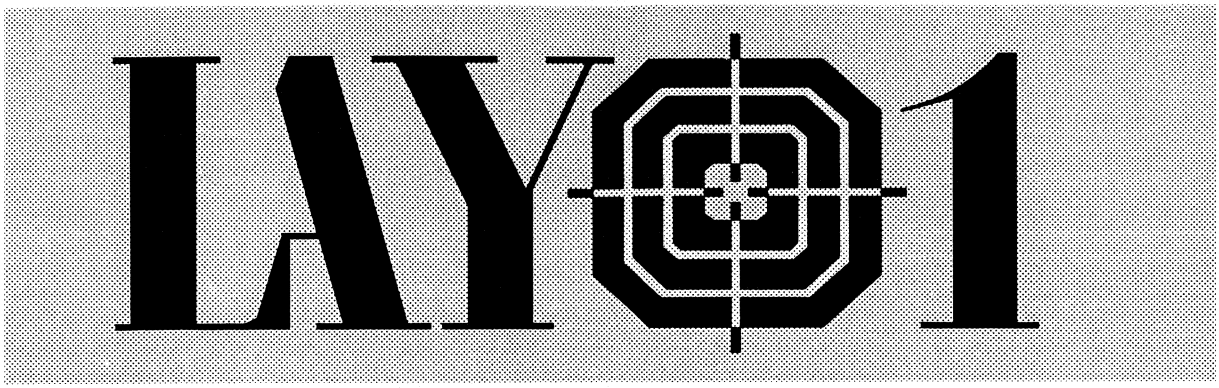


User Guide

Layo1 PCB CAD system

Layo1 Level 4

Registratie Nummer:

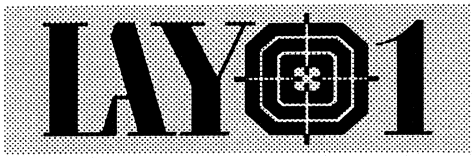
I403115322

Manual version 4 of 1/92. Manual layout and design: JamaSys Electronics, Po Box 223, NL-4760 AE Zevenbergen, The Netherlands.

Layo1 is a product of Baas Electronics BV, Rijksstraatweg 42, NL-3281 LW Numansdorp, Netherlands. Tel (+31) 1865-4211, fax (+31) 1865-3480.

C O N T E N T S

List of figures



CHAPTER 1

Introduction

This User Guide is designed to introduce you to Layo1, install the package on your computer and to get you started using it. You will find several chapters explaining how to make the best use of Layo1, including tutorials. This guide contains the following chapters:

Chapter 1 The chapter you are reading. Discusses the contents of the Guide, and introduces you to Layo1.

Chapter 2 This chapter guides you through the install procedure to get a running copy of Layo1 and its support programs on your computer. Although the install process itself is automatic, you have to make a few basic decisions and enter a few parameters. This chapter also discusses how to install Layo1 to work integrated with a schematic capture program from OrCAD.

Chapter 3 Chapter 3 discusses the use of the Project Manager in controlling and managing yourPCB design projects. If you are new to Layo1 it will get you started using the package. First we will introduce you to the Project Manager. If you have integrated OrCAD, we will show how to use Schematic Diagram data to set up Layo1.

The chapter contains a short tutorial to design a Layo1 component. Finally, the last section will tell you how to use a macro to store frequently used keystroke sequences under an **[Alt][key]** combination. This is not required reading, but sooner or later you will want to use this feature.

Chapter 4 This chapter is a tutorial that takes you through a complete PCB design cycle. We will use a schematic diagram we have prepared for you. There are two parts to this tutorial. The section titled 'Integrated design' is used if you have integrated Layo1 with OrCAD/SDT. We will use OrCAD to prepare the net- and component lists for Layo1. The section titled Stand-alone design is used if you work with Layo1 stand-alone, loading the components and building the netlist yourself.

After the initial preparations of the various files for Layo1, a single section titled Design layout will take you through the actual design, both for the integrated and for the standalone mode. Both methods lead to the same correct results. Even if you use the integrated method initially, you may want to review the stand-alone tutorial sometime later. It gives you another view of how Layo1 handles nets.

Chapter 5 Chapter 5 contains a small tutorial emphasizing various routing functions and creating a netlist. If you work through it you will get a feeling for the possibilities and aspects of using Layo1's built-in router to route your design for you. In this tutorial you will design a simple dual layer PCB.

Chapter 6 This chapter is a discussion and short tutorial for using the Forward Annotate function, to update your design after schematic design changes. If you use Layo1 integrated with a version of OrCAD/SDT, you can use a built-in function from the Project Manager, and this is what we will discuss. This function will preserve the design integrity.

Chapter 7 This last chapter is a description and tutorial for the separate Output Driver program. The outputdriver is a stand alone program that can convert Layo1 output files to industry standard output files. These files can drive penplotters, photoplotters and NC drilling equipment to generate documentation, layout films and PCB's for a professional end product.

How to use this guide

It is not necessary to work through this guide from the first to the last page before you can start working with Layo1. It is obvious that you should do the installation first. When you are new to Layo1, you should work through Chapter 3. Then you can jump right in by working through Chapter 4, Working with Layo1. If you get the feel for Layo1, you can learn about specific functions doing the tutorials in Chapters 5, 6 and 7. You will probably want to periodically review certain entries in the Reference Guide while you work. Therefor it is a good idea to at least leaf through the Reference Guide to familiarize yourself with its organization. That way, you can find your topics quicker while you work, making the interruptions as short as possible.

Using the Project Manager is not essential for designing PCB's with Layo1. However, Project Manager will help you to maintain a logical flow throughout your work, and also to control the configuration of the design(s) you work on. If you use a schematic capture program to 'feed' Layo1, a customized Project Manager menu system will keep you in tight control of the design status and help you manage changes and additions. Note that at install time you can specify to configure the Project Manager for integration with OrCAD/SDT III or IV. The Reference Guide contains extensive information on how to customize Project Manager for your own environment.

Layo1 history

Back in 1984 our PCB production facilities were stretched to the limits of the accuracy and speed of manual PCB design methods. We needed automation but could not find a commercial program that was affordable and had the functions we required. We wanted to be able to use both millimeter and inch dimensions, mixed in a single design. We also wanted general drawing functions to design equipment housings and fronts. We decided to see how far we could go by designing our own PCB design package.

We started with the following design specifications:

- Run under MS-DOS;
- Closely mimicking a manual design with tape and rub-off symbols;
- Be fast and flexible;
- Have extensive editing functions;
- High internal resolution of 1/1280 inch;
- Use millimeters and inches in a single design;
- At least 7 pen-widths, 7 drill diameters and 15 different pads;
- Manipulation functions to move, rotate, copy components and parts of a design;
- Output GERBER photoplotter files;
- Directly drive our EXCELLON NC driller.

The first results were very encouraging, and we felt we were on the right track. The quality of our products was no longer dependent on 'good' or 'bad' days at the tape table. The high internal database resolution enabled us to accept designs for any accuracy required. Implementing changes was fast, without loss of quality.

In 1985 the program was so stable and mature that one of our PCB customers was prepared to pay quite an amount for it to design his own PCB's. It then dawned on us that this program was a very powerful tool for any PCB designer. A small advertisement in a trade journal brought a few interested dealers together on a saturday, where we demonstrated the program - and sold several copies on the spot. We decided on the name Layo1: this is 2/3 of the word layout, and we felt that once you have Layo1, your design is more than half done...

Our philosophy

The most important feature of any PCB for any circuit is its track layout. Traces should be considered as additional 'components' (resistors, impedances and capacitances) in the circuit itself. If signals are weak and/or frequencies are high, the PCB layout may have a dominant effect on the operation of a circuit. In many cases, an optimal trace layout in this sense is different from an autorouter product. These facts led us to the following philosophy:

"Because the trace layout is an integral part of the circuit, it should be designed with experience, knowledge and creativity. A PCB design program should not override the designer's creativity with all kinds of 'auto' functions with limited results.

The program should stimulate an interactive design process of knowledge and experience coupled with PC power. A high response speed will motivate a designer to create a quality product and is therefor indispensable."

Somewhere in 1988 we really felt the need for a possibility to use the data from a schematic capture program in Layo1. Layo1 was a pure drawing program, with a minimum of overhead and high workspeed. We therefor developed a way to import and maintain connectivity information, without losing Layo1's inherent advantages. The users urged us not to transform Layo1 into a pure autorouter, and we have always honored this. The new version of Layo1 was called Layo1+ (+netlist). The integration with a schematic capture program offers the following advantages:

- Auto loading of part shapes;
- All electrical connections are always controlled, without losing the possibility for manual intervention;
- The ratsnest feature makes it very easy to find an optimal parts placement;
- The use of colors and rubberbanding makes for faultless manual routing;
- Real time indication of routings that are completed, partly completed or in error;
- If required, autorouting using a netlist.

Once we started selling the basic Layo1 package, we decided to do all further development in close cooperation with the users. This is probably the reason that many packages have been sold by direct recommendation from enthusiastic users. We therefor invite you to let us know your comments, remarks and ideas about Layo1. This is an unique opportunity to help shape the future direction of Layo1.

Quality assurance

Throughout this Guide you will find references to ways to control the design of the PCB you are working on. This cannot be overemphasized. Very often, last minute changes are necessary to prototypes and even production runs. If manual modifications are necessary, they are always very expensive. We don't even mention throwing away PCB batches because of some design error. Changes to prototypes cannot always be prevented, because not everything can be calculated and simulated. The layout of the PCB is part of the circuit, and thus must be tested in hardware. But what can be prevented is that the layout contains connection and hole pattern errors.

The worst thing you can do is to tape a diagram alongside the monitor, call up Layo1 shapes and start routing from the diagram. If you have available a supported Schematic Capture Program, *use it*. Don't think it will take to long. You will very quickly find out that if there is no time to do it right the first time, you will have to *make* time to do it over again!

These Schematic Capture programs can help you make sure that the PCB layout is exactly conforming to the schematic. Of course, you must make sure that the schematic is correct, but that must be done anyhow.

If you work with Layo1 in a standalone mode, more effort is required, but greater risks for errors are also present. At the very least, make a list of all components and select the required shapes from the library in the Appendix to the Reference Guide. If in doubt, place the actual component over the printout and verify that it fits the holes. Before you start routing, make up a netlist. Several ways are discussed in Chapter 3. Afterwards, check the netlist against the schematic diagram.

After it all checks out, *then* start routing. All these measures also indicate the difference between Quality Assurance and Quality Control. QA means organizing your work in such a way that you cannot help but producing quality products. QC means that you find out which products exhibit failures and throw them away. That means throwing *money* away. Layo1, and its integration facilities has all the features to do it right the first time. Its up to you to use them. And as a reminder, you can tape an enlarged copy of the following phrase next to you workplace...

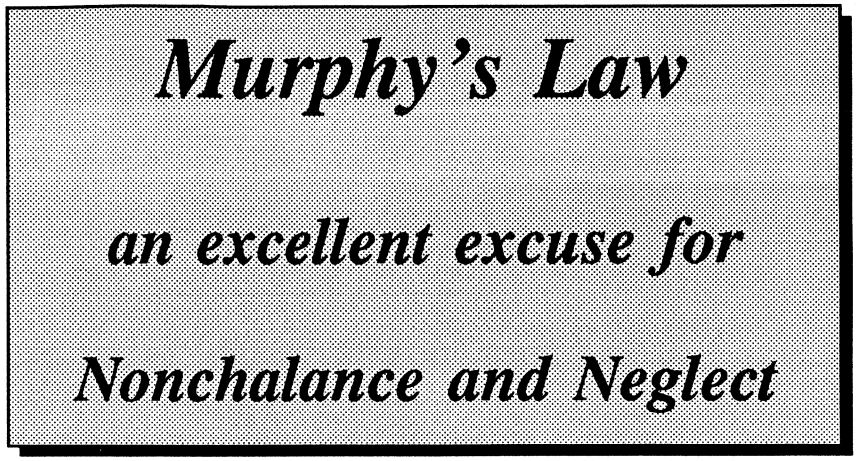


Fig. 1-1 Murphy's Law

PCB production

If you do not do your own PCB production in-house, there are some considerations you should make when designing your layout. You should visit your PCB producer to take a look at the production processes, to get an idea where things could go wrong if the data or plots you deliver are not optimal. If you contract out the production, do not do your own plotting. Deliver the output files (and the Output Driver if necessary) to the production site. Making the plots is a part of the production process and is best done by the same people that use the plots.

Making a prototype from a checkplot and then manually drill it certainly has its value. But with high volume and/or high value production runs, do not skimp on an 'official' prototype including machine-drilling. Many designers do not take this trouble, but even if the financial risk for an error lays at the production site, you can still be faced with a large time delay when errors pop up. As we indicated above in the section on quality assurance, try to be absolutely sure of *everything* before committing money and resources. We know that under pressure of time-to-market these principles often do not survive. But if at all possible, follow them.

Conventions

The following conventions are used in this Guide:

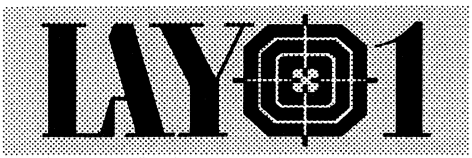
Keypresses Single keypresses are represented by **[keyname]**. Keyname may be one or more keys. If **[↵]** has to be added, this will be explicitly shown. Shift-key combinations are shown as in **[↑f]**, which is identical to **[F]**. Control-key combinations are shown as **[^F3]**, which means to hold 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. Also text in a `courier font` indicate text displayed on the screen or a command that you must enter.

▸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 the current 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

Installing Layo1

This chapter guides you through the installation of Layo1 on your system. Although the install process itself is automatic, you have to make a few basic decisions and enter a few parameters.

Upgrade

If you are upgrading from a previous version of Layo1, you must make sure that the drive and directory you will specify to the installation program are the same as your current system. If you do not, the install program will not be able to find your current projects. You can install to a different drive and/or directory, but then you must move all your current projects and maybe your custom component libraries, if any, into the new project and component directories.

Warning! Install will overwrite the old component banks with the new ones. Although the new banks may be larger then the previous ones, your current components will been retained unchanged. Therefor, this poses no problems at all, and gives you access to the latest component models.

However, if you have modified or created components, and saved them under existing names in the existing component directories, you must move them to a separate directory before doing the installation. Again, we stress that you should never modify components and save them under the same name. This will create compatibility problems sooner or later. Give them another name and put them in another directory like **'..\MYPARTS\'**

Integration

Layo1 is normally run from the Project Manager. The Project Manager is a programmable shell for managing your design and Layo1 programs. Its use and features are described in the next chapter and in the Reference Guide. However, before installing you must decide whether you want Project Manager to take care of the integration with the Schematic Capture Program or not. Currently, Layo1 supports integration with several versions of the popular OrCAD Schematic Capture Programs: SDT III versions 3.1 and 3.2, and SDT IV.

You can of course use Layo1 fully in its PCB design role without Schematic Capture Integration. But integration will make your designs much easier to maintain, with much less chance of errors and mistake. You will be able to use Layo1's built-in error checking facility, and will have the extra documentation from the schematic diagram.

If you do not want to do the integration now, you can always do it later, see ▸Install and Modify. If you would like to integrate with a not supported Schematic Capture Program, we suggest that you first install without Schematic Capture, and get acquainted with Layo1. Later you may want to review Chapter 5 in the Reference Guide on customizing the Project Manager, and Chapter 2 in the Reference Guide on the Layo1 net- and component list file formats.

System configuration

Layo1 is a complex program that can manipulate large amounts of data very fast. Although it is fully functional on a minimum system as described below, the recommended system will give you faster responses and better display performance.

Minimum system

A minimum system consists of:

- IBM (compatible) XT with EGA display adapter;
- 640 Kbyte main memory;
- MS-DOS 3.xx
- EGA monitor
- Harddisk with at least 5 Mbyte free space
- 3 button mouse;
- Laser printer.

Recommended system

We recommend the following system configuration for optimal performance:

- IBM (compatible) 386 with VGA display adapter;
- 1 Mbyte main memory;
- 2 Mbyte expanded memory
  (EMS 4.0 compatible);
- MS-DOS 5.xx
- VGA monitor (1024 x 768) non-interlaced
- Harddisk 40 Mbyte
- 3 button mouse;
- Laserprinter (for prototype and checkplots);
- Photoplotter if you do your own production.

Preparations

Before going into the actual details of the installation, there are some preparations to be make. You also should have decided on the integration issue. And don't skip the backup step!

First time install

One of the first questions the install program will ask you is whether you want to do a first-time install, or whether you want to change the directory structure or the integration with OrCAD/SDT. Select first-time install if this is the case and follow the procedure as outlined below. If this is not a first-time install, continue at the section ▸Install and Modify below.

Prepare for installation with the following steps:

- Use **DISKCOPY** to make a copy of the Layo1 diskettes. Store the original diskettes in a safe place.
- Use **CHKDSK** to make sure there is at least 3 Mbyte free space on your hard disk before starting the install procedure.

OrCAD/SDT users only

- If you have decided to integrate Layo1 with OrCAD/SDT, check and write down all path and directory names used by your version of OrCAD.

Normally, OrCAD/SDT 4 uses the defaults as shown below:

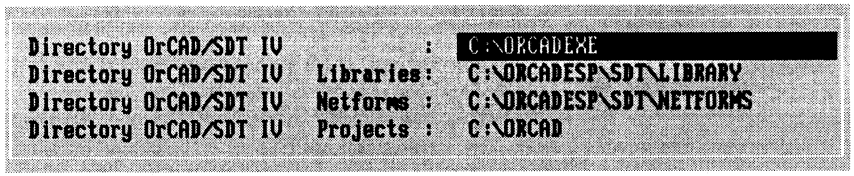


Fig. 2-1 OrCAD/SDT 4 default directory structure

The default OrCAD/SDT III structure will normally be different. Also, your configuration may differ anyway with regard to drive letter and/or directory structure. Just write down your current setup and the OrCAD version you have.

Specifying parameters

In a moment you will start the installation. Several screens will appear. Depending on your particular system set up, you might need to edit an entry or a path. In that case, place the highlight over the entry to be changed with the cursor keys **[→]** and **[←]** and press **[F9]**. You can now use the **[cursor keys]**, **[Home]**, **[Backspace]**, **[Ins]** and **[Del]** to edit the entry. If you want to install to another than the default directory, edit the path. You can change the default drive by pressing the new drive letter, as shown at the bottom of the screen.

Follow the directions on the screen. At the bottom of each screen you will find the keys to press if you want to abort the installation, to backup a screen, to select another drive to install to or to start again.

There will be various messages and questions on the screen. These will be mostly self-explanatory. You can also read the next section which gives additional information.

Drive, directory

The first install screen will welcome you to the install program. If you continue, the first question is in which directory you want to install Layo1. You can modify the drive to install to if necessary. *Only for very specific reasons should you install in an other directory.* If you accept the default, future updates, Project Manager customizing etcetera will be much easier. But, if you insist, set it to your preference.

Schematic integration

Place the highlight on the OrCAD/SDT version of your preference and press [↵]. The Project Manager will be automatically set up to provide the links to the Schematic Diagram of your projects.

Project Manager only

Select this choice only when you do an update for the Project Manager or modify the Schematic Capture integration. If you want to do that, you should follow the section 'Install and modify'.

Overwrite confirm

Normally you can set this to **Yes**. Only if you have custom libraries you may want to select **No**. In that case, the program will ask you if it can overwrite existing files during installation. If at that time you decline overwrite, that particular file will not be installed. But in that case it is better to rename the custom parts and/or move them to a separate directory and then install all the new files.

Note that in any case, the Project Manager menu structure and default user configuration files, if present, (PM.DAT and USR00.CFG) will automatically be saved to PM.LA1 and USER00.LA1.

Modify autoexec.bat

If you let the install program modify the autoexec.bat, you will be able to start the Project Manager from any disk and from any directory. The modification consists of adding the 'path' to the Layo1 programs. This change will not impact any of your other programs you might have. If the path is not in autoexec.bat, you can only start it when you first log on to the `..\layo1\pman\` directory.

Finally, the install program will present an overview of the choices you have made. A typical display is shown below:

```
Integrate Schematic Capture Program. OrCAD/SDT IV

Directory Layo1 files:              C:\LAYO1
Directory OrCAD/SDT IV            : E:\ORCADEXE
Directory OrCAD/SDT IV  Libraries: E:\ORCADESP\SDT\LIBRARY
Directory OrCAD/SDT IV  Netforms : E:\ORCADESP\SDT\NETFORMS
Directory OrCAD/SDT IV  Projects : E:\ORCAD

Update Project Manager files/menu's only?                    No
Overwrite existing files without confirm?                    No
Add Layo1 path to 'autoexec.bat'?                            No

                  Please verify the parameters.
  Start installation by pressing <Enter> or redo by pressing <Esc>.
```

Fig. 2-2 Typical install configuration

If you want to integrate with OrCAD, continue with the next section. Otherwise, continue at 'Install stand-alone' or 'Install and modify'.

Install and integrate

To start the install process, insert the copy of Layo1 diskette 1 in drive A: and type **a:install[↵]**. (Substitute b: if you use drive b:). Continue the installation until this is finished. Do not yet start the program, but follow the following section first.

SDT IV users only

To enable the Project Manager to start OrCad SDT IV with the correct display driver, add the following line to your **AUTOEXEC.BAT** file. (Review the DOS documentation if you are not sure how to do this). We assume that you keep the OrCAD device drivers in `c:\orcadesp\drv\`. If not, insert the correct path in the following command line:

```
set orcaddd=c:\orcadesp\drv\vga640.drv
```

Reset the computer to activate the new path.
If you see the DOS message: **'out of environment space'**, you must modify the **CONFIG.SYS** file to increase the environment space where DOS keeps its data (see the DOS documentation if you are not sure how to do this).

Adding the line:

```
shell=command.com /p /e:512
```

to the config.sys file increases the environment space to 512 bytes. If you had to do this, again reset the computer to activate the new setting.

Field stuffing

In the following section we will set up OrCAD to automatically provide component shape information to Layo1. OrCAD/SDT can automatically associate a Layo1 component shape with an SDT parts name. SDT maintains several data fields with each part used in a schematic diagram. One field contains the part name. Other fields can be filled in from a text file. The process is called **'fieldstuffing'**, and the text file to use as input is called a **stuff file**.

Stuff files

We have included several stuff files with Layo1. These contain the required parts names and corresponding component shapes. Later, when we start the Project Manager, you will see a menu page titled **Fieldstuffing**. The screen will show the files with the extension **.STF**. There will also be a user stuffile option called **USER.STF**. This can be used to customize the association of SDT parts with Layo1 shapes, for instance if you built your own unique component shapes. Mostly, however, this will not be necessary. Just select the correct stuff function from the Project Manager and SDT will do the rest.

You will have a chance to use this in the tutorial in Chapter 4. If you do want to make your own stuff file, use the user.stf option, or copy an existing stuff file to your new one and edit that file. Stuff files are described in the Reference Guide.

Stuff fields

What we will do is tell OrCAD which of the eight data fields for a part must be filled with the Layo1 shape name, so that Layo1 can find it. Layo1 will then look up the shape in its directories or libraries and substitute its own correct part shape to be used on the PCB design. For this we use the Key Field option, and set the field number to 'stuff' to '1'.
(ZIE BLZ. 2-13 NED. VERSIE)

Configure OrCAD

To configure OrCAD for Layo1, we need to do two things. First, we will add the **LAYO1.LIB** to OrCAD. This library contains the schematic symbols used in the tutorial, described later in this Guide. For a description of a library file, see the Reference Guide. The contents of the library (the components) are also in other libraries, but using LAYO1.LIB insures that the tutorial illustrations are the same as your screen displays.

Next we will set up the stuff parameters as explained above. Start the Project Manager by going to the **..\PMAN** directory and typing **PM[↵]**

SDT III users

Using **[PgUp]** and **[PgDn]** bring up the menu page titled **OrCAD/SDT Schematic Design Tools**. Select **'Configure OrCAD/SDT III'**. When the OrCAD screen appears, type **KF12[↵]**, next press **[1]**. This sets up the keyfield.

Press **[Q]** to return to OrCAD's main menu. To install the library, type **LFA1[↵]**. Now type **LAYO1.LIB[↵]**. Save the settings and leave the configuration program by typing **QUQ**. This finalizes the configuration of OrCAD/SDT III to work with Layo1. You can skip the next paragraph and continue with Chapter 3 to familiarize yourself with the Project manager and Layo1.

SDT IV users

Using **[PgUp]** and **[PgDn]** bring up the menu page titled **OrCAD/SDT Schematic Design Tools**. Select **'Configure OrCAD/SDT IV'**. When the OrCAD configuration screen appears, move to the **Key fields** options. From the field **Create Netlist** select **Module Value Combine**. Enter a '1' in the field, then press **[↵]**. This sets up the keyfield.

To install the library, move the cursor to **Library** options. From the window **'Available libraries'** select **LAYO1.LIB** and click the **Insert** button. This finalizes the configuration of OrCAD/SDT IV to work with Layo1. Save the new settings and exit by pressing **[Home][↵]**.

If later you want to change the directory structure or the integration with OrCAD, see ▸Install and modify. You can continue with Chapter 3 to familiarize yourself with the Project Manager and Layo1.

Install stand-alone

To start the install process, insert the copy of Layo1 diskette 1 in drive A: and type **a:install[↵]**. (Substitute b: if you use drive b:). Be sure to select `No Schematic Capture integration` as shown below:

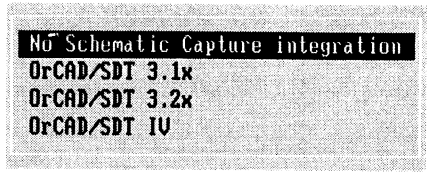


Fig. 2-3 Stand-alone integration

Continue the installation until this is finished. If later you wish to repeat the installation with Schematic Capture integration, see the section ▸Install and modify below.

Install and modify

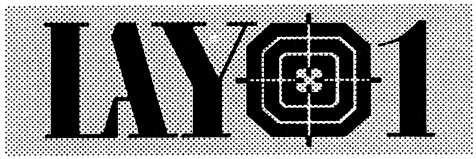
If you have installed Layo1 previously, but want to change the directory structure and/or the integration with OrCAD, you can use the install program to do this. Insert the diskette with the copy of diskette # 1 in a: and start install as described above. Answer **Yes** to the question `Update Project Manager files/menu's only?`

Change structure

You can modify the drive and/or directory where you install Layo1. The install program will save any previous versions of the Project Manager files it finds to the same name, but with extension **.LA1**. But, if you change the drive and/or directory, Layo1 may not be able to find your projects anymore. In that case, you must move the entire `..\proj\` directory under the `..\layo1\` directory. You can use a utility like Norton Commander or PC Tools. The process is called 'prune and graft'.

Change integration

Install will also ask if you want to integrate Project Manager with a version of the OrCAD Schematic Capture program. If you say **Yes** and enter the required directories, the install program will set up a new Project Manager structure. If you have already integrated OrCAD previously, and now say **No**, install will dissolve the integration.



CHAPTER 3

Getting Started

This chapter will get you started using Layo1. First we will introduce you to the Project Manager. If you have integrated OrCAD, we will show how to use Schematic Diagram data to set up Layo1. Finally, the last section will tell you how to use a macro to store frequently used keystroke sequences under an **[Alt][key]** combination. This is not required reading, but sooner or later you will want to use this feature.

Project Manager

Although you can run Layo1 from the DOS command line (see ▸Layo1 structure in the Reference Guide), this is not very practical. The Project Manager provides an elegant and efficient command shell for Layo1 and its utilities and related functions. This section will therefor assume you use Project Manager to run Layo1.

You start the Project Manager by entering `pm` from the DOS command line. Of course, DOS must be able to find the pm.bat file, which will be in the `..\pman\` directory. This will be no problem if you let the install program add the path to layo1.bat to your autoexec.bat file, or if you did that yourself.

Otherwise, you must first change to the ..\pman\ directory before staring Project Manager. The project Manager opening screen will be similar to the illustration below.

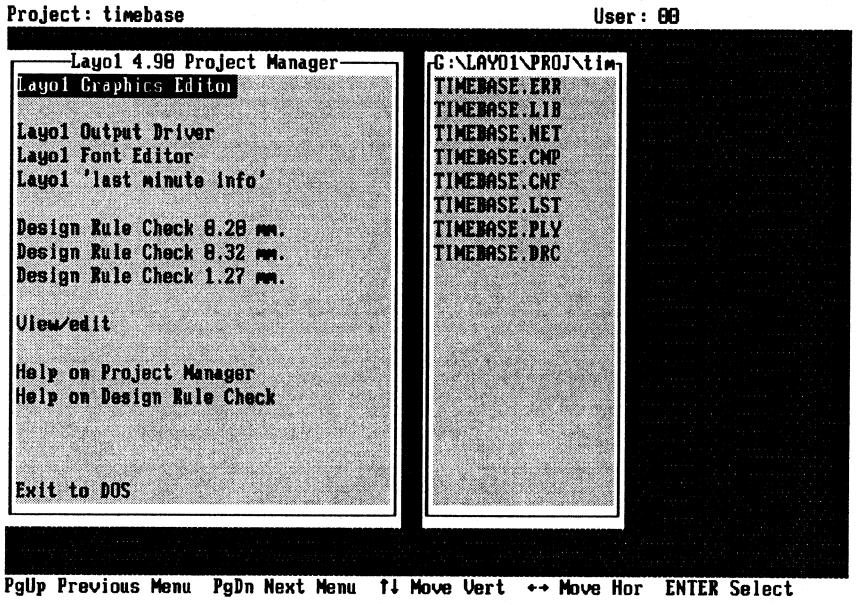


Fig. 3-1 Project Manager opening screen

At the top of the screen you see the labels for the active project name (**Project:**) and user number (**User:**).
The display itself is divided in three vertical columns.

- the left column or window contains the menu entries for the functions you can select;
- the middle column or window shows several files;
- the right column or window is not displayed at the moment, but would be similar to the middle one.

The bottom line shows the commands to move the highlight around in the windows.

Select menu

The display you look at is just one page of several menu pages. The menu title is shown on the top border of the left window. **[PgUp]**, **[PgDn]** and mouse □■□ will take you to the other menu pages. Press **[PgUp]** a few times to sample the other menu pages. Each page contains several related commands and functions. On other menu pages other files may be shown in the second and/or third window. (All this can be customized; see Project Manager Reference in the Reference Guide).

Select function/file

Within a menu page, you use the highlight to select a function or a file. The highlight is moved from window to window by moving the mouse horizontally, or with **[←]** or **[→]**.

Within a window, the highlight is moved by moving the mouse vertically, or with **[↑]** and **[↓]**.

You can also jump to a selection by entering its first character. You activate the selected function or file by pressing [↵] or by clicking ■□□. If the highlight is in one of the two directory windows this will start the built-in text editor with the selected file loaded. With the highlight in the left window, selection will execute the function.

Confirming a selection

There are many instances when you will have to confirm a selection from a list or menu, to accept a default or to start a function. This is generally called **confirming**. Confirming in Layo1 can be done in two ways. You can use the keyboard to press [↵]. If you prefer using the mouse, you confirm by clicking ■□□. There is no preference other than your personal one. In the remainder of this Guide, we will just say 'Confirm', and you chose the method that suits you best. Note that we often mention **Top Level**. This is the status of the program when no function is active. It is shown by the small Layo1 logo in the far left corner of the status line. You can always reach Top Level by repeatedly pressing **[Esc]**.

Last minute info

This is a good time to take a minute to read the last minute info that didn't make it into the documentation. Move to the menu page titled **Project Manager**. Move the highlight to **Layo1 last minute info** in the left window and press [↵]. This function will load the info file into the built-in editor, so you can scroll through it using the keys indicated at the bottom of the display. When you are done, quit with **[Alt][x]**.

Warning! If you open a non-text file with the internal text editor, you will see unreadable text and symbols on the screen. Press **[Alt][x]** immediately to exit. If the program asks to save the file, be sure to answer **N**. If you inadvertently change and save a non-text file, Layo1 (or any other program) may not be able to use it anymore.

You may exit Project Manager now by selecting the appropriate function from the left window. When you are new to Layo1, we strongly recommend that you continue with this chapter, a short introductory tutorial.

Create a component

The next section is a tutorial on creating a component shape. This will introduce you to most of Layo1's principal features and functions. It will not be anything revolutionary, but we strongly recommend to follow it before you jump into really fancy designs and projects. However, if you already know Layo1's basics, you could skip this and go to Chapter 4, which guides you through a complete project. The component shape we will create here is a standard 16 pin DIL.

Getting started

We assume that you are in the Project Manager. If not, start it as described in the previous section. The Status Line at the top of the screen will show a project name, probably NONAME. First we will clear this name, because we just want to built a component, not a full project.

Use **[PgUp]** or □■□ and display the menu page titled **Project & User Manager**. Select **Select project**. The cursor will move to the top left of the screen. Press **[Space]** to clear the existing name and then press **[↵]** to exit the function.

Using **[PgUp]**, **[PgDn]** or □■□ display the menu page **Project Manager**, place the highlight on the `Layo1 Graphics Editor` entry and press **[↵]**. This will start the Graphics Editor. Open the Main Menu with **[↵]** or □□■, place the highlight on **File** and press **[↵]** or □□■. Using the same procedure, select the **New Component** action from the file menu.

A window opens where you must specify a component bank and a component name. We will use **'DIL16'** as component name. To prevent any confusion with the original components, we will store DIL16 in a separate bank (really a directory) named **'TUTBANK'**.

As long as no filename has been entered, **'NONAME'** will be displayed by the program. Use **[Backspace]** to clear NONAME. Append to ...\LMC\: **TUTBANK\DIL16**. Press **[↵]**. Depending whether the directory TUTBANK exists or not, you may here a beep and see a message. If that is the case, press **[↵]** again. You will see Layo1's Graphics Editor screen again.

Setting grid

Because the pads in a standard DIL are placed in a .1 inch grid, we will set Layo1 to the same raster. Look at the status line at the top of the screen, to the left of the X. You will see a grid setting like 1/20. Change that to 1/10 using **[F9]** or **[F10]**. Now press **[>]** twice, this will display a helpgrid, and make it also .1 inch.

The last thing to do before starting the design is to snap the cursor to the .1 inch grid, and we do that by pressing **[*]** twice. (Do not try to memorize all these commands at this point. The idea is to get a feeling for the program. The Reference Guide contains a full description of all available commands and functions).

Placing pads

Next we will place the pad pattern for DIL16. Open the Main menu (press **[↵]**), select **Pad**, from the pad menu select **Place**. A pad will appear at the cursor. The cursor position in x and y is shown on the status line. Move the cursor to **0,0**.

Drop the pad by clicking ■□□. You may think the pad disappeared, but when you placed the pad, you hitched another onto the cursor. The two pads cover each other and are not visible. You will see it next: Move the cursor one step to the right (to **1,0**) and again click ■□□; this places the second pad. Repeat this six more times until you have a row of eight pads, .1 inch spaced. The cursor should now be at **7,0**.

We now will place the other row. Position the cursor to **7,-3** and confirm. Place the other seven pads one step to the left of each other. (Layo1 has a Repeat Copy feature which lets you place rows of pads much quicker. This function is explained in the Reference Guide). The last pad should be at **0,-3**. Now click □□■ to exit the pad place function. You should be back at the Top Level.

Pin numbers

Let us check the pin numbers assigned to the pads by Layo1. The pins are numbered in the order they are placed. Press **[n]** to display pin numbers. Verify that pins are numbered from 1 through 16 according to the standard ic numbering sequence. These numbers are small so they do not clutter the design. If you want, you can click □■□ and select a larger zoom scale. The display should be similar to the one below. Press **[n]** again to deactivate the pin number display.

Set temporary origin

By defining a temporary origin, we can continue to use positive coordinates. First position the pad pattern for a good overview. Place the cursor at **3,2**. Click □■□ and select zoom scale 6. Next move the cursor to **-1,-4** and press **[F6]**. The ruler at the status line now displays **0,0**.

Drawing the shape

Next we will draw a half-circle between the pins 1 and 16, to mark that end of the shape, as is customary for TTL ic's. Place the cursor at **4,3**, press **[z]** or click □■□ and select zoom **8**. Open the Main menu (press **[Esc]** or click □□■).

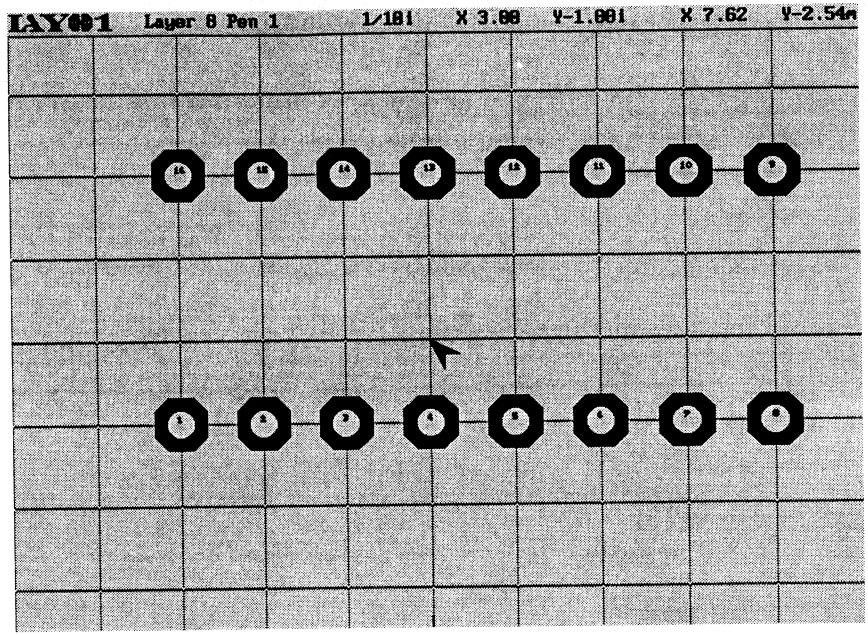


Fig. 3-2 Dil-16 pads pattern

From the Draw menu select **Circle**. The Status Line at the top of the display will show **Draw circle**. Now use **[F9]** or **[F10]** to set the drawing grid to 1/20 inch (displayed at the status line). Position the cursor to **0.5,2.5** and confirm. This sets the origin of the circle. Press **[→]** once and a box appears, showing the dimension of the circle. Confirm the size.

Window delete

Now we will delete the left half of the circle as follows. From the submenu: **MultiFunc/ Window** select **Delete**. We will draw a window that encloses the part of the circle to delete.

Position the cursor to **0,2** and confirm. Next position to **0.5,3**. The window defines the area that will be erased. Confirm. Answer **[Y]** to the question `Delete ?` Half the circle has been erased now. Return to top level (**[Esc]**). Press **[F5]** to redraw the screen.

Next we will draw the 'box' part of the shape. From the Draw menu select **Draw**. Position the cursor at **0.5,3**. This is the start of the box we will draw. Confirm. Move the cursor to **0.5,4** and again confirm. Continue this with the following positions until the shape is finished: **8.5,4**, **8.5,1**, **0.5,1**, **0.5,2**. Your drawing should now resemble the figure below. Press **[Esc]** to return to Top Level.

Editing the drawing

Now, looking at the drawing, we decide that the shape outline should actually run inside the pad pattern instead of through the pads. We will therefor edit the drawing. Set the grid size to **1/40** (F9 or F10). Position to **0.5,1** on the start of a line segment. Press all three mouse buttons simultaneously or press **[=]** to enter the edit function. Layo1 will blink the line and ask if this is the one you want to edit. Press **[Y]** or confirm. Position the cursor to 0.5,1.25 and note that the line segment is moved. Press **[-]** to step to the next node in the drawing (see Chapter 3 in the Reference Guide for more information on nodes). The cursor should now be at **8.5,1**. Move this node to **8.5,1.25**. Again press **[-]** to jump to the next node. Move the node at **8.5,4** to **8.5,3.75**, and the node at **0.5,4** to **0.5,3.75**. Return to Top Level and redraw the screen.

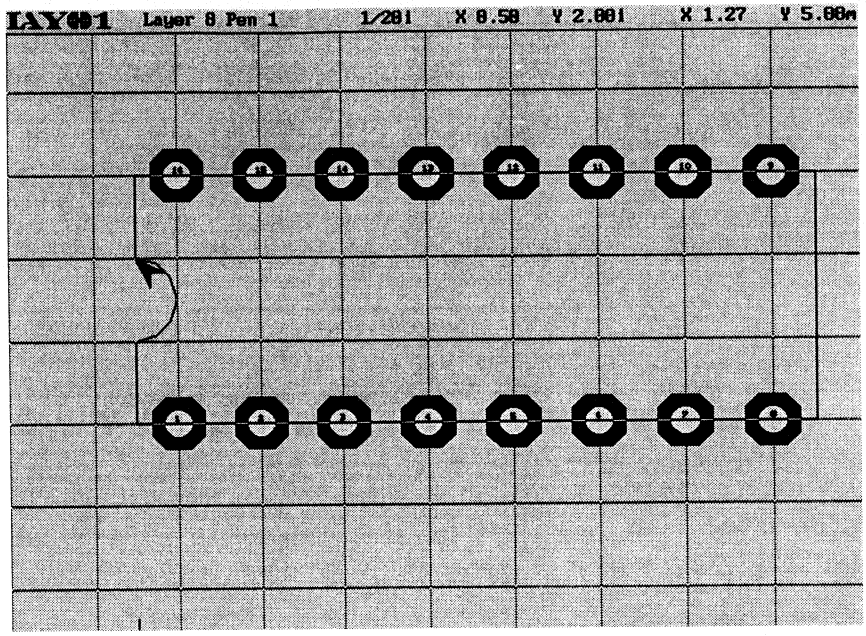


Fig. 3-3 First attempt on DIL16

The last thing we will do is to mark pin 1 of the IC. We do that by placing a small square around it. Decrease the grid size to **1/80 inch.** From the Draw menu select **Box**. Position to **0.63,3.63** and confirm. This is one corner of the square to be drawn. Let's make this a temporary origin, so we can place the other corner of the square more easily. Press **[F6]** to define a temporary origin. Now position to **0.75,0.75**. A window appears showing the size of the square. Confirm. The final result should look like the illustration below.

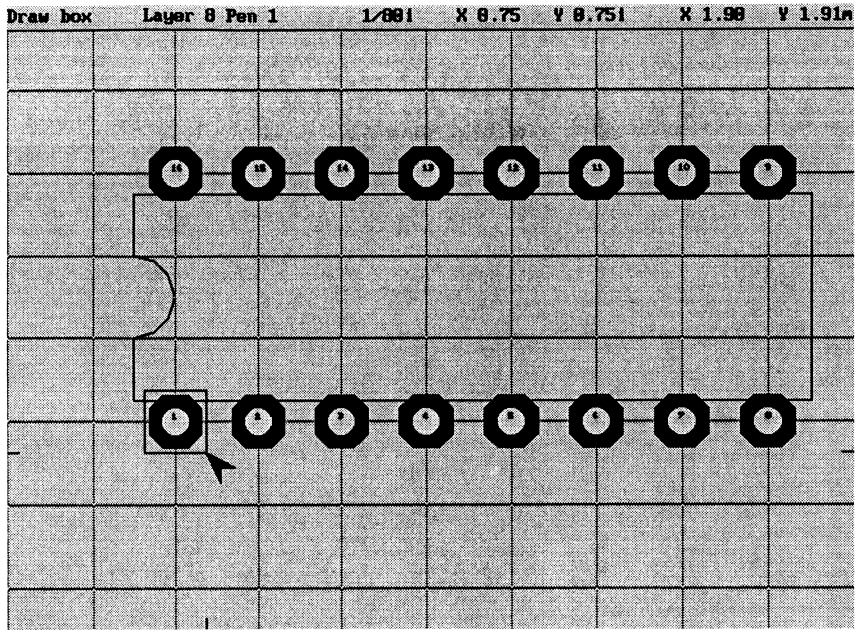


Fig. 3-4 Final DIL16 component

Area fill One other function we will exercise is area fill. Suppose that the shape we just designed is to be used in a sensitive circuit, and you want to have a ground plane under it. We will customize the current DIL16 to include this plane on the copper side.

Define area First center the design. Place the cursor at **3,-1** and select zoom step 7. Set the grid to 1/20 inch. To draw the plane, position the cursor to **-1,-4**. Make this a temporary origin with **[F6]**. Press **[l]**, and select layer 1.

Press **[t]** to enter the draw mode. This is shown at the left on the status line. Draw the plane outline by confirming the following positions from left to right (confirm with ■□□):

0,1.5;	**9,1.5;**	**9,0;**
10,0;	**10,5;**	**9,5;**
9,3.5;	**0,3.5;**	**0,5;**
-1,5;	**-1,0;**	**0,0;**

This should finish the outline. Press **[Esc]** till you are at the Top Level.

Fill area

Now we do the actual filling. The cursor should still be at **0,0**. Press **[F]** (capital F). Confirm the question. This identifies the drawn outline as the fill boundary to the program. Next place the cursor inside the area, for instance at **-.5,1**. Confirm this position and the area will be flooded with horizontal lines. If you see small blank horizontal lines in the area, don't worry. They will not show in the output. The result should be similar to the figure below.

Now clean up : Return to Top Level. Reset the grid size to **1/10 inch** and press **[*]** twice to snap the cursor to the grid.

Saving your work

When you want to save a component, it is good practice to place the cursor over pin 1. This will set the component's origin relative to the cursor to pin 1.

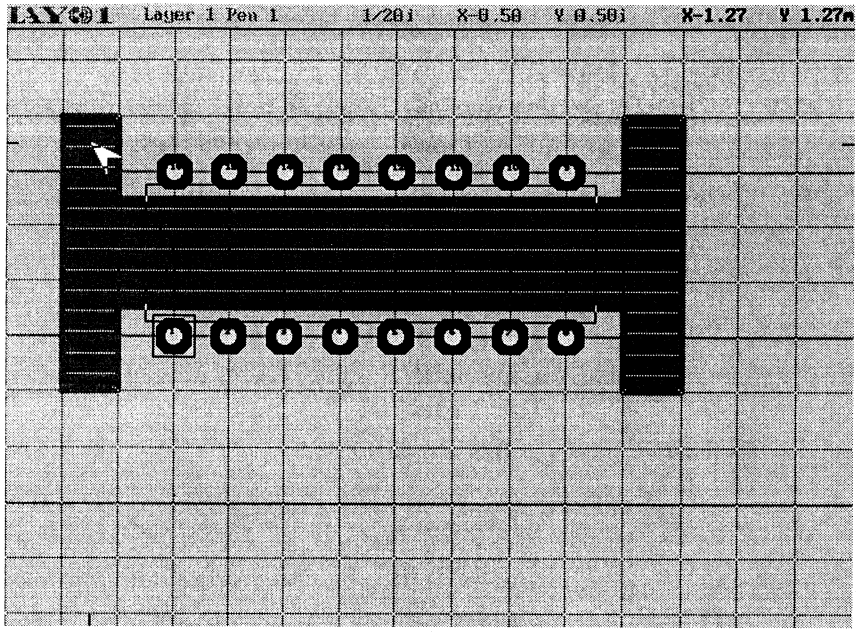


Fig. 3-5 DIL16 with ground plane

Then, when you later load the component, it will be attached to the cursor at the same spot. Position the cursor at the center of **pin 1** (the lower left pin). Press **[F4]** to save your work. The drawing is saved as `..\TUTBANK\DIL16`.

Well, that's it for this tutorial. You have just created a Layo1 component, and in the process executed many important Layo1 functions. We suggest that next you tackle the next chapter which will lead you through a complete PCB design cycle, using a schematic diagram we have prepared for you.

Macros

If you have to press the same key-sequence repeatedly, you may define and save such a sequence as a **macro**. One useful macro would be one that presses the keys for you when you want to load a component into a design. We will create such a macro here to illustrate the process.

From the File menu select **Reset Layo1** (this is not really necessary, but this way we start with a clean screen). From the Options menu select **Create Macro**. You are asked for the key-combination for this macro, press **[ALT][g]**.The macro will be saved and can be recalled later by pressing the **[Alt]** and **[g]** keys simultaneously.

Note: Mouse control is deactivated during a macro, so you must do all positioning from the keyboard.

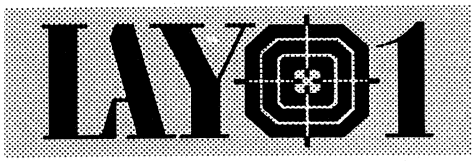
Press **[Esc]** to open the Main menu. Layo1 will always save the last menu selection. That implies you won't know which selection is active when you start the macro. You must therefor synchronize the menu selection by starting from the top position in each menu. You do this by pressing **[Home]**. Now you are sure to select the File menu from the Main menu, regardless of the previous selection.

- Press [↵] to open the File menu, press **[Home]** and select the Load menu;
- Press **[Home]** and select **Component LMC**;
- Press **[F8]**.

That is as far we will go, because from this point the actions depend on the particular component you want to load. End macro creation by pressing **[Alt][F10]**.

Return to Top Level and redraw the screen. Press **[Alt][g]** to try out your macro. You can use this macro every time you want to load a component.

This ends the chapter 'Getting Started'. You should have a good feel now how to use Layo1. You will probably discover, that the more you use Layo1, the more you appreciate its many advanced features. Not all of them are covered in this User Guide. If you work through Chapter 4, Working with Layo1, you get more experience. But you should also from time to time browse the Reference Guide for specific functions or functions group. Layo1 will meet your requirements as you get more and more experienced and efficient in its use.



CHAPTER 4

Designing a PCB

If you are new to Layo1, we strongly recommend that you work through the previous chapter, Getting Started, first. That will give you a working knowledge of several Layo1 function, which are used in this chapter. This chapter is a tutorial that takes you through a complete PCB design cycle. We will use a schematic diagram we have prepared for you.

There are two entry points in this tutorial. The section titled 'Integrated design' is used if you have integrated Layo1 with OrCAD/SDT. We will use OrCAD to prepare the net- and component lists for Layo1. The section titled 'Standalone design' is used if you work with Layo1 stand-alone, loading the components and building the netlist yourself.

After the initial preparations of the various files for Layo1, a single section titled Design layout will take you through the actual design, both for the integrated and for the standalone mode. Both methods lead to the same correct results. The only difference is that the integrated method is a bit faster, and helps you with preventing routing errors (if the schematic diagram is correct, that is).

Even if you use the integrated method initially, you may want to review the stand-alone tutorial sometime later. It gives you another view, and a extra understanding, of how Layo1 handles nets. The schematic design we use is shown on the next page. It is a simple digital timebase circuit, drawn in OrCAD/SDT IV.

Integrated design

Start the Project Manager if necessary. Make sure the TIME-BASE project is shown in the status line. If not, use **'Select Project'** to correct it. At this point, we have the schematic diagram in electronic form, but we must generate the files for Layo1 to use (netlist, componentlist).

Preparing the files

We assume that you have done the section ▸Configure OrCAD in Chapter 2, installation. Go to the menu page **Schematic Design Tools** (use **[PgUp]** or **[PgDn]**). Create the required files from OrCAD by executing the following function:

Annotate will annotate (number) the components;
Cleanup will remove any redundant objects;
ERC will check for electrical errors (shorts etc).

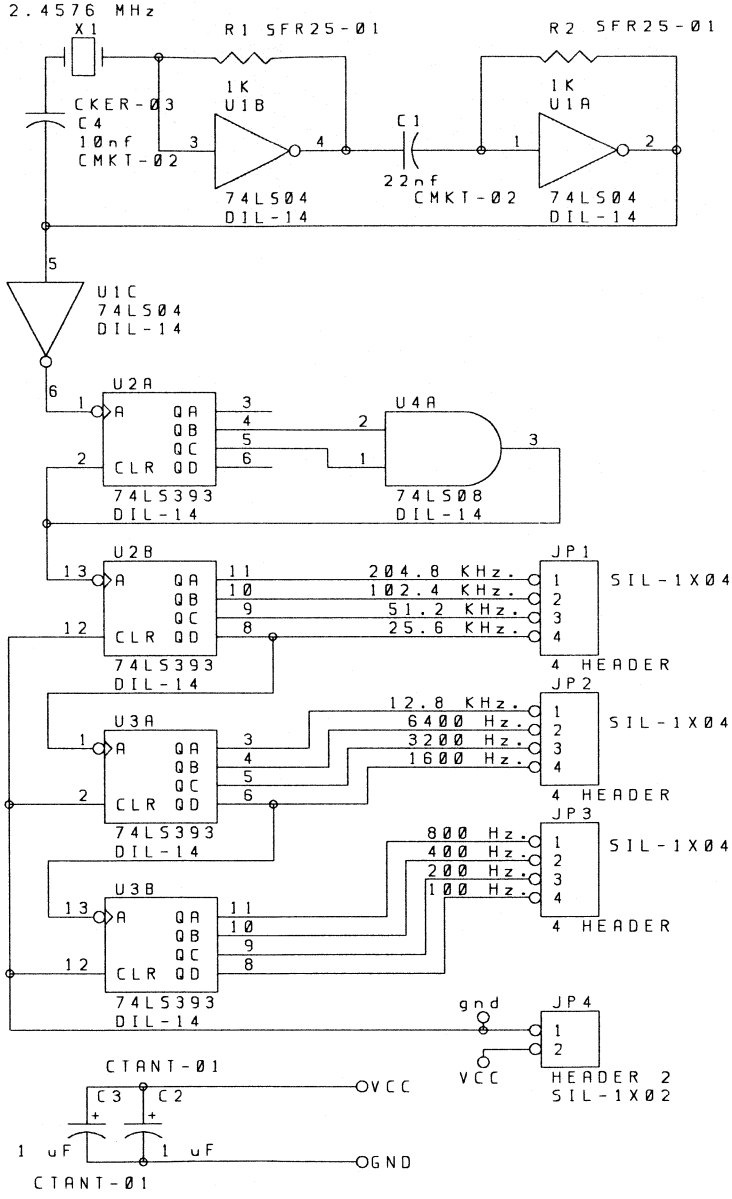


Fig. 4-1 Basic timebase circuit

Verify that there are no errors by opening **timebase.err** in the right window (place the highlight over the filename and press [↵]). Before we generate the component list, we must tell OrCAD to insert the Layo1 shape names for every part. This is called fieldstuffing. We already configured OrCAD for it, see ▸Configure OrCAD in Chapter 2, Installation.

Stuff fields

Go to the menu page **Fieldstuff** and select function **Fieldstuff all *STF files**. You will see the messages from OrCAD scroll over your screen, when it is extraction information from the stuff files and inserting it in the parts fields. If you are curious, you can open a stuff file and see how it is formatted.

Go back to the menu page Schematic Design Tools and select the function **MAKE: Netlist/Create PCB files**. This will create the required files and place them in the Layo1 timebase directory. Layo1 realizes you started a new design. The Graphics Editor will start and go straight to the Configuration screen. Before doing any editing, please turn to the section ▸Design layout and continue reading there. The next section is specifically for a design without schematic capture support.

Stand-alone design

In this tutorial we will design a PCB from the schematic diagram shown in the previous section without a direct link from the schematic files to Layo1.

You would probably tend to start by loading and placing the components. Next you would start to manually route each track, using the schematic diagram as a guide.

However, there is a much better method, which gives you access to Layo1's netlist checking features, and lets you use the ratsnest and the router in a controlled design environment. With this method, you would first create a net- and component list, and let Layo1 use this to control the design.

Starting the design

The first thing to do is to set up the directory for Layo1 to store all the files you will create in the course of the design. Start the Project Manager, and use □■□ to display the menu page titled **Project & User Manager**. Choose **Select Project** and at the status line enter **'TIMEBASE'**. Next select **Create Project Directory**, and you will see the project name TIMEBASE appear in the middle window.

Initial configuration

Start the Graphic Editor from the **Project Manager** menu page. Because this is a new design, Layo1 will present the design configuration screen as shown below so you can set among other things the PCB dimensions. We will initially set the PCB dimensions to 160 x 50 millimeters. The y-dimension is already set at 160 millimeter, so select x-dimension, answer the units query to input metric and enter 50. Then save the configuration. (you can always correct the settings later if necessary, both from this screen, as well as from the Graphics Editor).

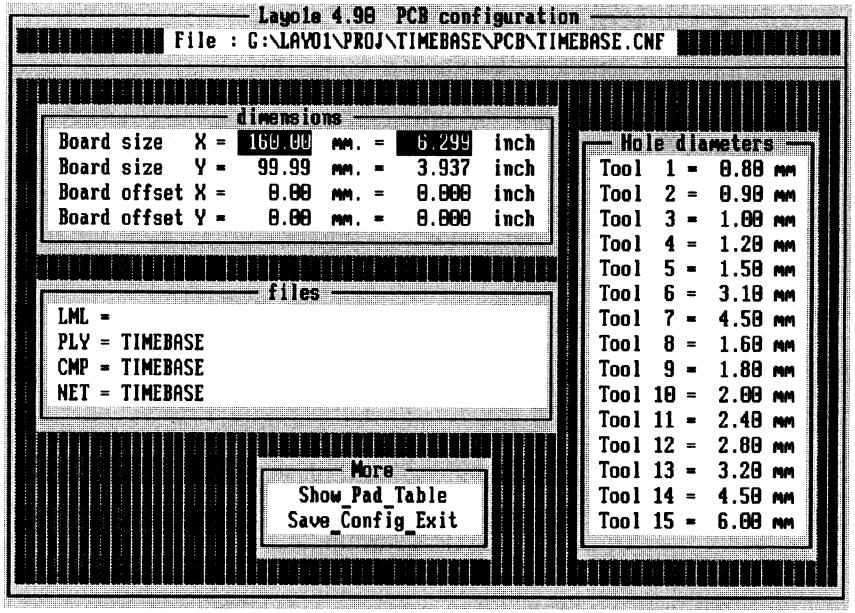


Fig. 4-2 Design configuration screen

Layo1 will automatically enter the Graphics Editor and try to load the design. But there is no component list, and you will see the message `..\proj\TIMEBASE\PCB\TIMEBASE.CMP not found`. Ignore that for the moment and press [↵] to continue. The screen will only show the corner lines, at the PCB boundaries. The PCB is automatically zoomed to just fill the screen, giving you maximum detail with a complete view of the design.

Observe *two* rulers at the top status line, one displaying inches and the other millimeters. Using the cursor, verify that the dimensions indeed are 160 by 50 mm.

Create component list

Next we will select the component models you need for the design. Using the Appendix from the Reference Guide, you would make a list of each component and the shape you wish to use for it. This should be done carefully, because it will be the basis for the design. To save time at this tutorial, you can skip this for once. We will use part of the component list we have prepared as shown below:

Shape	Type	Ref
DIL-14	74LS04	U1
CKER-03	2.4576 MHZ	X1
SIL-1X04	4 HEADER	JP1
CMKT-02	22NF	C1
SFR25-01	1K	R1

Be sure you are at the program's Top Level (the Layo1 logo displayed in the upper left corner), if not repeatedly press **[Esc]** until you are. Use **[F9]** or **[F10]** to set the drawing grid to 1/10 inch. Let us load the first part, the 74LS04. Click □□■ to open the Main menu, select **File**, and from the File menu select **Load/ Component LMC**. A window will open as shown below.

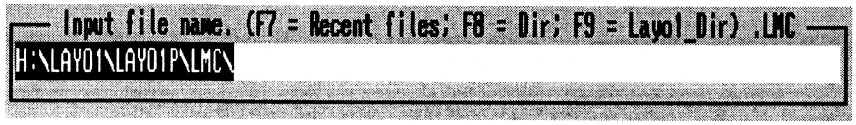


Fig. 4-3 File/ load component submenu

Loading components

Press [↵] to accept the suggested LMC master directory. From the subsequent display select the **ACTIVE** component bank. Yet another window will open, select **DIL-14**. You will return to the Graphics Editor screen with a 14 pin DIL shape attached to the cursor. A window will open requesting the component type. Enter **74LS04**. In the next window (Ref:) enter **U1**. Now move the cursor (with the shape following), to position 4.0,4.0. Click □□■. The ic will be placed and the cursor is released.

Again open **File/ Load/ Component LMC**. Backspace over **active\dil-14** and enter **passive** (don't forget the trailing \). Press **[↵]** and from the directory select the **CKER-03** shape for X1. In the Type: and Ref: windows enter the type (2.45776 MHZ) and the reference (X1). Place this part at 14.0,1.0. Repeat the File/ Load/ Component LMC selection and when you get the pathname display (which will be the previous name ..\passive\cker-03) backspace to ..\LMC\ and press **[F8]**. Navigate the displayed directories to the **CONNECT** directory. Select **SIL-1X04**, and place it at 19.0,1.0. Enter the type and reference as shown in the table above. In the same way, enter the other components, placing them as follows:

- load C1, from the passive directory, and place at 24.0,1.0;
- load R1, from the passive directory, but before placing click ■□■ to rotate the part, then place at 24.0,3.0.

When you are done, the screen should look as the one below.

Check attributes

Place the cursor between the components and press zoom (**[z]**, make sure caps lock is off!). Select a zoom two steps higher. The screen will enlarge around the cursor. Note that the reference PCB in the top left corner shows the PCB outline, the hole density and a small green cross at the cursor location. Press **[']** several times and note that the display cycles through displaying the component shape, type and reference.

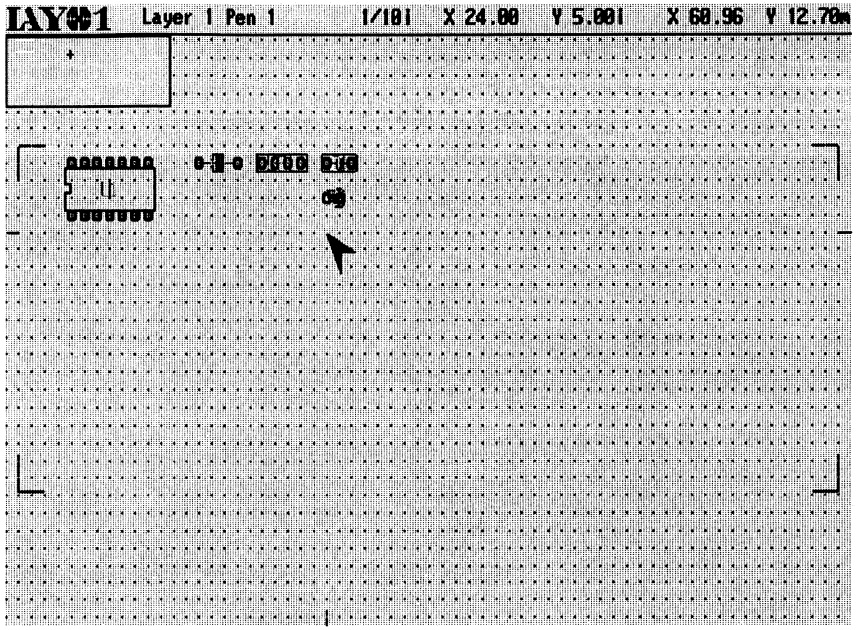


Fig. 4-4 Timebase project, few parts loaded

Save .CMP

Having entered part of the component list, we will save it. Go back to the Top Level, open again the File menu, and select **Save**. From the submenu select **Component list CMP**. The component list is saved. Reopen the File menu, select **Quit** and accept **Save?**. You return to the Project Manager and note that the right window now shows the file **TIMEBASE.CMP**. Place the highlight on this file and press **[↵]** to load it in the editor. Compare it with the list above to be sure there are no errors (the numbers in the last columns are for Layo1 internal use). Press **[Alt][x]** to exit the editor.

Create netlist

Now that we have successfully entered the component list, the next thing is to tell Layo1 the connections to be made. There are two basic methods to create the netlist. The first one is from inside the Graphics Editor by pointing with the mouse to the pins that are part of a particular net. The other method is to make a netlist with a text editor and loading this netlist in Layo1. We will show both of these methods here.

Using the net-editor

Start the Graphics Editor again. Layo1 will automatically load the TIMEBASE project and show the screen as you left it. Now open the Main menu, select **Options** and from the submenu select **Edit Netlist**. Layo1 will go into the net-edit mode as shown in the top left of the screen. We will create two nets for the components we have loaded. From the schematic diagram (see illustration at beginning of this chapter), you can make out the following nets:

```
net # 1: X1(2) U1(3) R1(2) C2(1);
net # 2: R1(1) U1(4) C1(1);
```

Create net

Locate X1. Make sure the component reference is displayed (use ['']) and zoom to scale 6). Toggle the pin number display to on with **[n]**. Place the cursor on pin 2 of X1 and click ■□□. Confirm **Create new Net?**. Notice that the selected pad is highlighted. Move the cursor to the next pin in the net (U1 pin 3) and click ■□□ again. Notice that the created net (now still only two connections) is highlighted.

Repeat this for R1 pin 2 and C2 pin 1. To exit from this net, click □□■. The color of the pads changes to indicate 'part of a net, but not routed yet'. Do this also for the second net. Exit the net editor by clicking □□■ once more. If at any time you make an error and cannot recover, just exit the Graphics Editor without saving anything and start over again.

Press **[z]** and go to scale 5. Press **[^F4]** (Control-F4), and Layo1 will show the ratsnest. Press **[F5]** to redraw the screen and clear the ratsnest. Open the File menu and select **Save/ Netlist NET**. Next press **[F4]** to save the design itself, and Quit the Graphics Editor.

Using netlist file

Normally you will not enter all nets manually from the net editor. A better way is to prepare a netlist with a text editor. Place the highlight on the file **TIMEBASE.NET** in the right window and press **[↵]**. You can see that the netlist is just a text file, and can be edited and created with Layo1's built-in text editor. But this is not a typing lesson. We have already prepared a netlist for this project for you to use.

Exit the editor. We will clear the timebase project we have constructed so far. We will then continue using the net- and component lists we prepared for you. Go to the menu page **Project & User Manager**. Verify that the Timebase project is selected. Select **Remove Project Directory**. Affirm the query. Now all that is left of the painfully constructed Timebase project is your experience... Next select **Create Timebase Project**.

You will see that all files are again displayed in the right window. Now continue with the actual layout design at the section titled **Design layout** below.

Design layout

If you are at this point not looking at the PCB Configuration screen, start the Graphics Editor. Because this is a new design, the configuration mode is automatically activated (you can access the configuration menu any time from the Options/Edit Design CNF menu). The **PCB configuration display** similar to the one below will appear:

Initialization

Through this menu you can define various parameters like PCB size, drill diameters and offsets. You can also display the **Pad Table**, which contains pad sizes and attributes. We will skip that for now, though. We will just set the PCB size: leave the PCB X dimension at 160 mm, and set the Y dimension to 50 mm.

Use the commands and keypresses you have used in Chapter 3 ([↵], confirm etc): Select **Board size Y**, answer the units query to use metric input and enter 50. Select **'Save Config'** at the bottom and save the configuration. At this point, you will enter the Graphics Editor.

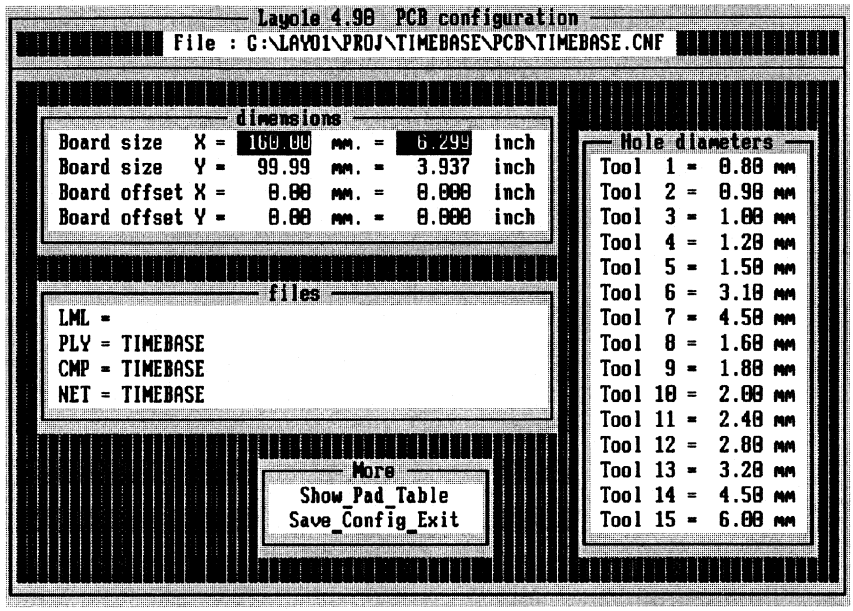


Fig. 4-5 PCB configuration screen

Project loading

Your timebase project will be loaded as follows. The corner lines are placed. These show the dimensions of the PCB. The components are placed in the PCB area. Next, the netlist is loaded and the **Ratsnest** set up. The ratsnest shows all connections present in the netlist as direct connections. Note that the pads that are not used (not in the netlist) have a different color then the others. At the top of the screen, in the Status Line, you will see the following information:

- the selected layer,
- the selected pen,
- the grid size,
- the X,Y coordinates in inches and millimeters.

The display should be similar to the one below

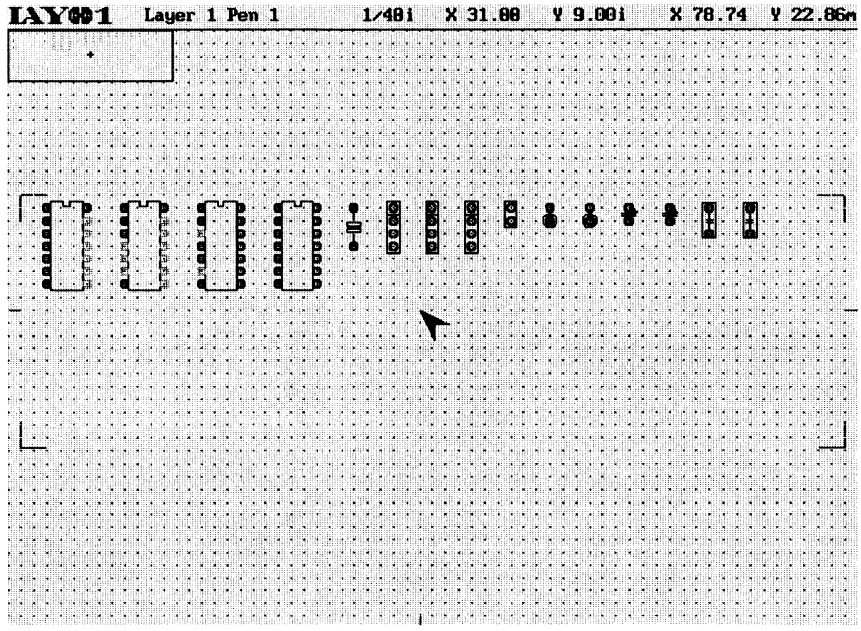


Fig. 4-6 Initial timebase layout

Set grid

Before we edit the PCB we first modify the grid size. Set the grid to .1 inch (**[F9]** or **[F10]**), and snap the cursor to it (**[*]**). A .1 inch helpgrid is displayed.

Select zoom twice (press **[zz]**) to make sure that the design is zoomed to just fill the display.

Positioning components

Activate the **Blockfunction** by pressing **[b]**. This will be used to move and/or rotate components. `Get block` is displayed at the left of the status line to indicate the current active function. Put the cursor on **pin 1 of U1**, this is the leftmost IC with coordinates **2,1** (X,Y) on the imperial ruler.

If you have difficulties with precise placement, use **[*]** to snap the cursor to the grid. Confirm when you have the cursor positioned. The component will be highlighted and the connections to other components are shown. This is very useful to position components to minimize track lengths and crossings. Next position the cursor at **8,9** and press □□■, to place the component.

Manual placing

Now move component **U3** from **20,1** to **14,9**. It is possible that the component name is different, but you should use the indicated position. This component is also hooked at pin 1.

Similarly grab **R1** at pin 1 (**41,1**) and move it to **2,11**, but do not place it yet. Layo1 always show the connections to the closest node in the net. Note that R1's two connections are crossed. We should rotate this component.

Press **[F1]** to verify the mouse buttons for rotate, clear with **[Esc]**. (You can also use **[F2]** to rotate). Rotate twice, note that the connections do not cross anymore and place at **6,12**.

Grab **R2** at **44,1** and temporary leave at **2,9**. Use rotate to optimize the connections and place **R2** at **6,10**.

Return to Top Level. Redraw the screen (**[F5]**). If your screen does not correspond to the figure below, exit to the Project Manager without saving (Select File, Quit). Start **Layo1 Graphics Editor** and work through this chapter again.

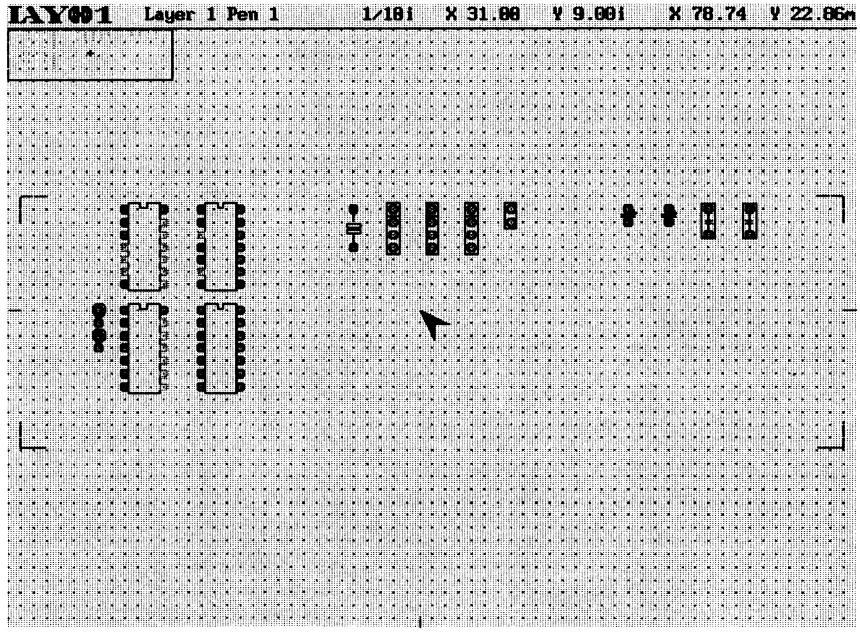


Fig. 4-7 Timebase components partly positioned

Predetermined placing

You should now know how to position Layo1 components. To continue with this tutorial, we will let Layo1 place the other components. Actually, we performed the placing and recorded the keystrokes in a macro (▸Macros) called **[Alt][1]**. When you replay this macro, it will duplicate the keystrokes to place the remaining components. Press **[ALT][1]** to start the macro. You will see the components being moved and placed, including the real-time rubberbanding. After the macro finishes, your screen should look as the one below.

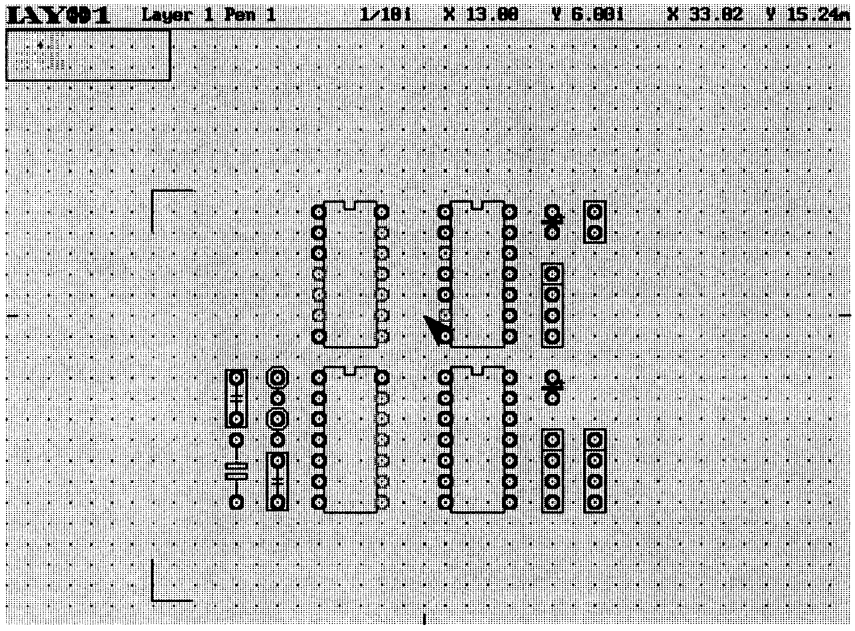


Fig. 4-8 Timebase fully positioned

Adjust PCB dimensions

Now we can adjust the corner lines that set the PCB boundary. Select **Corners** from the Draw menu. Position the cursor at **3,0** and confirm. This sets the top left corner. Note that Layer 15 is selected automatically. A window appears when you move the cursor. Position the cursor at **22,16** and confirm. The previous corner lines are replaced. The layout is automatically redrawn, centered and zoomed to fill the screen. The design now looks as follows:

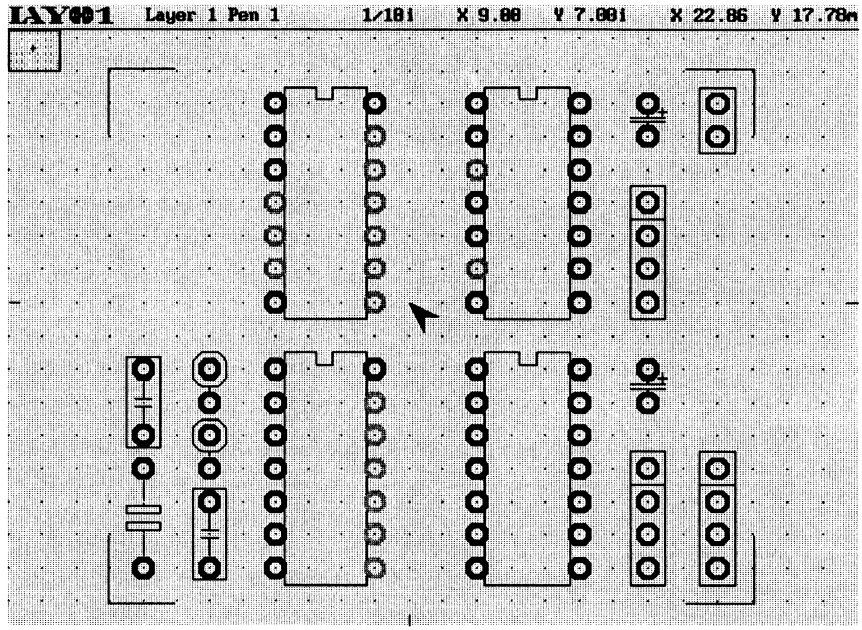


Fig. 4-9 Final timebase layout

Save your work

We recommend that you save your design regularly, to prevent losing your work because of a power failure or other mishap. Press **[F4]** to save the design. You may also activate the **Auto Save** option in the **User xx/ auto settings** submenu. This will automatically save your design at intervals you can specify.

Place text

Next we will place some text in the design. From the Draw menu select **Text**. A submenu will open, select **Input Text**. If this is the first time that text is used, you may get a request to select a font. Confirm the request for the default font by clicking ■□□ twice. Enter the following text: **Layo1**.

The text is attached to the cursor and may be moved across the screen. Pressing **[F1]** will show the various possibilities for placing and scaling text. Press **[+]** or **[-]** and you will see **Scale:** at the status line.

Set scale to **6** using the mouse or the **[+]** or **[-]** keys. Rotate the text **90°** and place at **4,3** by confirming.

After placing the text you can immediately enter another string. This time enter **Timebase**. Note that the rotation and scale selected above are kept. Place at **2,2**. Return to Top Level.

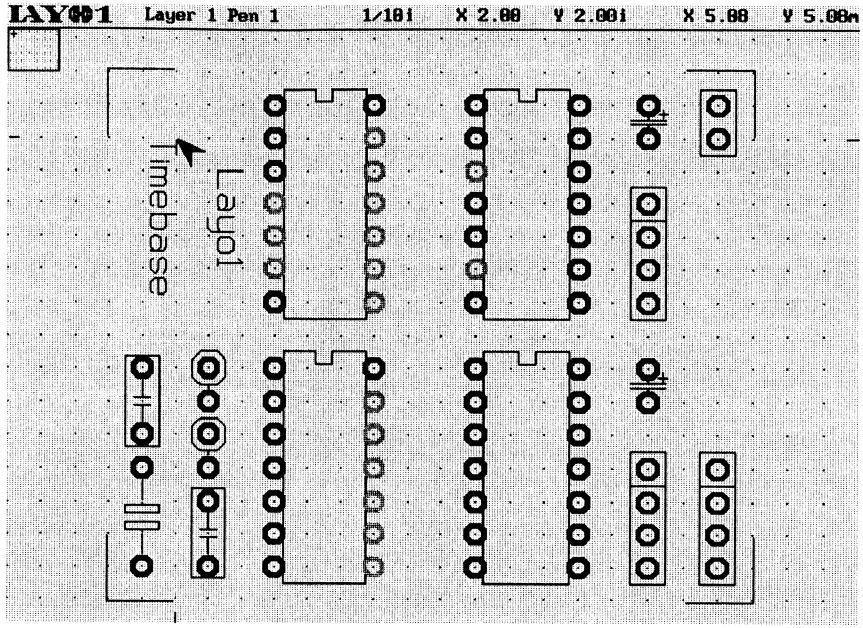


Fig. 4-10 Timebase with text placed

Using the ratsnest

Now on to the actual connections. First let us check the connections of a specific pad to other pads. Position the cursor at **3,11** and press **[^F2]**. Layo1 will highlight the connections as contained in the netlist. If you would use the router to route this net, these are the connections that would be routed. Do it once more at **18,1**.

If you want to check the connections of a specific component to other components, proceed as follows. Leave the cursor at 18,1 and press **[^F3]**.

Again, netlist information is used to show the connections. Finally, pressing **[^F4]** shows all connections in the design as a ratsnest. Only connections that would have been routed would not be shown. After this last action, the display should look as below:

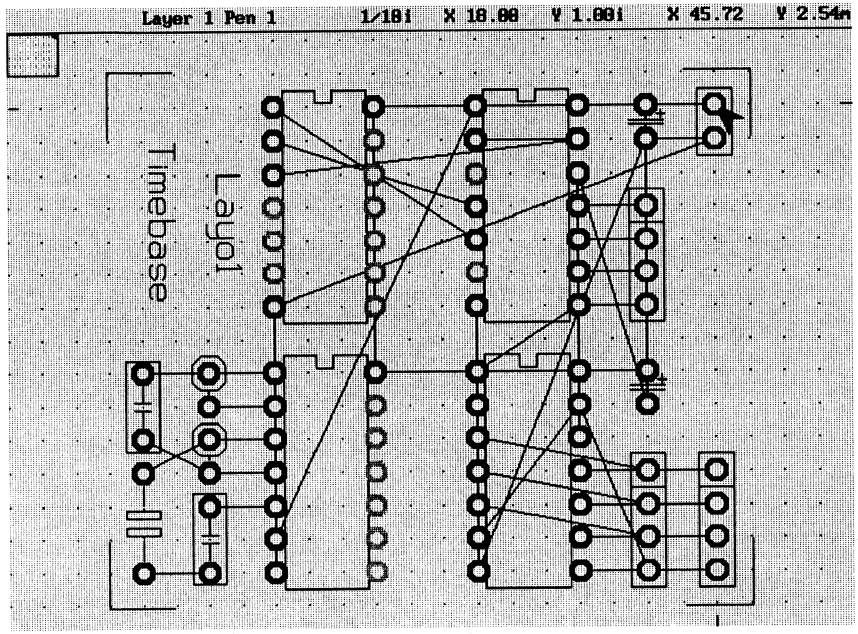


Fig. 4-11 Timebase ratsnest

Note that the ratsnest displays connections as contained in the netlist. This may not be the shortest route. However, the router will route the shortest routes depending on the way you set it up. Redraw the screen to clear the ratsnest.

Using the router

Layo1 contains a 'quick and easy' autorouter combined with optimum possibilities to manually edit track routes. (See the section 'Our philosophy' in Chapter 1). From the Router menu select **Route all**. A setup menu will be displayed.

Setup

Set the various options as follows (see the Reference Guide for additional information on the meaning of each of the options).

- Set `Grid size` to 1/40 using **[+]** or **[-]**. This is the grid that the router will use to make turns and corners in the traces.
- Set `Bus router` to No and `Lee router` to Yes. The Lee router is better suited for individual traces.
- Set `Via router` to 0; we will not let the router use vias to route on different layers.
- Set `45° routing` and `Pad to pad` to Yes. The last setting will restrict the router to connect only pads and not traces.
- Set `Horizontal` to 1 and `Vertical` to 2. This specifies the layers for traces in those directions.
- Set `Max. xy error` to 32, `Max. len error` to 16 and `Rout len test` to 0. These define the maximum length and strategy for each trace related to the ratsnest length. See ▸Router setup, Chapter 3, Reference Guide.

The final setup should be as shown below. Exit the setup menu with □□■.

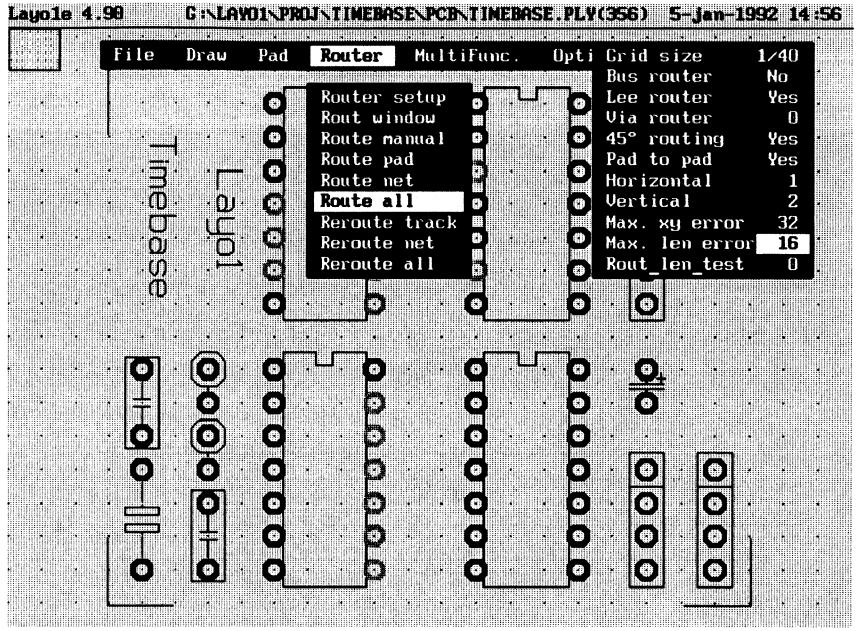


Fig. 4-12 Router setup menu

Route

Start the router with ■□□. `Route all` is displayed in the status line. Note that the color of a pad changes when it has been routed. When the router stops the top right of the screen will show `End....` The routed PCB is shown below. Any connections that had not been routed would also be displayed. Return to Top Level.

Edit tracks

Now, if you had done the routing, you would probably have routed some traces differently.

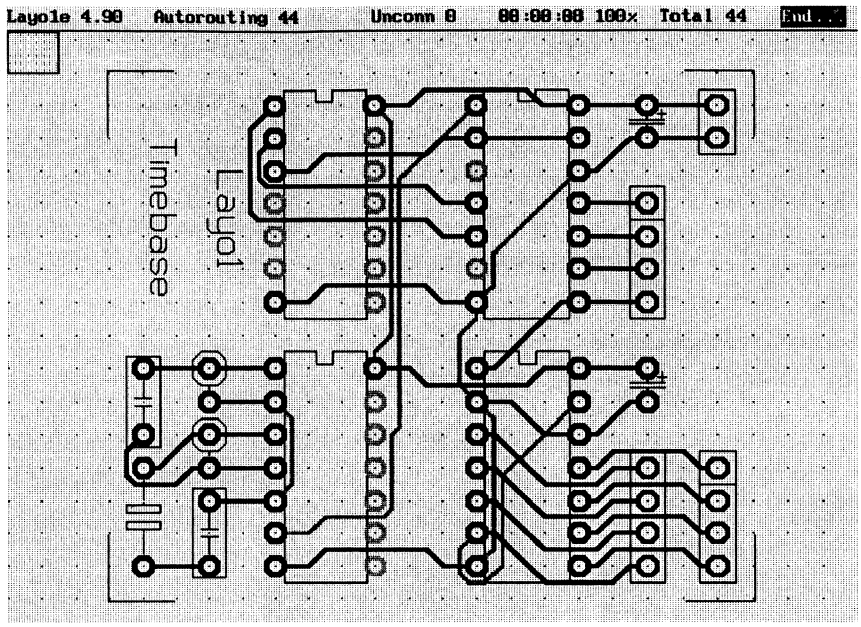


Fig. 4-13 Routed timebase layout

Lets make some adjustments to our own preference. Position the cursor at **11,1**. Set grid size to **1/40** (F9, F10). Pressing **[=]** or clicking ■■■ takes you into the edit mode. A track segment will blink. Confirm to `found?`.

We used this edit mode already in the short tutorial in the previous chapter creating the custom DIL16 shape. We have now 'hooked' a track section to the cursor. Moving the cursor will also pull the track segment to the new position. We can then use **[+]** or **[-]** to step to another segment, and pull that to another position.

Actually, we step through the datalines in the design (▸Vectors, Chapter 2, Reference Guide). Because the sections of the track we are working on have been placed consecutively, we can 'step' through the full trace run. Press **[-]**, the cursor will move to position **8.75,3.25**. Move the cursor to **9.5,1.5** and press **[-]**. Move the cursor to **9.5,13.5** and press **[-]**. Move the cursor to **6,14**. Return to Top Level. Press **[F5]** to redraw the screen.

Optimizing

You will also note some square corners. Position the cursor to **9.5,13.5** .This is such a corner. With the next function we will change any square corner in the design to a mitered corner. Set zoom to **7**. From the submenu `Options/Optimizing` select **Sq45 opt 1/40**. This will replace all square corners with 45° corners. The '/40' part of the function specifies a 1/40 inch grid for the miter, making it small. The result can be seen in the next figure.

Erasing tracks

Deleting tracks is just as easy as drawing them. Position the cursor to **11.5,11.5**. Press **[Del]**, then confirm. The track section will blink, and is then deleted. Another way is to first hook a track, and then delete it. Position to **14,10**. Erase this track by pressing ■□□ and confirming. Redraw the screen (F5).

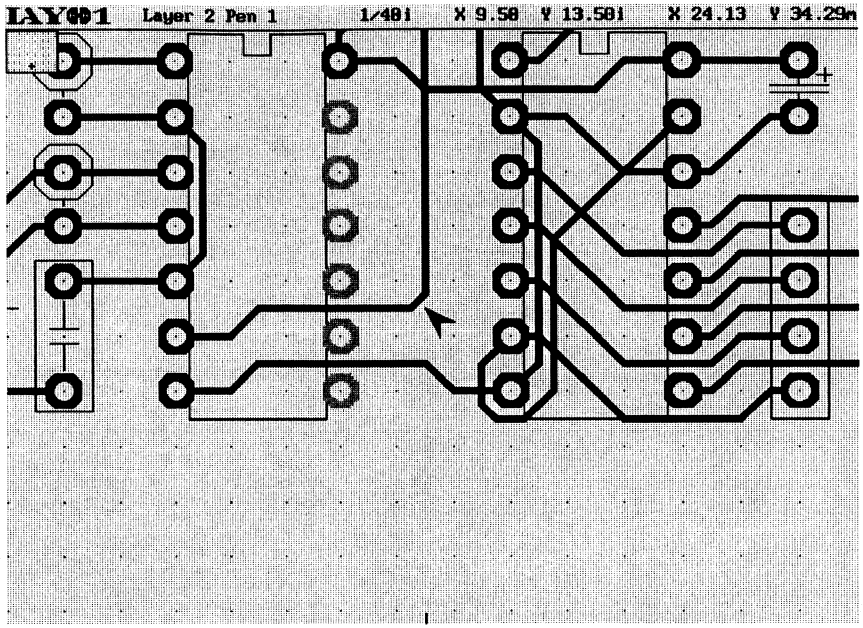


Fig. 4-14 Optimize mitered corner

Routing manually

We will now manually reroute the tracks we have just deleted. From the Router menu select **Route manual**. Position the cursor to 14,10 and confirm. Next move the cursor to 11,14 and confirm. The connection between these nodes will be routed. Press □□■. This will release the cursor for another 'hook' action. Similarly, connect **11,15** and **11,10**. Press **[z]** twice to zoom to full screen display. Return to Top Level.

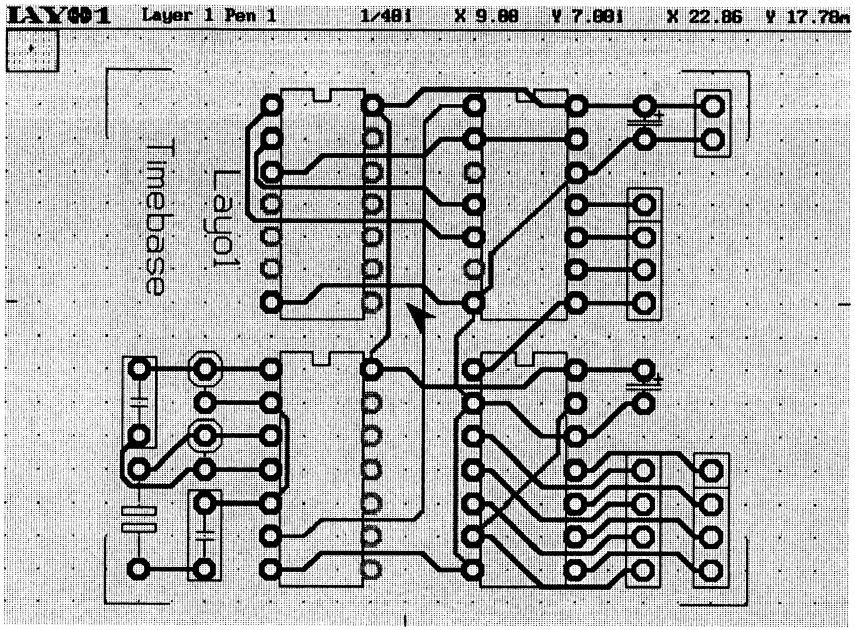


Fig. 4-15 Manually routed tracks

Defining pads

Another thing you might want to change is the shape and/or size of pads. From the Pad menu select **Define**.

A window will appear in the upper left corner showing the various pad definition options. Select **Adjust 4**. With this option, you can change all four sides of a pad simultaneously, so that the shape is preserved.

An edit window is shown in the upper left screen area displaying an enlarged image of the pad. The cursor is changed to a cross-hair and placed on one of the sides of the pad. This is the 'side-select' mode. With the **Adjust 4** mode it does not matter at what side the cursor is. The display will look as shown below.

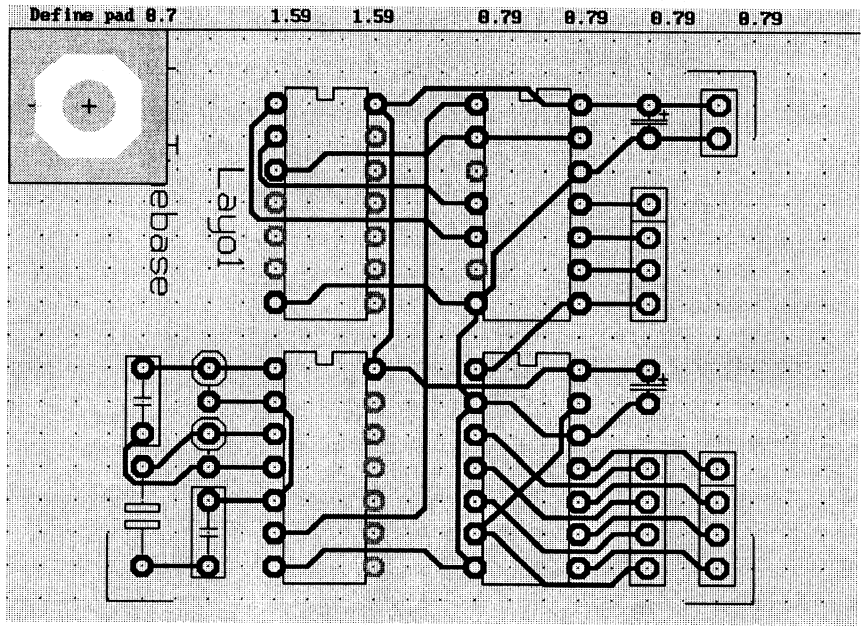


Fig. 4-16 Pad define mode

The status line displays the pad number: **Define Pad 0.7**. Next to it the pad size in millimeters: **1,59 1,59**, and the dimensions from pad-center to each of the sides: **0,79 0.79 0.79 0.79.**

Now confirm. The cursor changes to a small circle ('adjust-mode'). Use the cursor keys or the mouse to set pad size to **1,39**. Press **[Esc]** to get back to the define submenu. Select **Octagon 1/4**. Now all # 0.7 pads in the design are adjusted to the new dimensions. Return to Top Level.

Defining pen widths

Looking at the design we realize that the ground and power traces could have been a bit thicker. We will therefor change the penwidth and reroute a net with the new pen. From the Draw menu select **Define**. Select **pen 2**, change penwidth to **0.5 mm**. Return to Top Level. The top of the screen will show pen 2 active.

Rerouting a net

Now reroute a power net. From the Router menu select **Reroute net**. Place the cursor at 18,1 and confirm. The selected net will be re-routed with a wider track. Repeat this for the net at **18,2**. The result should look like the layout below. Return to Top Level.

Design rule checking

Layo1 can also check if there is enough clearance between tracks and pads, and if there are any wrong connections (as long as a netlist is available). First save your work with F4. Exit and return to the Project Manager. From the menu page Project Manager select **Design rule check .32 mm**. Layo1 will now check for .32 mm (1/80 inch) minimum clearance between elements (all combinations of tracks and pads).

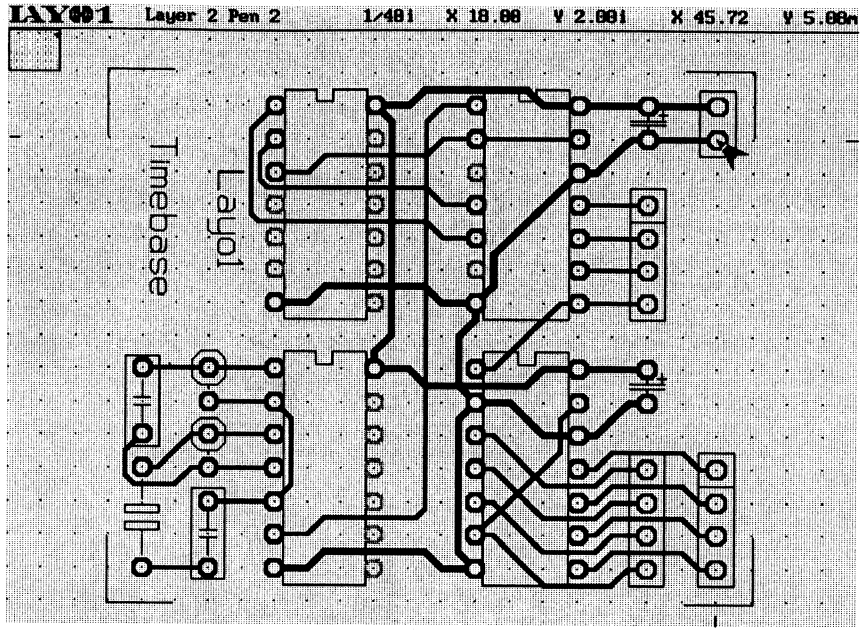


Fig. 4-17 Final timebase layout

DRC will show its progress on the screen and also write a report to the file **TIMEBASE.DRC**. This report can be viewed by loading it into the editor.

Checkplot

Finally satisfied with the design, we will make a checkplot on our printer. Restart the Graphics Editor if necessary. From the User xx menu select **Printer settings**. Select the printer you want to use. Check all settings for your particular setup. If you need to change any, place the highlight on it and press **[↵]**. Save the settings and exit.

Selecting layers

You may print layers separately using the display/hide layer feature. Press **[d]** and the following screen or one similar to it will be displayed.

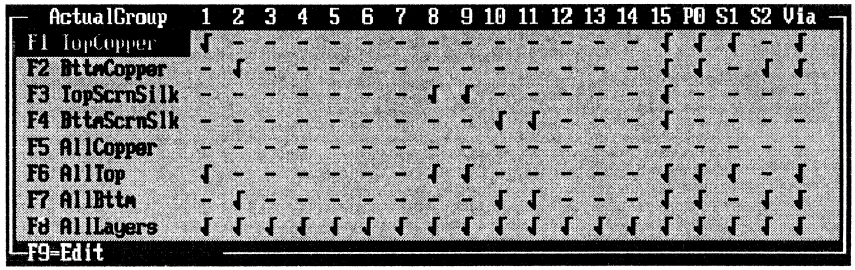


Fig. 4-18 Layer select menu

This gives you the option to define up to eight groups of layers, and select any group for display. This will also affect the checkplot output (however, if you use the separate Output Driver for production plots, you can select layers for output independent of this setting). The possible settings are all discussed in detail in the Reference Guide.

For now lets set the F4 group to display the pads (0), both track layers (1 and 2), the text layer (9) and the corner lines (15). Place the highlight on F4 and press F9. Now enter a meaningful name for this group. That will make it easier later to remember the layers in the group. Call this group **CheckPlot**.

Select display

Move the highlight to each of the entries on the line and toggle the checkmarks for layers 0, 1, 2, 9, and 15 to on. Reset all others. You can toggle the checkmarks with F9. When you are done, place the highlight on the F8 line and press **[↵]**. You will return to the Graphics Editor. All layers should be visible. Now press **[d][F4]**. Only the previously selected layers are now visible.

Making a checkplot

From the Options menu select **Printer checkplot**. A submenu with printoptions will open. Set **Plotscale** to **2**. Set **Pads open centers** to **Y**. Set **Plot comment** to **Y**, this will print the project name, date and time of the plot. Make sure your printer is loaded and on-line. Select **Start output**. The print will start, and you can see the progress at the top left of the screen. The result should be similar to the plot below.

When the print function has finished, save your design (F4) and exit to the Project Manager.

This ends this interactive tutorial. The next chapters will go into more detail on routing.

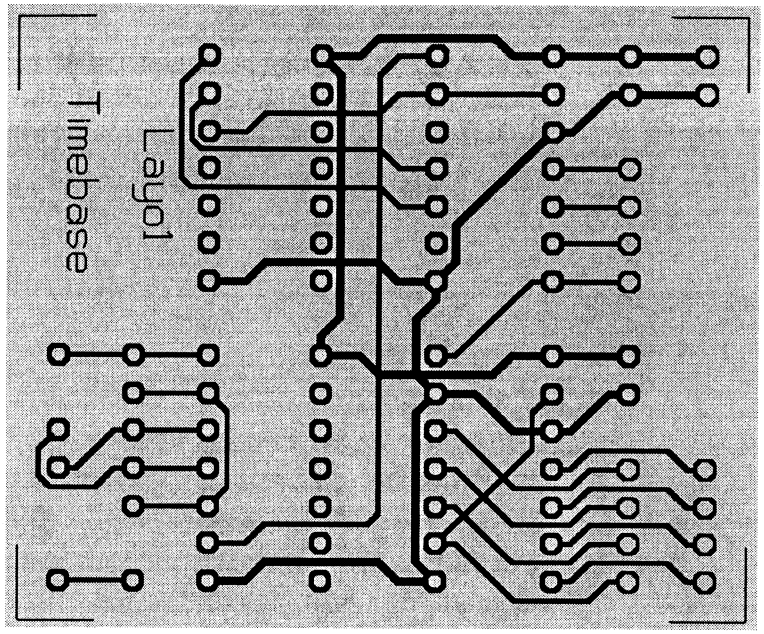
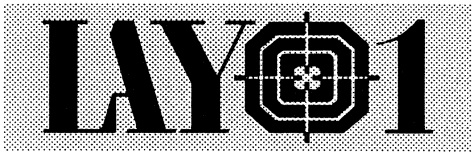


Fig. 4-19 Checkplot scale 2:1



CHAPTER 5

Routing functions

This chapter contains a small tutorial emphasizing various routing functions and creating a netlist. If you work through it you will get a feeling for the possibilities and aspects of using Layo1's built-in router to route your design for you. This will be a simple dual layer PCB. To be clear, lets review some terms.

In Layo1, manual routing means identifying the routes to be placed and letting the program do the drawing. Manual tracing means that you actually trace the connections yourself, inserting nodes wherever you change direction. In the fully automatic mode, the program will itself select the nets and draw the traces, if necessary changing layers and inserting vias on the fly. All this is controlled by the router setup parameters that you specify before starting the router.

Start the design

Start Project Manager and go to the Project & User Manager menu page. Activate **Select Project** and name it **TUTORIAL.** Next we must create the directories and default files for Layo1 to be used with the project 'Tutorial'. Make the required directories with **Create Project.**

Start Layo1 Graphics Editor. Because you defined a new project, you are automatically taken to the Design CNF screen. Set the PCB size to **90 x 50 mm** and save the configuration.

Layo1 will now try to load the component shapes for 'Tutorial' into the editor, so that you can position them. But in this case there isn't a componentlist (it would have been called `TUTORIAL.CMP`). Layo1 will generate a warning message: `file ..\TUTORIAL\PCB\TUTORIAL.CMP not found.` Ignore this message and continue by pressing **[↵]** or confirming.

Manual loading

We will manually load the parts for this design. First a standard 16 pin DIL. Set the grid size to 1/10 inch. Open the submenu File/ Load. Select: Component LMC. (You could also use the **[Alt][g]** macro created in chapter 2). A window opens with a pathname preset to `..\LMC\`. Press **[↵]** and from the resulting directories display select the `ACTIVE` directory.

From this directory select the shape `DIL-16`. Layo1 will load the shape at the cursor position and will open a couple of windows to request the component attributes. For 'Type' enter **TEST**, for 'Reference' enter **IC** (the attributes are also shown at the status line). The DIL-16 is positioned at the cursor position. Rotate it once and place at position **5,5** by clicking □□■ (use mouse help **[F1]** to review the rotate command).

Next we will load a connector with 2 x 17 pins to connect the board to some external device. We will get the shape for it from the **CONNECT** component bank. Position the cursor to **6,2**. Again open file/ load/ component LMC, now select directory CONNECT. Load the shape **SIL-2x17**. For 'Type' enter: **CONNECT**, and for 'Reference' enter: **J**. Rotate the component once and place at **30,2**.

We need another DIL-16. Because you have already loaded it once, it will be in the list of recent used shapes maintained by Layo1. Let us use this then. Click □■■. From the window that opens select **Directories**. Click □■□ and you will see recent accessed files. Select **\ACTIVE\DIL-16.LMC**, and for 'Type' specify **TEST** and for 'Reference' **IC**. Rotate and place at **11,5.**

Manual tracing

We will first put down some power traces with pen #2. Press **[p]** and select pen 2. Position the cursor at **30,2** and confirm. This will activate the draw function from the Top Level. Layo1 will show the net data at the status line, but because there is no net defined yet, the netnumber is zero, as is the number of pins in the net. Position the cursor next at **8,2** and confirm. Now let us continue at the other layer.

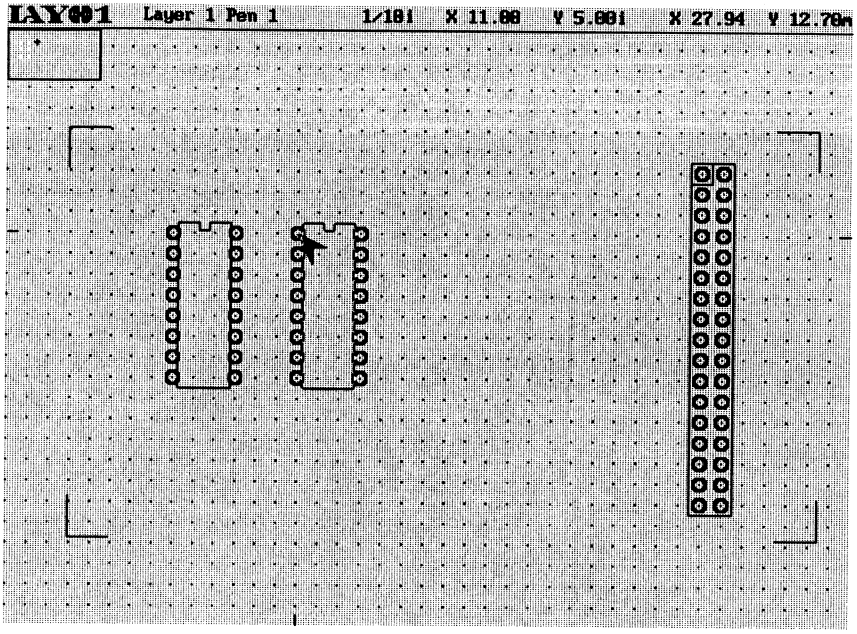


Fig. 5-1 Tutorial PCB, shapes placed

Check the status line for the current layer, probably 1. Press **[F8]** to change the layer and verify at the status line. Position the cursor at **8,5**. Press **[Esc]** to exit from the Draw function. Note that although the traces look connected, they are not because they are on separate layers. We will fix that later.

Start again at position **14,5** . Confirm and go to **14,2** and press **[Esc]**. We will place a via here to connect the two traces. Zoom in to level **7** to get a clear view. Press **[F2]** to place the via at the current position (14.0,2.0). Place another via at position **8.0,2.0**. Press **[z]** twice to zoom the design to fit the display.

Manual routing

Now let us use the services of Layo1's built-in router. We will identify a connection to be made, and let the router do the actual tracing. We call that manual routing; but in the literature it is sometimes called semi-automatic routing as well. From the Router menu select **Router setup**. Set the router options as shown in the illustration below (use **[+]** or **[-]**).

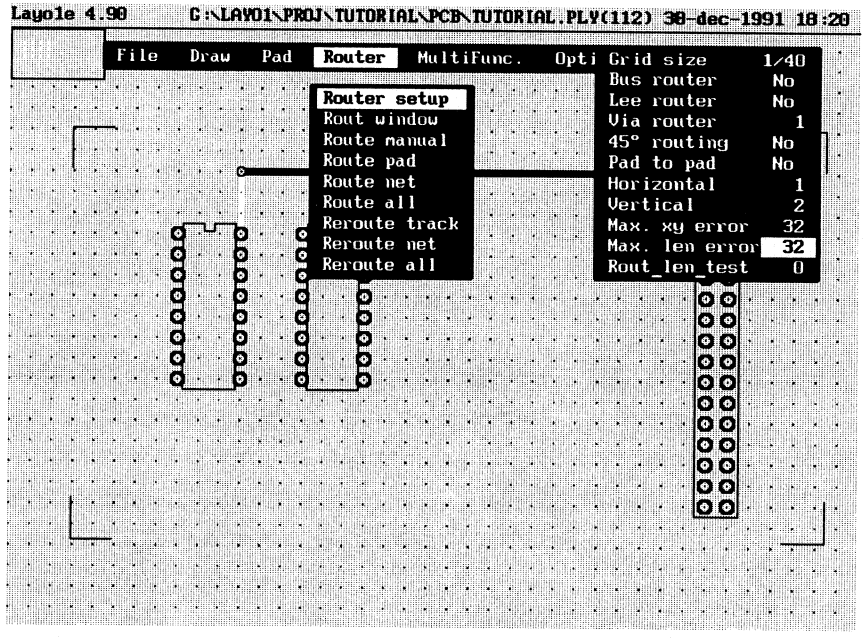


Fig. 5-2 Router setup for manual routing

Manually route trace

Exit the setup menu (**[Esc]**) and select **Route manual** from the Router menu. This is now shown at the status line. Confirm position **30,15** and move to position **11,12**. Layo1 will maintain a ratsnest line while you move the cursor. Confirm The track will be automatically routed between the two specified points. Because you allowed a via in the setup, Layo1 will use that to switch layers. A via will be automatically inserted at 11,15. Release the cursor by pressing **[Esc]** or clicking □□■. Using the same method, route a track from **11,15** to **5,12** and return to the Top Level.

Move the connector a little closer in, using the **Block** function. Position the cursor at **30,2**. Press **[b]** to activate the block mode. Confirm to 'hook' the connector and attach it to the cursor. The shape's color will change, and ratsnest connections will be shown. Move to **20,2** and press □□■. Redraw the screen.

Delete traces

Before trying yet another routing method, lets first delete some traces. We will do that by drawing a window and deleting all enclosed objects. From the Multifunc/ Window submenu select **Delete**. Confirm position **2,1**, this is one corner of the window. Move the cursor to **22,16** and confirm to set the window size. Reply affirmative to: **'Delete ?'**. All tracks inside the window will be erased. (We could also have delete the whole pattern at once using 'Delete layer'; see the Reference Guide). Return to top level and redraw the screen.

Full automatic routing is only possible if a **netlist** is available for the current design. You can generate a netlist easily using the **Net-editor**. We will create a short netlist and tell Layo1 to route the connections contained in it. You may also want to review the section titled 'Standalone design' in Chapter 4 on creating and using a netlist.

Warning! To maintain design integrity you **cannot** edit a netlist generated from a schematic capture program. If you try to call up the net editor in such a case, the message: **Project Manager active!** will be displayed. To insure design integrity you should make changes in the connections or the components to the Schematic Diagram only, and use the Forward Annotate function to transfer the changes to Layo1. Forward Annotate will be described in the next chapter.

Create netlist Now we will generate a netlist. From Options menu select **Edit netlist**. Confirm **20,2**. Because no net is active, and the clicked node is not in any net, Layo1 assumes that you want to start a new net. Answer **[Y]** to: `Create new net?` The selected node is highlighted. Also confirm **14,5** and **8,5**. These three nodes are now part of a net (see also the status line). They are all highlighted. Release the cursor using □□■. Now press **[Esc]** and answer **Yes** to `Quit Net Editor`. Go to the Top Level and press **[^F4]** to check the ratsnest.

In the same manner as above, again enter the Netlist Editor and define the following four additional nets (release the cursor with □□■ after each net):

- net # 2: **20,15 + 11,12 + 5,12.**
- net # 3: **20,6 + 11,6 + 5,6.**
- net # 4: **21,6 + 14,6 + 8,6.**
- net # 5: **20,10 + 14,10 + 8,8 + 5,11.**

Quit the net editor and return to Top Level.

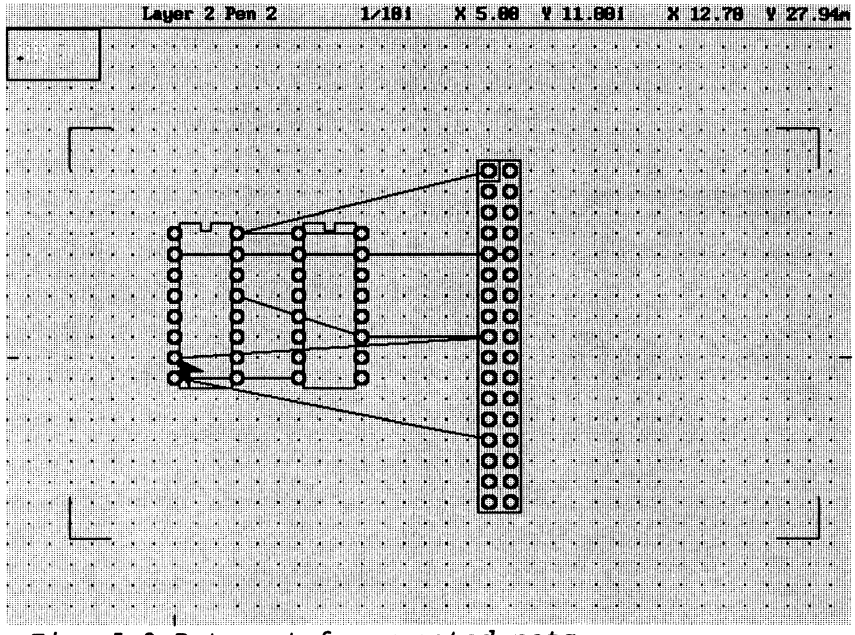


Fig. 5-3 Ratsnest for created nets

Manually route net

With manual routing of nets, we have full control of the order and way the connections are traced. We identify the nodes to be connected, and Layo1 merely does the drawing. We will show that here. Press **[p]** and select pen 1. From the Router menu select **Route manual**. Confirm **20,2**. This is the first node from net # 1. Next confirm **14,5** and **8,5**. In each case, the traces are drawn. Because the routing of 45° corners is disabled (see router setup above) Layo1 uses square corners.

Error detection

Lets now try another net, making a deliberate error. First set the router setup parameters as shown below, which among other things confines all traces to layer 1 and enables mitered corners.

Confirm the following nodes: **20,6 11,6** and **8,6**. Layo1 realizes that the first two are from net # 3, but the last one is from net # 4. Now if you want you *can* connect two nets. But Layo1 warns you with the question: `Route to other net?` Answer **No** and confirm **5,6** which is correct (see net # 3 above). Note also that now the track corners are 45°.

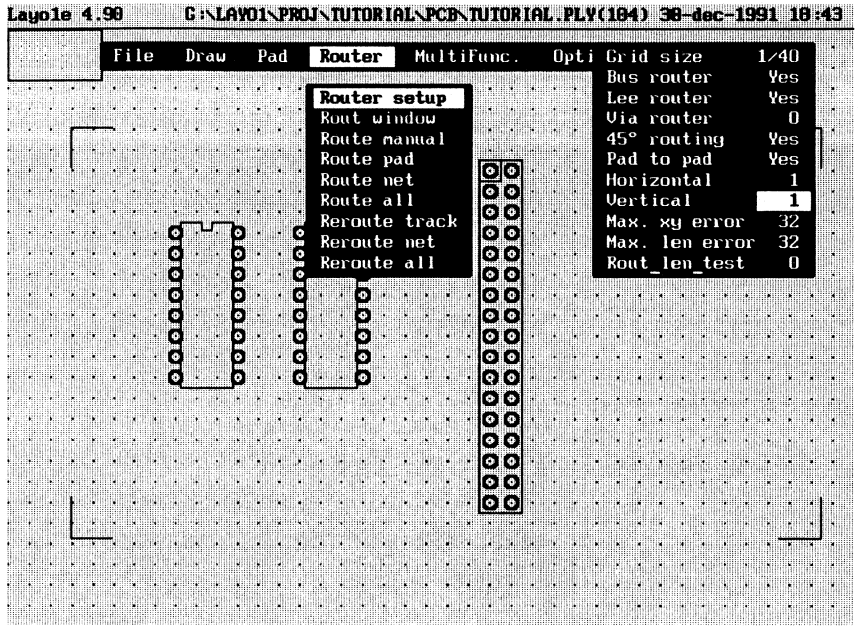


Fig. 5-4 Router setup single sided

Release the cursor and return to Top Level. Display the ratsnest and verify that the routed nets are considered completed. Their nodes also have a different color. This method of routing and the support that the program gives to it is very powerful if you need full control over the layout, yet maintain full control over the design. Now is a good time to save your work (F4). The result should be as shown below.

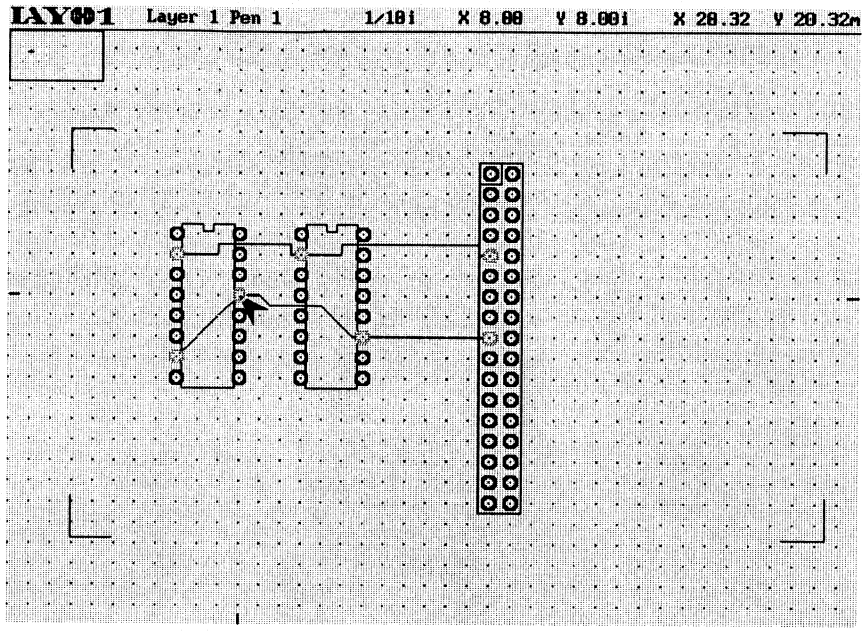


Fig. 5-5 Two nets routed

Full automatic routing

We will let Layo1 autoroute the remaining nets. We will keep the previous router setup, because it will be possible to route everything on just one layer (see router setup illustration above). This is the normal setting if you are building a single sided PCB. From the Router menu select **Route all**. The remaining connections are routed. At the end, the status line indicates 100% completion. The resulting layout is shown below.

Save the netlist by selecting **Netlist NET** from the File/ Save submenu. Quit the Graphics Editor. If you wish you can view the generated netlist from the Project Manager's editor.

This brings us to the end of this tutorial. When you have worked through the previous tutorials as well, you should now be ready to tackle more involved designs. Keep your Reference Guide handy. There are many more functions you will want to use; you will find that when you look up the function listing, that there is something for every situation.

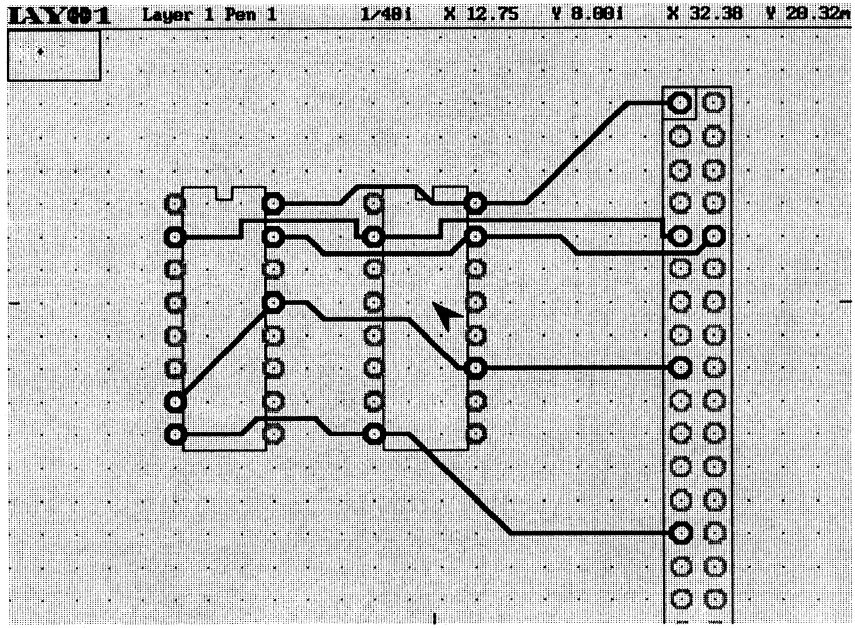


Fig. 5-6 Final tutorial PCB layout

Router tips

If you must route a complex PCB, take some time before you start to decide on the strategy to use. First determine whether you really need a double sided PCB. Single sided ones are much cheaper. You can do a quick test by turning the autorouter loose on the design. Even if it doesn't finish, you will have an idea whether a single sided PCB is feasible or not. Next determine if there are any nets (for instance power supply lines or high frequency lines) that are critical regarding their length and/or routing. Route these first by hand or semi-automatically.

Next you can use the router for the other nets. If you make a double sided PCB, use one layer for vertical and another for horizontal routes. Begin with no via's and small values for the length error parameters (see ▸Router setup in Chapter 3, Reference Guide). Then allow one via, and relax the error parameters. Check the layout between steps, because very often you can clear a path with just one or two manual changes.

Note that when you allow two via's, the router will tend place a via every time it changes from a horizontal to a vertical direction. A track going north, then east, then south will have two via's, and use two layers, even if it could have been routed all on one layer. This is not a shortcoming of the router, but part of the algorithm that assumes that you only allow vias if you want the router to make use of it as part of *your* strategy to design the layout. This consequent use of separate layers for horizontal and vertical traces will ensure a maximum completion rate for complex designs.

CHAPTER 6

Forward annotate

This chapter concerns itself with changing a design as a result of changes in the Schematic Diagram. If you use Layo1 integrated with a version of OrCAD/SDT, you can use a built-in function from the Project Manager, and this is what we will discuss. This function will preserve the design integrity. If you are using Layo1 stand-alone, you will have to do it manually. This is a little more involved. Connection changes can be made with the net editor (do not forget to save the modified net before leaving the Graphics Editor). You can remove components by deleting them, and add new ones by loading them. In both cases you have to adjust the netlist. The component list must also be saved separately.

Update the schematic

Start Project Manager and select the Timebase project. Select **Draft** from the Schematic Design Tools menu page. Place a capacitor (capacitor pol) from VCC to GND, to the right of C2. (don't forget the junctions!). The Schematic Diagram should now look similar to the illustration at the next page.

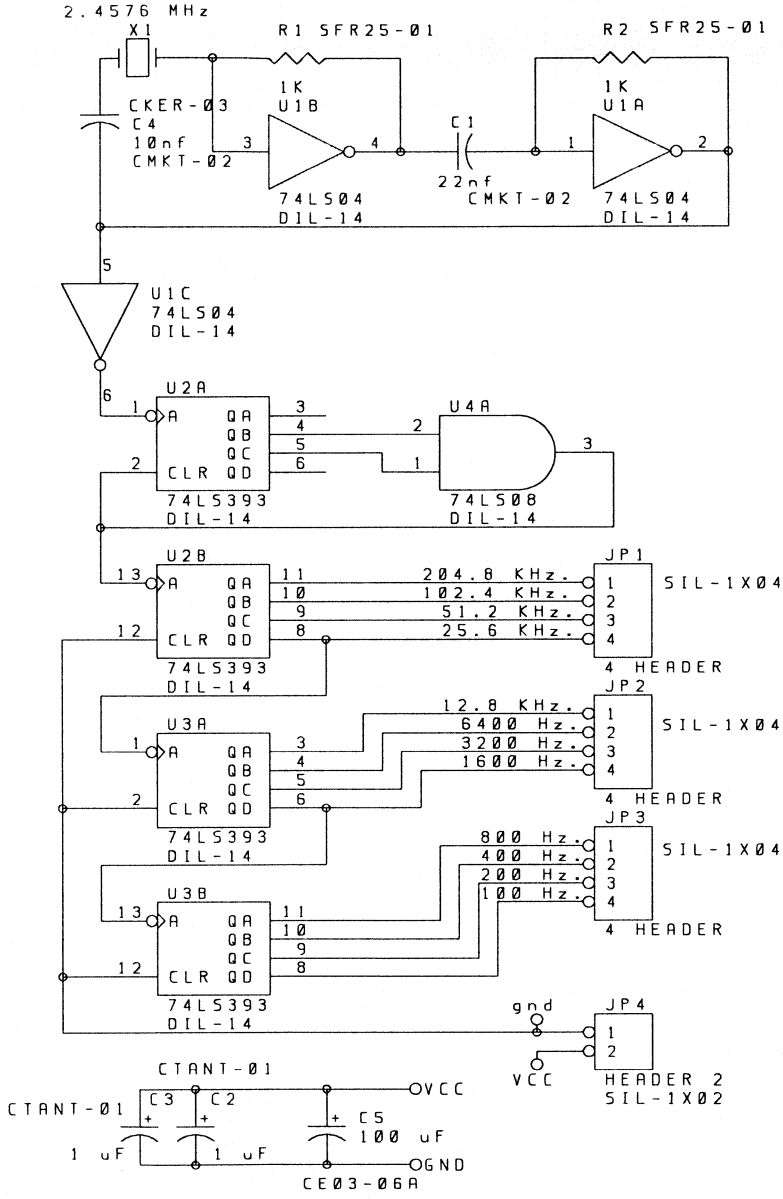


Fig. 6-1 Timebase schematic, C5 added

In the next section, use capitals for the shape name. Edit the new part's parameters as follows (consult the manual for OrCAD/SDT if necessary):

- part reference (Reference): **C5**
- part value (Part Value): **100 Uf**
- shape name (1st Part Field): **CE03-06A**

Exit the schematic capture program with **QUA**.

Update all files

Now we will have to update the net- and component list files. This is simple. From the menu page **Schematic Design Tools** select the function **Layo1 Forward Annotate**. You will see a lot of activity while the Project Manager first makes OrCAD/SDT generate the new net and component files, and then transfers the changes to Layo1.

Error log Note that if there are any errors encountered, Layo1 will save these in a file with the project name and the extension **.ERR**. The changes will not be implemented. But, if there are no errors, the Layo1 Graphics Editor will start up. The new component is positioned in the upper left corner. The ratsnest will show the connection to the new component.

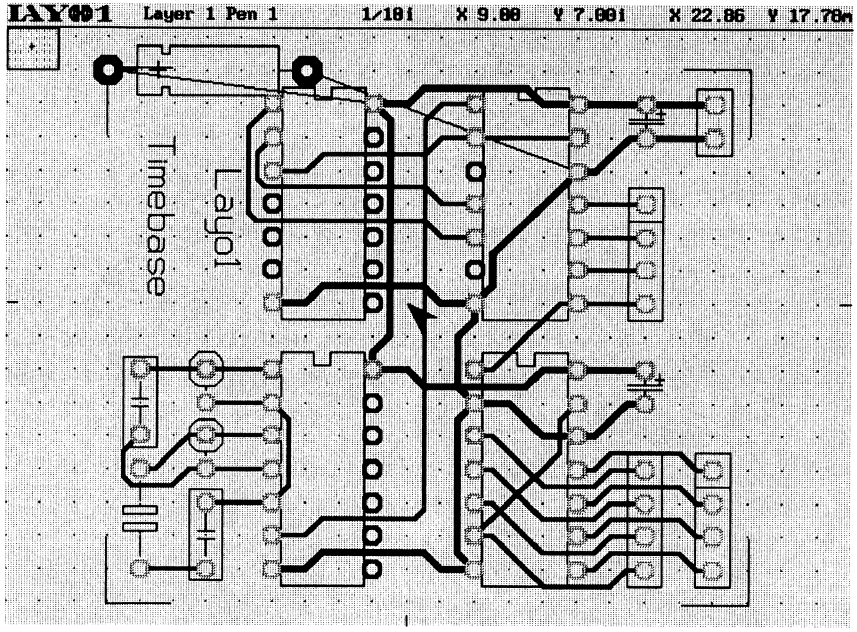


Fig. 6-2 Timebase layout, C5 loaded

We will place this capacitor at a free area on the PCB. Position the cursor at **18,10**. We will snap the component to this position by pressing **[u]**. This will open a window where you can specify the component to snap. Enter **C5**.

The capacitor moves to the cursor. Rotate it three times (use mousehelp, F1, if you are not sure how to do this). Place it by clicking □□■. Press **[F5]** to clear the screen.

Note that C5's pads have a different color, indicating that they still have to be connected to their nets. Because these are power traces we will again use a heavier trace. Open the pen menu by pressing **[p]** and select pen 2. Select **Route all** from the Router menu to connect the component to the existing layout. The final result is shown below. Save your work, quit the Graphics Editor and return to Project Manager.

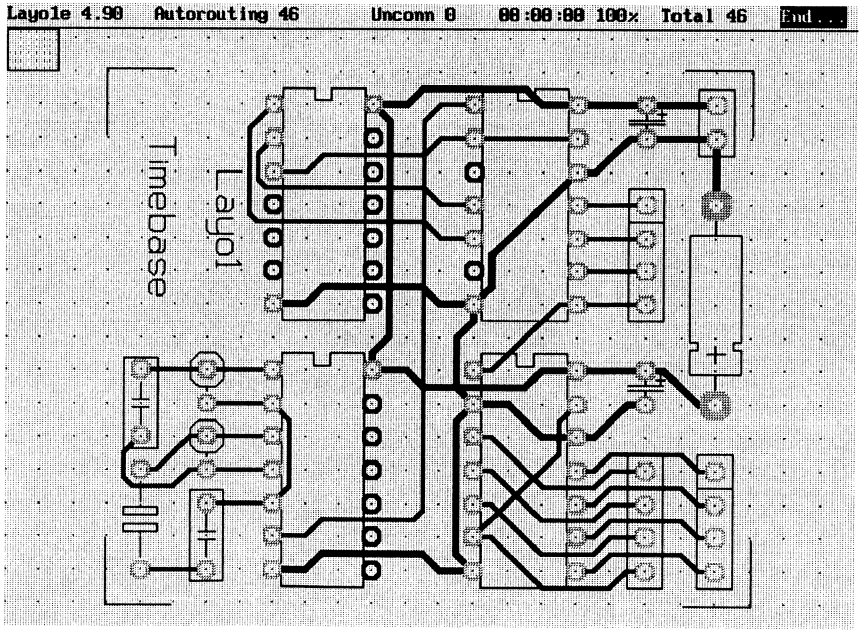


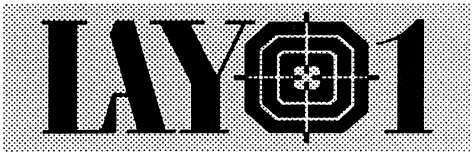
Fig. 6-3 Updated timebase PCB

Note 1: A change in component shape cannot be implemented with Forward Annotate because it is not an electrical change. You can change a shape by hooking the old component using the Block function and then replace the shape with **[^F3]**.

Note 2: **Never** change the references in your schematic diagram. Do not use the Annotate function from OrCAD if you have added a part, but rather set the reference for the new part manually. Part references are Layo1's means to keep the design intact. If you absolutely have to make changes, do as follows:

- Make a **'WAS-IS'** file as described in the OrCAD documentation;
- Edit the .NET file in Layo1's PCB directory for the project so that it corresponds to the WAS-IS file;
- Run OrCAD's **BACKANNO** function with the WAS-IS file to update the schematic diagram.

Note 3: OrCAD/SDT IV has a feature called **incremental annotation**. If your schematic change consists of added components *only*, you can use incremental annotate. OrCAD will number the new components but will not touch the references of the existing components. See your OrCAD/SDT IV Reference Manual for this feature.



CHAPTER 7

Output Driver

The outputdriver is a stand alone program that can convert Layo1 output files to industry standard output files. These files can drive penplotters, photoplotters and NC drilling equipment to generate documentation, layout films and PCB's for a professional end product.

The Output Driver program is not copy protected and may be copied and installed wherever it improves the production process. However, the copyright to the program is retained by Peter Baas.

Limited license

Peter Baas hereby grants you a limited license to duplicate and distribute the Output Driver program, on the condition that it will not in any way be modified or altered. You also may not request or receive any payment or other reimbursement from third parties for the use or distribution of this program.

The program contains 20 configuration pages that can be selected with **[PgUp]** and **[PgDn]** or through a selection window. Each configuration page contains all the data and parameters for a complete output job, including the device configuration and the data to be output. All configuration parameters are saved on disk.

We recommend that you use a consistent system for allocating the configuration pages. For example:

- Pages 1 - 10: testplots;
- Pages 11 - 18: photoplots;
- Pages 19 - 20: drill files.

Overview

The Output Driver has the following features:

- Menu driven;
- Selection of up to 40 recently used files;
- Most options can be directly verified and adjusted;
- Graphical preview of pads and layers;
- Extensive scaling options;
- Image move, rotate and mirror;
- Optional pad center holes to facilitate drilling;
- Automatic or manual pen change;

- Specification of any pen width and/or speed;
- Adjustable 'blackscale' (fill step) to generate normal or extra black plots;
- Scaling of all pads and pens for mask generation purposes;
- Drill program: a 'tool' selection for every drill;
- Via's, pads, SMD pads can be assigned a unique 'pen';
- 15 layers can be selected separately, each with a different pen;
- An extra pen selection for drawing a grid, the filename and a date/time stamp;
- Automatic flash/draw selection for GERBER output;
- Output directly to device or to disk.

Inputs and devices

There are many different files and datasets that can be selected. Also, the number of output devices and output destinations is large. That gives a very wide range of combinations for every user. You should therefor plan the output process carefully. But if you contract out the PCB production, leave the output plotting to the contractor. (See also ▸Quality Assurance in Chapter 1).

Input files

In the interest of maintaining upwards compatibly, the Output Driver accepts also output files from previous versions of Layo1. The Output Driver can handle any Layo1 file with extension .BNK, .LAY, .LMC, .LML (with or without its .CNF) or .PLY (with its .CNF).

Output devices

Several output standards are supported for maximum flexibility:

- DMPL Houston Instruments plotters;.
- HPGL Hewlett Packard compatible plotters;
- GERBER For photoplotters and some drilling equipment;
- EXCELLON Drilling equipment (US);
- SIEB&MEYER Drilling equipment (European);
- POSTSCRIPT PostScript compatible printers.

A **DXF** driver is planned for a future release. Note that prototype quality output can be generated directly from the Layo1 Graphics Editor to HP LaserJet and Epson compatible printers (see Chapter 4 and the Reference Guide).

Delivery by modem

A Layo1 file is quite compact and contains all information for producing a PCB and supporting documentation. The output files may be 50 to 200 times larger.

If you want to deliver your files to a production site by modem, you should send the original Layo1 files and have the output files generated at the production facility. You are allowed to copy and install the output driver program at the production site, see the limited license statement at the beginning of this chapter.

Setting up an output job

All parameters for a particular job are associated with a configuration page. To set up a page for a job, there are two basic data sets to be defined: the output device and the data to be output.

Select device

To configure a device for a job, you select **Device** from the Main window of the Output menu screen, and select a device. When you press **[F9]**, a window will open to let you edit the device specific parameters. You can find more info on this data in the Reference Guide, Chapter 7.

Select data You must of course also specify the data you want to output. There are currently three data sets you can select from the **Main** window from the Output screen:

- Output selected pad-groups and layers,
- Output the drill hole pattern,
- Output a drill file for NC drilling equipment.

After you have selected the data set, you can further refine the output and its appearance by editing the data in other windows of the Output screen.

The following section will discuss a typical case using the previously designed TIMEBASE PCB.

Output plot

Let's assume you have an HPGL compatible penplotter at parallel port LPT1. You wish to plot layer 1 (solder side), layer 2 (component side), layer 8 (part shapes) and layer 9 (text) as a stuffing guide for the timebase project.

Enter the Project manager and be sure that the Timebase project is selected. From the Project Manager menupage Project Manager select **Layo1 Output Driver**. You will be taken to the Output screen as shown below.

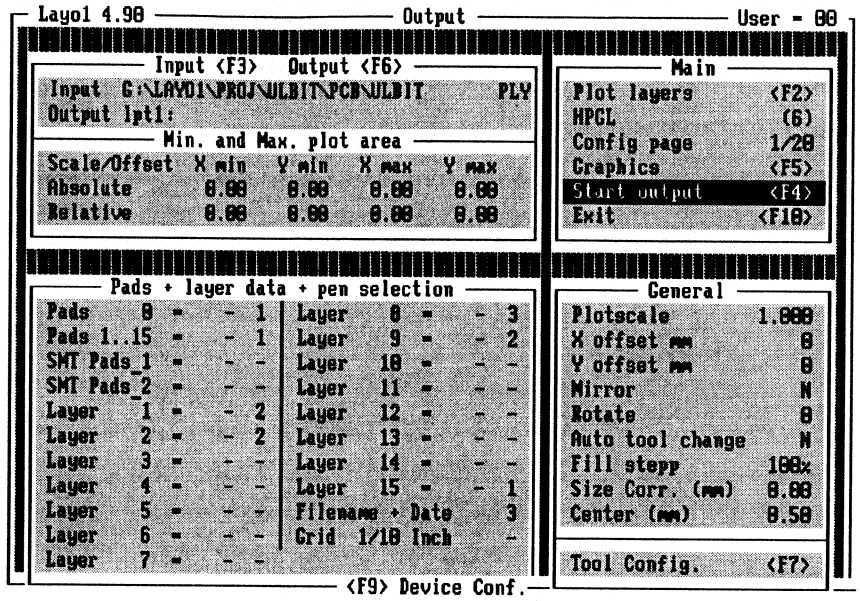


Fig. 7-1 Output Driver Output screen

From this screen select **Config page 1/20** and choose a page number, lets say page 5. Go to the second line in the Main menu, which default shows **HPGL**. Press **[F9]**, and you will enter the **Device configuration** screen. From the window titled **Device basics** select Driver type, from the window that opens select HPGL. Next select the Device name entry and enter a meaningful name for later reference, like **Stuffguide**. This name is only for reference purposes and has no effect on the output. Save and exit this menu and return to the **Output** menu.

File and data set

Next we will set the remaining entries in the two top windows. Select the **Plot Layers** entry from the Main menu. You will see a window with three choices. Select the choice **Plot layers**. Now open the **Input** entry in the Input/Output window (you can use F3 as indicated). From the **File type** window select **Design CNF PLY**. This tells the output driver that the input is a full Layo1 design complete with configuration file.

Note that you can also output single components (.LMC) and other files. These are explained in the Reference Guide. Next a window will open requesting the name of the design. Press **[F8]** and select the appropriate directories to navigate to the `..\proj\timebase\pcb\timebase.ply` file. Set the **Output** entry to **lpt1:** if necessary (use the highlight or F6).

Pen select

Now we will select a pen for the selected layers to be plotted. Move the highlight to the lower left window titled **Pads + Layerdata + pen selection**. Note that the number of data items present on each layer is shown. Select plotter pen # 1 for the following entries:

- Pads 1..15,
- Layers 1,2,8 and 9,
- Filename and Date.

If any other layers seem to contain data (layer 15), select pen 0 for them, effectively excluding them from being plotted.

General data

From the lower right window titled **General** set and modify as necessary for your particular plotter setup:

- Plotscale: 2.00;
- X and Y offset: 0;
- Rotate, Mirror: N;
- Auto tool change: Y;
- Fill step: 100;
- Size Corr: 0.00;
- Center: 0.5.

Fill step

Fill step specifies the overlap of the lines drawn. At 100%, there is no overlap. If you want to know more of the other parameters, see the relevant entries in the Reference Guide. Now that al parameters have been set, press **[F5]** (or select Graphics from the main window) for a graphical preview of the output. This gives you a last check point before starting an output process, which can be very time consuming with some devices. The final setup is also shown below.

Switch the plotter on-line and verify that pens and medium are loaded. Press **[F4]** to start the output. You can stop the output anytime by pressing **[↵]**.

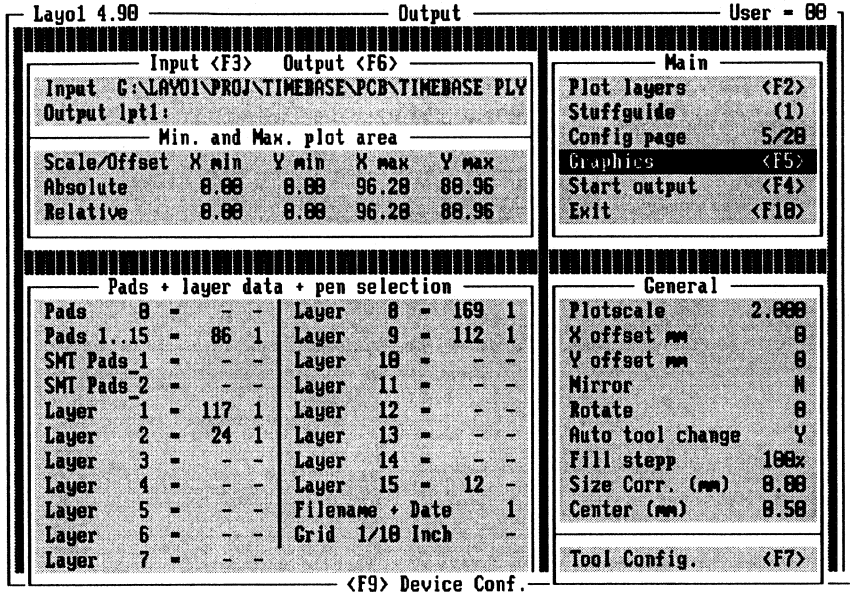


Fig. 7-2 Final Output Driver setup

Note 1: If no output is generated, check all cabling and connections. Press **[Esc]** to cancel the output.

Note 2: For GERBER files, use round pads if possible. Such files are more compact and are processed much quicker.

I N D E X

C

D

E

F

G

H

I

I

K

L

M

N

O

P

Q

R

S

T

U

V

W

Z