trace 3-70
Recent files 3-37
Redraw screen 3-71
Reference name 3-92
Reference PCB 3-71
Rename 3-72
Rename file 3-72
Replace
  editing command 4-6
  options 4-6
replace pad 3-64
Reroute 3-73
Reroute all 3-73
Reroute net 3-73
Reroute track 3-74
Reset program 3-74
rotate
  90-degree 3-75
  design 3-75
  output 6-9
Rotate object 3-75
Route all 3-76
Route len test 3-81
Route manual 3-77
Route net 3-77
Route pad 3-77
Route window 3-76
router
  45o routing 3-80
  Bus 3-80
  constraints 3-76, 3-77
  Grid size 3-79
  Horizontal 3-80
  Lee 3-80
  length ordered 3-81
  Max xy error 3-81
  menu 3-81
  netlist ordered 3-81

R

Pad to pad 3-80
Reroute all 3-73
Reroute net 3-73
Reroute track 3-74
Route all 3-76
Route len test 3-81
Route manual 3-77
Route net 3-77
Route pad 3-77
Route window 3-76
Router setup 3-78
strategy 3-81
to traces 3-81
Vertical 3-81
Via router 3-80
Router menu 3-81
Router setup 3-78
Rubberband 3-83
real-time 3-83
Ruler
imperial 3-42
metric 3-42
origin 3-83
select, font editor 7-9

S

save
.CMP 3-70
.CNF. 3-84
.LML 3-84
.NET 3-70
.PLY 3-84
.SMB 3-67, 3-70
as .LMC 3-50, 3-51, 3-52, 3-85
at exit 3-70
current settings 3-84
design 3-84
file 3-85
penwidth 3-67
text file 4-4
timed 3-84
user configuration 3-84
Save/exit 3-84
scale
attribute, select 3-93
components, different 3-86
design 3-86
display 3-99
output 6-8
text 3-91
Schematic capture 2-4
Screen
text editor 4-1
Screen redraw 3-71
Search
editing command 4-6
options 4-6
Select
Block 3-5
Select layer 3-46
select pad 3-65
Select scale 3-93
Select tool 3-95
Sensitivity
Mouse 3-53

setup
  Router 3-78
Shape 3-86
Shape display 3-86
Shape name 3-93
Size correction 6-9
SMD pads 3-62
Snap component 3-87
Snap cursor 3-87
Snap design 3-86
Step nodes 3-88
strategy
  Router 3-81
string
  draw 3-89
Strings
  entering 1-3
subdraws elements 2-3
Swap block 3-88

T

Text
  attribute tips 3-93
  attributes 3-91
  default layer 3-94
  display 3-94
  display toggle 3-94
  draw 3-89
  Enlarge 3-90
  font 3-89
  input text 3-90
  layer 3-90
  Mirror 3-90
  move 3-93
  options 3-91
  overwrite, attribute 3-93
  pen 3-91
  Rotate 3-90
  scale 3-91
  select layer 3-90
  select pen 3-91
  Shrink 3-90
  text objects 3-89, 3-91
text editor
  Cmd field 4-2
  Col field 4-3
  colors 5-10
  commands 4-4
  filename field 4-3
  Indent field 4-3
  Insert field 4-3
  Line field 4-2
  Project Editor command 5-7
  Save 4-4
  screen 4-1
  Status line 4-2
  Tab field 4-3
tool
  default 3-96
  select 3-95
Toolbox
  menu 3-94
  scale design 3-86
Toolbox menu 3-94
TOOLS
  separate utilities 2-9

Top Level 3-50
Traces
  Optimize 3-57
Type name 3-92

U

update 2-6
update utility 2-6
User configuration
  save 3-84
User xx menu 3-96
USRXX.CFG 2-13
UT 2-6
Utilities
  toolbox 3-94
utility
  update 2-6

V

Vectors 2-1
  character set 7-1
Vertical router 3-81
Via
  place 3-97
  remove 3-97
  router 3-80
Via router 3-80
Video board 3-11

W

Window
  define 3-98
  delete 3-98
  delete in .PLY 3-98
  electrical significant 3-98
  empty 3-98
  input/output 6-6
  Manipulate 3-51
  Rotate 3-75
  Route 3-76
  save as .LMC 3-51, 3-85

X

Xy offset 3-56

Z

Zoom 3-99
  fill display 3-99
  font editor 7-9

[.] 3-41, 7-7, 3-41
[,] 3-86
[:] 3-71, 3-72, 3-71
['] 3-3
[+] 3-52, 3-79, 3-88, 3-90, 3-91, 3-93, 3-79
[-] 3-52, 3-79, 3-88, 3-90, 3-91, 3-93, 3-79
[**] 3-87
[*] 3-87
[^F1,2,3,4,6] 3-70
[^F1] 3-70, 3-83
[^F2] 3-70
[^F3] 1-3, 3-70

[^F4] 3-70
[^F6] 3-7, 3-71, 3-7
[^F7] 3-46, 3-64, 3-46
[^F8] 3-46, 3-64, 3-46
[^F9] 3-43
[=] 3-29, 3-35, 3-74, 3-35, 3-74
[Alt][key] 3-49
[a] 3-19, 7-2, 7-6, 3-19
[b] 3-5, 3-20, 3-50, 3-5, 3-20, 3-50
[c] 3-6, 7-5, 3-6
[Del] 3-20, 3-21, 7-8, 3-20
[d] 3-43
[F10] 3-42, 3-43, 7-7, 3-42
[F1] 3-6, 3-44, 3-75, 7-7
[F2] 3-52, 3-75, 3-90, 3-97, 7-5, 3-52, 3-75
[F3] 3-47, 3-48, 7-6, 3-47
[F4] 3-39, 3-56, 3-84, 3-85, 7-7, 3-84, 3-85
[F5] 3-70, 3-71, 6-5, 7-8, 3-70, 3-71
[F6] 3-59, 3-83, 7-8, 3-59, 3-83
[F7] 3-20, 3-37, 3-46, 3-52, 3-64, 6-3, 6-4, 7-4, 3-52
[F8] 3-20, 3-37, 3-44, 3-46, 3-52, 3-64, 7-4, 3-52
[F9] 3-37, 3-42, 3-43, 3-44, 6-5, 6-9, 7-7, 3-42
[f] 1-3, 3-40, 3-41, 3-40
[Ins] 3-17, 3-78, 3-90, 7-7, 3-78
[i] 3-42
[l] 3-46, 3-90, 3-46
[m] 3-42
[n] 3-67
[p] 3-17, 3-65, 3-91, 7-8, 3-65
[q] 3-23, 7-5, 3-23
[r] 3-83, 7-9, 3-83
[s] 3-63, 3-64, 3-65, 5-7, 3-63, 3-65
[t] 3-95, 7-6
[u] 3-87
[w] 3-76
[x] 3-15, 3-56, 7-9, 3-15
[y] 3-15, 3-56, 7-9, 3-15
[zz] 3-99
[z] 3-99, 7-9
[↑>] 3-41
[↑F2] 3-88, 3-90
[↑F5] 3-8, 7-8, 3-8
[↑F9] 3-42