



# **Ranger2**

**Installation  
&  
Getting Started Guide**



## Contents

### Chapter 1

INTRODUCTION & INSTALLATION.....	1
Introduction .....	1
About this manual .....	1
Conventions used in this manual .....	1
Minimum hardware requirements .....	1
Installing the Ranger2 software .....	2
Loading the product license .....	2
Enabling the Cooper & Chyan Specctra Interface .....	2
Notes for existing Ranger1 and Ranger2 for DOS users .....	2

### Chapter 2

RUNNING RANGER2 & OVERVIEW .....	4
Running Ranger2 for Windows.....	4
Opening an existing job for viewing or editing .....	4
Viewing/editing the circuit diagram .....	4
Viewing/editing the parts/wiring list .....	5
Viewing/editing the physical outlines.....	6
Viewing/editing the artwork .....	6
Closing Ranger2 .....	7

### Chapter 3

CREATING A DESIGN .....	8
Creating a new design .....	8
Starting the design .....	8

### Chapter 4

SCHEMATIC DEVICE CREATION.....	9
Example 1 .....	9
Creating the device and its text definition.....	9
Adding Graphical Information.....	10
Drawing lines .....	10
Adding circles .....	11
Adding arcs.....	11
Zooming in, out, panning etc. ....	11
Modifying the outline .....	11
Defining the regulator .....	12
Moving the datum cross.....	12
Adding the Terminals.....	12
Adding the Text.....	12
Assigning Pin Numbers .....	12
Rotating the Device .....	13
Saving the Device .....	13
Example 2 .....	13
Adding the Graphical Information.....	14
Adding the Terminals .....	15
Adding the Text .....	15
Assigning Pin Numbers .....	15
Defining equivalent pins .....	15
Rotating the Device .....	15
Saving the Device .....	15
Leaving Ranger2 .....	15
Modifying devices in use on the schematic.....	16

# CONTENTS

---

## Chapter 5

SCHEMATIC CAPTURE .....	17
Changing the Sheet Size .....	17
Adding Symbols to the Circuit .....	17
Rotating, Flipping and Moving Symbols .....	20
Saving the design .....	20
Allocating Reference Designators, Pin Numbers and Values .....	21
Manual Allocation .....	21
Automatic Allocation .....	21
Assigning Values .....	21
Adding Connections .....	21
Modifying Wires .....	22
Buses .....	22
Adding Signal Names .....	23
Viewing and Editing Other Pages .....	23
Non-Electrical Information .....	23
Window Move, Rotate, Save, Copy, Delete .....	23
Leaving the Circuit Diagram .....	24
Enabling gate and pin swapping .....	24
Copying Devices to the Master Library .....	24
Creating a parts/wiring list from the schematic .....	24
Viewing the Compiled Parts/Wiring Lists .....	24
Listing unused pins .....	25
Leaving the parts and wiring list menu .....	25

## Chapter 6

CHECKING THE BOARD PARAMETERS .....	26
Sizes table .....	26
Viewing or editing the master sizes table .....	27
Power rail names .....	27
Plane assignments .....	28
Part code table .....	29
Viewing or editing the part code table .....	29
Viewing or editing the master part code table .....	29
Important Information .....	29

## Chapter 7

DEFINING THE BOARD PROFILE .....	30
Defining the board profile .....	30
Drawing lines, circles, etc. ....	30
Grid snapping/visibility .....	30
X/Y Readout .....	31
Grid .....	31
Zooming in, out, panning etc. ....	31
Modifying the outline .....	31
Drawing the profile .....	32
Modifying the outline .....	32
Router & Keep-out information .....	33
Saving and backing up your design .....	33

## Chapter 8

CREATING A PARTS & WIRING LIST .....	34
The Parts List .....	34
Typing in the parts list .....	35
The Wiring List .....	36
Typing in the wiring list .....	36

Chapter 9	
CREATING COMPONENT OUTLINES .....	38
Example 1 .....	38
Adding the pins.....	38
Modifying the pins/identifying pin numbers .....	39
Adding the silk-screen outline .....	39
Example 2 .....	40
Defining the grid .....	40
Adding the pins.....	40
Moving the datum .....	41
Adding the silk-screen outline .....	41
Adding text to outlines.....	41
Adding components to the master outline library .....	41
Additional Information and Tips .....	41
Chapter 10	
PLACING THE PARTS ON THE BOARD.....	43
Viewing the component pads and signal connections .....	43
Viewing and reconnecting the power connections.....	44
Other commands in the part placement editor .....	44
Gate swapping .....	44
Pin swapping .....	44
Part fixing and unfixing .....	45
Additional Information and Tips .....	45
Chapter 11	
LINE/PAD DIGITISING .....	46
Digitising pads .....	46
Digitising tracks.....	46
Additional Information and Tips .....	47
Chapter 12	
ROUTING THE BOARD .....	49
Auto-routing .....	49
Setting up the auto-router.....	49
Auto-routing the power supplies.....	50
Manual routing .....	50
Converting connections for manual routing.....	50
Manually routing/modifying tracks .....	51
Important points/tips to bear in mind when manually routing .....	51
Routing the signal connections.....	52
Deleting the artwork and starting again .....	53
The Rip/retry Auto-router .....	53
The Specctra Auto-router Interface.....	53
Modifying a Specctra routed design .....	54
Routing a board with power planes .....	54
Chapter 13	
COMPLETING THE DESIGN .....	55
Design rule checking .....	55
Silk screen generation .....	56
Colour and layer options .....	58
Finding out what pad and track sizes have been used .....	59
Adding copper areas to the artwork .....	59

# CONTENTS

---

## Chapter 14

OUTPUTS .....	60
Output to pen plotters .....	60
Plotter configuration .....	60
Pen plotting the circuit schematic .....	61
Pen plotting the artwork .....	61
Pen plotting the drill drawing .....	62
Sending files to the plotter .....	63
Output to photoplotters (Gerber) .....	63
Photoplotter basics .....	63
The output procedure - Artwork layers .....	63
DCode Table .....	65
The output procedure - Soldermask .....	65
Output to printers .....	66
Printing the circuit schematic .....	66
Printing the artwork .....	67
Output to NC drill machines .....	68
Producing a print-out or HP-GL file of a solder mask .....	68

## Chapter 15

ADDITIONAL INFORMATION .....	70
Moving parts on a digitised design .....	70
Adding extra parts and/or connections to a completed design .....	70
Adding tooling, fixing, mounting holes etc. ....	71
Removing parts and/or connections from a completed design .....	71
Renumbering parts once they have been placed on the board .....	71
Designing a multi-layer board .....	72
Reconstructing the netcodes .....	75
Window Move, Rotate, Copy, Delete .....	75
Changing the board profile .....	76

## Appendix

MASTER DEVICE LIBRARY LISTING .....	A1
-------------------------------------	----

Copyright Seetrax CAE Ltd 2004

Printed November 1997. Updated Feb 1998, June 2000, March 2002 and May 2004.

All brand and product names are trademarks or registered trademarks of their respective companies.

If there are any inaccuracies, ambiguities or omissions in this document, Seetrax CAE and its consultants and distributors cannot accept responsibility for any loss or damage these errors may cause.

Seetrax CAE reserve the right to alter the specification.

Seetrax CAE Ltd, 28 Vine Farm Close, Poole, Dorset, BH12 5EJ, England

Telephone: + 44 (0)1202 528686      Fax: + 44 (0)1202 528848

Web site: <http://www.seetrax.com>

email: [sales@seetrax.com](mailto:sales@seetrax.com)      or      [support@seetrax.com](mailto:support@seetrax.com)

## Chapter 1

# INTRODUCTION & INSTALLATION

### Introduction

Welcome to RANGER2, a suite of software for **INTEGRATED** schematic and PCB design.

Ranger2 has evolved from the successful Ranger1 software, which was originally launched over 17 years ago. Ranger is now well established in over 1000 companies all over the world.

Ranger2 includes schematic capture with automatic parts and wiring list generation linked to the artwork layout. Features such as graphical board profile definition, gate and pin swapping with automatic back-annotation to the schematic, auto-routing, track hi-lighting, automatic power plane generation, filled copper areas and design rule checking (DRC) are all included as standard, as are output tools for the production of Gerber and NC drill files.

All the **RANGER** series of PCB CAD software is written near the South coast of England, where Seetrex is based.

### About this manual

This manual should be used for all versions of the Ranger2 software that can be downloaded from the Seetrex web-site or supplied on CD. It assumes a licensed copy of the software is being used.

The unlicensed software will allow you to use all the facilities of Ranger2, but the design cannot be saved. The Lite licensed software allows you to use all the facilities of Ranger2 (except the rip/retry auto-router and the Specctra auto-router interface) provided the design does not exceed the specified limits. If the limits are exceeded, the design will not be saved.

This manual can be used as a guide in order to achieve a completed design with the required output files. It includes a worked example which explains all the stages required to design a printed circuit board, starting from either a schematic diagram or a typed in parts and wiring list and includes details on how to create library parts and perform subsequent modifications.

If you are working on your own design, use the worked example as a guide to point you in the right direction.

This manual does not describe each individual command in detail, as it is a guide for self-training. All the commands are described in detail within the on-line help that is available when Ranger2 is running.

### Conventions used in this manual

Chevrons enclose special keys on the keyboard. For example:

<Enter> is the carriage return key (labelled Return, or on some keyboards ↵)

<F1> is the function key labelled F1

In the graphical editors, you will often be required to select a command, which produces a pull down menu as shown in Figure 1. 1, from which another command should be selected. Some commands also have *icons* in the toolbar that act as shortcuts to them.

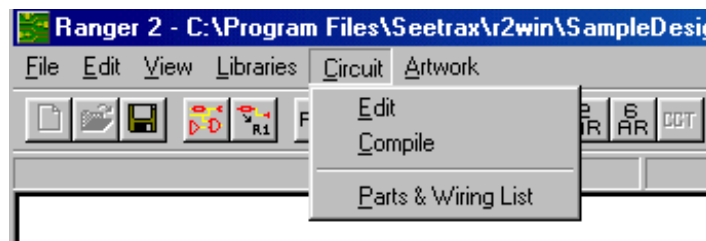



Figure 1. 1

This will be described as, select  or **Circuit** ⇒ **Edit**. This means either select the matching icon from the toolbar or the **Circuit** command from the command bar, followed by the **Edit** command from the pull down menu.

### Minimum hardware requirements

IBM PC 486 (or 100% compatible)  
 running Windows 95, 98, NT, 2000, ME or XP  
 8 Mbyte base RAM (Win 95/98, more is required for NT, etc.)  
 5 Mbyte hard disk space  
 SVGA colour graphics card and monitor (800x600 min, 1024x768 preferred)  
 Microsoft Mouse (or compatible)

## INTRODUCTION & INSTALLATION

---

### Installing the Ranger2 software

You will need approximately 5 Mbytes of disk space in order to install Ranger2. (Additional space will be required for designs, output files, etc.)

Ensure Windows is running and all other applications are closed.

(If the software was downloaded from our web-site, follow the installation instructions supplied on the web-site.)

Insert the Seetrax CD into the appropriate drive and close the door. The installation program will run automatically.

(If it doesn't, select **START**, followed by **RUN**. Select the **Browse** button, then locate the CD drive. Select the file **SETUP.EXE** from the window that appears, then select **Open**. Select **OK** to continue.)

A readme.txt file is supplied and can be viewed from the installation program. It contains the latest information so should be viewed/printed before continuing with the installation.

When you are ready to install the software, select **Install Ranger2 for Windows**, read carefully and then follow the instructions given on the screen. Default answers are supplied in all cases - if you change them, then you must remember the answers you supply for future reference.

When the process is complete, Ranger2 is ready to be used. If you experience any problems with the installation, please contact your supplier.

Until the Ranger2 product license is loaded, Ranger2 will operate as a demonstration system.

### Loading the product license

The product license is typically supplied on paper, but it can be faxed or emailed.

Once Ranger2 has been installed it must be run to allow the registration details to be entered, select **Start** ⇒ **Programs** ⇒ **Seetrax** ⇒ **Ranger2 for Windows**. On-screen prompts indicate how to proceed with the registration.

Type in the **Customer Name** and **Keycode** in the spaces provided. Take care to match every character correctly - you must not change any of the characters (including the customer name) or the license will not work.

Once the details have been typed in, select **OK** to continue. If the details were typed in correctly, Ranger2 is ready to be used as a licensed copy. If the details were not typed correctly a window will indicate a registration error, select **OK** to continue. You can then choose to continue as an unregistered version or re-enter the product registration details.

To close Ranger2, select **File** ⇒ **Exit**.

The registration details can also be accessed once Ranger is running by selecting **File** ⇒ **System Setup**, then selecting the **Product Registration** button from the window that appears.

If you experience any problems with Ranger2, please contact your supplier.

### Enabling the Cooper & Chyan Specctra Interface

Ranger2 has an interface to the Specctra auto-router software. (Specctra is available as a separate purchase). To enable the interface, with Ranger2 running, select **File** ⇒ **System Setup**, tick the box alongside **Cooper & Chyan Technology Specctra Autorouter**. Select **OK** to close the window.

### Notes for existing Ranger1 and Ranger2 for DOS users

- Refer to the file **readme.txt** that was placed in the `..\seetrax\r2win` folder when the software was installed. It contains important and useful information for users wishing to use existing Ranger1/2 master libraries and also details the differences between the DOS and Windows version of Ranger2.
- In Ranger2 (Windows) each job is held as one file, with an extension of `.r2`, instead of many files with different extensions (ie `J01.NCR`, `J01.NDL`, `J01.OLB`, etc.)
- Jobs created in Ranger1 or Ranger2 for DOS can be loaded into Ranger2 for Windows, however they will be saved in the new one file format (you will be prompted for a new name). This means that your original job remains as it is, but the newly saved job can only be loaded into Ranger2 for Windows.
- Opening a Ranger1 or Ranger2/DOS job:

With Ranger2 (for Windows) running, Select **File** ⇒ **Open**, a window similar to the one in Figure 1. 2 appears.

Select the down-pointing arrow (A) and from the browser that appears, locate the appropriate drive/folder where your old Ranger jobs are held.

Now change the **File type** by selecting the down-pointing arrow (B), to *DOS Ranger1 & 2 job index (job.idx)*.

The file *job.idx* will be listed (provided the folder contains a Ranger1 or 2 job index - if the *job.idx* file does not exist then the jobs cannot be opened). When it is double selected, the job index appears listing all the existing jobs that can be loaded. Select the job followed by **Open**.

When saving the jobs, they will be saved as a *Ranger2 for Windows job (.r2)* in any folder you select.



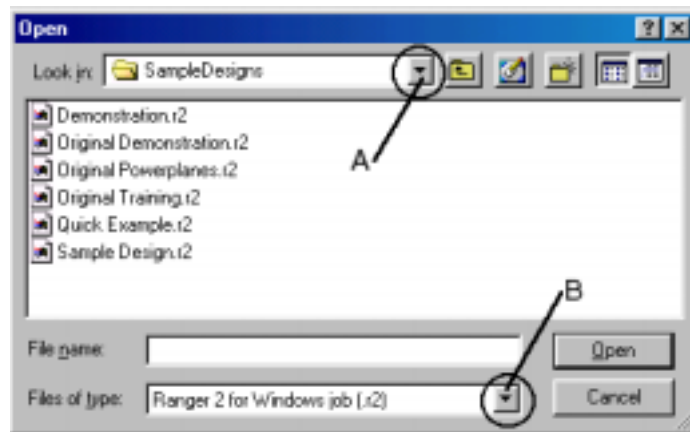


Figure 1. 2

- Converted Ranger1 jobs will not have internal netcodes in the artwork, which are required by new routines such as copper fill, track hi-lite, etc. First, open the job in Ranger2 and check the artwork (*Artwork* ⇒ *Trace/Check artwork*, select *Clearance Check & Connectivity Tracer* followed by *Connectivity Checker*). Correct any short circuit errors and re-run the checks. Then select *File* ⇒ *Maintenance* ⇒ *Reconstruct netcodes*.
- There is **no auto-save** facility within Ranger2/Windows - so you **MUST** take regular saves and backup copies. Ask yourself this simple question, "How much work am I prepared to lose?" Your answer will tell you how often to save your work and how often to copy your work.

### Chapter 2

## RUNNING RANGER2 & OVERVIEW

It is assumed Ranger2 has been installed and Windows is running. This chapter explains how to run up Ranger2 for Windows, how to select items, zoom in/out and gives a very brief look at some of the editors in Ranger2.

### Running Ranger2 for Windows

Select **START** ⇒ **Programs** ⇒ **Seetras** ⇒ **Ranger2 for Windows**. The program loads and a window similar to the one shown in Figure 2. 1 appears, but with a registration window super-imposed over it.

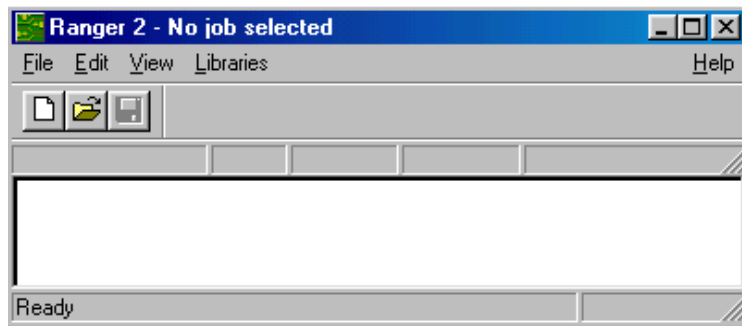



Figure 2. 1

The *title bar* across the top of the window, shows that this window is *Ranger2* and that *No job is selected*.

### Opening an existing job for viewing or editing

Within Ranger2 (for Windows), all the data that makes up the complete PCB design, for instance the circuit schematic, the artwork, library parts, etc. is held in one file, that from now on will be referred to as the *job*. We will look at an existing job to start with.

Select  or **File** ⇒ **Open Job**. The folder for Ranger2 jobs opens and the jobs that have been supplied with Ranger2 appear. The job names will always have an extension of *.r2* to indicate they are Ranger2 for Windows jobs.

Select the job called **Demonstration.r2** followed by **Open**. It will open and two additional commands **Circuit** and **Artwork** appear in the command bar and more icons appear in the toolbar as shown in Figure 2. 2. The title bar now indicates which job is loaded.

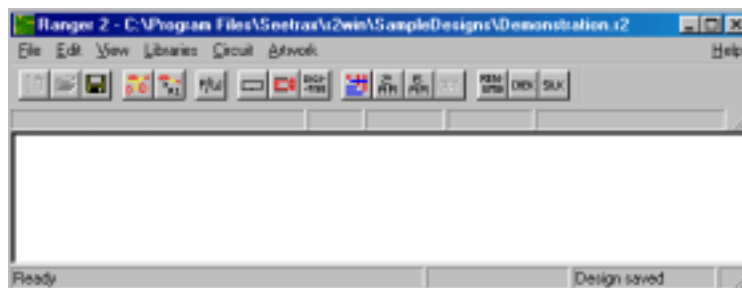




Figure 2. 2

You now have access to the complete design or *job*. Maximise the Ranger2 window for clarity (maximise button top right of the window).

### Viewing/editing the circuit diagram

Select  or **Circuit** ⇒ **Edit** to open the schematic editor and display the circuit diagram. The circuit editor is used to define the parts and their connectivity for the design. A parts and wiring list is extracted from the circuit for artwork design.

To zoom in to an area of interest, select  or **View** ⇒ **Zoom In**, point at the area of interest and click the left button to give one level of magnification, or the right button to give two levels. *Zoom in* remains active until another command is selected, or maximum zoom in is achieved.



(**View ⇒ Zoom Out**) operates in reverse. **View ⇒ Full** restores the display so that the complete sheet can be seen.

To speed up movement around the screen the following keys can be used instead of selecting the **View** menu commands.

- <F1> Zoom (one level, about the cursor position)
- <F2> Reduce (one level, about the cursor position)
- <F3> Full
- <Space bar> Pan (about the cursor position, whilst zoomed in)

Zoom in to the top right corner of the page of circuitry.



To add a part: select **(Symbol ⇒ Place)**, a list of parts appears on the right side of the screen, known as the *Tray*.

Point at **4011** from the list, then click the left button. As the cursor is moved onto the page, the symbol for the 4011 appears attached to it. Click the left button again to release the part, a second 4011 appears and can be positioned with another click of the left button, alternatively another part can be selected from the tray, or the right button can be clicked to cancel more symbols.



To move a part: select **(Symbol ⇒ Move)** from the command bar, point at the middle of a part, then click the left button. As the cursor is moved, the part and any attached connections move with it. Click the left button again to release the part. (You do not need to hold the button down whilst moving things in Ranger2.)

Select another part and move it, but this time click the right button to release the part - notice that the part returns to its original position - a right click cancels the move.



Adding a connection: select **(Wires ⇒ Insert Connection)**, point at a terminal on the symbol just added (zoom in if necessary), then click the left button. Move the cursor, a connection appears attached to the cursor that maintains a straight or right-angled line. Additional corners can be inserted in the connection with clicks of the left button, to allow you to thread the connection around obstacles. To terminate the connection, point at another terminal or existing connection, then click the left button. The connection is released from the cursor. More connections can be added in the same way.



To move a corner in a connection: select **(Wires ⇒ Move Point)**, point at the corner to be moved and click the left button. Move the cursor, then click the left button again to release the corner (right button cancels the move). As you move the corner, Ranger tries to maintain the segments at either side, horizontal or vertical. This is not always what you want, so uncheck the tick-box alongside *Wire Lock* in the tool bar to disable wire locking. (When running at 800x600 screen resolution, only the check-box is visible on the far right, but it can be selected.) Now select **Wires ⇒ Move Point** again and move a corner in a wire, notice the difference.

If another connection is added now, the connection will move freely until wire locking is enabled again.

We will not be saving the changes you have made, so don't worry about making a neat circuit at this stage. Let's now take a look at the parts and wiring list produced from the circuit diagram.

Select **File ⇒ Close Circuit** to leave the circuit editor. (The changes made will be held in memory until you decide to save or cancel them.)

## Viewing/editing the parts/wiring list



Select **(Circuit ⇒ Parts & Wiring List)**, a window appears similar to the one in Figure 2. 3, containing the parts list, and behind that, the wiring list. You may want to resize the window in order to see the complete page of parts or wires.

Parts & Wiring Lists

File

Edit

Sort

Tools

Window

Help

Wiring List

Sig. Name					
TR1 . 4	IC3 . 1	R5 . 1			
IC3 . 2	R10 . 1	IC2 . 5			

Parts List

ID	Description	Outline	Code	X	Y	Rot	
IC1	74193, 40193	DIL16		5.000	3.400	0	
IC2	4070, 4070	DIL14		3.900	2.100	0	
IC3	40106, 40106	DIL14		5.000	2.150	0	
IC4	LM324, LM324	DIL14		2.900	4.000	0	
IC5	4072, 4072	DIL14		6.100	2.750	0	
IC6	4013, 4013	DIL14		6.100	3.400	0	
IC7	4069, 4069	DIL14		5.000	4.050	0	
IC8	4011, 4011	DIL14		6.100	4.050	0	
IC9	4011, 4011	DIL14		5.000	2.750	0	
IC10	4023, 4023	DIL14		6.100	2.100	0	
IC11	4013, 4013	DIL14		3.900	4.000	0	

Figure 2. 3

The parts and wiring lists for this design were automatically generated from the circuit diagram that you just saw, including the *Outline* names. The *outline* is the physical representation of the part, for instance IC1 requires a 16 pin dual-in-line package which is called DIL16 in the outline library.

It is possible to type in a parts and wiring list if you don't want to draw a circuit diagram. If a circuit had not been created, the parts list would be empty and you would have to fill in the appropriate information - more about this later in the worked example.

## RUNNING RANGER2 AND OVERVIEW

When the list is initially created, the X & Y position fields are set to 0. As parts are placed on the artwork, these co-ordinates are updated - you are looking at a completed design so the parts have already been placed on this artwork.

To bring the wiring list to the front, click anywhere in the Wiring list window.

Each line in the wiring list editor forms a connection or net. A connection (net) is made between two or more pins. The pins in a connection are often called nodes. Pins on the same line are connected to one another.

Taking the first line in the wiring list as an example: **TR1.4 IC3.1 R5.1**

This indicates that **TR1** pin **4** is connected to **IC3** pin **1** and **R5** pin **1**.

Scroll through the list to the end (using the scroll bar on the right hand side of the list). Notice that some connections have a signal name assigned to them, some of which are highlighted.


Any signal name that is highlighted means that Ranger2 regards that connection as a power rail, so it will be treated differently to signal type connections (routed with a thicker track, auto-routed with a different strategy or defined as a power plane).

The GND and VCC connections have many pins in them, so they require more than one line. The word CONTD-> appears, to indicate that the line is a continuation of the previous line.

Select **File ⇒ Exit Editor** to leave the parts/wiring list.



### Viewing/editing the physical outlines

Select **Libraries ⇒ Component Outlines**. This opens the library of outlines held within the design. To edit an outline,

select  (**File ⇒ Open Outline**). The list of outlines in the job library appears, select **DIL16** followed by **Open**. The DIL16 outline appears on the screen.

This is the outline that IC1 will use on the artwork. We will create some outlines later in the training exercise, so select **File ⇒ Close Outline**, **File ⇒ Close Library** to close the outline library.

### Viewing/editing the artwork


Select  (**Artwork ⇒ Editor**) to display the artwork. Zoom in a couple of times as explained previously (, <F1> or **View ⇒ Zoom in**). Tracks on one side of the board are shown in red, tracks on the other side are shown in blue. The silk-screen is green. (These colours can be changed.)


*Swapping a track from one layer to another:* select  (**Mroute ⇒ Layer Swap**), select a track. Notice it swaps to the other layer and vias (through plated holes to link a track from one side of the board to a track on the other) are inserted or removed automatically. Select it again to restore it to its original layer.

*Viewing the grid:* items that are moved snap to a grid or half grid point when they are released. The current grid size is displayed in the *information bar* across the top of the working area, as shown in Figure 2. 4.



Figure 2. 4

To see the grid, which you will find useful, press the <g> key on the keyboard, or select  (**Grid ⇒ On/off**).

*Moving corners:* select  (**Mroute ⇒ Corner**), point at a corner or a via in a track, then click the left button. Move the cursor and attached corner, click the left button to release the corner (right button to cancel). A via can be moved in the same way.

*Adding corners:* whilst **Mroute ⇒ Corner** is still active, point at the middle of a track segment (not on a corner) and click the **right** button. Move the cursor, you have now inserted a corner in the track. Release it with a further click of the **right** button (left button to cancel).

*Controlling which layers are visible:* at the moment all the layers of the artwork are visible which can be confusing. If for instance you were completing the silk-screen, you probably would not want to see the tracks.

The design can have up to 16 layers that can be used for copper or silk-screen information. When using the outlines that are supplied with Ranger2, layer 1 is always the top (or component) side of the board, and layer 2 is always the bottom (or solder) side. Layer 0 always contains the drilled pads (component pads & vias).

Select **View ⇒ Layer Properties**. A window similar to the one shown in Figure 2. 5 appears.

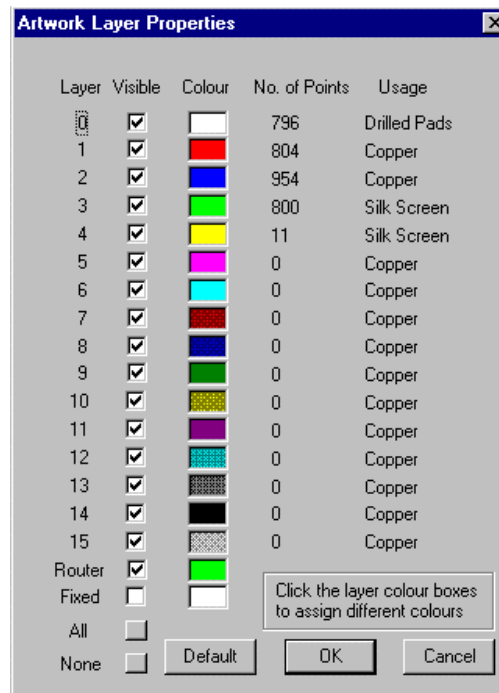



Figure 2. 5

Each line in the window represents a layer on the artwork. So for instance, layer 1 is visible and will be displayed in red (this is hard to see in black & white!!). It has 804 points on it and it is a copper layer. If a layer has 0 points, then it is empty. (Your example will be different if you have added some corners to tracks or layer swapped some tracks because a *point* is a pad, a via, a corner in a track or line, etc.)

In this example the silk-screen has been added to layers 3 & 4. We know this, because the usage column indicates those layers are silk-screen layers. Also there are some *points* on those layers.

So to make layer 1 invisible, position the cursor over the tick in the *visible* column of layer 1, then click the left button. The tick is removed. Repeat for layer 2. Select **OK** when you are ready to move on. Notice that only the silk screen layers and pads are now visible because we made layers 1 & 2 invisible.

*To move the silk screen labels:* zoom in if necessary. Select  (**Text ⇒ Move**), now point at the lower left corner of a text string and select it with a click of the left button. Move the cursor and reposition as required.

This stage of the tutorial is now complete and you should have obtained a brief introduction to Ranger2. You are now ready to start your first design.

## Closing Ranger2


Assuming you are still in the artwork editor, select **File ⇒ Close artwork, File ⇒ Close Job, No, do not save changes, File ⇒ Exit**.

### Chapter 3

## CREATING A DESIGN

Run up Ranger2 for Windows (select **START** ⇒ **Programs** ⇒ **Seetrex** ⇒ **Ranger2 for Windows**).

### Creating a new design

To start a new design or “job”, select  (**File** ⇒ **New Job**). The job is created and opens automatically, the window is similar to that in Figure 3. 1. The title bar shows that the job is unnamed, and two additional commands, *Circuit* and *Artwork* have appeared in the command menu along with more icons in the toolbar.

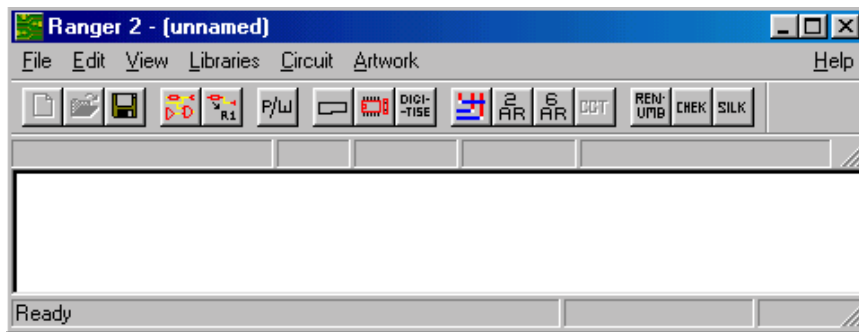


Figure 3. 1

### Job Creation Information

When a new job is created, the master *outline* library and a set of design defaults are copied into the job. The design defaults consist of pad and track sizes, part pre-fixes, etc. The library and design defaults may be changed within the job without affecting any other jobs or the master set.

(The master libraries are accessible before a job is opened, via the *Libraries* ⇒ *Master Libraries* menu, and the design defaults via the *Edit* ⇒ *System Defaults* menu. Both these menus are only accessible if a job is not open. It is suggested that you do not start to alter the master libraries or system defaults until you have some experience with Ranger2.)

### Starting the design

A worked example follows, however if you have your own design in mind, simply follow the same procedure but using your own design instead.

Decide now whether you intend designing the board from a circuit schematic, or by typing in a parts and wiring list.

If you do not want to create a circuit schematic, move on to chapter 6, page 26, titled *Checking the board parameters* and continue from there.

If you intend creating a circuit schematic, continue with the next chapter overleaf, titled *Schematic Device Creation*.

## Chapter 4

### SCHEMATIC DEVICE CREATION

In this chapter, we create schematic devices ready for use on the schematic – if you prefer to skip device creation, move onto chapter 5, titled *Schematic Capture* on page 17.

Schematic devices can be made at any time prior to use, so you could decide to start the schematic diagram, then create the devices once you know which ones are unavailable from the master library - it's not necessary to make them at the beginning of the design.

Example 1 can be used as a guide when creating any parts that have one "symbol". For example, diodes, transistors, single resistors, capacitors, integrated circuits that are shown as one part or connectors where all the pins are shown together.

Example 2 (page 13) describes how to create parts with more than one element/gate such as 7400's (4 x 2 i/p nand gates), resistor packs where the resistors are shown individually or connectors where each pin is shown individually. It also includes details on how to define equivalent pins and gates.

The examples assume that you know how to run up Ranger, select items and have already created a job. The job design menu should be visible as shown in Figure 4. 1.

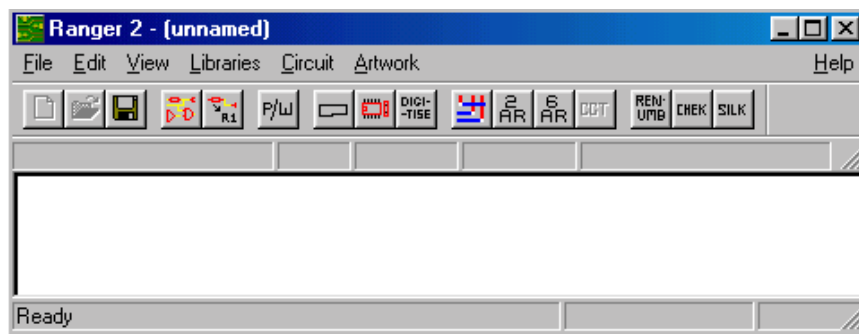


Figure 4. 1

#### Example 1


For the worked example a voltage regulator is required that doesn't already exist in the master device library.

It is good practice to make new library parts in the job library and not directly in the master library. In this way, the devices may be used and verified before being added to the master library, which would enable them to be used by other people.

From the job design menu select **Libraries ⇒ Circuit Devices**. The job library will be empty because devices are added to the job library (from the master library) as they are used on the circuit.

The circuit diagram in the next chapter (pages 19 and 20) will be created. The voltage regulator (IC1) has to be created.

#### Creating the device and its text definition

Select  (**File ⇒ New Device**) a window appears similar to the one shown in Figure 4. 2. The cursor is blinking in the *Device name* entry field. Type in the device's name as shown (VREG-). This is the name that you (and Ranger) will use when referring to it.

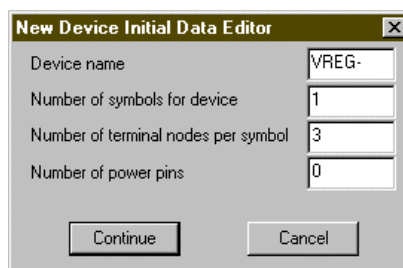


Figure 4. 2

Press the <tab> key after the name has been typed, which moves the cursor to the next field. Type in the remaining values as shown, pressing <tab> after each one. Their relevance is explained below.



## SCHEMATIC DEVICE CREATION

Once this window is closed, only the device name can be changed, so please ensure you get it right!

*No. of symbols for device* - the number of identical symbols or gates (apart from the pin numbers) in the device, in this example, 1. For example, there would be 6 symbols (inverters) in a 7404.

*No. of terminal nodes/symbol* - the number of connection points on each symbol or gate. Example: there are 2 terminals on an inverter, 2 on a capacitor, 3 on a transistor, etc.

*No. of power pins* - the number of pins that are always connected to the same power rails. These pins will not be seen, but they will always appear in the wiring list connected to the specified rails. For example there are 2 power pins in a 7404 (pins 7 & 14), but none in diodes, resistors, transistors, etc.

When all the details have been typed, either press <enter> or select **Continue**. The graphical editor appears.

We need to enter some more information about the voltage regulator before continuing. Select **Device Properties** from the toolbar across the top of the window. Enter the details as shown in Figure 4. 3.

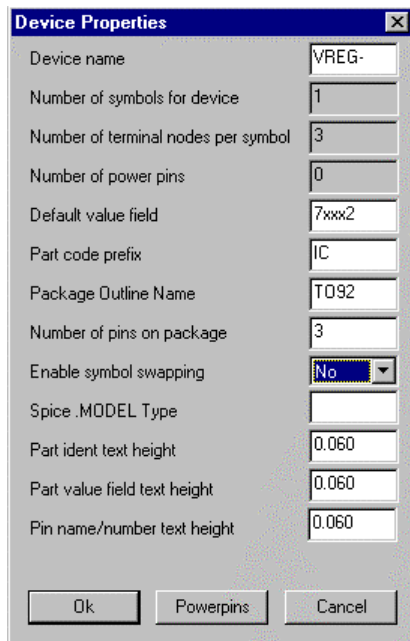


Figure 4. 3

*Device name* - the name the device is stored under in the library. This can be changed provided the device is not in use on the circuit schematic.

*Default value field* - the text entered here appears on the symbol's graphical representation. It can be changed as required on the circuit schematic. It typically contains a common value, i.e. 220k, 10uF, 74LS00.

Parts that have the same value field can have their gates swapped during part placement, with gates from other parts with the same value field.

*Part code prefix* - part of the components unique reference, i.e. IC, C, R, D, etc.

*Package outline name* - the name of the physical outline (footprint) that will be called up when the schematic is compiled into a parts/wiring list. The outline does not have to exist until the part placement editor is loaded.

*No. of pins on package* - Ranger calculates the minimum number of pins required on the device, based on the information already given. Some devices do have unused pins, so the actual number of pins required may be higher, in which case enter the highest used pin number.

*Enable symbol swapping* - If Yes is selected, then gate and pin swapping will be permitted for the device during part placement. This device doesn't have any gates or equivalent pins, so it should be set to No. (Example 2 gives an example with equivalent gates and pins.)

*Spice Model type* - This field can be left blank unless a PSpice format output file is required, in which case the PSpice model name should be entered for the device. (Consult the PSpice documentation for model types and names.)

*Part ident text height* - defines the height of text used for the device's ident, i.e. R1, R2, R3, IC1, IC2, etc. Set as required or use the existing value.

*Part value field height* - defines the height of text used for the device's value field, i.e. 220k, 10uF, etc. Set as required or use the existing value.

*Pin name/no. text height* - defines the height of text used for the device's pin numbers and pin names. Set as required or use the existing value.


Select **OK** or press <enter> to continue.

### Adding Graphical Information

One of the next things to be defined in the device is the shape of the schematic symbol. The schematic shape has no significance for Ranger, it's purely used to identify the function of a part in the schematic. (Sometimes users become confused and think the schematic shape is the physical shape of the part, but think about a transistor. On the circuit a transistor is typically shown as a circle, with the emitter, base and collector pins, but the physical shape can vary enormously.)


Here we describe how to "draw" lines for the schematic shape of the device. Experiment with drawing them until you are "happy" and ready to define the outline. These lines will be deleted prior to drawing the regulator.


### Drawing lines

Select  (**Outline ⇒ Add Line**). Across the bottom of the window the status bar indicates which command is currently active (OUTLIN:LINE) and what you should do next to action the command (Select line start position). Move the cursor into the working area and click the left-hand mouse button to start a line. Move the cursor and click the left button again to insert corners (or points) in the line. Click the right-hand button to release the line. The line is made up of a series of straight-line segments.

Grid snapping is switched on automatically. This means that points (corners) in the line always snap to the nearest grid or half grid point. The current grid pitch is displayed in the information bar across the top of the working area.




Zoom in (, <F1>, **View ⇒ Zoom In**). As you zoom in, the grid dots on screen remain approximately the same distance apart. This is because an *auto-pitch* grid is active. An auto-pitch grid changes as you zoom in and out, so that the grid does not obliterate everything else on the screen, or become too widely spaced to be useful. The actual distance between dots is always given in the information bar across the top of the working area.

Now draw a few more lines (, **Outline ⇒ Add Line**). You should notice that the points are snapping to the grid or half grid.


It is very seldom in schematic or artwork design that you work with grid snapping switched off. However should you wish to do so, select **Grid** and notice the tick alongside *Snap to Grid*. This means that grid snapping is switched on. If you select **Snap to Grid** the tick is removed and the window closed. Notice the words *SNAP OFF* appear in the information bar. Now when you draw lines, the points are released wherever the cursor is positioned, the finest resolution being 0.001". Switch grid snapping back on (**Grid ⇒ Snap to Grid**).

In the *Grid* menu you also have the ability to select *Metric auto-pitch* grids and *fixed grids* that are fixed at the size specified at all zoom levels. A user-defined grid can also be selected, you will be prompted for the X & Y pitch. We will come back to this menu later when it is more relevant. Leave the grid set to **Inch (Autopitch)** and grid snapping on.

### Adding circles



Select , (**Outline ⇒ Add Circle**). Get used to following the prompts in the status bar. Select circle centre position, move the cursor to stretch the circle, then click the left-hand mouse button to release the circle at the required radius. If the right button is clicked whilst stretching the circle, it is cancelled. The command stays active so another circle could be defined.

### Adding arcs

Select , (**Outline ⇒ Add Arc**). Select the start point of the arc, move the cursor and select the end point of the arc, now move the cursor to stretch the arc and click the left-hand mouse button to release it. More arcs can be added until the right button is clicked after an arc has been released.

If the right button is clicked whilst stretching the arc, a straight-line segment is introduced. This allows a series of arcs and straight lines to be added at the same time.

### **Zooming in, out, panning etc.**


The zoom-in, reduce, pan, etc. commands are accessed from the **View** menu or   icons for zoom in/out. The following shortcut keys are available:

<F1>	Zoom (one level, about the cursor position)
<F2>	Reduce (one level, about the cursor position)
<F3>	Full
<Space bar>	Pan (about the cursor position, whilst zoomed in)


### **Modifying the outline**

Existing lines, arcs and circles can be modified with the following commands. The status bar gives guidance on how to action the commands. The following tips may also be useful to you.


#### Moving points (corners) in lines

Select , (**Outline ⇒ Move Point**). An existing point (corner) in a line should be selected. Whilst the point is being moved it can be cancelled with a click of the right button.

#### Adding points (corners) to lines

Select , (**Outline ⇒ Add Point**). When selecting the line in which the point will be added, select the line away from an existing point (corner) i.e. in the middle of a segment - you can then be sure the point is added in the segment you want. Whilst the new point is being moved it can be cancelled with a click of the right button.

#### Moving and adjusting the size of circles

Select , (**Outline ⇒ Move Point**). Select the edge of a circle to change its size, select its centre to move it.

#### Deleting existing points in a line

Select **Outline ⇒ Delete Point**. The endpoints of lines cannot be deleted if there is only one segment left in the line. (Use the **Outline ⇒ Delete Line** command to delete complete lines.)

#### Modifying arcs, converting lines to arcs

Select **Outline ⇒ Adjust arc**. Select the arc or line, move the cursor, then click the left button to release the line/arc. If the right button is clicked whilst moving an arc, the arc is converted into a straight segment.

Repeated clicks of the right button converts the last selected segment between a line and an arc. The position of the cursor about the previously selected line determines which way the arc is adjusted.

Once you are happy using these commands, delete all the lines, circles and arcs using the **Outline ⇒ Delete Line** command. If you have trouble selecting a line, point at a corner or an end point of the line, or zoom in if necessary.

## SCHEMATIC DEVICE CREATION

Select **View ⇒ Full** or <F3> to restore the screen to its original scale. If any of your practice lines are still present, delete them.

### Defining the regulator

Now draw the regulator outline as shown in Figure 4. 4, the small cross represents the datum of the device, the blue cross on the screen. Use the grid to draw the device to scale.

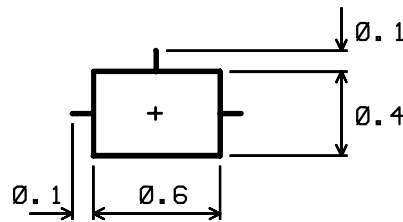


Figure 4. 4

The grid spacing is shown in the information bar. Provided you have **Grid ⇒ Inch (Autopitch)** selected, the grid changes size as you zoom in and out.

Zoom in/out so that you have a 0.1" grid visible.

### Moving the datum cross

When drawing symbols, try to draw them centrally over the datum (blue cross) to save you the extra task of moving the datum afterwards. If you do need to move the datum, select **Edit ⇒ Set Datum**, then indicate its new position with a click of the left-hand mouse button. The datum is moved.

### Adding the Terminals

Once the outline has been drawn, you are ready to add the terminals. On the circuit diagram connections can only be started from terminals so they are very important.

Select **Terminals ⇒ Place**. The status bar across the bottom of the screen indicates which terminal is being placed. These terminal numbers are NOT pin numbers, they are just a reference number. Position the cursor over the end of a stalk, then click the left-hand mouse button. Repeat for the other two terminals. The status bar will indicate when all the terminals

have been added. They can be moved using  (**Terminals ⇒ Move**).

### Adding the Text

Text is added to the device for identification purposes. At this stage only its position is defined, as the actual text may only appear on the circuit, for example, its name IC1 or IC2, etc.

**Pin Number Positions** - Select **Text ⇒ Place Pin Number**, select a terminal, then select the position for the pin number.

Pin numbers have not been assigned yet, so NNN will appear, as shown in Figure 4. 5

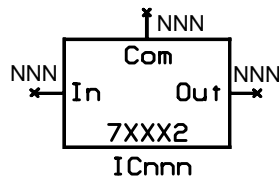



Figure 4. 5

Repeat for the other terminals. Use  (**Text ⇒ Move**) to reposition the text strings if required.

**Value and Ident Positions** - The position for the symbol's identification, IC1, 2, etc. is defined by selecting **Text ⇒ Place Ident**, then selecting the position required. (Refer to Figure 4. 5 for positional information.) At this stage ICnnn will appear as the numbers are assigned when the part is used on the circuit diagram.

Repeat for the value field, using the **Text ⇒ Place Value** command. The content of the value field (7xxx2) was entered in the device's text definition earlier on (*Device Properties* from the tool bar).

**Free text** - Free text is added purely for user information, Ranger does not use it. Select **Text ⇒ Insert Free Text**, then enter **IN** in response to the prompt. Position the text string as shown in the diagram, near the input pin.

The prompt re-appears, so enter **COM** and position the text string as shown, near the common pin. Position the text string **OUT** near the output pin. Select **Cancel** to terminate text entry.

If necessary, adjust the text positions using the  (**Text ⇒ Move**) command. (The height of new free text strings is controlled by the **Text ⇒ Change Height** command.)

### Assigning Pin Numbers

Pin numbers must be assigned to terminals to ensure the correct connectivity on the circuit and PCB.

The tool bar displays **Symbol** set to 1 and the **Next free pin number** as 1, (Figure 4. 6).



Figure 4. 6


Note: at 800 x 600 screen resolution the *Next free pin number* setting hangs off the edge of the screen; the toolbar it is in can be moved (hold down the left button and drag it) and placed underneath the icon toolbar so that it becomes visible.

The **Next free pin number** arrow buttons may be selected to show the available pin numbers for the device. The **Symbol** box number will not change, as only one symbol was originally defined for this device.

Set the **Next free pin number** box to pin 1. Select **Pin Assignment** ⇒ **Set Pin Number**, then select the terminal on the right. The pin number 1 replaces the *NNN* characters. Pin 2 appears in the **Next free pin number** box. Select the upper terminal, followed by the left terminal when pin 3 appears in the box. \*\*\* appears in the **Next free pin number** box to indicate that all the pins have been assigned.


Use **Pin Assignment** ⇒ **Deassign a pin** if the wrong number has been assigned, then assign it again.

### Rotating the Device

If the device will be rotated on the circuit, the text positions for those rotations need to be defined. Select the **Rotate Symbol** button from the toolbar or **Text** ⇒ **Rotate Symbol** - the symbol is rotated by 90 degrees and the text is no longer in the correct location. Position the text as required using  (**Text** ⇒ **Move**). Do not alter the symbol outline or terminal positions (unless you want to alter the original as well). Repeat for the other two orientations if required.

### Saving the Device

The device has now been defined and should be saved in case of power failure. Design changes are held in memory until the job is saved.

Select **File** ⇒ **Close Device**, followed by **File** ⇒ **Close Library**. Select  (**File** ⇒ **Save**). The Windows browser appears, with the Ranger2 jobs folder, *Sample Designs* open. You can navigate around to locate a different folder if required. Because the job has not been saved previously, you must enter a job name in the *File name* box, for instance **My Worked Example**. Press <enter> or select **Save** to action the save and return to the Ranger2 window.

### Example 2

This example shows how to define a part that has more than one element (gate) and how to define equivalent gates and pins within a device to enable gate and pin swapping on the layout. It also shows how to define power pins without including them as terminals on the device.

(This example assumes you have worked through the first example and the job is still open.)

Select **Libraries** ⇒ **Circuit Devices** then  (**File** ⇒ **New Device**) and enter the details as shown in Figure 4. 7. Use <tab> to move the cursor between fields.

Figure 4. 7

This indicates that the device's name is MY7400, it has 4 equivalent symbols (gates), each with 3 terminals and the device has 2 pins that are always connected to the same power rails.

Select **Continue**. The graphical editor loads.

Select **Device Properties** from the tool bar. Enter the details as shown in Figure 4. 8.

Figure 4. 8

Because symbol swapping has been enabled, the four symbols (gates) can be swapped for other identical symbols during the part placement stage of the design, to reduce connection crossovers and assist routing. Pin swapping is enabled at the same time, but the equivalent pins have yet to be defined.

Devices of this type are usually connected to the same power rails, so it is common practice to imply them on the circuit, i.e. they are not shown. Of course, Ranger has to know which pins are power pins and which power rails to connect them to.

Select the **Power pins** button from this window to define the power pins for the device, a window similar to that shown in Figure 4. 9 appears. Fill in the details as shown.

Number	Power Name
14	VCC
7	GND

Figure 4. 9

Select **OK** to continue, then **OK** again.

### Adding the Graphical Information

Add the graphical information for a 2 input nand gate, as shown in Figure 4. 10. (The ampersand can be added as free text, 0.2" high.)

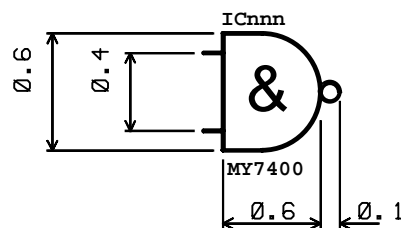


Figure 4. 10

Move the datum (blue cross) into the centre of the device if necessary, using the **Edit** ⇒ **Set Datum** command.

### Adding the Terminals

Add the terminals on the ends of the stalks and outside of the circle, using **Terminals ⇒ Place**. Remember the numbers shown in the status bar are NOT pin numbers, they are just a reference number. The bar should indicate that all the terminals have been added.

### Adding the Text

Position the pin numbers using the **Text** menu (**Text ⇒ Place Pin Number**), use the prompts in the status bar to assist you. Position the *ident* (**Text ⇒ Place Ident**) and *value* (**Text ⇒ Place Value**) as required.

### Assigning Pin Numbers

Select **Pin Assignment ⇒ Set Pin number**. On this device, the arrows in the box alongside the **Symbol** box in the tool bar can be selected to show the four symbols (gates) that were defined. The pins for each of these four symbols have to be assigned.

Set the **Symbol** box to **1** and the **Next free pin number** to **1** in the tool bar, then select the appropriate terminal, as shown in Figure 4. 11 (you will only see one of the gates at any time). Pin 1 has now been assigned to the terminal. Pin **2** now appears in the tool bar, so select the next terminal and so on.

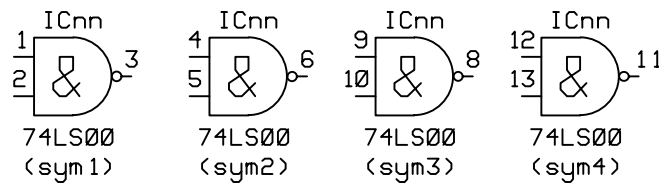


Figure 4. 11

Once all the pin numbers have been assigned to the first gate in the package, set the **Symbol** box to **2**, the second gate appears, then assign the correct pin numbers to its terminals.

Repeat for the 3rd and 4th gates. \*\*\* will appear in the **Next free pin number** box when all the pins have been assigned. Use **Pin Assignment ⇒ Deassign a pin** if the wrong number has been assigned and then assign it again.

### Defining equivalent pins

On this device the two input pins are equivalent, so they can be swapped one for the other during the part placement stage, if required. They are defined as being equivalent within Ranger as follows:

Select **Pin Assignment ⇒ Swap Group 1** followed by the two input terminals. They will both turn **yellow** to indicate they belong to **Group1**.

If there were other terminals on the symbol that were equivalent to themselves, but not to the terminals in Group1, they could be assigned to Swap Group2 or 3. Pin swapping is only permitted between pins of the same Group within the same symbol.

### Rotating the Device

If the device will be rotated on the circuit, the text positions need to be defined for those rotations. Use **Text ⇒ Rotate Symbol**, then position the text as required in the rotation shown.

### Saving the Device

The device has now been defined and should be saved in case of power failure.

Select **File ⇒ Close Device**, followed by **File ⇒ Close Library**. Select  (**File ⇒ Save**). If you have been following these instructions, you will already have saved the design and given it a name, in which case the previous copy is now overwritten.

If you want to maintain a separate backup copy, use the **File ⇒ Save As** command and supply a different filename. Press <enter> or select **Save** to action the save. Refer to the title bar across the top of the screen to find out which design is loaded and you are working on. (To load the previous job, you will need to close the current job (**File ⇒ Close Job**), then open the previous one (**File ⇒ Open**).

### Leaving Ranger2

If you want to take a break now before continuing (or at some other stage) ensure you have saved the design, then select **File ⇒ Close Job**, **File ⇒ Exit**.

When you are ready to continue, run up Ranger2, then select **File ⇒ Open**, locate then select the design using the browser (ensure the **Files Of Type** is set to **Ranger2 for Windows**), select **Open** and you are ready to continue.

The next chapter describes how to define the schematic diagram.

## SCHEMATIC DEVICE CREATION

---

### Modifying devices in use on the schematic

If the graphics (lines, arcs) within a device are modified or terminals are moved, the symbol on the schematic sheet is automatically updated - connections attached to the moved terminals will become disconnected.

The number of terminals, symbols and power pins cannot be changed and all the terminals should be placed in the device.

If the datum is moved, this will cause dis-connections - the datum remains where it is on the schematic, so it will look like the symbol has moved.

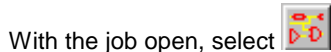
If the pin numbers are changed, the device will need to be de-allocated, then re-allocated on the schematic.


## Chapter 5

### SCHEMATIC CAPTURE

The worked example is a simple one, but serves to illustrate **most** of the features used during the schematic capture process.

The schematic comprises two pages of circuitry drawn on A5 sheets.



With the job open, select  (**Circuit ⇒ Edit**) to load the circuit schematic editor. We are about to draw the circuit schematic shown on pages 19 and 20 which comprises of two pages of inter-connected circuitry.

#### Changing the Sheet Size

The white rectangle on the screen represents the drawing sheet, in this case an A4 sheet. Two A5 sheets will be required for this design. (The design may consist of up to eight inter-connected pages of circuitry.)

To change pages 1 and 2 to an A5 sheet size, select **Circuit Setup ⇒ Setup Editor**. A window appears as shown in Figure 5. 1, divided into sections. Move the cursor into the *Page sizes* section, then select the down-pointing arrow alongside page **1**, **A4**. A list of the page sizes available appears (A5 to A1), select **A5**. Repeat for page 2.

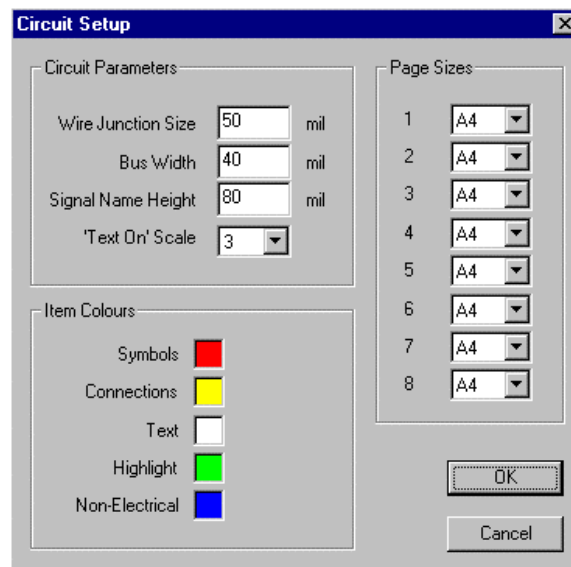


Figure 5. 1

Sheet sizes can be changed at any time during a design, but if a smaller sheet size is selected some of the data may appear outside of its area so it would have to be made larger again.

Circuit parameters and colours may also be changed from this window, but for this exercise they can be left at their original settings. Select **OK** to implement the changes.

#### Adding Symbols to the Circuit

Symbols are first added to the tray from the master or job library, then placed on the sheet.

#### Adding Symbols to the Tray

The master library is divided into sections known as volumes, numbered from 0 to 99. Resistors are held in one volume, capacitors in another, etc., etc. (Refer to the appendix for a list of the contents of each volume). Parts that are used in the job are copied to the job's own library. There are a number of ways to transfer parts from the library to the tray, use the method that's easiest for you.

If you don't know which volumes the parts are in, select **Tray ⇒ Add From All Volumes**. The tray appears on the left-hand side of the display and a window appears listing all the parts in numeric/alpha order, from all the volumes held in the library directory. (The volume numbers are given as well for reference.) Use the scroll bar on the right-hand side of the window to view all the parts.

Parts can be selected or de-selected with a fast double-click of the left button. \*\* appears alongside selected parts to indicate they are selected. (If you have difficulty double-clicking quickly, select the part, followed by the **Select** or **Deselect** button from the bottom of the window.

## SCHEMATIC CAPTURE

Select the following parts from the list.

<b>PTO18</b>	(volume 30)	<b>R0.25W</b>	(volume 40)
<b>1N4002</b>	(volume 32)	<b>CAPA10</b>	(volume 45)
<b>BRIDGE02</b>	"	<b>CAPR4D8E</b>	"
<b>BZX83C</b>	"	<b>PL10-01</b>	(volume 50)

Now select **Add to tray** to add the part names to the tray on the right side of the display and close the window.

When adding parts to the tray, it is possible to search for particular entries as follows. With the **Tray ⇒ Add From All Volumes** window open, type some or all the characters from the part name in the text entry box between *Select* and *Filter*, as shown in Figure 5. 2 (00 has been typed in this example), then select **Filter**. Only the parts with the specified characters somewhere in their name will appear in the list. Select **Unfilter** to restore the original list.



Figure 5. 2


To add the voltage regulator, *VREG-*, from the job library (if you created it), select **Tray ⇒ Add From Job Library**, select it in the same way, then add it to the tray.

Symbols may be added to the tray at any time. It's not typical to add them all at the beginning of a design. Don't worry if you've added other symbols to the tray accidentally as they can be removed - select the part in the tray followed by **Tray ⇒ Delete entry**.


Symbols that have not yet been placed on the sheet have a + sign alongside them in the tray.

### Placing Symbols on the Sheet

Once the symbols appear in the tray, they can be positioned on the sheet, and then moved, rotated or flipped into their final positions. Refer to Figure 5. 3 for circuit information.

Select  (**Symbol ⇒ Place**). Select the connector **PL10-01** from the tray, then move the cursor with the attached part towards the left-hand side of the sheet where the connector will be positioned. Locate it in its approximate final position and click the left-hand mouse button to release it. As it is released, a second symbol of the same type appears on the end of the cursor. Notice the + has now been removed from the part in the tray because it is now used in the job. As we do not need a second connector, move across to the tray and select the next symbol required - **CAPA10**.

Four CAPA10's (C1-4) are needed on page 1, so release these capacitors one after the other in their approximate positions.

Use the **Grid** menu (or  or the <g> key) to toggle the grid on/off. When symbols are released, their datum snaps to the nearest grid or half-grid position. Use the **View** commands or **icons** to zoom in/out and pan, or use the following keyboard keys if preferred.

<F1>	Zoom (one level, about the cursor position)
<F2>	Reduce (one level, about the cursor position)
<F3>	Full
<Space bar>	Pan (about the cursor position, whilst zoomed in)
<g>	Grid (toggle on/off)

When zooming in/out, the grid dots remain approximately the same distance apart. This is because an *auto-pitch* grid is active; it changes as you zoom in and out, so that the grid doesn't obliterate everything else on the screen, or become too widely spaced to be useful. The actual distance between dots is given in the information bar .

Place the rest of the symbols in the following list, in their approximate positions.

R0.25W	R1-4
BRIDGE02	BR1
CAPR4D8E	C5-6
BZX83C	D1
PTO18	TR1
1N4002	D2
VREG-	IC1 (if this device wasn't created, leave it off or select a different part)

Once all the symbols have been added, either press the right-hand mouse button, or select another command.



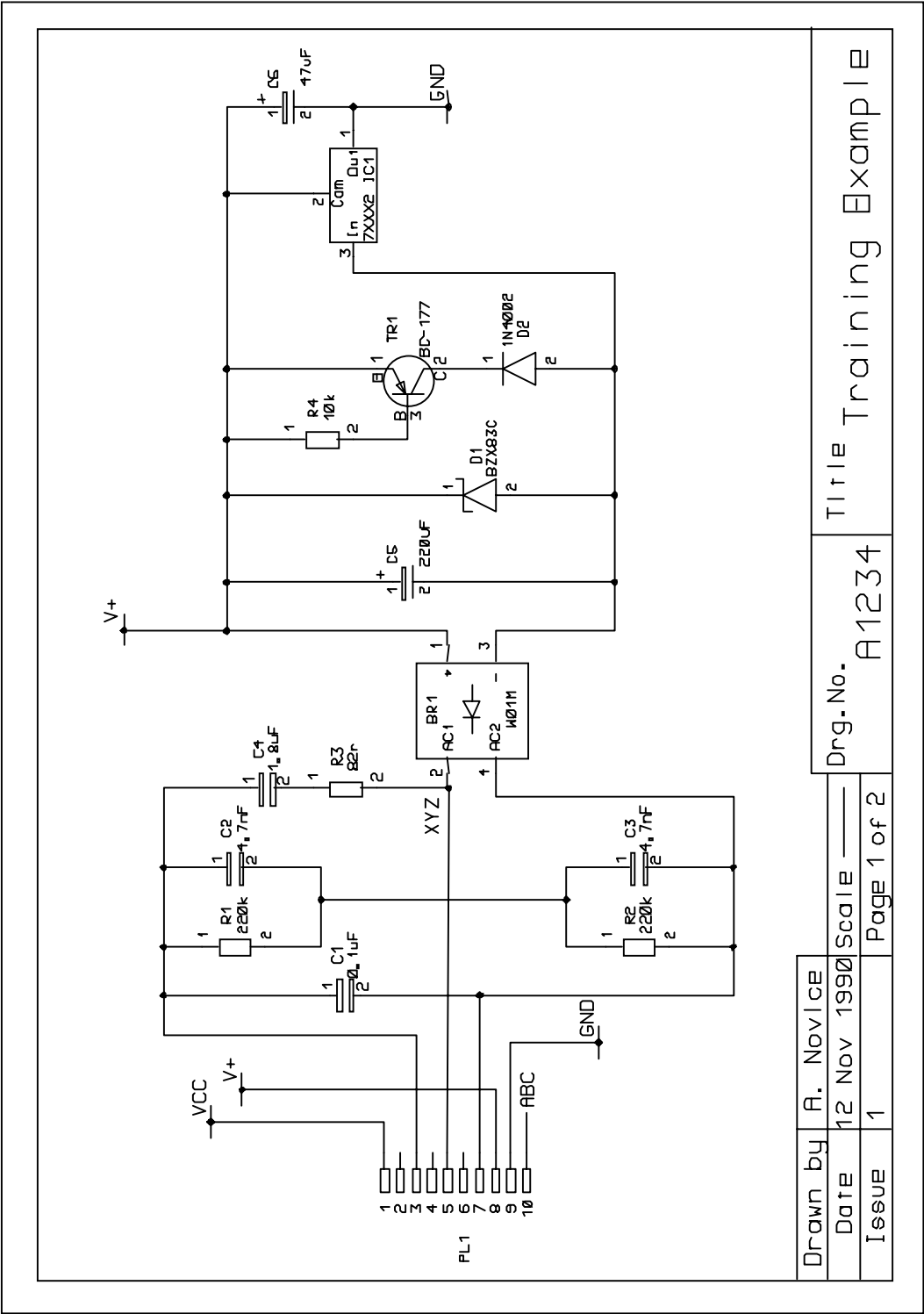


Figure 5. 3

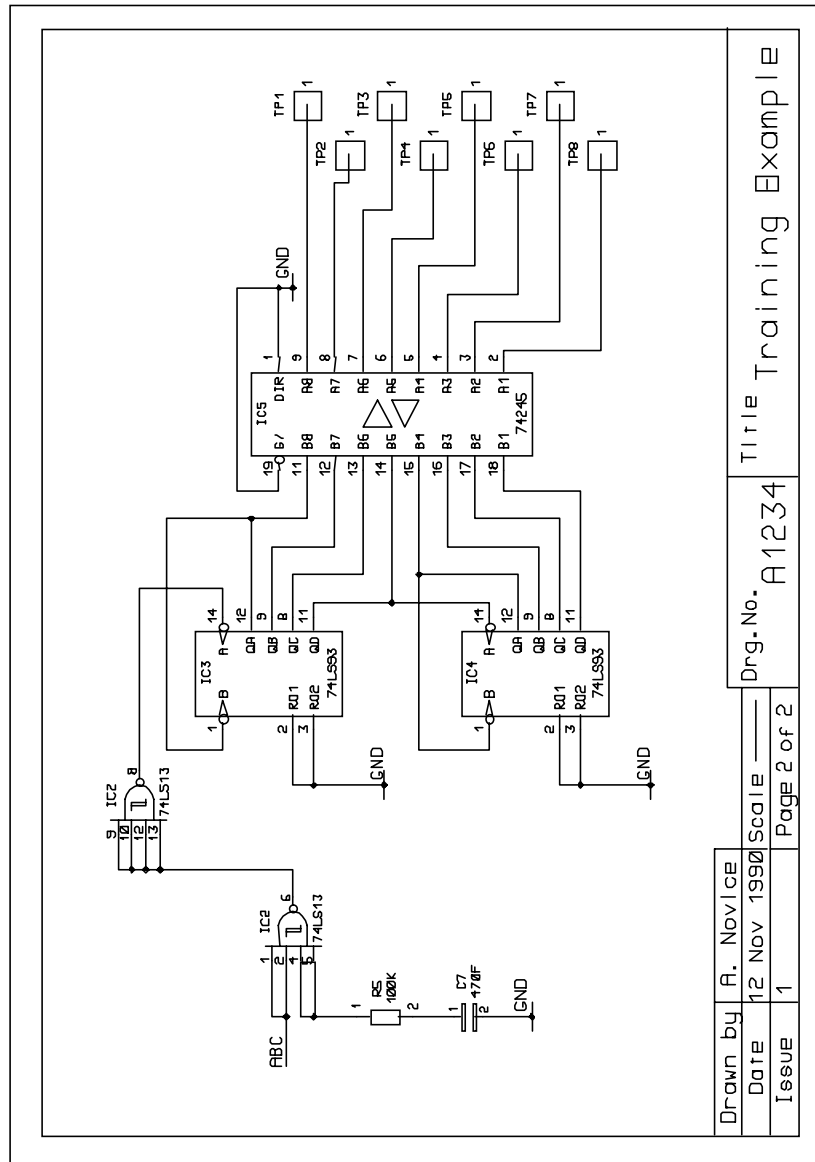



Figure 5. 4

## Rotating, Flipping and Moving Symbols


Once some symbols have been placed on the sheet, you may wish to start rotating, moving, flipping (mirroring) and aligning them to their approximate final positions. Symbols should be selected by their datum point, which is usually in the middle of the symbol.

### Rotating a Symbol

Select  (**Symbol** ⇒ **Rotate**), then select a part, such as a resistor. It rotates in 90 degree increments about its datum (the centre) each time it is selected.

Rotate the other symbols as required. (TR1, BR1 and PL1 need to be flipped as well.)

### Moving a Symbol

Select  (**Symbol** ⇒ **Move**), then select the symbol with the left-hand mouse button. Move the symbol to its new position and release it by pressing the left button again.

The right-hand mouse button cancels the movement and returns the symbol to its original position.

### Flipping (mirroring) Symbols

Flip is used to mirror the symbol. This is useful for transistors and connectors, as can be seen from this example. The results of flip are best seen on non-symmetrical symbols, or after the pin numbers have been assigned.

Select **Symbol** ⇒ **Flip**. Select the symbol to be flipped (PL1, BR1 and TR1).


## Aligning Symbols

Align is used to align symbols in either the X or Y axis, using their datum as reference.

Select **Symbol** ⇒ **Align X** (or *Align Y*). Select a position for the alignment axis with the *left-hand* mouse button. A dashed line appears. Re-position as required.

Select the symbols to be aligned with the *right-hand* mouse button. They move so that their datum is over the axis.

## Saving the design

Because it may have taken you some time to get to this stage, save the design in case of power failure. Select  (**File** ⇒ **Save**).

If the job has not previously been saved, the Save window will appear, with the Ranger2 jobs folder, *Sample Designs* open. You can navigate around to locate a different folder if required. A job name must be typed in the *File name* box, for instance **My Worked Example**. Press <enter> or select **Save** to action the save and return schematic.

Repeat this to guard against power failures as often as you think fit - it's entirely up to you how much work you are prepared to lose!

## Allocating Reference Designators, Pin Numbers and Values

Before a parts/wiring list can be obtained (compiled) from the schematic, all the symbols must be assigned to packages, and reference numbers and pin numbers added.

Ranger will not allow symbols to be packaged incorrectly. For example, it won't allow a 7400 in the same package as a 74LS00 or a 7404. Nor will it allow the same designator to be used twice. You won't find two R1's for example.

Automatic or Manual allocation may be used, or a combination of both. The *Allocate* menu is used to perform this operation.

### Manual Allocation

Select **Allocate** ⇒ **Auto Allocate Selected**. Select a symbol for allocation. This assigns the symbol to the next available package. Repeat as required, but leave some others unallocated for the rest of the allocation exercise.

or

Select **Allocate** ⇒ **Key Allocate Selected**, then select an unallocated symbol. Enter the number required for the part in the window that appears. This method allows you to choose the reference designator, and also the gate if so required, for the selected symbol. For example, when page 2 has been drawn, there will be two, four-input NAND gates that belong in the same type of package. Select one of them and enter *20.10* in response to the prompt. This will assign the gate to IC20, using pin 10 and associated pins.

### Automatic Allocation

Select **Allocate** ⇒ **Auto Allocate Everything** to allocate all the parts in the design. All the *unallocated* symbols on *every sheet* will be assigned to packages and pin numbers added. Priority is given to symbols on the first page in the top right-hand corner, numbers are then assigned vertically from left to right. Zoom in so that you can read the values.

Don't worry if your circuit is allocated differently to the circuit supplied, it just means your symbols are in a slightly different position on the sheet. You could manually allocate to obtain the same results.

### Assigning Values

Most symbols have a *value* assigned to them, taken from the symbol definition. This value may be changed on the circuit.

Typically it is used to change the default value of discrete components. For example, R1 automatically appears with a value of 100K, which according to Figure 5. 3 needs to be changed to 220k.

In the tool bar across the top of the graphical area as shown in Figure 5. 5, select the **Value** field box and enter the value required, i.e. 220k.

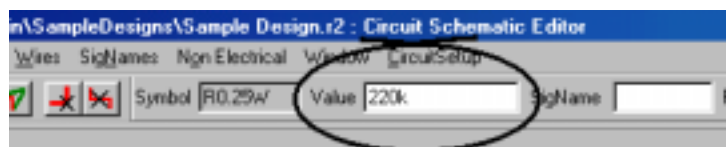


Figure 5. 5

Select **Symbol** ⇒ **Set Value**, select **R1** and its value will change.

Change the values of other components as required. Where more than one symbol has the same value, select the other symbols at the same time to save re-typing.

## Adding Connections

Connections may be made between the following:


terminal - terminal

## SCHEMATIC CAPTURE

terminal - wire  
wire - terminal  
terminal/wire - free space

Junction blobs are automatically added when connecting to existing wires. Connections left in free space should have a *signal name* added to create connectivity.

### Adding a connection

Select  (**Wires ⇒ Insert Connection**). To make a connection between C1 and R1 as shown in the Figure 5. 3, proceed as follows. (Switch the grid on to assist you and zoom in/out to a sensible level – so that you can see!)

Point at, and select the upper terminal of C1. As the cursor is moved, a green orthogonal wire appears on the end of it. Move the cursor up and to the right, (above R1) and insert a corner by pressing the left-hand mouse button. Now bring the wire down and position the cursor over the terminal of R1. Press the left-hand button again to terminate the connection. If the wire remains attached to the cursor, the terminal was not selected. You may need to zoom in.

Start the next connection on the right-hand corner of the connection just added and terminate it on the top terminal of C2. Notice a junction blob appears automatically. Add the rest of the connections as shown.


If additional corners are needed as a connection is being added, simply click the left-hand mouse button. The <space bar> can be used to pan whilst the connection is being added.

Some connections need to end in free space, in which case insert a corner, then release the wire with the right-hand mouse button. These connections will need to be given a signal name before the circuit is complete, to maintain connectivity to other connections with the same name. Adding signal names is described later.

The *symbols* for VCC, V+ and GND are not symbols at all, they are made from connections. Start the connection from a terminal, for instance C6 pin 2, then move the cursor down and to the left and insert a corner.

(You may need to toggle the grid on now, press the <g> key. The grid size is shown at the top left of the screen.)

Move the cursor 0.1 inches to the left and insert another corner. Move the cursor away and press the right-hand mouse button to release the connection. Now make a connection from the corner of the wire, away 0.1 inches to the right, insert a corner and then release the wire. You should have a wire in the shape of an upside down T. Repeat for the other power or ground connections.

If you make an incorrect connection simply delete it using  (**Wires ⇒ Delete Connection**). The complete connection between terminals or junction blobs is removed.

Connections that are not connected to another component, like the VCC and GND connections, must be given a signal name to create connectivity to other like-named connections.

If any symbols are moved, the connections remain attached to the terminals. You may need to tidy up the wires.

### Modifying Wires

Wires may be modified using the    icons (**Wires ⇒ Add Point, Move Point or Delete Point**). (Ranger refers to corners as *points*).

The wires always remain locked at right angles when they are moved if the **Wire Lock** tick-box in the tool bar has a tick alongside it. (Only the tick-box may be visible on the far right of the toolbar when running at lower screen resolutions.)

When the box is unticked (by selecting it to toggle it on/off) the connections may be moved at any angle. If wires are added with the *Wire Lock* off, then they will not 'square' up.

Wire junctions (junction blobs) are best moved with *wire locking* off.

### Buses

Buses should be added in the same way as all the other connections. Once added, the bus part of the connection should be thickened up using the **Wires ⇒ Toggle Signal/Bus Width** command to form the bus, as shown in Figure 5. 6. This command toggles a wire between a signal (thin line) and a bus (thick line) when the wire is selected.

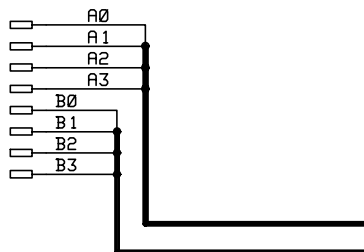


Figure 5. 6

Buses are not required in this exercise, but if you are creating your own circuit, ensure that all connections (thin lines) entering a bus have a signal name attached to them before the circuit is compiled, as shown in Figure 5. 6.

If required, the bus (thick line) may be given a name. The name *must* have a colon (:) in it somewhere to allow Ranger to differentiate it from signal names. Signal names should not have a colon in them .

(Ranger ignores any connections that have been thickened up - they are used to make the circuit easier to read. Connectivity is maintained by the signal names assigned to the signal (thin) connections.)

## Adding Signal Names

All connections may be drawn if required, but diagrams can get very cluttered or untidy. It is also necessary to maintain connectivity across sheets. *Signal names* ensure connectivity between all like named wires, including across sheets.

There are some signal names on the circuit (Figure 5. 3) that need to be added.

### Adding a signal name

First a signal name has to be specified. Select the **Sig Name** box in the tool bar and enter the signal name to be added, for example **GND**, as shown in Figure 5. 7.

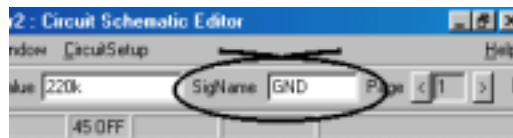


Figure 5. 7

Select **SigNames** ⇒ **Add**, select the correct wires (as shown in Figure 5. 3) with the *left-hand* mouse button. The name appears and will remain assigned to that wire, even if it is moved (using **SigNames** ⇒ **Move**). It can of course be deleted (using **SigNames** ⇒ **Delete**).

More connections may be selected to assign the same name, providing **SigNames** ⇒ **Add** is still active. Add all the signal names shown in Figure 5. 3.

When naming wires that increment by 1, i.e. A0, A1, A2, etc., start with the lowest number (A0), and use the *right-hand* mouse button to select each wire. The signal name increments by 1 each time a wire is selected.

## Viewing and Editing Other Pages

All the information Ranger2 requires has now been added to page one. Page two has yet to be designed.

Notice the *Page* setting in the tool bar (also shown in Figure 5. 7) and the arrows alongside it. These arrows can be selected to travel backwards and forwards between pages 1 to 8 of the design. Pages 2 to 8 are blank at the moment. To start page 2, ensure 2 is visible in the page button. Page 2 appears and may be designed.

Using the techniques already learnt, create page two. The following additional components are required:

7413	(volume 10)	IC2
7493	"	IC3, 4
74245	(volume 12)	IC5
TESTPIN	(volume 51)	TP1-5

Don't forget to save your design at frequent intervals, to allocate the symbols (*Allocate* menu) and to add the signal names (*SigNames* menu).

## Non-Electrical Information

Information added using the **Non Electrical** commands is ignored as far as connectivity is concerned.

Non-electrical data was used to add the drawing blank, titles, etc., shown in Figure 5. 3. Standard drawing blanks may be created and stored for future use using the *Window* commands.

## Window Move, Rotate, Save, Copy, Delete

The window must first be defined.

### Defining the window

Select **Window** ⇒ **Define**, select a corner of the yellow box that appears and drag it into the required position. Repeat with the other corners until the area is defined.

### Moving, copying, deleting, etc. the window


Once the window has been defined, it can then be moved, rotated, etc. Select the desired command, **Window** ⇒ **Move**, **Copy**, etc. If you're moving/rotating the window select the new position for it until it is in the correct location. Then select **Window** ⇒ **Go**. (The *Window* ⇒ *Copy* command does not include the signal names – use the Macro Generate command if they are required.)

## SCHEMATIC CAPTURE


### Saving an area of design - Macro Generation

Once a window has been defined, it may be saved. Select **Window ⇒ Macro Generate** followed by **Window ⇒ Go**. Enter a filename without an extension when prompted. The area of circuitry is saved on disk ready for future use. It includes everything except the current part allocation.

### Using Macros


To use the macro on any circuit, transfer it to the tray using **Tray ⇒ Add Macros** then place it on the circuit using  (**Symbol ⇒ Place**). Macros in the tray have \* alongside them to indicate that they are macros.

### Leaving the Circuit Diagram

To leave the circuit editor, select **File ⇒ Close Circuit**, the circuit is held in memory it is not saved. Use  (**File ⇒ Save**) to save the design.

### Enabling gate and pin swapping

The libraries supplied contain the information required for gate and pin swapping, but it is disabled. Normally the master library would be modified, but we will just change the job library.

Select **Libraries ⇒ Circuit Devices**. Select  (**File ⇒ Open Device**), you will see a list of all the symbols used on the circuit, plus any others that have been created.

Double-select **7413** to edit it. Enable symbol swapping by selecting **Edit ⇒ Properties**, then toggling **Enable symbol swapping** to **Yes**. Select **OK** to close the window.

Select any command from the **Pin Assignment** menu and you will see that all the input pins belong to the same **swap group** because they are all yellow (pink or blue). This means that connections can be swapped between these pins on the PCB.

Select **File ⇒ Close Device** to leave the device. Repeat for any other appropriate devices.

On this design, we are allowed to pin swap the connector, PL10-01. By default, it is not set up for swapping. Enable it for swapping, then define all the pins as being equivalent. (Refer back to *Defining equivalent pins* on page 15, if you can't remember how to do it.)

The resistors and non-electrolytic capacitors can also be enabled for swapping. You will have to define the equivalent pins on those as well.


Select **File ⇒ Close Library** to leave the device library.

### Copying Devices to the Master Library

Only devices that have been tried and tested should be copied to the master library. The copy is actioned from within the job library, select **Libraries ⇒ Circuit Devices**. Select **File ⇒ Copy ⇒ Device to Master Library**. Choose the volume where the device should be transferred to, then select the device to be transferred, followed by **Copy**. The status bar will indicate the copy has taken place.


More devices can be transferred to that volume. Select **Cancel** when all the required devices have been transferred. Another volume or **Cancel** can then be selected. Select **File ⇒ Close Library** to leave the device library.

### Creating a parts/wiring list from the schematic

Once the circuit has been designed, it must be compiled to produce a parts and wiring list. Select  (**Circuit ⇒ Compile**), then **Compile**.


Each page of the circuit is analysed and a parts and wiring list produced. Any reported errors should be investigated and the schematic corrected. Select **Close** or **OK** to continue.

For instance if unallocated parts are found, return to the schematic and allocate those parts (**Allocate ⇒ Auto Allocate Everything**). Ambiguous bus spurs are caused by signal names (without a colon (:)) being assigned to buses, or bus names (with a colon (:)) being assigned to signals. Delete the offending name (or connection if you're not sure) then re-name it. Refer back to *Buses* and *Adding Signal Names* (pages 22/23) for more information.

Once the changes have been made, select  (**Circuit ⇒ Compile**) then **Compile** again. The message **Compilation complete** should appear. Select **Close** to continue.

### Viewing the Compiled Parts/Wiring Lists

It's a good idea to view and check the parts/wiring list before commencing the artwork. This ensures you've drawn what you think you've drawn! It also allows you to check there are no invalid entries, for example outlines that are called up but not defined in the outline library, pin numbers that do not exist in an outline – these will all be highlighted. A list of unused pins can also be obtained.

Select  (**Circuit ⇒ Parts/wiring list**). If the parts or wiring lists contain invalid items an error window appears, select OK to continue and the invalid entries are highlighted in red. Corrective action will need to be taken.

For example if an outline is highlighted in the parts list, does the outline exist in the outline library? If a pin is highlighted in the wiring list, does the outline contain this pin number (i.e. IC25 calls up DIL14, if IC25.15 appears in the wiring list it will be invalid (only 14 pins in the outline). No pin 0's should exist in the wiring list, if they do, it is likely that you haven't assigned the pin numbers in the circuit device - edit the device and correct it, edit the circuit, de-allocate the part and re-allocate it, compile the parts/wiring list again.)

The *Parts List Editor* is displayed on the screen. It is possible but not practical to change any of the fields by selecting the desired field and retyping.

To find a specific part, select **Edit ⇒ Find Part**.

Compare this parts list with the parts list shown in Figure 8.3 (page 35), they should contain the same information. If they are different, correct the circuit and compile it again.

Behind the parts list window lies the wiring list window. Select the window to bring it forward.

Each node in the wiring list is connected to nodes on the same line, but not to other lines, unless the word *Contd>* appears in the *Sig Name* column of the next line. Signal names appear in the first column.

Pins that were defined as power pins in the device definitions will appear connected to the relevant power rail, i.e. VCC, GND, etc. Power names should always appear highlighted in the wiring list. If they are not, then they will be treated as signal connections. Later in this exercise, we will make V+ a power rail so that it is routed with thicker tracks.

Compare this wiring list with the wiring list shown in Figure 8.6 (on page 37). They should contain the same information. If they are different, correct the circuit and recompile it.

### Listing unused pins

A list of the unused pins within this job can be obtained, by selecting **Tools ⇒ Find unused pins** from the top of the parts/wiring list window. Select **File ⇒ Exit** to close the window.

### Leaving the parts and wiring list menu

Select **File ⇒ Exit Editor** to return to the job design menu.

The schematic is now complete. Now is a good time to save the job (**File ⇒ Save**) and also take a separate backup copy of the design for future use (**File ⇒ Save As**). The title bar indicates which job is open, you may want to close the backup copy and open the original job (**File ⇒ Close Job**, **File ⇒ Open Job**).

## Chapter 6

## CHECKING THE BOARD PARAMETERS

Whether or not you have drawn a circuit diagram, start designing the printed circuit board artwork here.

Where the procedure for artwork design differs depending on whether or not a schematic has been drawn, the heading *Relevant to Schematic Entry Only* or *Relevant to Parts/wiring list Entry Only* appears. Where the text becomes relevant to both methods again, the heading *Relevant to Both* appears.

The example board design will initially be double sided, with instructions given on how to change it to a single-sided or multi-layer design.

When the job was created, a set of design defaults information was copied into it from the master default tables of information. These defaults can be changed within the job without affecting other jobs or the master set.

Typically, these changes would be made at the beginning of the artwork layout, although they can be changed at any time (the artwork will be updated automatically).

If the job isn't open, open it now, select  (**File ⇒ Open Job**), select the job you saved previously (*My Worked Example*).

### Sizes table

From the job menu, select **Edit ⇒ Sizes table**. A table appears listing the track sizes assigned for this particular job, similar to the one shown in Figure 6. 1.

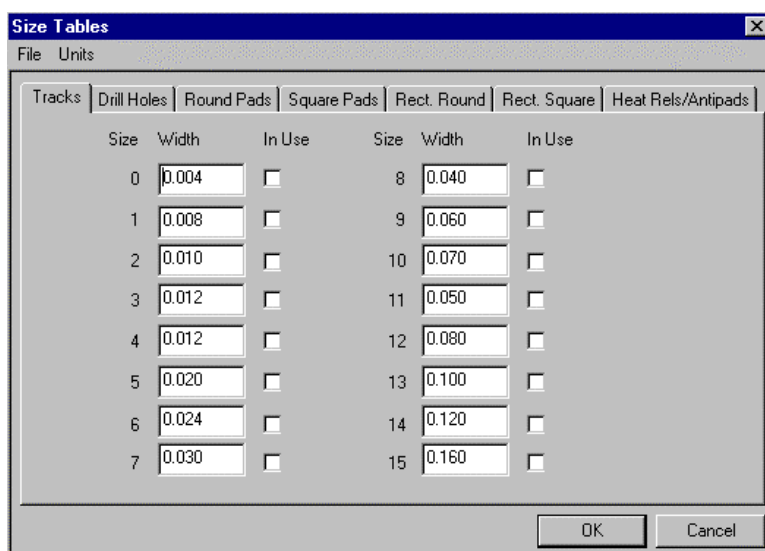


Figure 6. 1

Although the term *track* sizes is used, the sizes in the table are also used when drawing silk-screen lines and for the characters in text strings.

Up to 16 different track sizes can be defined for each job, which are numbered from 0 to 15. These numbers are referred to as the *Size*, although they are sometimes referred to as the *code* or *size-code*.

Alongside the *Size*, is the actual size assigned to the track. For instance, size 4 is assigned as 0.012". The *Units* command at the top of the sizes window can be used to toggle between imperial and metric units as required. Imperial (inch) units are displayed with a dot (.) as the decimal point, whilst metric units (mm) use a comma (,) as a decimal point.

Please note: the finest resolution of Ranger2 is 0.001" (or 0.0254mm).

You can see whether any size is currently in use within the artwork, by a tick being present in the *In-use* box. The artwork design has not been started yet, so no ticks are present.

Any of the sizes may be changed using the standard Windows editing facilities, the changes are updated in the artwork automatically.

Before you start to change these sizes, be aware that track codes 4 & 11 are the default sizes for signal and power tracks respectively. If you do decide to change these values, your design will be different to the one in the example.

Later in the exercise we will need a 0.026" track in order to create a solid copper fill area on the board, so change the code 6 track to 0.026".



Select the **Round pads** tab from the top of the window. A table appears similar to the one in Figure 6. 2, listing the 16 round pad sizes that can be used within this design.

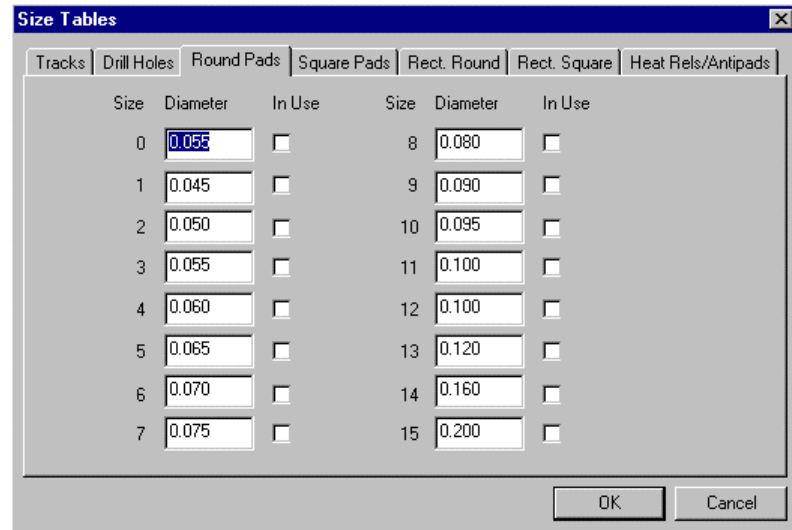


Figure 6. 2

Again the values may be changed, but it is wise to maintain the values assigned until you know which pads are in use by the supplied library outlines, then change the ones not in use.

Note: Size 0 round pads are always used for vias, so ensure that code 0 round pads are set to the size you require for vias, changing it in the table if necessary.

Code 0 pads should not be used for anything other than vias. If a component pad requires the same size, use a separate pad code to define it.

To check the hole sizes defined for the pads, select the *Drill Holes* tab from the top of the window. Although a drill hole size is specified for each of the 16 pads, the pads are not necessarily drilled when they are used, the pads can either be drilled or undrilled.

There are four standard pad shapes available (round, square, round-ended rectangular and square-ended rectangular), their sizes can be viewed by selecting the appropriate tab from the top of the window. Heat-relief and anti-pads are used on internal power planes, which we will discuss later.

The drill hole size relates to all pads with the same size code. For instance, if the size 2 drill hole is changed, it affects all four standard pad shapes.

All the sizes we require for the worked example are present. Most of the pad sizes listed are used by the outlines in the supplied outline library, but it is unlikely that all the outlines will be used in one design. Once the parts have been placed and the pads digitised, the used pad sizes are indicated, so it is possible to identify unused pad sizes and change those if required.

The dimensions can be entered in inches or millimetres, after choosing the appropriate units (*Units* command).

Select **OK** to implement any changes and close the window. Select *Cancel* if you have not made any changes or do not wish to implement the changes.

### Viewing or editing the master sizes table

A master size table is supplied, which may be modified and stored. All new designs will have this table copied into them when they are created. To modify the master table, close the job if one is loaded, then select **Edit** ⇒ **System Defaults** ⇒ **Sizes Table**. Existing jobs will not be affected by this change.

**Important Note:** changes made to the master size table will affect the master outline library that uses the master size table. If you intend modifying the master size table, proceed with caution!

### Power rail names

Ranger needs to know which connections are power connections so that they can be displayed independently of the signal connections, routed with thicker tracks, biased in a particular direction, auto-routed with a power routing strategy, or added to an internal power plane.

Select **Edit** ⇒ **Define Power Names**, the window shown in Figure 6. 3 appears. Up to 8 power rails can be defined within a job.

## CHECKING THE BOARD PARAMETERS

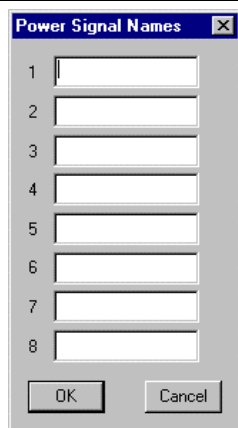


Figure 6. 3

### Relevant to Schematic entry only:

If a schematic has been drawn and compiled into a parts/wiring list, the power rails defined in devices used on the schematic, appear in this table automatically. (VCC & GND if you are following this worked example)

If you require additional named connections to be treated as power rails, their names should be added to this table. We require V+ to be treated as a power rail, so add it to this list.

If you do not want a named net treated as a power rail, remove it from this list.

For this design, VCC, GND & V+ should appear in this table. If you are working on your own design, ensure all the power rail names you require are entered.

### Relevant to parts/wiring list entry only:

Before typing in a wiring list, the signal names that you require to be treated as power rails should be defined in this table.

Enter the following power rail names for the worked example. If you are working on your own design, ensure all the power rail names you require are entered.

**GND    VCC    V+**

### Relevant to Both:

Do not change the order of the names in this table once the wiring list has been compiled from a schematic or typed in. You will find the names changed in the wiring list too.

If a name is removed from the table, the power name will also be removed from the wiring list although the connection will remain, but unnamed.

Select **OK** when the power rails have been defined.

## Plane assignments

The *Plane Assignment* table gives control over the artwork layers. Ranger2 has 16 layers available for copper and silk-screen information. They are numbered from 0 to 15.

Select **Edit ⇒ Plane Assignments**, a window similar to that in Figure 6. 4 appears.

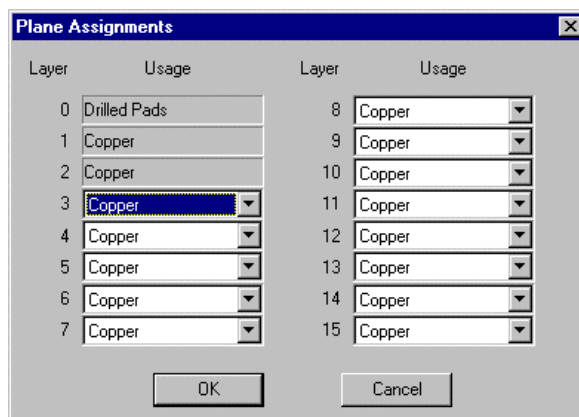


Figure 6. 4

Layer 0 is always used for pads with drilled holes (conventional component pads and vias) and nothing else (usually).

Layers 1 and 2 are used for the outer copper (tracks) layers of the board. Surface mounted component pads also appear on these layers.

If you use any of the supplied outline shapes, layer 1 is the *top* copper (component) side and layer 2 is the *bottom* copper (solder) side.

The other layers can be changed to silk-screen or power plane layers as required.

Selecting any of the layers from 3 onwards produces a list of all the options available for that layer, one of which can be selected. The options will be *Copper*, *Silk-screen* and each of the defined power rail names, i.e. *VCC*, *GND* or *V+* in this example.

If a power name is selected, it means the layer becomes an internal power plane for that particular power rail.

The silk-screen layers will be generated automatically later on in the design, and this table will be updated automatically.

Ranger2 performs design rule checking on all layers except silk screen layers. So if you want to use a layer for notes, assembly instructions, measurements, etc. that you do not want to use as part of the finished artwork, assign the layer as a silk-screen layer.

Return the layers to copper layers as we are designing a double-sided board (the extra layers won't be used).

Select **OK** to save the changes.

### Part code table

Before parts may be entered in the parts list, their prefixes must appear in the job's *part code table*. For example, will *IC* or *U* be used for integrated circuits? A default set is supplied.

#### Relevant to Schematic entry only:

If a circuit has been drawn and compiled, any additional part codes will have been inserted in the job's part code table automatically, so you do not need to do anything.

#### Relevant to Parts/wiring list entry only:

If you haven't drawn a circuit, the additional part codes should be entered now, although no additional part codes are required for this worked example.

However, if you are creating your own design, check that all the prefixes you need are present.

#### Relevant to Both:

Note: once the parts list has been compiled or typed in, the order of codes in the part code table must not be changed. If you do change this table, you will alter your parts/wiring list and it will no longer be correct.

### Viewing or editing the part code table

Select **Edit** ⇒ **Part Prefix Codes**. A window similar to that in Figure 6. 5 appears.

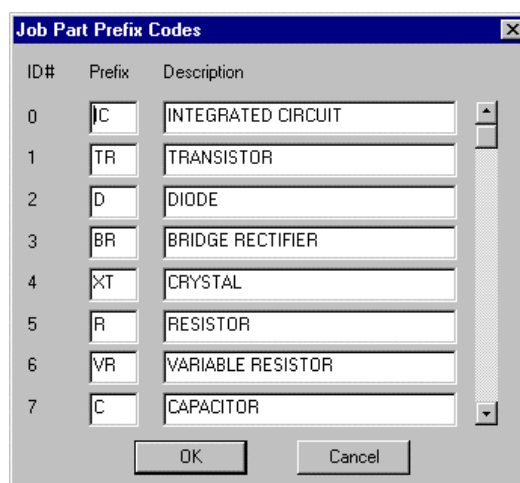


Figure 6. 5

Up to 32 part codes and their descriptions can be defined, they are numbered from 0 to 31. If a part code you intend using does not appear, scroll through the list to an empty part code and enter the code required.

The table is case-sensitive, most codes should be entered using upper case characters. A description can be entered in the column alongside, but Ranger does not refer to it.

Note: once the parts list has been compiled or typed in, the order of codes in this list must not be changed. Only new prefixes should be added, and at the end of the existing entries.

Select **OK** when you have finished, to update any changes.

You are now ready to define the board profile.

### Viewing or editing the master part code table

A master part code table is supplied, which may be modified and stored to suit your company standards. All new designs will have this table copied into them when they are created. To modify the master table, close the job if one is loaded, then select **Edit** ⇒ **System Defaults** ⇒ **Part Prefix Codes**. Existing jobs will not be affected by this change.

### Important Information


Do not alter the order of names in either the *Part code table* or the *Power names table* once the parts or wiring lists have been created. If you do, you will also be changing the parts/wiring list and it will no longer be correct.

### Chapter 7

## DEFINING THE BOARD PROFILE

The board profile acts as a guide for you when manually placing parts, routing tracks, etc. The automatic routines such as auto-routing, copper fill, etc. also operate within the area defined by the profile. Keep-out and router lines can also be added in this editor.

### Defining the board profile

Select  (**Artwork ⇒ Define Profile**), the graphical profile editor appears and looks similar to Figure 7. 1.

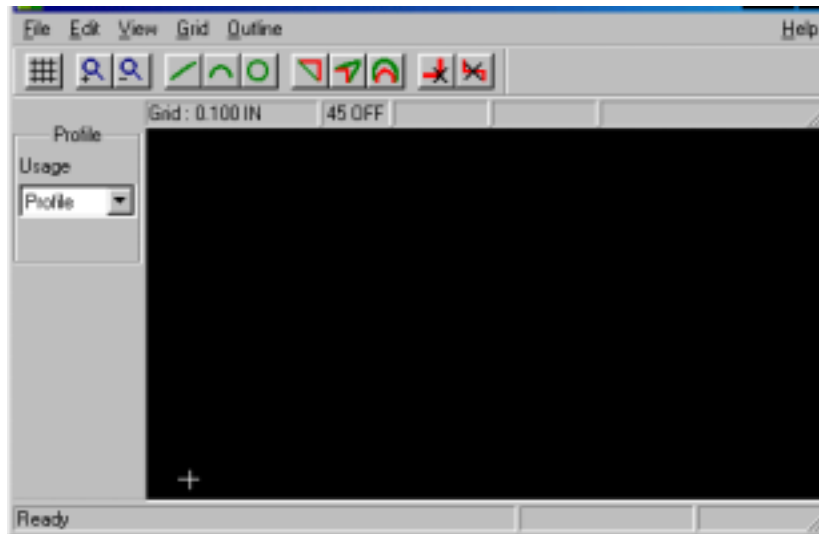


Figure 7. 1

The command menu and toolbar can be seen across the top of the editor.

An information bar appears above the working area, it indicates that the grid pitch is currently set to 0.100", and that auto 45 degree snapping is switched off (45 OFF). Other information will also appear in this bar as you start to do things.

A dialogue bar appears down the left hand side of the screen, with a pad titled Profile Usage, which is currently set to Profile.

Towards the lower left corner of the working area, a pink cross can be seen, this is the datum of the editor.


The status bar across the bottom of the screen indicates *Ready*.

If you created the devices for the schematic, then you will know how to add and modify lines, arcs, circles, etc. If you are confident with your drawing skills, then move on to the heading *Drawing the profile*.

If you started the design with parts/wiring list entry in mind, then you will not have used Ranger's graphical editors yet. So before defining the profile, take a few minutes to become familiar with drawing in Ranger2. Experiment drawing lines, circles, arcs, etc., as explained next, until you are happy to continue. The practice lines will be deleted before continuing with the rest of the exercise.


### Drawing lines, circles, etc.


#### Drawing lines

Select  (**Outline ⇒ Add Line**). Across the bottom of the window the status bar indicates which command is currently active (Profile, Add line) and what you should do next to action the command (Select line start position). Move the cursor into the working area and click the left hand mouse button to start a line. Move the cursor and click the left button to insert corners (or points) in the line. Click the right hand button to release the line. The line is made up of a series of straight line segments.


#### Grid snapping/visibility

At the moment, grid snapping is switched on, even though you can't see the grid. This means that the cursor and therefore points (corners) in the line always snap to the nearest grid or half grid point.

You will find it easier to notice what is happening if you switch the grid on. Select  (**Grid** ⇒ **On/off** or press the <g> key) to toggle the grid on. The grid covers the working area - lines cannot be added outside of this area.

Zoom in,  (**View** ⇒ **Zoom In**), then select the position you wish to zoom-in to. As you zoom in, the grid dots remain approximately the same distance apart. This is because an **auto-pitch** grid is active.

An auto-pitch grid changes as you zoom in and out, so that the grid does not obliterate everything else on the screen, or become too widely spaced to be useful. The actual distance between dots is always given in the information bar across the top of the working area.

Now draw a few more lines (, **Outline** ⇒ **Add Line**). You should notice that the points now snap to the grid or half grid.

### X/Y Readout


An X-Y readout can be displayed in the top information bar - select **View** ⇒ **X-Y Readout**, the co-ordinates step in increments of the grid if grid snapping is switched on. (The readout is in millimetres if the units are set to Metric (**Edit** ⇒ **Units** ⇒ **Metric Units**) and a metric grid is selected (**Grid** ⇒ **Metric**.)

### Grid


It is very seldom in schematic or artwork design that you work with grid snapping switched off. However should you wish to do so, select *Grid*, notice the tick alongside *Snap to Grid*. This means that grid snapping is switched on. If you select *Snap to Grid* the tick is removed and the window closed. Notice the words *SNAP OFF* appear in the information bar. Now when you draw lines, the points are released wherever the cursor is positioned, the finest resolution being 0.001". Switch grid snapping back on (*Grid* ⇒ *Snap to Grid*).

In the *Grid* menu you also have the ability to select *Metric auto-pitch* grids and *fixed grids* that are fixed at the size specified at all zoom levels. A user-defined grid can also be selected, you will be prompted for the X & Y pitch. We will come back to this menu later when it is more relevant. Set the grid back to *Inch (Autopitch)* and the Units back to Inches if you changed them.

### **Adding circles**

Select  (**Outline** ⇒ **Add Circle**). Get used to following the help in the status bar. i.e. select circle centre position, move the cursor to stretch the circle, then click at required radius position with the left hand mouse button to release the circle. If the right button is clicked whilst stretching the circle, it is cancelled. The command stays active so another circle could be defined.

### **Adding arcs**

Select  (**Outline** ⇒ **Add Arc**). Select the start point of the arc, move the cursor and select the end point of the arc, now move the cursor to stretch the arc and click the left hand mouse button to release it. More arcs can be added until the right button is clicked after an arc has been released.

If the right button is clicked whilst stretching the arc, a straight line segment is introduced. This allows a series of arcs and straight lines to be added at the same time.


### **Zooming in, out, panning etc.**

A quick method of zooming in and out, panning, etc. can be performed by pressing keys on the keyboard. These keys can be used whilst you are drawing lines, etc. Position the cursor over an item, such as a circle you've drawn, then press <F1> to zoom in, <F2> to reduce, and <Space bar> to pan. <F3> restores the complete profile to the display. The keys operate about the cursor position.


### **Modifying the outline**

Existing lines, arcs and circles can be modified with the following commands. Refer to the status bar for guidance on how to action the commands. The following tips may also be useful to you.


#### Adding points (corners) to lines

Select  (**Outline** ⇒ **Add Point**). When selecting the line in which the point will be added, select the line away from an existing point (corner) i.e. in the middle of a segment - you can then be sure the point is added in the segment you want. Whilst the new point is being moved it can be cancelled with a click of the right button.

#### Moving points (corners) in lines



Select  (**Outline** ⇒ **Move Point**). You must select an existing point (corner) in a line. Whilst the point is being moved it can be cancelled with a click of the right button.

#### Moving and adjusting the size of circles


Select  (**Outline** ⇒ **Move Point**). Select the edge of a circle to change its size, select its centre to move it.

## DEFINING THE BOARD PROFILE


### Deleting existing points in a line

Select  (**Outline ⇒ Delete Point**). The endpoints of lines cannot be deleted if there is only one segment left in the line.  
(Use  (**Outline ⇒ Delete Line**) to delete complete lines.)

### Modifying arcs, converting lines to arcs

Select  (**Outline ⇒ Adjust arc**). Select the arc or line, move the cursor, then click the left button to release the line/arc. If the right button is clicked whilst moving an arc, the arc is converted into a straight segment.

Repeated clicks of the right button converts the last selected segment between a line and an arc. The position of the cursor about the previously selected line determines which way the arc is adjusted.

Once you are happy using these commands, delete the practice lines and arcs using  (**Outline ⇒ Delete Line**). If you have trouble selecting a line, point at a corner or an end point of the line, or zoom in if necessary.

Select **View ⇒ Full** or <F3> to restore the screen to its original scale. If any of your practice lines are still present, delete them.

Now draw the outline as shown in Figure 7. 2, using the *Grid* commands and X/Y readout to draw the device to scale as described.

## Drawing the profile

The board profile for this example is shown in Figure 7. 2.

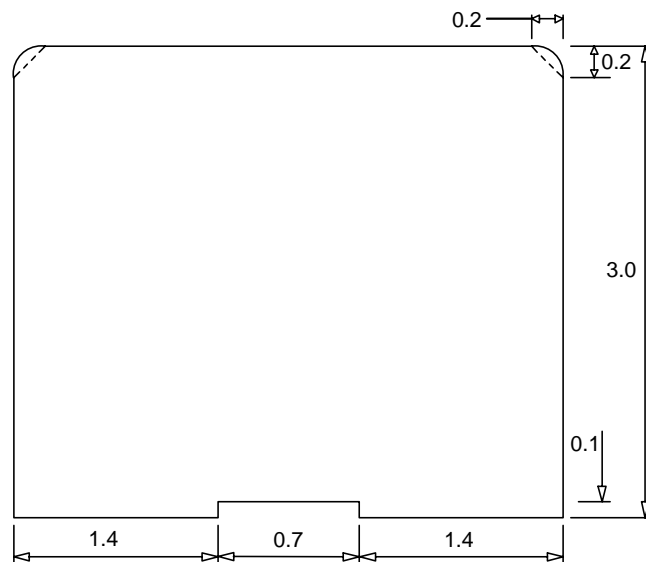



Figure 7. 2

Ensure the grid is visible, grid snapping is switched on and an *Inch auto-pitch* grid is selected (**Grid** menu).

Turn on the X-Y co-ordinate readout, (**View ⇒ XY Readout**) - as you move the cursor, its position is displayed in the information bar.

The pink cross is usually used as the lower left corner of the board profile. It is 1" in and up from the corner of the working area (0,0).

Never start the profile along the edges of the working area (zero in either axis) as those lines will be removed automatically when you leave the editor.


Select  (**Outline ⇒ Add Line**). Position the cursor over the datum point (at x=1.0, y=1.0), start drawing a line, you may need to zoom in <F1> or pan <space bar>.

Move the cursor to the right, to X=2.4, Y=1.0", then click the left button to insert a corner. The first segment is now 1.4" long. Move the cursor up by 0.1", to X=2.4, Y=1.1", and insert the next corner. Continue to add corners in order to define the shape shown in Figure 7. 2. Ignore the curves at the top of the profile for the time being, instead insert a diagonal line as indicated by the dashed lines.

Click the right hand button to release the line when the outline is complete.

### Modifying the outline

Use the **Outline** icons/commands if any adjustments need to be made.

Use  (**Outline ⇒ Adjust arc**) to convert the 45 degree lines into arcs. Select the segment with a click of the left-hand button, move the curve to the required position and click the same button again.

To ensure the curve is a quadrant of a circle, select the curve with a click of the left hand button, followed by a click with the right button. The curve is converted to a straight line. If the cursor is now moved to one side of the line, and the right button clicked again, the last selected line is converted to a quadrant of a circle. Repeated right clicks, toggles the arc to a line and back again. (If the cursor is moved to the other side of the line, the quadrant is created in the opposite direction.) The board profile should now be correct.

### Router & Keep-out information

Router lines are used as the path for the NC routing tool. (No width is assigned to the path, as this will vary depending on the cutter in use.)

Keepout lines are used to stop the automatic routines, such as the auto-router and copper fill crossing into areas where you do not want them to go.

Keepout and router lines are defined in the same way as the board profile, except that *Keepout* or *Router Path* should be selected from the **Profile Usage** dialogue bar on the left side of the screen.

Lines drawn in keepout mode are displayed in yellow and router lines pink. Profile lines are blue.


Select **File** ⇒ **Close Profile** to close the profile and return to the job design menu.

### Saving and backing up your design

(If you drew the circuit diagram you should have saved your design quite a few times already and taken a back-up copy.)

All your work is held in memory until you actually save it. This means that if there is a power failure, none of your work will be saved. It is VERY IMPORTANT that you get into the habit of saving your work and also taking back-up copies. A back-up copy is quick to take and may save you HOURS of repetitive work if you go down the wrong path. Please take regular backups.

#### Saving the design

Select  (**File** ⇒ **Save**). If you have already saved the design, then the save takes place. This will overwrite the previous save. If the design has not previously taken place, then you will be prompted for the location and name for the design in a window similar to that shown in Figure 7. 3.

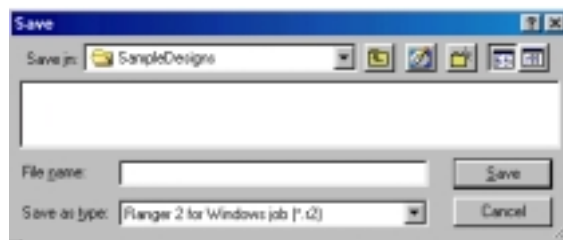


Figure 7. 3

The *Save in* bar indicates which folder the job will be saved in, this can be changed using the browse facility. The name that you want to assign to the job should be entered in the *File name* bar, for example *My Worked Example*. The *Save as type* bar will always be set to *Ranger2 for Windows job (\*.r2)*.

Select **Save** when you are ready to save the job. You can continue to work on the job.

#### Taking a back-up copy of the design

The procedure for taking a backup copy of the job is the same as for saving the job as explained above, except you must select **File** ⇒ **Save As** instead of **File** ⇒ **Save**. Make sure you choose a different name for the job.

When the save is complete, the newly saved file will be open (the *Title bar* indicates which job is open) so you may want to close this job (**File** ⇒ **Close Job**) and open the original job (**File** ⇒ **Open Job**) to continue working.




## Chapter 8

### CREATING A PARTS & WIRING LIST

If a circuit diagram has been drawn and compiled into a parts/wiring list, move on to the next chapter titled **Creating component outlines**.

If you have not drawn a circuit diagram then you must type in the parts/wiring list. Ranger2 does not lend itself to creating an artwork simply by adding pads and tracks in the artwork (although it can be done).

Select  (**Circuit ⇒ Parts & Wiring list**). A window similar to that shown in Figure 8. 1 appears. It consists of the main window with the command bar, plus a window for the parts list and behind that, the wiring list.

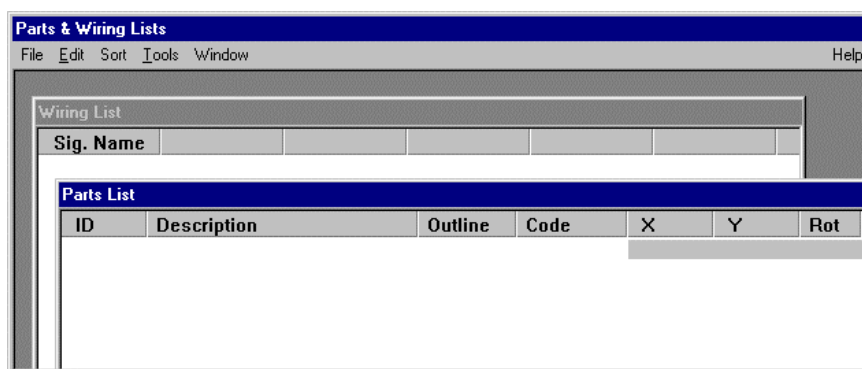


Figure 8. 1

### The Parts List

The parts that are required on the design must appear in the parts list window as shown in Figure 8. 2. As can be seen, the editor is divided into *fields*, consisting of *ID*, *Description*, *Outline*, *Code*, *X*, *Y* and *Rot*.

Parts List						
ID	Description	Outline	Code	X	Y	Rot
IC1	VREG-, 7XXX2	T092		0.000	0.000	0
IC2	7413, 74LS13	DIL14		0.000	0.000	0
IC3	7493, 74LS93	DIL14		0.000	0.000	0
IC4	7493, 74LS93	DIL14		0.000	0.000	0
IC5	74245, 74245	DIL20		0.000	0.000	0
TR1	PT018, BC-177	T018		0.000	0.000	0
D1	BZX83C, BZX83C	DO35		0.000	0.000	0
D2	1N4002, 1N4002	DO41		0.000	0.000	0
BR1	BRIDGE02, W01M	BRIDGE02		0.000	0.000	0

Figure 8. 2

The *ID* and *Outline* fields must have an entry, the other fields are optional so can be left blank. They are described below, so you can decide whether to spend time typing them in (not recommended for this exercise).

#### ID field

The *ID* is the part reference, for instance IC1, IC2, R1, etc.

#### Description field

The *Description* field is optional but it makes the link to the schematic parts that will be required if you intend using gate and pin swapping on the artwork.

**Note:** the gate and pin swapping database for each part is held within its schematic device definition. If a schematic has not been drawn and you wish to perform gate and pin swapping on the artwork, ensure the relevant devices exist within the job device library by adding them to the empty schematic sheet. The first part of the field, before the comma, must correspond to the schematic device name. The second part, after the comma should correspond to the value field of the device. Only gates with the same description field can be swapped one with another. If the field is left blank, gate and pin swapping cannot be performed on that part.



## Outline field

The *Outline* field must contain the name of the library outline that defines the physical representation of the part. (A separate booklet available if required, contains a print-out of all the outlines available.)

## Code, X, Y and Rotation fields

The *Code*, *X*, *Y* and *Rotation* fields are optional, so if you're typing in the data you can leave them blank - recommended for this exercise.

*Code* - this field is not used by Ranger, but it could be used to enter a company stock number for the part. (The parts list can be output to an ASCII file (*File* ⇒ *Save as Text*) which could be converted into a bill of materials, or similar.)

*X*, *Y* and *Rotation* - these fields are updated as parts are moved on the board. (The co-ordinates can only be entered or modified after selecting **Edit ⇒ Allow Position Edit** from the Parts and Wiring List command menu.)

## Typing in the parts list

The cursor should be blinking in the *ID* field waiting for data entry, if it isn't select anywhere in the first horizontal entry line. For the worked example, type in **IC1** and press <tab> to move to the next field (*Type*).

This is an optional field, so type **VREG-,7XXX2** if required, then <tab> to move to the *Outline* field.

To save you remembering and then typing the outline names, press <tab> again. Because an entry must be made in this field, Ranger presents the list of outline names from the library, from which one must be selected. Scroll through the list, then double select **TO92** (for the worked example). The name appears in the outline field. (If you find this long winded, you can always just type the correct outline name in.)

Press <enter> when all the data required for IC1 has been typed.

If you are following the worked example, type in the details for IC2 and IC3 as shown in Figure 8. 3.

Part	Type	Outline	Code	X	Y	Rot
IC1	VREG-,7XXX2	TO92	-	0.000	0.000	0
IC2	7413,74LS13	DIL14	-	0.000	0.000	0
IC3	7493,74LS93	DIL14	-	0.000	0.000	0
IC4	7493,74LS93	DIL14	-	0.000	0.000	0
IC5	74245,74245	DIL20	-	0.000	0.000	0
TR1	PTO18,BC-177	TO18	-	0.000	0.000	0
D1	BZX83C,BZX83C	DO35	-	0.000	0.000	0
D2	1N4001,1N4001	DO41	-	0.000	0.000	0
BR1	BRIDGE02,W01M	BRIDGE02	-	0.000	0.000	0
R1	R0.25W,220k	RESA10	-	0.000	0.000	0
R2	R0.25W,220k	RESA10	-	0.000	0.000	0
R3	R0.25W,82r	RESA10	-	0.000	0.000	0
R4	R0.25W,10k	RESA10	-	0.000	0.000	0
R5	R0.25W,100K	RESA10	-	0.000	0.000	0
C1	CAPA10,0.1uF	CAPA10	-	0.000	0.000	0
C2	CAPA10,4.7nF	CAPA10	-	0.000	0.000	0
C3	CAPA10,4.7nF	CAPA10	-	0.000	0.000	0
C4	CAPA10,1.8uF	CAPA10	-	0.000	0.000	0
C5	CAPR4D8E,220uF	CAPR4D8E	-	0.000	0.000	0
C6	CAPR4D8E,47uF	CAPR4D8E	-	0.000	0.000	0
C7	CAPA10,470F	CAPA10	-	0.000	0.000	0
PL1	PL10-01,10WAY	IDC10	-	0.000	0.000	0
TP1	TESTPIN,TP	SIL1	-	0.000	0.000	0
TP2	TESTPIN,TP	SIL1	-	0.000	0.000	0
TP3	TESTPIN,TP	SIL1	-	0.000	0.000	0
TP4	TESTPIN,TP	SIL1	-	0.000	0.000	0
TP5	TESTPIN,TP	SIL1	-	0.000	0.000	0
TP6	TESTPIN,TP	SIL1	-	0.000	0.000	0
TP7	TESTPIN,TP	SIL1	-	0.000	0.000	0
TP8	TESTPIN,TP	SIL1	-	0.000	0.000	0

Figure 8. 3

Once IC3 has been entered, use the *Repeat part* command as follows, to enter IC4. Position the cursor over one of the fields of IC3, then double select it quickly to select the complete line. The line highlights to indicate it is selected. Select **Edit ⇒ Repeat Part**, a window similar to Figure 8. 4 appears.

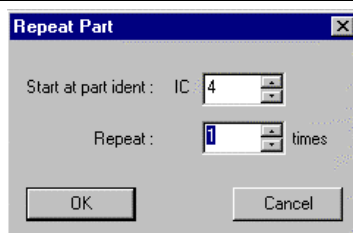


Figure 8. 4

Enter the details as shown then select **OK**. *IC4* appears. This command will obviously be more useful when there are a lot of repeated parts, like the resistors, capacitors and test points.

If a mistake is made, or *Descriptions* need to be edited on repeated parts, simply select the string to be edited and make the changes using the normal Windows editing tools.

Complete the parts list as shown in Figure 8. 3, pressing <tab> between fields and <enter> at the end of the line.

Be aware that the changes are not saved until you leave the parts and wiring list editor (**File ⇒ Exit Editor**) and select **File ⇒ Save**. It is suggested that you use this command regularly. Select **Circuit ⇒ Parts & Wiring List** to continue.

### The Wiring List

Assuming the parts list is on the screen, click the wiring list to bring it forward. The inter-connections between the parts in the parts list should be entered here, as shown in Figure 8. 5.

Wiring List					
Sig. Name					
C3 . 2	BR1 . 4	R2 . 2	PL1 . 7	C1 . 2	
C4 . 2	R3 . 1				
C2 . 1	C4 . 1	R1 . 1	C1 . 1	PL1 . 3	
C2 . 2	R1 . 2	R2 . 1	C3 . 1		
C5 . 2	BR1 . 3	D1 . 2	D2 . 2	IC1 . 1	

Figure 8. 5

The editor is divided into fields like the parts list. The first field is reserved for signal names or continuation characters. The remaining fields are used for *nodes* in a connection. All the nodes on one line are connected together to form a *connection* or *net*.

#### Typing in the wiring list

The cursor is blinking in the first node position. Type in **C3.2** (for the worked example, or type in a node required for your own design) then press <tab> to move to the next field. Do not press <enter> until the complete connection has been typed in.

Type in **BR1.4** (or a node for your own design) then press <tab> to move to the next field. Continue to type in the net as shown in Figure 8. 6 (or the net for your own design), pressing <tab> between nodes. Press <enter> when the net is complete, i.e. after *C1.2* has been typed.

Because IC was the first entry in the part code table, whenever a node is preceded by IC, you do not have to type in the part code. For instance, when you have to type in *IC1.1*, simply type *1.1*, the *IC* is added automatically. (Do not be tempted to adjust the order of the part code table to take advantage of this facility once the parts list has been started or you will alter the part prefixes in the parts/wiring lists as well.)


When entering signal names, notice that the power names are highlighted to differentiate them from signal connections. If they do not highlight, then they have not been defined as a power rail. Return to the chapter *Checking the board parameters* for details.

If a mistake is made, the entry can be removed by selecting the node, then pressing the *Delete* key. Alternatively the node can be selected and modified. Ranger will not allow you to make invalid entries - like using the same pin twice or referring to a part that's not in the parts list.

Continue typing the remaining connections as shown in Figure 8. 6.

Sig Name	:	:	:	:	:	:
	C3.2	BR1.4	R2.2	PL1.7	C1.2	
	C4.2	R3.1				
	C2.1	C4.1	R1.1	C1.1	PL1.3	
	C2.2	R1.2	R2.1	C3.1		
	C5.2	BR1.3	D1.2	D2.2	IC1.1	
	TR1.2	D2.1				
	R4.2	TR1.3				
	TP1.1	IC5.9				
	IC5.8	TP2.1				
	TP3.1	IC5.7				
	IC5.6	TP4.1				
	TP5.1	IC5.5				
	TP6.1	IC5.4				
	TP7.1	IC5.3				
	IC5.2	TP8.1				
	IC3.12	IC5.11	IC3.1			
	IC5.18	IC4.11				
	IC4.8	IC5.17				
	IC5.16	IC4.9				
	IC4.12	IC5.15				
	IC5.14	IC4.14	IC3.11			
	IC5.13	IC3.8				
	IC3.9	IC5.12				
	IC3.14	IC2.8				
	IC2.6	IC2.13	IC2.12	IC2.9	IC2.10	
	R5.1	IC2.4	IC2.5			
	C7.1	R5.2				
XYZ	BR1.2	PL1.5	R3.2			
ABC	PL1.10	IC2.2	IC2.1			
V+	PL1.8	C5.1	BR1.1	D1.1	R4.1	TR1.1
Contd->	IC1.2	C6.1				
GND	IC2.7	IC3.10	IC4.10	IC5.10	PL1.9	IC1.3
Contd->	C6.2	IC5.1	IC5.19	IC3.2	IC3.3	IC4.2
Contd->	IC4.3	C7.2				
VCC	IC2.14	IC3.5	IC4.5	IC5.20	PL1.1	

Figure 8. 6

Select **File** ⇒ **Exit Editor** once all the connections have been typed in to leave the editor. Select  (**File** ⇒ **Save**) to save the design. Consider taking a back-up copy of the design now - do you really want to type the parts and wiring list again if you accidentally delete it?!

### Chapter 9

## CREATING COMPONENT OUTLINES

When a new job is created, it has the entire contents of the master outline library copied into it to form its own outline library. Changes to the job library do not affect other jobs or the master library.

The component outline defines the component's pad positions, the pad codes used, and its identification silk screen mask.

The supplied outline library has been defined as viewed from the top of the board. i.e. outlines are defined as though you are looking down on them. Additional outlines should follow this convention - if you don't, you may end up with a board where the parts cannot be inserted correctly.

As a rule of thumb, when components are drawn horizontally always put the pad of pin 1 on the left, and when drawn vertically put it at the top. Note: The pad numbers relate to the pin numbers on the schematic and the physical devices, so they should always be placed in the correct relative position to one another.

Tip: add the pads first, then add the silk-screen around the pads - not the other way round.


Try not to place silk screen lines or text over the centres of pads because when the board is produced, the silk screen ink will go down the drilled hole.

If in doubt always make the silk-screen outline a little bigger rather than smaller than the dimensions stated, this will always avoid components interfering with each other.

The names used for component outlines in the supplied library try to convey the function of the component. Examples of how the names apply are given below.

<b>CAPA12E</b>	This is a <u>C</u> APacitor with <u>A</u> xial leads with pads <u>12</u> x 0.05" apart (0.6" apart) and it is an <u>E</u> lectrolytic.
<b>CAPR2D4E</b>	This is a <u>R</u> adial <u>C</u> APacitor with pads on a <u>2</u> x 0.050" pitch (0.1" apart), the <u>D</u> iameter of the can is <u>4</u> x 0.050" (0.2") and it's <u>E</u> lectrolytic.
<b>CAPR4W2</b>	This is a <u>R</u> adial <u>C</u> APacitor with pads on a <u>4</u> x 0.050" pitch (0.2" apart), the outline is rectangular, the <u>W</u> idth being <u>2</u> x 0.050" (0.1").
<b>PTO18</b>	This is a <u>T</u> O18 can, called up by a <u>P</u> NP transistor on the circuit.
<b>NTO18</b>	This is a <u>T</u> O18 can called up by a <u>N</u> PN transistor on the circuit.
<b>D15WYSK</b>	This is a <u>15</u> <u>W</u> aY <u>D</u> -type <u>S</u> ocKet.
<b>DIL14</b>	This is a <u>D</u> ual- <u>I</u> n- <u>L</u> ine package with <u>14</u> pins.


Select **Libraries ⇒ Component Outlines** to load the outline library editor. Existing outlines can be viewed/modified by

selecting  (**File ⇒ Open Outline**), then selecting the outline to be opened. The outline is displayed on the screen and could be modified. Select **File ⇒ Close Outline** to close the outline (this is done automatically when another outline is opened or created). We are about to create a new outline.

### Example 1

The example component will be a crystal in a metal can, the leads of which are 0.5" apart and 0.026" diameter. The can base is 0.7" x 0.4" overall, but the ends are semi-circular. Pad sizes will be 0.060" diameter with a 0.030" hole, which in the pad sizes table is size code 4. It will have a name **XTR10** where *XT* is the reference for crystal *R* for Radial and *10* for 10 pitches of 0.05".

(To check the pad size table, you will need to close the library, then select **Edit ⇒ Sizes Table**.)

Select  (**File ⇒ New Outline**). A dialogue window appears requesting a name for the outline. Type in **XTR10** followed by <enter>.

The outline is created and the graphical editor appears. The grid is visible, its pitch is displayed in the information bar. A dialogue bar appears down the left side of the screen, containing three sections - *Pads*, *Outlines* and *Text*.

A blue cross is visible in the centre of the screen. This is the datum point of the outline and becomes the point by which the component will later be 'picked up' for moving and rotating. It is usual to build the component symmetrically about this cross.

Once the outline has been defined, the datum can be moved if necessary.

### Adding the pins

The pins will be added according to the settings in the *Pads* section in the dialogue bar on the left hand side of the screen. The example shown here in Figure 9. 1 indicates that pads will be added to layer 0, at size code 4 and they will be round.

To change the layer, select the spin controls (the up or down arrows alongside the layer number) to increase or decrease the number. Only layers 0, 1 & 2 are available.



Figure 9. 1

With **layer 0** selected - the pads will appear on all layers in the artwork with a drilled hole. As required by conventional components.

With **layer 1** selected - the pads will appear on layer 1 only of the artwork (top/component side) without a hole, as required by SMD's.


With **layer 2** selected - the pads will appear on layer 2 only of the artwork (bottom/solder side) without a hole, as required by SMD's.

The pad Size can be changed in the same way, size codes 0 to 15 are available. These size code numbers correspond to the sizes table we looked at in the chapter titled *Checking the Board Parameters*.


Remember, round code 0 pads will be used for vias, so do not use this size code for component outline pads.

There are four *pad styles* to choose from, *Round*, *Square*, *Round finger* or *Square finger*. If one of the finger pads is selected, its orientation is defined by the *Finger pad orientation* setting.

For this exercise they should be set to the values shown in Figure 9. 1.

Move the cursor to the left of the blue cross. Zoom in  (<F1>, **View ⇒ Zoom in**), so that you can see easily. Notice that the grid size changes from 0.050" to 0.020" as you zoom in. To make life easier, choose a fixed 0.1" grid at all zoom levels (**Grid ⇒ 0.1" fixed**).


Select **View ⇒ XY Readout** to display the cursor position in the information bar (if it's not already), it is given with respect to the datum's position.

Select  (**Pins ⇒ Add**). Pins are always added in numerical order so pin **1** will be added first, followed by pin **2**, pin **3**, etc. The status bar indicates what to do and which pin is being added.

Locate the cursor 0.25" to the left of the datum, using the read-out or counting 2.5 grid spaces, then click the left hand mouse key to add the pad. Now move the cursor 0.25" to the right of the datum and click the left hand mouse key to add pin 2. The second pad is added. These two pads have been added at the size specified in the dialogue bar.

## Modifying the pins/identifying pin numbers

If for some reason the pad size or shape/orientation was incorrect, the correct values should be set in the *Pads* section of the dialogue bar, then the **Pins ⇒ Change Style/size** command selected. Select the pad that has to be changed to implement the change.

If the pad is in the incorrect position, select  (**Pins ⇒ Move**). Select the pad with the left hand mouse button, it highlights and its pin number is displayed in the status bar. Select its new position with the left hand key to move the pad. To cancel the move after selecting the pad, click the right hand mouse key.

Selecting a pad to move it, then cancelling the move is a way of finding out the pin numbers of pads.

When pads are deleted (**Pins ⇒ Delete**) it is always the last pin added that is removed, irrespective of the pad being selected. It is not possible to have an outline with missing pads. (It is possible to remove pads from individual outlines once they have been placed on the artwork.)

## Adding the silk-screen outline

Silk screen lines are added at the line size specified in the *Outline* section of the dialogue bar (Figure 9. 2). Ensure the line width is set to the size number required for silk-screen lines, for this example we will use code 3.

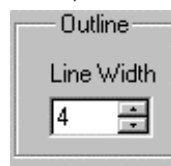


Figure 9. 2

Remember - codes 4 and 11 are used for signal and power tracks, so it is wise not to use those codes for the silk-screen lines. If you do for instance use size code 4 for the silkscreen lines, then later in the design decide to increase the size of the size code 4 signal tracks, then you will affect the silk-screen lines drawn with that code as well.

Once the line size has been set, add the component's silk-screen outline that is shown in Figure 9. 3, using the icons or commands in the **Outline** menu. These commands work in the same way as the commands you used to define the board profile.

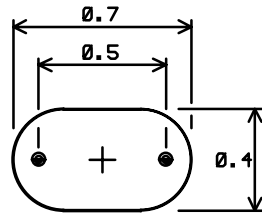




Figure 9.3

(Either draw a rectangle 0.3" wide by 0.4" high using  (**Outline ⇒ Add Line**), then use **Outline ⇒ Adjust Arc** to adjust the arcs on the ends of the rectangle, or use  (**Outline ⇒ Add Arc**) to define the outline in one go.)

If the datum is not in the centre of the outline, use **Edit ⇒ Set Datum** to re-position it.

When the component outline has been completed select the **File ⇒ Close Outline**. You are now ready to create the second example.

Your outlines are not saved until you save the design.

### Example 2

We will use this example to create a surface mounted as shown in Figure 9.4. The pads are on a pitch of 0.050" x 0.220".

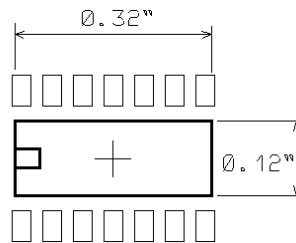


Figure 9.4


Gold plated edge connectors that are used on PC plug-in cards can be regarded as SMD components as the pads exist on one layer only and they are not drilled.

When pads are added to SMD outlines, the pads should be added to layer 1, the top component side of the board. Do not add the pads to layer 0 or those pads will be drilled and the pad will appear on all copper layers.

Ranger does not output drill information for pads on layer 1 (or 2), even though a drill diameter may be specified in the pad size table.

In the supplied library, layer 1 is the top, component side and layer 2 the solder side.

Once created, the parts can be flipped from one side of the board to the other, so it is only necessary to create one outline that can be used on both sides of the board.

Once the component outline editor has been loaded, (**Libraries ⇒ Component Outlines**), select  (**File ⇒ New Outline**) to create the outline. Enter a name of **MYSO14**.

### Defining the grid


Many outlines have pins on a non-standard grid pitch. In this example the pads are on a 0.050" x 0.22" pitch. Select **Grid ⇒ User Defined**, enter the pitches, X=0.05" and Y=0.22" followed by **OK**. Zoom in if necessary. The pads can now be added easily and accurately using the grid points.

Metric values can be entered after switching to Metric mode (**Edit ⇒ Units ⇒ Metric Units**). Bear in mind that the resolution of Ranger2 is 0.001", so metric values will be rounded to the nearest 0.001". (This does not lead to cumulative errors when adding a row of pads. Each pad will be out of position by no more than 0.0005".)

To assist you, ensure the co-ordinate readout is visible (**View ⇒ XY Readout**).

### Adding the pins

Set the **Pads** dialogue bar to **Layer 1**, **Size 4**, **Finger Square** and **Vertical** (set to 0.050" x 0.030" in the sizes table).

Select  (**Pins ⇒ Add**), then add the pins. Making sure that pin1 is positioned at the bottom left of the outline shown in Figure 9.4 and the remaining pins added in an anti-clockwise direction to ensure that connectivity will be correct. Use the grid points as pad positions. (The dimensions given in Figure 9.4 are for the centre lines of the silk-screen lines.)

Notice the pads are now blue to indicate they are on a different layer to those on layer 0 (white).

(Typically in the outline editor, pads on layer 0 are white, those on layer 1 are blue and those on layer 2 are red. Be aware that the colours can be changed, so do not assume that blue pads are always on layer 1.)

## Moving the datum

The datum cross should be placed in the centre of the pads. Because Ranger snaps to the grid and half-grid, the grid does not need to be changed. Select **Edit ⇒ Set Datum**. Point at the middle of the pins and click the left button. The blue cross moves to the middle of the pads.

## Adding the silk-screen outline

The grid will need to be changed using the **Grid** commands to define the outline required. For this exercise, the **Inch Autopitch** grid should be adequate (zoom in so that a 0.020" grid is visible).

Use the **Outline** icons/commands to define the silk screen outline. (The co-ordinate read-out of the lower left corner of the silk screen will be x=-0.160", y=-0.060" and top right will be x=0.160", y=0.060".)

When the component outline has been completed select **File ⇒ Close Outline**

Select **File ⇒ Close Library** to leave the library. Have you saved your work?

## Adding text to outlines

Any text that is added to an outline will appear on every occurrence of that outline. Typically things like pin numbers at either end of a long row of connector pads are added.

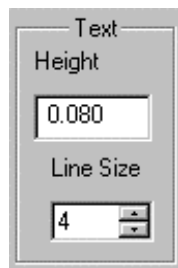


Figure 9. 5

The **Text** section in the dialogue bar (Figure 9. 5) controls the height, and width of line used to draw the characters.

Experiment adding the same text string (**Text ⇒ Add**) using line size 2 and size 15 to see the difference. (Then delete the strings unless you want to keep them - text strings have to be selected by their lower left corner.)

The silk-screen labels (or reference designators IC1, IC2, IC3, etc.) are added automatically to the parts once they have been placed on the artwork, centrally over the datum. (Do not move the datum to what you think will be a better position for the label - it won't be.)

The datum should be in the middle of the part to make it easy to select it for moving/rotating/flipping. The labels can be moved into suitable locations later.

## Adding components to the master outline library

Once components have been completed and they are known to be correct (i.e. used on a manufactured board) they can be added to the master outline library so that everyone has access to them.

To add the outline to the master outline library: From the job outline library, select **File ⇒ Copy ⇒ Outlines to Master Library**. Select the part to be copied, followed by **Copy**. A message appears in the status bar to indicate the part has been transferred. Select **Close** once all the required outlines have been transferred.

## Additional Information and Tips

The following information is given for future reference - you will gain a better understanding if you return here after you have completed the training exercise.

### Pad/Line Sizes

When a pad or line is added to an outline, only its size code is stored in the outline, not the actual size assigned to the size code. This means that one outline, when used on different jobs could have differently sized pads, depending on the sizes assigned in the size table for that particular job.

For this reason, take care when adjusting the pad, track and drill sizes table, so that existing outlines are not affected by the change.

### Modifying outlines on an artwork that has been started

Once the artwork has been started (digitised) some changes to outlines in the library will not be updated automatically on the artwork as follows:

#### Replacing pads

If pad codes or pad shapes in an outline are changed after it has been placed on the board and the pads digitised in, the artwork is NOT updated (for example you have changed a code 4 pad to a code 5 pad, or a round pad to a square pad).

You will need to delete the changed pads from the artwork (use **Amend ⇒ Delete Pad**), then digitise the pads again. The pad digitiser only adds pads that are not on the artwork - if a pad is already in the correct position then it is not digitised again even if it is the wrong code or shape.

#### Moving pads

If pads in an outline are moved after it has been placed on the board and the pads digitised in, the artwork is NOT updated.

## CREATING COMPONENT OUTLINES

---

You will need to delete the moved pads from the artwork (use **Amend ⇒ Delete Pad**), then digitise the pads again. Tracks connected to the old pad positions will need to be re-connected to the centres of the new pad positions (use **Amend ⇒ Move Point**).

### Removing pads

If pads in an outline are removed after it has been placed on the board and the pads digitised in, the artwork is NOT updated.

You will need to delete the removed pads from the artwork (use **Amend ⇒ Delete Pad**).

### Datum position

Don't move the datum once the outline has been placed on the board - or be prepared for the parts to "move". The position of each outline is held in the parts list, according to the X & Y co-ordinate of the datum. If you move the datum within the outline, the co-ordinates of the datum position in the parts list remain the same. (If the datum is moved after the pads have been digitised, then you will need to delete the pads and re-digitise.)

### Changes to the sizes table

If the pad sizes table is changed the changes are updated automatically. (For example changing a round code 2 pad to 0.100").

### Changes to the silk-screen outlines/text


Once the silk-screen has been generated on the artwork, any changes you make to the outline will not be updated. There are various approaches that can be used to update the artwork. The easiest, provided you have placed the silk-screen outlines on one layer and the labels on another, will be to simply re-generate the silk-screen outlines on the board.



## Chapter 10


### PLACING THE PARTS ON THE BOARD

Once the parts and wiring list, the required component outlines and the board profile have been defined, you are ready to start placing the parts on the board.

Select  (**Artwork ⇒ Place Parts**). The editor loads and the board profile can be seen. If component outlines have been called up by the parts list that do not exist in the job's component outline library, the editor will not load and a list of the undefined outlines is given. These outlines will need to be created in the job's outline library before part placement can commence.

#### Viewing the component pads and signal connections


In preparation for part placement select **View ⇒ Part Pads**, **View ⇒ Drill Holes** and **View ⇒ XY Readout**, in order to see those items as parts appear on the board. (Notice that a tick mark appears alongside the items that are set to visible. Selecting those items again will remove the tick mark and set them to invisible.) You will also want to see the signal

connections between parts so select  (**Nets ⇒ Show Signals**).

Switch the grid on to assist in aligning parts. Try to position parts on a coarse grid, so that the pads lie on an increment of the routing grid to be used. In this example, a 0.050" routing grid will be used, so try placing the parts with a 0.1" fixed grid set, the half grid will be 0.050" (**Grid ⇒ 0.1" (fixed)**).

#### Placing parts on the board

##### Placing parts

Select  (**Parts ⇒ Place**). As you move the cursor you will see a part attached to it. Its reference name is displayed in the toolbar (IC1 in Figure 10. 1). The parts appear in the order they are listed in the parts list. The part can be changed by selecting the single or double arrows alongside the reference name.

The single left and right pointing arrows travel up and down through the parts in the parts list. The double arrows can be used to travel up and down through groups of parts (by prefix) in the parts list.

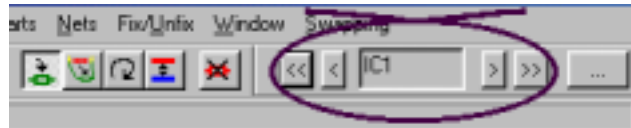


Figure 10. 1


The selected part appears on the end of the cursor as you move it. The part can be rotated by pressing the <R> key on the keyboard whilst it is being moved.

The component can be released in its current position and orientation by clicking the left hand mouse key. The next component in the parts list appears on the end of the cursor ready to be positioned. It appears at the same orientation as the previous component was released. Connections between it and the part already placed (if there are any), stretch as the part is moved. The shortest connection path is always shown. At any time you can halt the placement by clicking the right hand mouse key.

Continue placing components in the order required, until they have all been added. The tool bar will indicate **All Placed** once all the parts are placed.

An example layout is shown in Figure 10. 2. Spend time achieving a good part placement. On real designs this will ensure the design is easier, quicker and possible to route.

##### Moving parts

Use the  (**Parts ⇒ Move**) command to move parts once they have been placed. Parts should be selected by their datum point, which is indicated by the label.

Notice that as parts are moved, for instance R1, their connections *reconnect* to the nearest pin on the same net. This assists you when trying to find the optimum position for the part. The status bar also gives a before and after net length when the part is selected and released. This simply tells you the total length of all the nets attached to the pins - of course you could place all the parts on top of one another to reduce this number, but the board would prove impossible to route and make!

To move a part to a known co-ordinate use the **Parts ⇒ Key Move** command.

## PLACING THE PARTS ON THE BOARD

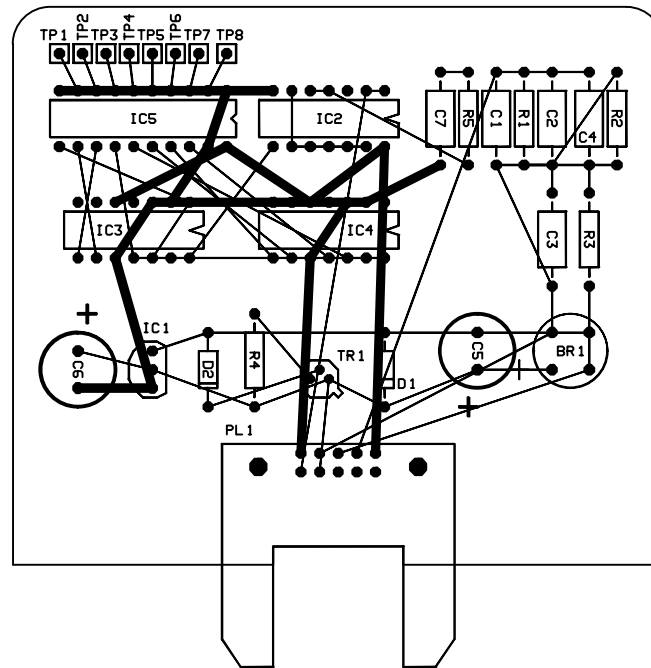



Figure 10. 2

The **Parts ⇒ Align X** command is used to align parts by their datum points in a vertical column. (Align Y works horizontally.) Select the alignment position with the left button and parts to be aligned with the right button.

### Flipping parts


The  (**Parts ⇒ Flip**) command is used to flip parts (usually SMD's) to the other side of the board, mirroring them as they are flipped so that the pads are still in the correct positions. The outlines are shown as dashed lines to indicate they are flipped.

### Removing parts

Parts that are removed are not removed from the design. They are simply added to the list of parts waiting to be placed.

, **Parts ⇒ Remove.**)

### Viewing and reconnecting the power connections

The power connections may be displayed by selecting  (**Nets ⇒ Show Power**). The power connections do not automatically reconnect to the nearest pin when parts are moved, so it is necessary to force a reconnect as required. Select the **Nets ⇒ Power Reconnect** command after moving some parts that are connected to the power rails. This command assesses the power pins that are joined together and rejoins them using the shortest point to point route.

### Other commands in the part placement editor

The **Swapping** menu contains commands to allow gate and pin swapping (provided a circuit device library exists containing the appropriate devices). All the gate/pin swapping changes are back-annotated to the parts/wiring list and schematic.

#### Gate swapping

First turn the pads off to make the gates more apparent (**View ⇒ Part Pads**). Select **Swapping ⇒ Gate Swap**. Select a part. A message appears in the status bar if the part is not enabled for swapping. In this example only IC2 the 74LS13 contains swappable gates. As the part is selected the gates are highlighted and can be swapped one for another. If more 74LS13's were on the board, all the equivalent gates would highlight and could be selected.

#### Pin swapping

Select **Swapping ⇒ Pin Swap**. Select a part. A message appears if the part is not enabled for swapping. In this example IC2 plus some of the discrete parts contain swappable pins. As the part is selected, the pins in the same equivalent group are highlighted and can be swapped one for another.

Gate and pin swapping will affect the parts/wiring list and the circuit schematic because the pin numbers on packages change. The electrical connectivity remains the same though. The parts/wiring list and schematic are automatically updated. This is called automatic back annotation.

When a satisfactory layout has been achieved, select **File ⇒ Close Layout** to return to the job design menu.

Note: all the parts should be placed before attempting to route the board. It is not practical to place some parts, route them, then place some more, because the routed tracks cannot be seen from the placement editor. Once the artwork has been digitised, always move parts from within the artwork editor.

Now is a good time to take a back-up copy of your design. If the design proves difficult to route, you can re-arrange the part placement then try again, knowing you still have the original to fall back on if it turns out to be better after all.

### Part fixing and unfixing

The **Fix/Unfix** menu can be used to fix parts so that they're not accidentally moved. Useful for things like connectors, tooling holes, etc. that have to be in fixed positions. The same menu is used for unfixing parts so that they can be moved or deleted.

### Additional Information and Tips

The following information is given for future reference.

Once the pads and tracks have been digitised on the board, never move existing parts using this editor. You will find that you cannot see the existing tracking and when you return to the artwork editor, the pads and tracks of the moved parts will not have moved. Instead, always move the parts from the artwork editor (**Parts** ⇌ **Move**).

## Chapter 11

### LINE/PAD DIGITISING

During part placement a graphical representation of the components and their interconnections could be seen, but neither actually exist as copper on the board. For instance the connections were shown as lines without thickness.

Ranger has to convert this graphical information into copper pads and tracks on the board. It has to know what size codes to use for the interconnections, and whether the board will be auto-routed or manually routed, or a combination of both. This information is supplied by the *digitiser*.

Digitising converts the pad information held in the part placement file into copper pads with holes in them (where required) and it converts the interconnections into auto-router information or copper tracks, using supplied track codes.

The actual sizes of the pads, holes and tracks used, will be taken from the **Sizes table** which relates the size codes to physical sizes.

To start digitising, with the job open select  (**Artwork ⇒ Pads & Tracks Digitise**) from the job design menu. A window similar to the one shown in Figure 11. 1 appears.

#### Digitising pads

To convert or *digitise* the pads, from the *Pad Insertion* section of the window, select **Digitise Partpads**. The pads are digitised and the counter bar (alongside *Ready* at the bottom of the window) fills up block by block until pad digitising is complete. There is no need to *Digitise Partpads* again, so it is no longer selectable (until next time).

If you were to select **OK** from the digitising window now and have a look in the artwork editor (select **Artwork ⇒ Editor**) you would see the board profile and the pads, but no connections or tracks.

If the pads are digitised again, the existing pads are not overwritten - this has implications when modifying the layout - see the tips at the end of this chapter.

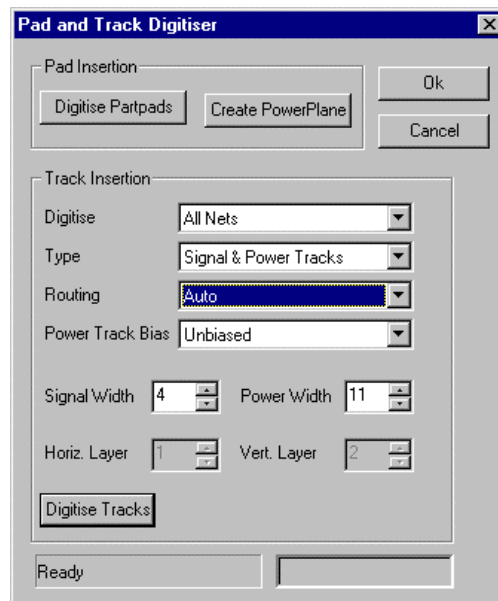


Figure 11. 1

#### Digitising tracks

In this design example we will digitise and auto-route all the power connections on the board. The results can be manually adjusted if required before routing the signal connections. Any connections that the auto-router cannot complete can be routed manually.

(If you don't want to auto-route the design, continue as described here, then in the next chapter skip the paragraphs headed *Setting up the auto-router* and *Auto-routing the power supplies*.)

This approach is used because the power connections are usually routed with thicker tracks and the routes taken are often critical. They would be more difficult to route after the signal connections have been completed. If there are other critical connections, such as clock lines, etc. they could also be routed before the signal connections.

To allow for these different routing strategies the digitiser can be set to digitise different combinations of interconnections, so we first of all need to set up the parameters that the digitiser will use, which are shown in Figure 11. 2. This will digitise all the power connections for auto-routing, without biasing them in a particular direction, i.e. the shortest point to point path will be used.

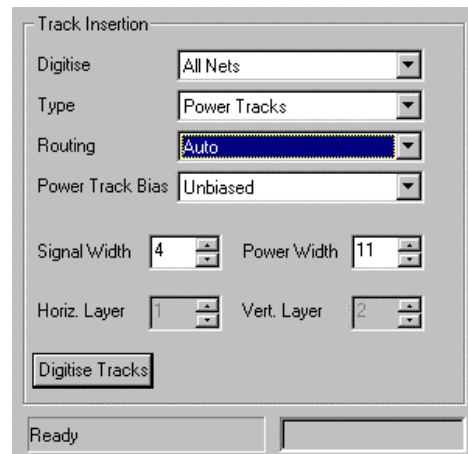


Figure 11. 2

If the parameters are not set as shown in Figure 11. 2, they need to be changed. To change any of the parameters, select the down-pointing arrow alongside it, then select the required setting from the list that appears.

The *Signal width* and *Power width* parameters control which track code is used for that particular type of connection. They can be changed as required, codes 4 & 11 are used by default.

Once the digitiser has been set up to match the values shown above, select **Digitise Tracks**. The counter bar fills up block by block until track digitising is complete.

If a mistake was made during the digitising process, select **Cancel** and start again.

Do not digitise the same connections more than once as the connections will be duplicated on the artwork - this is a common mistake made by new users - if two sets of tracks appear on your artwork then this is what you have done.

When both tracks and pads have been digitised, select **OK** to return to the job menu. You are now ready to start routing the board.

Notes: even if you intend to manually route a board it is better to digitise for auto-routing as it is easier to see what has and hasn't been routed.

### Additional Information and Tips

Once digitising has been performed, do not move existing parts from within the part placement editor because the digitised pads and tracks will not move. Always use the artwork editor - the positional changes are back-annotated to the part placement editor.

The following information is given for future reference - you will gain a better understanding if you return here after you have completed the training exercise.

#### Digitising pads

Before adding the pads, the digitiser checks the artwork to find out if a pad already exists in the location where it wants to add the pad. If there is, then the pad is not added.

This means that once the pads have been digitised, if you alter the pad codes or shapes in a component outline, then the digitiser will not update the artwork accordingly. You will need to open the artwork, delete the pads that have been altered (**Amend** ⇒ **Delete Pad**), then digitise the pads again.

#### Digitising tracks

Unlike the pad digitiser, the track digitiser does not check the artwork before digitising as defined in the window. This means that if you select **Digitise Tracks** twice, then you will get two sets of connections/tracks.

Ensure the track digitiser has been set up as you require it, before digitising.

The *digitise remaining* option allows you to digitise in connections that have not been routed successfully. (This means that any connections that are only partially complete will also get digitised.) The digitise remaining option uses the information supplied by the artwork checking routines to determine what hasn't been routed and is therefore remaining. So before choosing the *Digitise remaining* option, you will need to run the artwork checks.

#### Digitising power planes

The power plane digitiser always deletes everything from the selected power rail layer before creating the plane. This is to ensure the plane will be correct.

## LINE/PAD DIGITISING

---



If you want to add an isolation barrier around the edge of the board (to stop the planes shorting into metal casings, etc.), or to add text (in relief) to the plane, then it's a good idea to add this to another unused layer. The power plane layer and this other layer can then be output together to produce the results you require. This will save you having to re-enter the tracks/text if you ever create the planes again.

Anything you add to a power plane layer (or output with the power plane layer) will be "not copper", i.e. it will be in relief in the plane.

In order to give you flexibility, it is possible to add heat-relief or anti-pads to the plane using the **Enter ⇒ Add Pad** command in the artwork editor. They can also be deleted **Amend ⇒ Delete point**.

## Chapter 12

# ROUTING THE BOARD

After digitising, you are ready to start routing the board. If you are going to auto-route the design, select  (**Artwork ⇒ 2 Layer Auto-router**), if you intend manually routing the design select  (**Artwork ⇒ Editor**), the appropriate editor loads. The artwork should now contain the board profile, copper pads for all the parts, and the point-to-point inter-connections for the power rails, which are ready to be routed. The connections are not on a layer yet, as the auto-router (or you) will decide how to route the connections and on which layer, according to the rules you assign.

Select the **View** menu and ensure a tick mark is alongside **Lines at Width** to allow the true sizes of pads and tracks to be seen.

If you intend manually routing the design, move onto the heading Manual routing now - bear in mind you will already be in the artwork editor and all the power tracks will need to be completed.

### Auto-routing

#### Setting up the auto-router

Select **Autoroute ⇒ Setup** to display the auto-router setup window, similar to the one in Figure 12. 1.

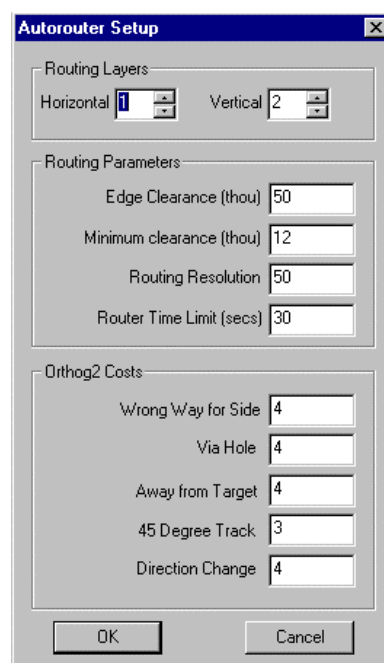


Figure 12. 1

The **Routing Layers** tell the auto-router in which direction to lay tracks on the layers selected. To change the layers, use the spin controls alongside. This example will create a double-sided design, placing horizontal tracks on layer 1, and vertical tracks on layer 2.

To route single-sided boards, both directions should be set to layer 1 (top), or both set to layer 2 (bottom).

The **Routing Parameters** control the space between the edge of the board and the routed tracks (*Edge clearance*), the space between pads and the routed tracks (*Minimum clearance*), the grid that the router is allowed to place tracks and vias on (*Routing resolution*), and the length of time allowed per connection - 30 seconds is ample for all designs (*Router time limit*).

You may need to change the two clearance values if you are working on your own design, they are fine for the training example.

The routing algorithms generally give the best results if the routing resolution is set to 50 (thou). However, you could always copy the design and try a finer resolution. Depending on the design 25, 20 or 10 are values you could try.

The **Orthog2 Costs** control the way in which the router operates. They can be varied between 1 and 9 as required, using the following notes for guidance.

## ROUTING THE BOARD

Ranger finds as many possible paths between inter-connected pads as it can. It chooses the path that gets the least number of points, according to the costs assigned in this table.

In this example, each time the router places a track that goes in the “wrong direction” as defined by the *Routing layers*, 4 points are added to the cost of the route, each time a via is used, 4 points are added to the cost of the route, etc. Ranger adds the costs of all the items that make up each path and it chooses the path with the least points.

The settings in Figure 12. 1 are suitable for double-sided boards. To route single-sided boards, set all the Orthog2 costs to 1, except for via holes, which should be set to 9. (This ensures that every time a via is used, the path becomes very expensive so it is less likely to be chosen.)

The auto-routing algorithms have now been set up and are ready for use, select **OK** to continue.

### Auto-routing the power supplies

There are four routing algorithms available, they appear in the toolbar, as shown in Figure 12. 2, they also appear in the *Autoroute* menu.



Figure 12. 2

Each one can be selected and de-selected by selecting the box alongside it. The tick indicates the algorithm(s) that will be used.

Each algorithm is geared up to route a particular type of connection. For instance the power algorithm only attempts power connections (as defined by the *Edit ⇒ Power Names Table*), whilst the other algorithms only attempt signal type connections.

Because we only have the power rails to auto-route, just select the box alongside **Power** to tick it. The power router strategy has now been selected and auto-routing can be started by selecting **Start routing**.


As the router proceeds, a progress report is given in the status bar showing the number of tracks routed and those refused for the current pass. Each successfully routed track appears on the screen. When the auto-router has attempted every power connection on the board, it re-draws the board and a report appears in the status bar indicating the number of connections left.

If any of the power connections has not been successfully routed, they are still displayed as point-to-point connections. These can be completed using manual routing techniques. If all the power connections have been routed, go to the next heading **Routing the signal connections**.

If any power connections have been left by the auto-router, they should be completed before attempting the signal connections - they will be more difficult to route once the signal connections have been routed.

### Manual routing


Manual routing is performed from the artwork editor. If the auto-router editor is open, select **File ⇒ Close Autorouter**,

followed by  (**Artwork ⇒ Editor**).

Switch the grid on to assist in routing the tracks and select the *Inch (Autopitch)* grid so that the grid gets finer as you zoom in.

### Converting connections for manual routing

The connections that have been digitised for auto-routing that haven't been auto-routed have to be first converted for

manual routing. Select  (**Mroute ⇒ Convert to ManRoute**). Locate the cursor over an unrouted connection, zoom in if necessary <F1>, then click the left hand mouse key. The connection changes colour and appears as a track on one of two specified layers. (If the connection does not change colour, zoom in further, if you still can't select it, select it on one end.)

The *Routing* section of the right hand dialogue bar, as shown in Figure 12. 3, controls which layers the converted tracks appear on.



Figure 12. 3



By default, layer 1 is the Primary routing layer and layer 2 the secondary routing layer. These layers can be changed by using the spin controls alongside them. Thus any layers from 0 to 15 can be used to make a multi-layer board.

Note: layers 1 and 2 are always the top and bottom sides of the board respectively. (SMD's get flipped between layers 1 and 2, so it is essential that these two layers are used as the outer layers of a board. In a multi-layer design, any additional copper layers (for instance layers 3 to 15) would be used as inner layers in the board.)

Any connection that is horizontal or less than 45 degrees from the horizontal is placed on the primary layer when it is converted to a track. Any connection that is vertical or more than 45 degrees from the horizontal is placed on the secondary layer. For single-sided routing, the Primary and Secondary layers can both be set to the same layer.

Once the connection has been converted to a track, it is ready for manual routing. (Tracks cannot be converted back to connections for auto-routing.) It is suggested you convert a few connections at a time, so that you can see what has and hasn't been completed - the display will be less cluttered.

## Manually routing/modifying tracks


The commands in the **Mroute** menu will only operate on tracks that are on either the Primary or Secondary layers, so when working on a multi-layer design, you may need to spin these layers first. For this exercise, they should be set to 1 and 2.

The *Mroute* commands should now be used to stretch the track so that it avoids obstacles (other pads and tracks).

Try to maintain the horizontal tracks on layer 1 and vertical tracks on layer 2 so that when the auto-router is used to route the signal connections it does not find too many obstacles in its way.

## Adding/Moving corners

The *Corner* command is used to add corners to tracks, or to move existing corners in tracks. The left and right buttons on the mouse control whether add or move is activated.

Select  (**Mroute** ⇌ **Corner**), this command will remain active until you select another one - refer to the status bar to find out which command is active and to remind you how to use it. Zoom in so that you can see clearly.


*Adding a corner* - point at the track (not a connection), then click the **right** hand mouse button. Move the cursor and the attached corner into the position required, then click the *right* button again to release it. (Clicking the left button cancels the new corner.)

*Moving an existing corner* - point at a corner in the track, then click the **left** hand mouse button. Move the cursor and the attached corner into the position required, then click the *left* button again to release it. (Clicking the right button cancels the move.)

When adding or moving corners try to avoid placing one corner directly on top of another, unless you want them to be joined together. (The only way to separate them is using the *Amend* ⇌ *Move point* command.)

## Swapping a track segment from one layer to another

Tracks are usually routed so that horizontal segments appear on one layer and vertical segments appear on the other layer. Track segments can be swapped from one side of the board to the other, vias being introduced where necessary, as follows.

Select  (**Mroute** ⇌ **Layer Swap**), then select the segment to be swapped. Providing the track is on either the primary or secondary layer, it will swap to the other layer. Multi-layer boards can be designed by changing the primary and secondary layers. (Layers 1 and 2 are always the outer layers of the board.)

Convert and manually route the remaining connections, using the commands just described. Don't forget to take regular saves of the design as you work.

## Changing the thickness of a track

The thickness of a track can be increased or decreased over a specific distance if required. This is typically used to reduce the size of a power track so that it will pass between two pads.

In the *Routing* section of the dialogue bar (shown previously in Figure 12. 3), set the track *Neck Size* to the code required for the necked section, i.e. 3, using the spin controls. (This code relates to the size table).

Now select **Mroute** ⇌ **Neck**. You are prompted for the length of the necked track in inches. If a value, i.e. **0.1** is entered, then just that length of track will be necked down, if <enter> is pressed without a value, then a complete selected segment is necked down.

Once <enter> has been pressed, point at the centre of the section of power track that you want necked, and click the left hand mouse button. The section is changed in size, either side of the cursor.

To "un-neck" a track, use *layer swap* twice on the segment, or *neck* the track back to its original size. (If layer swap is used, one end of the necked track must be at the original thickness, otherwise it doesn't know which size to restore.)

The **Amend** ⇌ **Reset Track Size** command can be used to change a length of track between vias or pads.

Once all the connections have been routed, leave the editor by selecting **File** ⇌ **Close artwork**.

## Important points/tips to bear in mind when manually routing

The *Mroute* menu should give you all the commands you require to manually route or modify tracks. This menu will not allow you to delete tracks or disconnect the ends of tracks from the centres of pads or vias. However there may be occasions when you need to route a connection in a different order to that found by the shortest path, or you may quickly

## ROUTING THE BOARD

want to re-insert a track that you accidentally deleted (the checking/digitising routines will flag missing connections and re-insert them if you weren't aware they had been deleted).

In these situations and others, the *Amend* and *Enter* menus are used. These menus allow you to do whatever you want, so should be used with some caution as you can wreck the artwork!

Provided you always ensure that the **ends** of tracks start and stop on the **centres** of pads or vias (even 0.001" off centre will not do), then connectivity will be maintained. If a track just touches a pad or via then Ranger does not regard this as a completed connection, so it will be reported as a connectivity error (or short circuit if they are not supposed to be touching at all).


The checking routines will indicate whether or not you have made any mistakes.

Please note: if curved tracks are used, the checking routines do not recognise them - the minimum clearance around them will not be checked and they will be reported as connectivity errors (i.e. unrouted).

### Routing the signal connections

When all the power routes have been successfully routed, the signal connections can be routed. They have to be digitised first, so leave the artwork editor (**File** ⇒ **Close Artwork**).

#### Digitising the signal connections

Assuming the job is open, select  (**Artwork** ⇒ **Pads & Tracks Digitise**). Set the track digitiser parameters as shown in Figure 12. 4 in order to digitise all the signals for auto-routing. (Even if you intend manually routing the connections, you'll probably find digitising for auto-routing is more convenient than digitising for manual routing. You could always copy the job first, then try both methods to see which you prefer.)

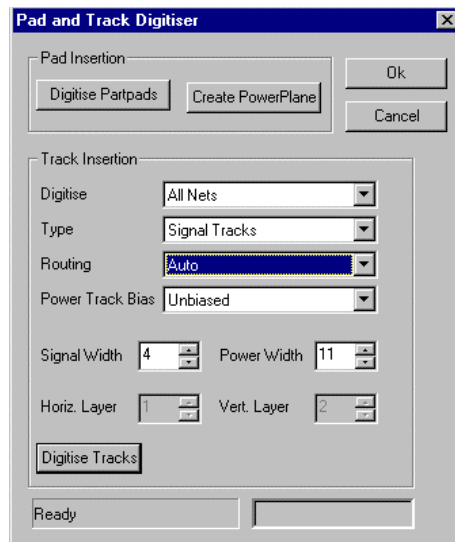




Figure 12. 4

Select **Digitise Tracks** and when complete select **OK**. If you intend manually routing the signal connections, open the artwork editor (, **Artwork** ⇒ **Editor**) then manually route the connections as described previously.

#### Auto-routing the signal connections

Select  (**Artwork** ⇒ **2 Layer Autorouter**). All the signal interconnections can now be seen in green (or whatever colour is chosen). The auto-router setup window will still be set as we left it. (Depending on the design, you might have to change the setup parameters to get the best results, but for this design they can be left as they are.)

Select the **Memory**, **Orthog1** and **Orthog2** algorithms as shown in Figure 12. 5. These algorithms only operate on signal type connections as opposed to power connections.

Select **Start routing** to start the auto-router.

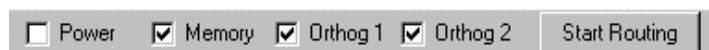



Figure 12. 5

Any connections that remain can be manually routed using the techniques already described for the remaining power connections. Once the artwork is complete, it is ready to be checked to ensure that it is the same as the parts/wiring list that was created from the circuit diagram or by typing.

Select **File** ⇒ **Close Autorouter** (or *Close Artwork*).

## Deleting the artwork and starting again

Sometimes you may not like the results that the auto-router gives. You might like to adjust the part placement, or in the case of the training exercise, simply have another go at manually routing the artwork. In this situation the existing artwork has to be deleted and the pads/tracks digitised again.

Get used to taking back-up copies of your design in order to pursue alternative design approaches. If the alternative method doesn't work, you can always return to the original design. Save your job now  (**File ⇒ Save**), then take a copy **File ⇒ Save As**.

### Deleting the artwork

Check the title bar to ensure the correct job is open.

Select **File ⇒ Delete ⇒ Artwork**. You will be asked to confirm this selection. Once confirmed, everything (pads, tracks, silk-screen, router data, etc.) from the artwork will be removed.

The part positional information is held in the parts list editor, so the parts will remain placed. For this reason never delete the parts/wiring list or all the part placement information will be lost - unless that is what you intend. When you modify a circuit diagram and convert it to a parts/wiring list, the existing parts/wiring list is always updated so should never be deleted.

To just remove the point to point connections that are ready for auto-routing, select **File ⇒ Delete ⇒ Router data**.

Select **Artwork ⇒ Pads and Tracks Digitise**, followed by **Digitise pads**. Ensure the digitiser parameters are set as you require them, then select **Digitise tracks**.

Remember, do not digitise tracks using the same parameters more than once, or the connections/tracks will be duplicated.

Even if you intend manually routing a design, it is better to digitise the connections for auto-routing, then convert them for manual routing when you are ready to route them. Using this method, it is easier to see what has and hasn't been routed.


## The Rip/retry Auto-router

This auto-router is supplied in addition to the standard 2-layer auto-router. It can route on up to 6 layers at one time and attempts to move unfixed tracks around in order to find space for connections that have yet to be routed. (Tracks can be fixed from within the *2 layer auto-router* editor (**Fix/Unfix** commands).

If you have access to the Specctra auto-router, it will give superior results, especially on boards containing surface mounted components or where finer routing grids are required.

### Using the rip/retry auto-router

Assuming any existing artwork has been deleted, digitise the pads. There is no requirement to digitise the connections.

Select  (**Artwork ⇒ Rip/retry Autorouter**).

Select **Setup router** from the toolbar. A window appears listing the routing strategies (or algorithms) available for power and signal tracks which can be enabled/disabled as required.

Each strategy has a setup associated with it. Under normal circumstances, only the *routing pitch* parameter should be altered, if required.

To autoroute all the connections, enable all the routing strategies.

To auto-route just the power connections, enable just the **Power pattern, Power cost, Power rip-retry** and **Interim optimise** strategies.

To auto-route just the signal connections, enable just the **Signal memory, Signal cost, Signal rip-retry** and **Interim optimise** strategies

The **Final optimise** should only be run when everything has been routed as it will reduce the number of vias in use, thus causing difficulties for subsequent routing passes.

Ensure the minimum clearance requirements are set as you require them, then select **OK** to continue.

Select **Setup Layers**. Enable or disable the layers that you want the auto-router to use and select the preferred direction for the enabled layers. Select **OK** to continue.

Select **Start Routing** to start the auto-router. As the router proceeds, a progress report is given in the status bar. Each successfully routed track appears on the screen.

When all the routing passes have been attempted or the routing is complete, a window appears allowing the routing results to be saved or rejected. Select the option required. Any unrouted connections will be visible as point-to-point connections.

Select **File ⇒ Close autorouter** to leave the rip-retry router, then continue as required.

The artwork checking routines **MUST** be run to ensure that all the connections have been completed successfully. Refer to the chapter titled **Completing the design** for information.

## The Specctra Auto-router Interface

The Specctra Auto-router is an optional extra that can be purchased to operate with Ranger2. The interface within Ranger has to be enabled as described in chapter 1 of this manual. Once enabled, designs can be submitted to the Specctra auto-router from Ranger. Refer to the on-line help within the Ranger software for full details on the interface.

## ROUTING THE BOARD

---

### Modifying a Specctra routed design

If you intend modifying the auto-routed design, valid netcode information should be created. This will allow all the Ranger commands such as track high-lighting, copper fill, etc. to operate correctly.

#### Constructing the netcode information

The artwork checking routines (**Artwork ⇒ Trace/Check Artwork**) must be run and any reported short circuit errors corrected, then the checks run again until no short circuits are present.

Then select **File ⇒ Maintenance ⇒ Reconstruct netcodes**. Select **Start**. The window will indicate when the process is complete. All the tracks will have been validated by Ranger and can now be modified as required.

### Routing a board with power planes

Before autorouting, ensure the required layers have been defined as power plane layers (**Edit ⇒ Plane Assignments**).

Auto-route the design using Specctra. When auto-routing is complete, return to the pad/track digitiser and create the power planes (select **Artwork ⇒ Pads and tracks digitise**, followed by **Create power plane**. Enable the planes required, followed by **Create**).

Check the artwork to ensure it is complete and correct.

If the design contains SMD's that should be connected to the plane, provided there are no short circuit errors, construct the artwork netcode information as described above, then create the power planes again. Re-check the artwork and correct if necessary.


## Chapter 13

### COMPLETING THE DESIGN

#### Design rule checking

When an artwork has been completed, it should be checked to ensure that no design rules have been violated during routing, and to ensure the board is complete connectivity wise. Do not attempt to manufacture the board until these checks have been performed and reported errors corrected. Proceed only when no errors are reported.

Please be aware that curved tracks are not checked so can cause clearance errors that are not flagged. The connection containing the curved track will be reported as a connectivity error as Ranger sees the curved section as an open circuit.

The checks available within Ranger are found as follows, with the job open select  (**Artwork ⇒ Trace/Check Artwork**). A window similar to the one in Figure 13. 1 appears.

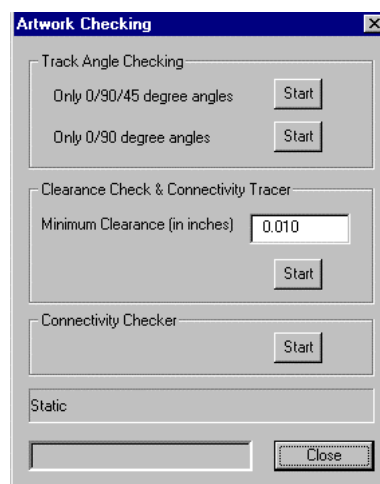


Figure 13. 1

#### Track Angle Checking

These checks are for your information only. It doesn't matter to Ranger2 what angle the tracks are at. Many designers like to see the tracks at 0, 45 or 90 degrees, or there may be a design requirement that specifies tracks must only be at 0 or 90 degrees.

If this check is performed, angle error markers are inserted against tracks that are at any other angle than those specified. To activate this check, select **Start** from alongside the *track angle checking* option required. A message within the window will indicate how many angle errors were found.

If no angles were found, run the *Clearance check and connectivity tracer*.

If angle errors were found, to view them select **Close**, then  (**Artwork ⇒ Editor**). Small green flags with the letter A (for angle) will be seen alongside the offending tracks. You can choose to change the angles or return to the checking routines.

Any angle error flags that were added will be removed before the next artwork diagonals check is run. (All error flags can be removed if required by selecting **File ⇒ Delete ⇒ Error Flags** once the artwork editor has been closed.)

#### Clearance check and connectivity tracer

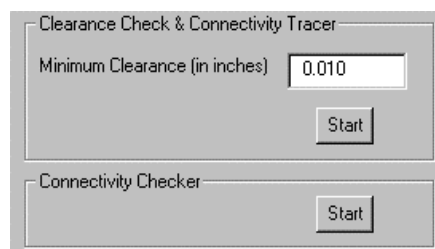


Figure 13. 2

## COMPLETING THE DESIGN

It is **essential** that **both** the checks shown in Figure 13. 2 (the **Clearance check & connectivity trace** and the **Connectivity checker**) are performed before the design is output for manufacture. Any errors that are reported must be corrected and the checks run again until no errors are reported. Failure to complete these checks may result in an incorrect design being manufactured.

Enter the minimum clearance required between copper entities on the design, **0.010"** in this case. Select **Start**.

If gap errors are found, the number of errors will be reported in this window and a gap error marker flag will be placed against the offending pieces of track.

These flags (containing the letter G for gap error) can be seen from the artwork editor. Correct any gap errors reported, then run this check again until no gap errors are reported (or you've chosen to ignore them for valid design reasons).

This check also generates an internal wiring list, the *connection trace*, from the artwork. Once all the gap errors have been corrected and the *Clearance check & connectivity trace* run again, select **Start** from alongside the **Connectivity checker**. (Don't start this check until the *Clearance check & connectivity trace* has been run after changing the artwork, or the results will be invalid.)

This check compares the last *connection trace* against the original wiring list and flags any differences. These can range from short circuits to un-routed tracks (connectivity errors).

The following messages are given as examples from an error report:

-----Shorted Signals-----

Signal

C4.1 R2.1 IC1.4

--- shorted to signal ---

C3.1 C2.1 PL1.5

Short circuit(s) detected at:-

X 3606 Y 2798 (flagged)

This message indicates that the two signals listed are shorted to one another at the location X 3.606", Y 2.798", and a short circuit flag has been added. In some situations Ranger is unable to calculate where the short circuit has occurred. If so, a report indicates which nets are shorted, but not where. i.e. they are not flagged.

The following message is given when connections have not been routed for whatever reason:

-----Connectivity Errors-----

Error in net

R2.2 C4.2 BR1.4 C1.2 PL1.7

Subnet R2.2 C4.2

not connected to

Subnet BR1.4 C1.2 PL1.7

This indicates that there is an error in the net that is listed out (R2.2 C4.2 BR1.4 C1.2 PL1.7). It then indicates that the error is caused because the net has been split into two halves or "subnets".

i.e. R2.2 and C4.2 are connected together, as are BR1.4, C1.2 and PL1.7, but there is a track missing between the two "subnets".

When connectivity errors are reported, the missing connections can be added to the artwork using the *digitise remaining* option from within the *Pads & track digitise menu*. They should of course be routed and the artwork checks run again.

Return to the artwork editor and clear any problems that have been indicated. Once they have been cleared, run **both** checks again to ensure further errors have not been introduced. The original gap or short circuit marker flags will be removed automatically at the beginning of the check. (A common mistake is to alter the artwork, then just run the *Connectivity checker* again. This simply compares the last trace file (which will now be out of date) against the parts/wiring list, so you will get the same error report you got last time.

All errors should be corrected and the checks re-run until no errors are reported. Once no errors are reported, the board may pass safely to manufacture. (The output files can be produced at any time, even if errors have been reported.)

## Silk screen generation

Once the parts have been placed on the board, the silk-screen information can be generated. This is usually left until the design is complete, as the silk-screen can clutter up the screen and until the design is complete you won't know where to place the labels.

The silk-screen outline as defined in the component outlines, and their reference designators (labels) will be added to user specified layers of the artwork. It is recommended that the outlines and labels are placed on separate layers in order to make subsequent modifications less time consuming. Once added to a layer, the labels can be moved into suitable positions.

(The labels that can be switched on and off in the artwork editor using the *View* command are not the labels that appear on the silk-screen, they are purely used for reference purposes before the silk-screen has been generated.)

With the job open, select  (**Artwork ⇒ Silkscreen Generation**). The silk-screen generation window appears as shown in Figure 13. 3.

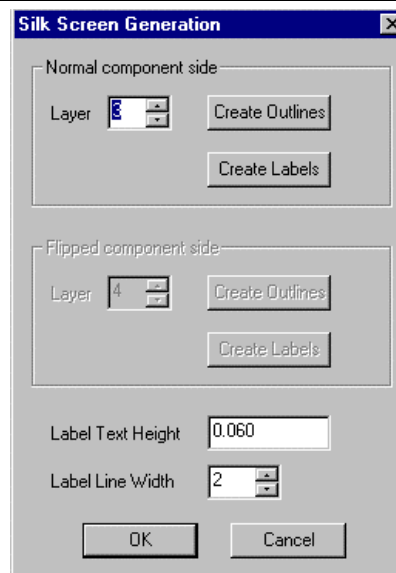


Figure 13. 3

The window is divided into two areas, *Normal component side* for parts that have not been flipped and *Flipped component side* for parts that have been flipped. (If no flipped parts exist, this area is unavailable for selection.)

The two areas allow component outlines and/or labels to be generated on either side of the board independently.

The silk-screen outlines and labels are added to the layer defined by the *Layer* box.

Set the *Layer* box to **3**, the default layer for text manipulation.

Ensure the *Label text height* & *Label line width* at the bottom of the window is set to the height and line width code you require.

Select **Create Labels**. The labels will be placed on layer 3. (Labels are added to the specified layer - so do not generate the labels twice.)


Set the *Layer* box to **4**. Select **Create Outlines**. A set of component outlines has now been added to layer 4 (everything from that layer will be deleted first).

**We advise that you always add the outlines and labels to different layers because this will make any subsequent board modifications easier to perform.**

If you have been working on your own design and some parts have been flipped, generate the silk-screen outlines and labels for those parts now, using layers 5 and 6.

Select **OK** when you are ready to continue.

Note: the silk-screen generation menu re-assigns the layers used for silk-screen data, as *Silk-screen* layers. This means that Ranger does not include them in the artwork checking routines. Provided you only modify the silk-screen layers, it is not necessary to check the artwork again. However, if any additional text is added to the copper layers, it may cause short circuits because the text will be formed from copper. In this case, perform the checking routines again. If in doubt, **always run the checks**.

Select  (**Artwork ⇌ Editor**) to view the completed artwork.

The artwork now contains drilled component pins and vias on layer 0, tracks (and SMD component pins if used) on layers 1 and 2, component labels on layer 3 and outlines on layer 4.

The labels are placed on the component outline's datum. This is not usually where they are required, as the datum is very often under the device and would not be seen on an assembled board. (Do not return to the component outline editor and move the datum or you will alter the position of the part as well.)

The labels can be moved using the Text menu. The *Text* section in the right hand dialogue bar controls text selection, addition, modification, etc.

Ensure the *Layer* in the *Text* section of the dialogue bar (Figure 13. 4) is set to the layer that the text you want to move is on.


The *Height* and *Line size* buttons are used when the *Text* ⇌ *Add*, *Change Width & Height* or *Get Label* commands are used.

The *Rotation* button determines the orientation of the text string as you move it, in degrees clockwise.





Figure 13. 4

Select  (**Text ⇒ Move**). Position the labels where required (select the label by its lower left corner). To rotate a text string, ensure the **Rotation** button in the dialogue bar is set to the angle required before moving the string.

The tracking layers can be made invisible to assist with the positioning of the labels, using the **View ⇒ Layer Properties** command (see below).

Additional text can be added as required to any layers (the artwork checks should be run if added to copper layers). When the text has been positioned satisfactorily then the artwork is complete and ready for output.

### Colour and layer options

The *Layer Properties* window controls the colours used for particular layers, and which layers are visible. It also contains information relating to the layer usage and the number of points per layer.

With the artwork editor loaded, select **View ⇒ Layer Properties**. A window appears as shown in Figure 13. 5.

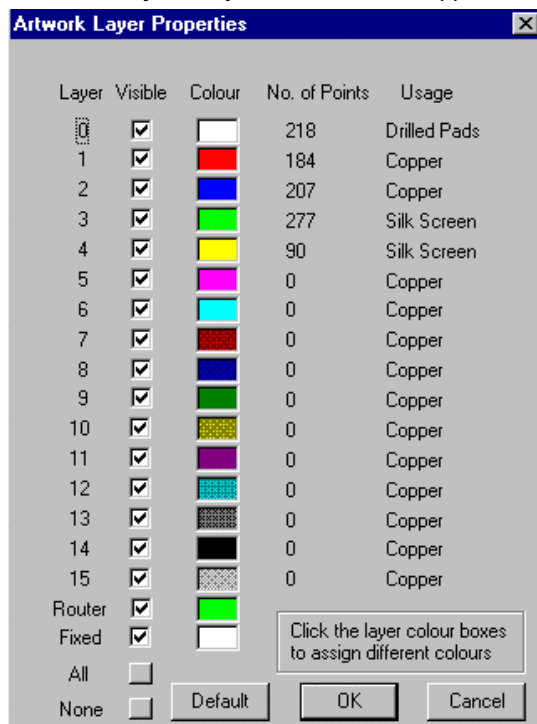


Figure 13. 5

The layers can be made visible or invisible by ticking the box in the *Visible* column alongside the layer. Notice that layers 3 and 4 have been changed from copper layers to silk screen layers in the *Usage* column and they now contain some *Points* which indicates they are used.

The colours assigned to each layer can be changed by following the instructions within the window.

Make layers 1 and 2 invisible, then select **OK** to return the artwork display. The tracks on layers 1 & 2 are now invisible. They can be made visible again from the *View ⇒ Layer Properties* window.



## Finding out what pad and track sizes have been used

From the job design menu (the artwork editor will need to be closed), select **Edit ⇒ Sizes table**. The table that we looked at at the beginning of the design example appears. Some of the pad and track sizes now have ticks alongside them to indicate that they are used in this particular job.

If you change a used size, the change is implemented immediately on the artwork. The checking routines would need to be run again if the size was made larger.

To find out the sizes of individual pads and tracks, load the artwork editor. Ensure the track layer in the left hand dialogue bar is set to the layer the track is on, then select **Amend ⇒ Show Size**. Select the pad or track to be identified (select tracks on a corner or end point). The status bar indicates what has been selected and its size code/shape.

## Adding copper areas to the artwork

On some designs it is necessary to add large areas of copper to the board, for shielding purposes, etc. This can be achieved automatically with Ranger. The copper area is created from horizontal and vertical tracks that form a cross-hatched or solid area, its shape being a user-defined polygon with keep-out areas if required. The copper area is joined to an existing pad or track that the operator specifies as a datum.

### To fill an area

Select **ARR** (**Artwork ⇒ 2 Layer Auto-router**). When the artwork has been loaded, select **Copper Fill ⇒ Setup**. A window similar to the one in Figure 13. 6 appears.

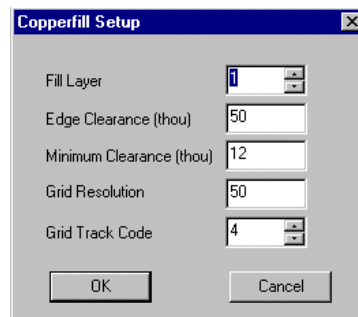


Figure 13. 6

Set the parameters as shown. These settings will create a copper area on layer **1**, the distance between the copper and edge of the area (edge clearance) will be **0.050"**, the distance between the copper itself and existing copper on the board (except the datum) will be **0.012"** (minimum clearance), the horizontal and vertical lines that form the fill will be **0.050"** apart (grid resolution) and the thickness of the lines will be track code **4** (grid track code).

Select **OK** to close the window. Now you must define the area to be filled. For this exercise we will fill the complete board.

Select **Copper Fill ⇒ Add Line**. Draw a polygon shape around the outside of the board profile. When the outline has been closed, release the line with a click of the right hand mouse key. Don't worry too much about following the exact shape of the profile - it is only possible to draw straight lines and this line will disappear when you leave the copper fill menu. (The fill will stop 0.050" from the edges of the profile as defined in the setup window when it is added.)

The line can be modified as necessary using the Copper fill menu commands.

Now a datum has to be chosen which the copper will be connected to.

The datum can be either a pad, track or an existing piece of copper, but it has to be inside the polygon area. If the datum is a track or SMD pad, it has to be on the same side as the copper fill being added.

Select **Copper Fill ⇒ Fill Datum**, then select a **GND** track from layer **1**. A message should appear in the status bar to indicate a valid datum point has been selected. Once this message appears, select **Copper Fill ⇒ Start Filling**. Messages appear in the status bar to indicate what is happening. The copper fill will appear, notice it is cross-hatched.

If you zoom in, you will see that the copper fill is connected to the datum, but not to anything else.

Now try deleting the fill and adding it as a solid area. Select **Copper Fill ⇒ Delete Fill**, then select a point on the fill. In the copper fill setup window, change the *grid resolution* to **0.025"** and the *Track code* to **6** (0.026"). This will ensure the lines in the fill actually overlap, rather than butt up. Add the fill as before (choose a datum, then start filling).

The copper fill can also be modified if required, using the *Amend ⇒ Move Point* command of the artwork editor.

When using copper fill, the copper is added inside either the board profile or the polygon outline, whichever is smaller. Keep-out zones for copper fill can be created by adding extra polygons inside of the main polygon, or by defining *keep out* areas within the board profile editor.

Select **File ⇒ Close Autorouter** when you are ready to continue.

## Chapter 14

## OUTPUTS

Once the design is complete it is ready for output, though it can be output at any stage throughout the design. Various output options and formats are available. Move to the heading in this chapter that describes the type of output you wish to produce. The headings are:

**Output to pen plotters** (on page 60) - to output the circuit, artwork or drill drawing in an HP-GL or Houston Instruments format file (alternatively you can use the *Outputs to Printers* if you have a Windows driver for the pen plotter).

**Output to photoplotters (Gerber)** (on page 63) - to output the artwork or solder mask to a Gerber format file.

**Output to printer** (on page 66) - to output the circuit or artwork to a printer (or any device with a Windows output driver).

**Output to NC drill machines** (on page 68) - to output the artwork hole positions or routing profile in an Excellon or Sieb & Meyer format file.

These are the only output routines available. You may wish to produce a print-out or pen plot of the solder mask, this is described at the end of this chapter under the heading **Producing a print-out or HP-GL file of a solder mask** on page 68.

### Output to pen plotters

Select **File** ⇒ **Outputs** ⇒ **Pen Plotting**. The window shown in Figure 14. 1 appears.

This window has a **Plotter Configuration** button and is divided into two main areas, entitled **Artwork and Drill Sheet Plots** and **Circuit Plots**.

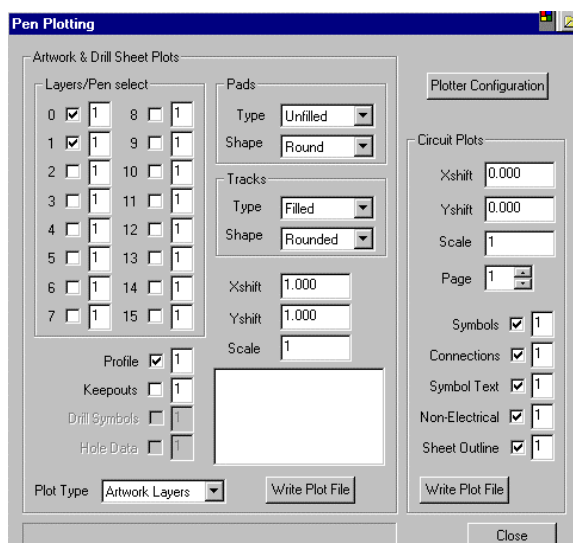


Figure 14. 1

### Plotter configuration

Before plotting, check that the plotter configuration is set correctly for the plotter in use. Select the button **Plotter Configuration** from the window, a window similar to the one in Figure 14. 2 appears.

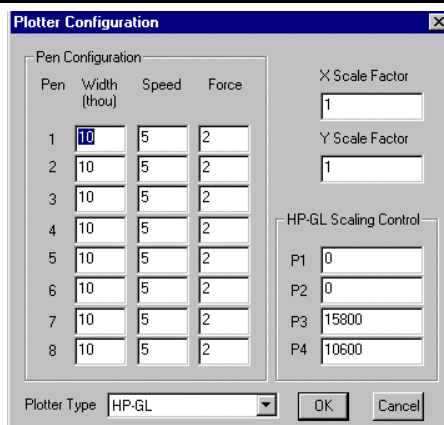


Figure 14. 2

This window is used to define the type of plotter in use, pen diameters, pen speed and force.

Set the **Plotter type** to *HP-GL* to drive Hewlett-Packard pen plotters (or compatibles, for instance Roland), or to *Houston Instruments* to drive Houston Instruments pen plotters.

If the plotter type is set to *HP-GL*, then ensure the correct *HP-GL Scaling Controls* have been entered (P1 & P2 values). These values can be obtained from your plotter manual.

The X & Y scaling factors are used to compensate for any inaccuracies in the plotter. The output is scaled by the value entered.

Select **OK** once the window has been set to suit the output type required. Now move on to the heading that describes what you want to plot, i.e. **Pen plotting the circuit schematic** (page 61), **Pen plotting the artwork** (page 61) or **Pen plotting the drill drawing** (page 62).

### Pen plotting the circuit schematic

Refer to the *Circuit Plot* area of the Pen Plotting window as shown in Figure 14. 3.

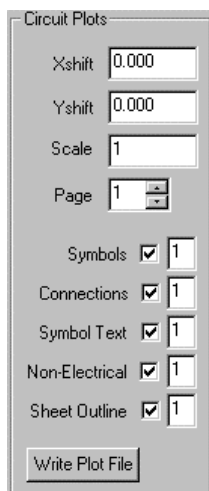


Figure 14. 3

The **X** and **Y shift** entries can be used to move the schematic around on the paper.

The **Plot scale** is used to scale the output. Enter 1 for a 1:1 plot, 0.5 for a 1/2:1 plot, 2 for a 2:1 plot, etc.

Note: because pen plotters cannot usually plot to the edges of the paper, you will need to scale down the output of an A4 circuit onto A4 paper, A3 onto A3, etc. A scale of 0.9 is usually OK.

The **Page** number indicates which page of the schematic will be output. Use the spin controls to change it, or re-type it.

A list of categories from the schematic follows (**Symbols, Connections, etc.**), select the box alongside them to include or exclude them from the plot (the tick indicates the category will be plotted).

The number in the box alongside each category indicates which pen in the pen plotters carousel will be used to plot it.

Once the window is set as required, select **Write Plot File** to commence the output. The *Save As* window appears, use the browser to select the folder you want the file to be placed in and supply a filename. The file is saved as a text file that can be sent to the pen plotter. Select **Close** when you are ready to continue. Refer to the heading **Sending files to the plotter** (page 63) for details on what to do with the file that's produced.

### Pen plotting the artwork

Refer to the *Artwork and Drill Sheet Plots* area of the Pen Plotter window as shown in Figure 14. 4.

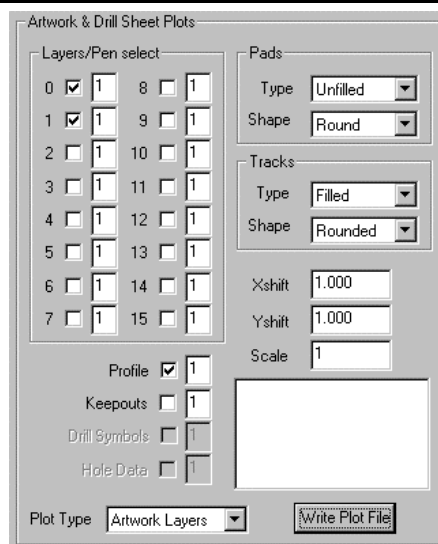


Figure 14. 4

The **Plot type** should be set to **Artwork layers**.

First of all decide which layers you want to plot and with which pen in the plotter. Combinations of layers are usually output, depending on the type of output required. Selecting the box alongside the layer includes or excludes it from the plot (the tick indicates the layer will be plotted).

Layer **0** contains all the drilled component pads and vias.

Layer **1** contains top mounted, surface mounted component pads (if used) and tracks (if used).

Layer **2** contains bottom mounted, surface mounted component pads (if used) and tracks (if used).

Therefore, layers **0 and 1** should be output together to produce an artwork for the **component side of the board**.

Layers **0 and 2** should be output together to produce an artwork for the **solder side of the board**.

If the silk-screen was created as suggested in the worked example, layers 3 and 4 should be output together for the top silk-screen mask, and layers 5 and 6 together for the bottom silk-screen mask (if flipped parts exist).

If you make a board with **power planes**, then **only** the **power plane layer** should be output (unless you have added some text, an isolation track, etc. on another layer in which case it should also be output – do not include layer 0). The actual output plot will need to be photographically reversed in order to achieve the film required.

The board profile and keepout areas can be included as required.

The number in the box alongside each layer indicates which pen in the pen plotters carousel will be used to plot the layer.

The **Pads** and **Tracks**, **Type** and **Shape** can be changed as required by selecting the arrow alongside them.

The **X** and **Y shift** entries can be used to move the artwork around on the paper.

The **Plot scale** is used to scale the output. Enter 1 for a 1:1 plot, 2 for a 2:1 plot, 0.5 for a 0.5:1 plot, etc.

Once the window is set as required, select **Write Plot File** to commence the output. The **Save As** window appears, use the browser to select the folder you want the file to be placed in and supply a filename. The file is saved as a text file that can be sent to the pen plotter. Select **Close** when you are ready to continue. Refer to the heading **Sending files to the plotter** (on page 63) for details on what to do with the file that's produced.

## Pen plotting the drill drawing

The drill drawing indicates the position of all the drilled holes and their size, using symbols to represent different hole sizes.

Change the **Plot type** setting in the Pen Plotting window to **Drill Drawing**, this will enable the drill drawing parameters and disable the artwork layers, as shown in Figure 14. 5.

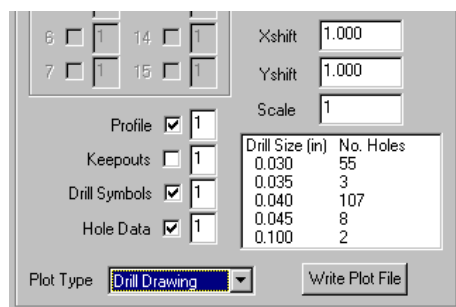


Figure 14. 5

The boxes alongside each category (**Profile, Keepouts, etc.**) indicate which items will be output. Selecting the box includes or excludes it from the plot (the tick indicates the category will be plotted).

The number in the box alongside each category indicates which pen in the pen plotters carousel will be used to plot it.

The **X** and **Y shift** entries can be used to move the drill drawing around on the paper.

The **Plot scale** is used to scale the output. Enter 1 for a 1:1 plot, 2 for a 2:1 plot, 0.5 for a 1/2:1 plot, etc.

Information regarding drill size and quantities is given.

Once the window is set as required, select **Write Plot File** to commence the output. The **Save As** window appears, use the browser to select the folder you want the file to be placed in and supply a filename. The file is saved as a text file that can be sent to the pen plotter. Select **Close** when you are ready to continue.

## Sending files to the plotter

### Plotter connected to parallel port (LPTnn)

At a DOS prompt, type:

**COPY FILENAME LPTnn**

(where **FILENAME** is replaced by the full name of the file (path included) and **nn** is replaced by the parallel port number)

### Plotter connected to serial port (COMnn)

Ensure a mode suitable for your plotter is set, typically type the following at a DOS prompt:

**MODE COMnn:9600,N,8,2,P**

(where **nn** is replaced by the serial port number)

Then send the file to the plotter by typing:

**TYPE FILENAME > COMnn**

(where **FILENAME** is replaced by the full name of the file (path included) and **nn** is replaced by the serial port number)

## Output to photoplotters (Gerber)

Either the *artwork layers* can be output (to create an artwork, silk-screen or power plane output), or a *solder mask* can be output.

Before we describe the output procedure, let's take a few minutes to describe how a Gerber photoplotter works. Most queries regarding photoplotting are to do with DCodes.

### Photoplotter basics

A photoplotter exposes specific areas of photographic film to light. The areas are exposed by shining a light through an aperture and onto the film. The size and shape of the aperture therefore controls the size and shape of the exposed area. Because there is more than one aperture available, the apertures are each given a unique name as a reference. This is known as its "DCode".

Typically the apertures are round or square, although other shapes are available.

Round pads are produced on the film by moving the film into the correct position under the light source, selecting a round aperture of the correct size, then switching the light source on and off. The film is then moved and the process repeated. This is known as *flashing*.

Tracks, text and round-ended finger pads are produced in a similar way, except the film is also moved whilst the light is switched on. This is known as *drawing*.

For instance to produce a round-ended finger pad of 0.030" x 0.080", a 0.030" round aperture is selected. The light is switched on, the film moved in the required direction by 0.050" and then the light switched off. This exposes an area 0.030" wide by 0.080" long, with rounded ends.

Square pads are *flashed* using a square aperture, and rectangular pads are *drawn* with a square aperture.

The Gerber file basically contains X/Y co-ordinates, aperture selection and draw/flash information - it doesn't include pads, tracks or holes.

The list of aperture sizes and shapes, and their associated DCodes are held in a table known as a DCode Table.

Each plotter has its own set of apertures and DCode numbers (there is not a standard) so consequently the DCode table information has to be supplied each time a Gerber file is produced.

When Ranger creates a Gerber file, it has to map the pad and track sizes used on the artwork to DCodes. It obtains the DCode information from the DCode table - which you have to supply.

Before the Gerber file is produced, Ranger checks that the DCode table contains the required aperture sizes, and that they have a DCode number assigned. If an aperture is missing, or it doesn't have a DCode number assigned, the Gerber file will not be produced and a message appears on the screen indicating the apertures required.

The necessary apertures and DCodes should be added to the DCode table. DCode numbers should start from 10 onwards. It is essential that the same DCode is not used for different sized apertures or different shaped apertures.

### The output procedure - Artwork layers

Before selecting the Gerber photoplot output tools, ensure the units are set to *Inches* (**Edit ⇒ Units ⇒ Inch Units**).

## OUTPUTS

Select **File** ⇒ **Outputs** ⇒ **Gerber Photoplot**, a window similar to the one shown in Figure 14. 6 appears. Ensure the **Plot type** in the Gerber Photoplotting window is set to **Artwork Layers**.

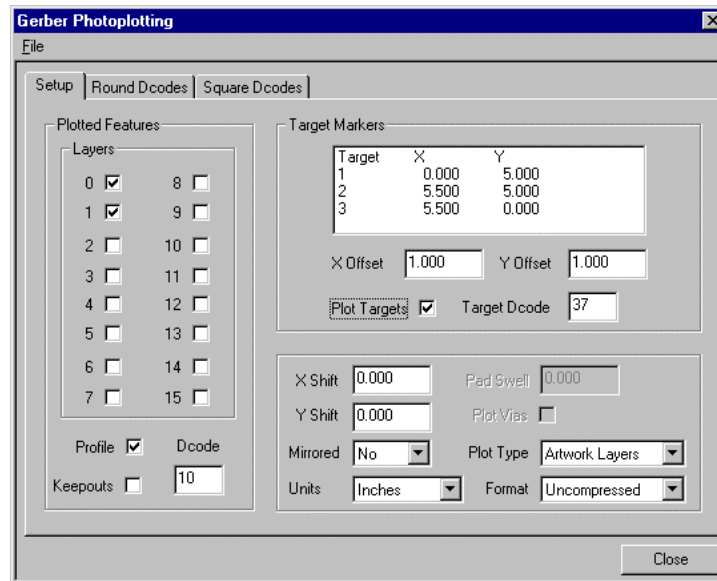


Figure 14. 6

The Gerber Photoplotting window is divided into three main *areas* (*Plotted features*, *Target markers* and *Other variables*).

**Plotted Features** - this indicates which layers from the artwork will be output.

Combinations of layers are usually output, depending on the output required. Selecting the box alongside the layer includes or excludes it from the plot (the tick indicates the layer will be output).

Layer **0** contains all the drilled component pads and vias.

Layer **1** contains top mounted, surface mounted component pads (if used) and tracks (if used).

Layer **2** contains bottom mounted, surface mounted component pads (if used) and tracks (if used).

Therefore, layers **0 and 1** should be output together to produce an artwork for the **component side** of the board. Layer **0 and 2** should be output together to produce an artwork for the **solder side** of the board.

If the silk-screen was created as suggested in the worked example, layers 3 and 4 should be output together for the top silk-screen mask, and layers 5 and 6 together for the bottom silk-screen mask if flipped parts exist.

If you make a board with **power planes**, then **only** the **power plane layer** should be output (unless you have added some text, an isolation track, etc. on another layer in which case it should also be output – do not include layer 0). The actual output plot will need to be photographically reversed in order to achieve the film required.

The board profile and keepout areas can be included as required. If they are output, the DCode specified alongside them will be used - ensure it is set to an appropriate size in the DCode table.

**Target Markers** - this area controls whether three auto-generated target markers (used to line up the layers during manufacture) are output. A tick alongside *Plot targets* will output them. (They cannot be seen from the artwork editor.)

The **X and Y Offsets** control how far away the targets are from the edge of the board. A positive value moves the targets away from the profile. Their absolute positions are given in the table above the offsets - this table is updated automatically as the offsets are changed.

The same offsets should be used for all output files for one design, typically 0.5" to 1.0" is used.

If the targets are output, then a DCode has to be specified for them - ensure it is set to an appropriate size/number.

**Other Variables** - the remaining area, shown in Figure 14. 7 is described now.

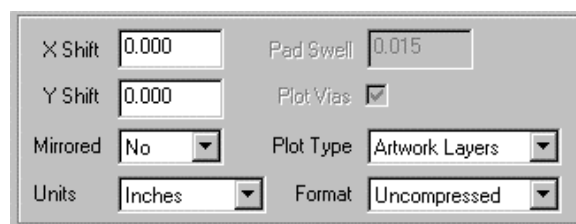


Figure 14. 7

The **X and Y shift** entries can be used to move the artwork around on the film. The lower left corner of the area occupied by the board profile and target markers is the datum position of the design.

The **Mirrored** option can be used to mirror one of the outer layers, so that all the films can be placed emulsion up, or emulsion down when the board is manufactured.

The **Units** parameter specifies whether the output will be produced in Inches or Metric (mm).

The **Format** should be set to **uncompressed** unless you know the plotter can take compressed formats.

Once the window is set as required, select **File ⇒ Write Gerber File** from the top of the window to commence the output. The Save As window appears, use the browser to select the folder you want the file to be placed in and supply a filename.

When the Gerber data is created, the pads and tracks on the selected layers are mapped to a size and DCode held in the DCode table. If an exact size match cannot be found, a report highlights the sizes missing from the DCode table and the output fails. The required sizes and DCodes must be added to the DCode table.

Note: working in metric units when producing the Gerber files can produce an incorrect report of the missing sizes. If the report is displayed in metric units (decimal points are displayed with a comma (,)) then close the report and output windows, change the units to inches (**Edit ⇒ Units ⇒ Inch Units**) and continue with the output.

If a size is highlighted in the *round*, *track*, *rwidth*, *anti-pad* or *heat-relief* columns, the size should be added to the *round* DCode table.

Sizes in the *square* and *swidth* columns should be added to the *square* DCode table.

Once all the sizes have been entered as required, select **File ⇒ Write Gerber File** again. A window appears to indicate the output file has been created.

## DCode Table

Ranger2 needs to know the DCode numbers that it should use in the Gerber file. It obtains this information from a table that you fill in.

Select the **Round DCodes** tab from the top of the Gerber Photoplot window, a window similar to that shown in Figure 14. 8 appears.

Size	Dcode	Size	Dcode	Size	Dcode	Size	Dcode
0.000	10	0.000	10	0.000	10	0.000	10
0.000	10	0.000	10	0.000	10	0.000	10
0.000	10	0.000	10	0.000	10	0.000	10
0.000	10	0.000	10	0.000	10	0.000	10
0.000	10	0.000	10	0.000	10	0.000	10
0.000	10	0.000	10	0.000	10	0.000	10
0.000	10	0.000	10	0.000	10	0.000	10
0.020	10	0.000	10	0.000	10	0.000	10

Figure 14. 8

The table contains pairs of columns, **Size** and **DCode**.

Enter the sizes required in the size column, and a unique number from 10 upwards in the corresponding DCode column.

Round DCodes are used to create round pads, round-ended rectangular pads, anti-pads, heat-relief pads and tracks.

Square DCodes are used to create square pads and square-ended rectangular pads.

Do not alter the sizes or DCodes whilst producing a set of films for one design.

Repeat this exercise for the required square DCodes after selecting the **Square DCodes** tab.

A copy of the DCode table should be supplied with the Gerber files. Select **File ⇒ Write DCodes file** from the top of the window to output the DCode sizes to a text file, or **File ⇒ Print DCodes** to obtain a print-out.

## The output procedure - Soldermask

This option is used to produce an oversize pads only, Gerber output that is used to produce a solder mask for the board. (The output is photographically reversed to produce the required film.)

Ensure the **Plot type** in the Gerber Photoplot, Setup window is set to **Solder mask** (Figure 14. 9).

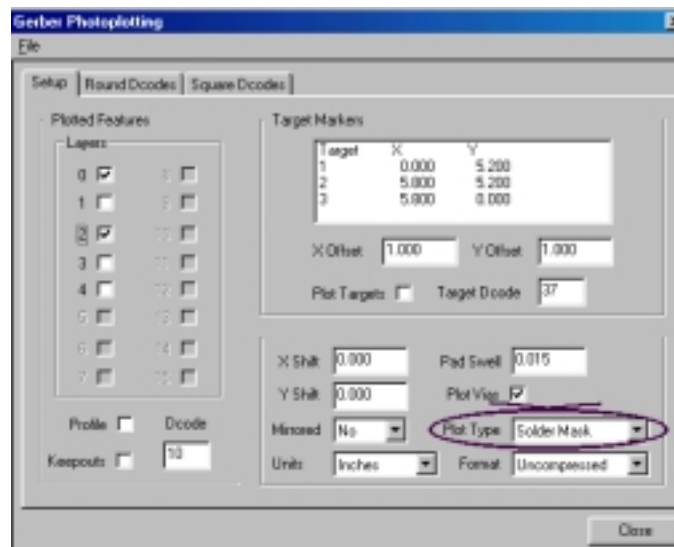


Figure 14. 9

When selected, only layers 0, 1 and 2, plus any silk screen layers are available for output.

**Note:** size 0 tracks and text are not output when the *Plot type* is set to *Solder Mask*. Size 0 component pads are only output if Vias are included in the output (size code 0 should be reserved for vias only).

Typically, layers 0 and 2 are output together for the solder mask for the bottom of the board. (Layers 0 and 1 for the top of the board.) Ranger does not include any tracks from layer 1 and 2.

**Pad swell** - The pads are output with the pad swell specified in order to make them slightly larger than the copper pads. So a 0.065" pad with a 0.015" swell will be output at 0.080".

**Plot vias** - If vias are included, code 0 pads are output which means the solder mask does not cover via holes.

Once the window is set as required, select **File** ⇒ **Write Gerber File** to commence the output. The **Save As** window appears, use the browser to select the folder you want the file to be placed in and supply a filename.

For the solder mask, Ranger does not require an exact match in the DCode table. It looks for an exact match but if one is not found then, then it uses the nearest DCode to within +0.010". So for instance, if it is looking for an 0.080" round aperture and one is not defined, then it will use one up to 0.090".

## Output to printers

Either the circuit diagram or artwork can be output to a device that has a Windows driver.

### Printing the circuit schematic

Open the circuit diagram  (**Circuit** ⇒ **Edit**). Select **File** ⇒ **Print Setup**. Ensure the window is set to the type of printer/device you require, paper size etc. Select **OK**.

Select **File** ⇒ **Print**, the window will change and appear similar to that shown in Figure 14. 10.

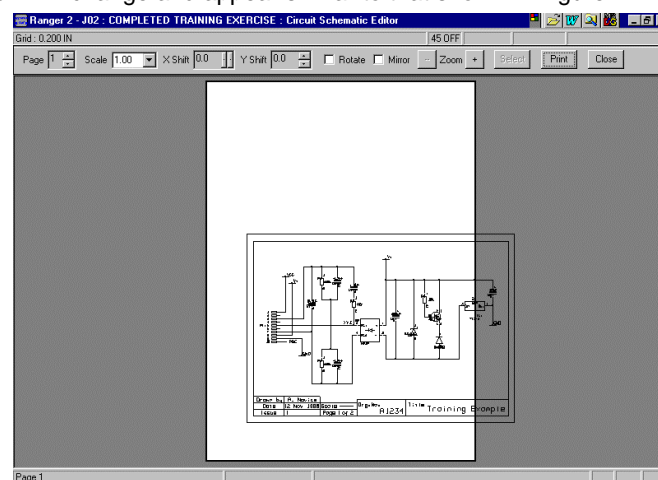


Figure 14. 10

The white area represents the paper that the circuit will be printed on. The tool bar across the top indicates how the circuit will be printed and these parameters can be changed as required, as follows.



- Page** - indicates the page that is on display and that will be printed. Use the spin controls to change it.
- Scale** - controls the scale of the output. A variety of pre-set scales are selectable, or the value can be selected and a user-specified value entered. Note: to print an A4 circuit onto A4 paper, change the scale to 0.9 because printers cannot usually print right to the edges of the paper.
- Shift X & Y** - controls the position of the circuit on the sheet. Use the spin controls to increase or decrease the values.
- Rotate** - the circuit can be rotated by 90 degrees if required. Select the box alongside to "tick" it. The tick indicates 90 degree rotation.
- Mirror** - the circuit can be mirrored if required. Select the box alongside to "tick" it. The tick indicates mirroring has occurred.
- Zoom ++ & --** - these buttons allow you to zoom in up to two levels and back out again.
- Select** - when selected a window appears that allows you to control what items are printed from the circuit, i.e. symbols, connections, etc. The *Plot in black* parameter controls whether a black & white or colour output is produced. (If a colour print is produced, the colours are as defined in the *Circuit Setup* ⇒ *Setup Editor* window.)
- Print** - when selected sends the circuit to the output device window, which can then be used to start the output.
- Close** - to return to the schematic editor.

## Printing the artwork

Open the artwork  (**Artwork** ⇒ **Edit**). Select **File** ⇒ **Print Setup**. Ensure the window is set to the type of printer/device you require, paper size etc. Select **OK**.

Select **File** ⇒ **Print**, the window will change and appear similar to that shown in Figure 14. 10, except the board design will be visible instead of the circuit schematic.

The white area represents the paper that the artwork will be printed on. The artwork can be rotated, scaled up/down, mirrored, etc. using the controls in the tool bar, until it is set as you require it. These commands were explained under the heading **Printing the circuit schematic** so will not be repeated. However the **Select** command is different.

**Select** - when selected, a window similar to that shown in Figure 14. 11 appears.

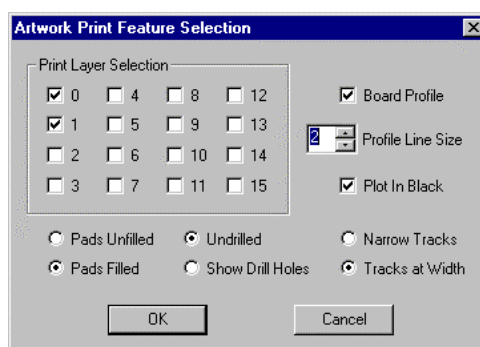


Figure 14. 11

**Print layer selection** - the ticks in the boxes indicate which layers will be output. Most artwork print-outs are a combination of layers.

Layer 0 contains all the drilled component pads and vias.

Layer 1 contains top mounted, surface mounted component pads (if used) and tracks (if used).

Layer 2 contains bottom mounted, surface mounted component pads (if used) and tracks (if used).

Therefore, layer **0 and 1** should be output together to produce an artwork for the **component side** of the board. Layer **0 and 2** should be output together to produce an artwork for the **solder side** of the board.

If the silk-screen was created as suggested in the worked example, layers 3 and 4 should be output together for the top silk-screen mask, and layers 5 and 6 together for the bottom silk-screen mask (if flipped parts exist).

If you make a board with power planes, then only the power plane layer should be output (unless you have added some text, an isolation track, etc. on another layer in which case it should also be output – do not include layer 0). The actual output plot will need to be photographically reversed in order to achieve the film required.

**Board profile** - the board profile can be included as required. If it is, you must specify a track code with which it should be printed

**Profile line size** (the profile has no thickness assigned to it).

**Pads unfilled or Pads filled** - one or the other can be selected to control the appearance of pads.

**Undrilled or Show drill holes** - one or the other can be selected to control whether the hole is printed or not.

**Narrow tracks or Tracks at width** - one or the other can be selected to control the appearance of tracks.

**Plot in black** - the tick indicates the output will be produced in black. (If a colour print is produced, the colours are as defined in the *View* ⇒ *Set Colours* window.)

## OUTPUTS

Select **OK** to continue.

**Print** - select this button to send the artwork to the output device window, which can then be used to start the output.

**Close** - select this button to return to the artwork editor.

### Output to NC drill machines

This option is used to produce files that can be used to drive an NC drilling machine. Select **File ⇒ Outputs ⇒ Drill Data Production** a window similar to that shown in Figure 14. 12 appears.

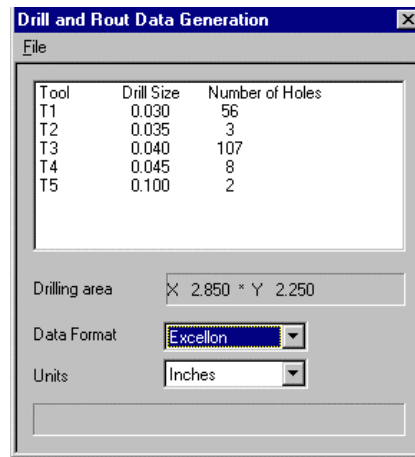


Figure 14. 12

**Data format** - Two formats are available, either *Excellon* or *Sieb & Meyer*. Select the format required.

**Units** - Two formats are available, either *Inches* (fixed 6 digit) or *Metric* (fixed 5 digit). Select the format required.

Select **File ⇒ Save Drill Information** from the top of this window to produce a file containing the drilling information or **File ⇒ Save Rout Information** to produce a file containing routing information.

The **Save As** window appears, use the browser to select the folder you want the file to be placed in and supply a filename. The file is saved as a text file.


Select **File ⇒ Save Tool Information** to output the tool sizes to a text file, or select **File ⇒ Print Tool Information** to print it out.

Select **File ⇒ Close** to continue.

### Producing a print-out or HP-GL file of a solder mask

A soldermask can be automatically produced if it is required in a Gerber format. A work-around can be used to drive a printer or pen plotter, as follows:

#### Solder mask to cover via holes:

- \* Copy the job (**File ⇒ Save As**).
- \* Ensure the copied job is open, then delete all the artwork (**File ⇒ Delete Artwork**).
- \* Digitise the pads in  (**Artwork ⇒ Pad & Tracks Digitise, Digitise Pads**).
- \* **Edit ⇒ Sizes Table** and increase all the used pad sizes by the amount required for the oversize pads (i.e. add 0.015" to the diameter or length and width of the pads).
- \* If the board contains no SMD's, produce a print-out or pen plot of layer **0**. This will produce an output of all the component pads, increased by the amount specified. A photographic reversal will be required.  
If the board contains SMD's, then layer **0 and 2** should be output together (for the solder-side solder mask) and layers **0 and 1** together for the component-side solder mask if required.

#### Solder mask to expose vias:

- \* Copy the job (**File ⇒ Save As**).
- \* Ensure the copied job is open, use the window delete commands in the artwork editor to delete layers 1 and 2 (**Artwork ⇒ Edit, Window** commands).
- \* If the board contains SMD's, digitise the pads to restore any SMD pads on layers 1 & 2 (**Artwork ⇒ Pad & Tracks Digitise, Digitise Pads**).
- \* **Edit ⇒ Sizes Table** and increase all the used pad sizes by the amount required for the oversize pads (i.e. add 0.015" to the diameter or length and width of the pads).

- \* If the board contains no SMD's, produce a print-out or pen plot of layer **0**. This will produce an output of all the component pads, increased by the amount specified. A photographic reversal will be required.  
If the board contains SMD's, then layer **0 and 2** should be output together (for the solder-side solder mask) and layers **0 and 1** together for the component-side solder mask if required.

### Chapter 15

## ADDITIONAL INFORMATION

It is likely that modifications will need to be made to completed designs, or that a multi-layer board will have to be designed. This chapter explains how to approach these situations using the training exercise to work on (or your own design).

Whenever a board is modified, it is good practice to work on a copy of the design. If you make a mistake you will still have the original.

It is also good practice to make a copy of a design as the design progresses so that it is possible to back-track if required.

### Moving parts on a digitised design

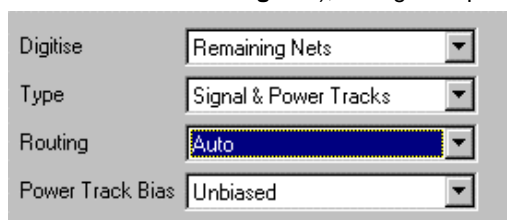
Once digitising has been performed, always move parts from within the artwork editor, using the **Parts** or **Window** menus. The positional changes are automatically back annotated to the parts list and part placement editors.

If parts are moved in the part placement editor (or part list editor) then the pads and tracks in the artwork editor do not move. This will lead to errors when the design is checked because the part positions will not line up with the digitised pads and tracks.

### Adding extra parts and/or connections to a completed design

This example assumes the board has been completed and no errors are reported by the artwork checking routines.

- On the circuit schematic add the extra parts and connections, allocate the parts, then re-compile the parts/wiring list.  
Do not de-allocate the existing parts to renumber them sequentially (use the part renumbering facility which is explained later). It is essential that the parts that have been placed on the board do not have their reference designators changed on the schematic because this will cause the parts to shuffle around on the board, leaving their pads and tracks behind.  
Do not delete the existing parts/wiring list or you will lose all the part placement information for the board. (The digitised pads and tracks will remain though, so you could re-position the parts in exactly the same position they originally occupied, but this would be time-consuming.)
- If a circuit was not drawn, add the additional parts and connections to the parts and wiring lists.
- Within the part placement editor, place the new parts just outside the board profile because the existing tracks cannot be seen. Do not be tempted to move any of the existing parts.
- Now perform the artwork checking routines. i.e. select **Artwork ⇒ Trace/Check Artwork**, followed by the **Clearance checker** and **Connectivity checker**. A list of connectivity errors will appear, which corresponds to the connections that have just been added. This list will be used by the digitiser to add the *remaining* connections.
- Return to the digitising menu (**Artwork ⇒ Pad & Tracks Digitise**), change the parameters so that they are set to:



Digitise	Remaining Nets
Type	Signal & Power Tracks
Routing	Auto
Power Track Bias	Unbiased

Then select **Digitise tracks**.

- Return to the artwork editor. You should now be able to see the new parts with their attached connections. Use the **Parts** or **Window** menu commands to make space on the artwork for the new parts if required, then move the new parts into position. Route the new tracks.
- If the silk-screen layers have already been created, use the **Text ⇒ Get Label** command to add the new labels required - set the *Text* dialogue bar as required first. Don't use the automatic silk-screen *label* generation option again because all the parts will have labels added, duplicating the ones already there.
- Create the silk-screen *outlines* again on the appropriate layer using **Artwork ⇒ Silk-screen generation** (everything on the selected layer is removed first - that's why you put the labels and outlines onto different layers).
- Now perform the artwork checking routines once more. No errors should be reported.

## Adding tooling, fixing, mounting holes etc.

There are various methods that can be used to add additional pads/holes to the artwork.

Whichever method is in use, the *Sizes table* has to be modified to include the pad/hole size required, select **Edit ⇒ Sizes Table**. The used sizes have a tick alongside them provided the pads have been digitised in. Choose an unused pad size and change it to the size required - if a copper ring is not required around the hole, set the pad diameter and hole diameter to the same size. (This is because the checking routines check against the pad size, not the hole size.) The copper ring will disappear when the hole is drilled out, leaving just a hole.

### Method 1 (the quick method)

Add a pad to the artwork using the **Enter ⇒ Pad** command. Ensure the pad dialogue box is set as required - the pad must be added to layer 0 to ensure that it is drilled.

This method has the disadvantage that the tooling holes will be removed if a modification necessitates the artwork being deleted and the design re-routed - you may forget to add them again.

### Method 2 (the preferred method)

Create a device in the schematic device library editor, with one terminal that calls up an outline with one pin.

Add the device to the circuit schematic and allocate it. (If preferred, add it to an unused page so that it is part of the design but doesn't appear on the print-outs.) When the circuit is compiled, the part representing the tooling/mounting hole appears in the parts list, calling up the appropriate outline.

Create the outline in the *Outline library*. The silk screen outline can be shown smaller than the pad's hole size if you do not want to see the silk-screen outline on the finished board.

The part should be placed in the usual way.

This method has the advantage that the holes can be accurately positioned using the **Parts ⇒ Key Move** command in the part placement editor, and they cannot be removed accidentally if the *delete all artwork* command is selected. They do have the disadvantage of appearing in the parts list.

## Removing parts and/or connections from a completed design

This example assumes the board has been completed and no errors are reported by the artwork checking routines.

- On the circuit schematic remove the parts and connections. Re-compile the parts/wiring list.  
Do not de-allocate the existing parts to renumber them sequentially (use the part renumbering facility which is explained later). It is essential that parts that have been placed on the board do not have their reference designators changed on the schematic because this will cause the parts to shuffle around on the board, leaving their pads and tracks behind.  
Do not delete the existing parts/wiring list or you will lose all the part placement information for the board. (The digitised pads and tracks will remain though, so you could re-position the parts in exactly the same position they originally occupied, but this would be time-consuming.)
- If a circuit was not drawn, remove the connections then the parts from the parts and wiring lists.
- There is no need to view the part placement. The parts are automatically removed from the design when the parts list is modified. Do not be tempted to move any of the existing parts using the part placement editor - the pads and tracks will get left behind - use the artwork editor **Parts ⇒ Move** command instead.
- In the artwork editor, locate the pads, tracks and silk-screen that still exist for the deleted part. Use the **Amend** commands to remove them.
- Now perform the artwork checking routines. No errors should be reported.

## Renumbering parts once they have been placed on the board

Once the parts have been placed on the board, never de-allocate them in the schematic editor in order to re-number them. If you do, the parts on the board will swap positions, but the pads and tracks will remain where they are. Use the **Artwork ⇒ Parts renumber** command instead. This will update the artwork, parts/wiring list and circuit diagram.

Always renumber the board before adding and moving the silk-screen labels into their final positions.

To assist with the renumbering, it's useful to produce a temporary silk-screen layer with labels and outlines on it. This layer can be printed along with layer 0 in order to produce a plot that can be marked up with the number changes. (The temporary silk-screen labels & outlines can be removed using the **Window** commands in the artwork editor - make sure that only that layer is removed though.)

With the job loaded, select **Artwork ⇒ Parts renumber**, a window similar to the one shown in Figure 15. 1 appears.

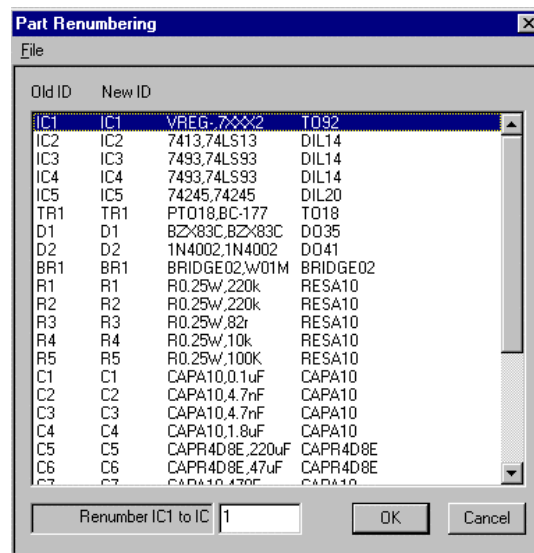


Figure 15. 1

The first two columns show the old part reference (*Old ID*) and the new one that you want assigned (*New ID*). The remaining columns are for description only.

The selected part, IC1 in Figure 15. 1, is displayed in the *renumber* box at the bottom of the window. Enter the new number required for the selected part in this box.

It is not possible to have two parts with the same number - Ranger2 will renumber the part to clear the conflict. For example if you try to renumber IC1 to IC2 in this example, IC2 will become IC1.

When <enter> is pressed the next part in the list is selected. To select a different part, simply select it with the cursor.

When you have completed renumbering the parts, select **OK** to implement the renumbering (select **Cancel** to cancel the changes). The parts are renumbered throughout the design (back-annotated), except for the original silk-screen labels on the artwork (if present) which will have to be removed because they are now incorrect.

## Designing a multi-layer board

Use a copy of the worked example to practice on. Delete all the artwork (*File* ⇒ *Delete* ⇒ *Delete All Artwork*), then digitise the pads, followed by the tracks for auto-routing, even when manually routing (*Artwork* ⇒ *Pads and Tracks Digitise*).

### Inner copper layers

Using additional tracking layers is quite straightforward. In the artwork editor, simply change the *Primary* and *Secondary* routing layers in the *Tracks dialogue bar* to the layers that you intend working on.

For instance:

To convert a connection onto layers 3 or 4, change the *Primary* and *Secondary* layers in the *Tracks dialogue bar* to layers 3 & 4.

To convert a connection onto a particular layer, change the *Primary* and *Secondary* layers to the same particular layer.

To swap a track segment from layer 2 to layer 5, ensure the either the *Primary* or *Secondary* layer is set to 2, and the other to 5 (layers 2 & 5 are now active). Select *Mroute* ⇒ *Layer Swap*, then select the segment that is on layer 2. It swaps to layer 5. Tracks on layers 2 & 5 can now be modified (Layer swap, Corner, Delete point, etc.) until the routing layers are changed.

Remember:

- *Mroute* ⇒ *Convert to ManRoute* converts predominantly horizontal connections on to the *Primary* layer, and predominantly vertical connections to the *Secondary* layer.
- *Mroute* ⇒ *Layer swap* swaps track segments between the *Primary* and *Secondary* layers.
- Layers 1 and 2 are always the outer layers of the board because they include any surface mounted pads. The other copper layers should be regarded as inner copper layers.
- When producing output files, layer 0 should be included with each copper layer (not power plane layers), to include the pads with holes in them.
- The *2 Layer auto-router* can be used to route any two layers together. However, it is not aware of the tracks on any other layers, so is likely to insert vias that will cause short circuits. Use the auto-router as a double-sided router for layers 1 & 2, then route the remaining layers as single sided layers. (Refer back to chapter 10 for details on single-sided auto-routing.) Alternatively use the *Rip-retry router* to auto-route on upto 6 layers at a time.
- Ensure the extra layers are assigned as copper layers (*Edit* ⇒ *Plane Assignments*).

### Inner power plane layers

Internal power planes are an inner layer that is formed from a solid sheet of copper. All holes through the board are connected to the copper layer unless the holes have a clearance area formed around them. The clearance area is formed from an anti-pad that is larger than the hole - the pad is called an anti-pad because it is a pad in reverse, i.e. it is not a copper pad.

A sample power plane is shown in Figure 15. 2. The round white areas are the anti-pads around holes.

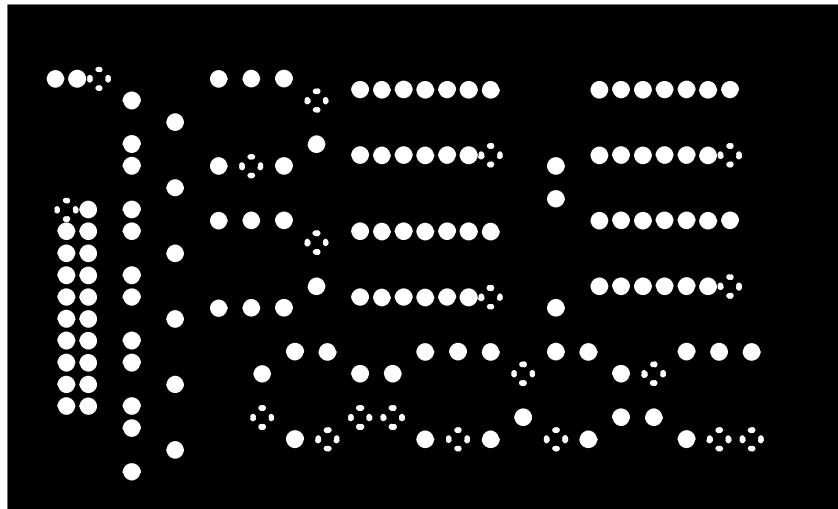


Figure 15. 2

The four straight clearance areas that surround other holes are heat-relief pads. They are not required for connectivity, but are added to stop the heat dissipating into the power plane when the board is soldered.

Internal power planes are generally specified before the artwork is started. They could be specified later, but it would mean removing any power tracks that had already been routed.

If you are using a copy of the completed worked example to practice on, you will need to delete the existing artwork. Ensure the correct job is open, then select **File ⇒ Delete ⇒ Delete all artwork**. Confirm the deletion when prompted.

You now have a job with all the parts placed on the board but nothing on the artwork that can be used.

So let's assume a design has been created, the circuit schematic drawn and converted into a parts/wiring list.

- Select **Edit ⇒ Define Power Names**. Ensure the power rails required as power planes are listed in this table, for example *VCC* & *GND*.
- Select **Edit ⇒ Plane Assignments**. Layers 0, 1 & 2 cannot be changed. On the worked example, layers 3 & 4 will be assigned as silk screen layers. The others are probably defined as copper layers, unless you have created additional silk-screen layers.

Select the arrow alongside a copper layer that has not so far been used, for instance layer 5 and change it to the power rail that you intend to create as a power plane (Figure 15. 3).

If you cannot remember which layers have been used, return to the artwork editor, select **View ⇒ Layer Properties** - any layers with 0 points are unused.

Layer	Usage	Layer	Usage
0	Drilled Pads	8	Copper
1	Copper	9	Copper
2	Copper	10	Copper
3	Silk Screen	11	Copper
4	Silk Screen	12	Copper
5	GND	13	Copper
6	Copper	14	Copper
7	Copper	15	Copper

Figure 15. 3

Repeat for the other layers to be converted to power planes, ie *VCC*. Select **OK** to return to the job design menu. Ranger now knows that the power rails *VCC* and *GND* (in this example) should not appear as tracks, but should be connected on the internal power plane layers.

- Check that the specified power rails are highlighted in the wiring list, which indicates that Ranger recognises them as power rails. Select **Circuit ⇒ Parts & wiring list**. Pop the wiring list window to the front, then scroll down through the list, until the power rails *VCC* and *GND* appear. If they are not highlighted return to the **Edit ⇒ Define Power Names** table and ensure the names have been entered correctly.

## ADDITIONAL INFORMATION

- Ensure that all the parts are placed on the board, **Artwork ⇒ Place Parts. All placed** should appear in the tool bar.
- Select **Artwork ⇒ Pad and Track Digitising**. Select **Digitise Part Pads**, followed by **Digitise Tracks** using the parameters required in order to route everything except the power rails that are being created as a plane.  
(Even if *digitise power* is selected, Ranger will not include the power rails that have been defined as power plane layers, but any other power rails will be included.)
- Route all the digitised connections as required. For speed in this example, use the auto-router.
- Once all the connections have been routed, you are ready to create the internal power planes.
- Select **Artwork ⇒ Pad & Track Digitising ⇒ Create Power Plane**. Individual power plane layers can be created, or all the power planes can be created at the same time. Ensure a tick appears alongside each of the power plane layers to be created (Figure 15. 4), then select **Create**. The window closes, select **OK**.



Figure 15. 4

- Load the artwork editor. Select **View ⇒ Layer Properties**. Notice that the layers assigned as power plane layers now have some points on them because the planes have been created. Make all the layers invisible (select **None** from the bottom of the *Visible* column), then make one of the power plane layers visible. Select **OK** to continue. Zoom in so that you can see the pads.  
You are looking at the power plane in reverse. The power plane layer will be manufactured as a solid sheet of copper, except for the pads that are displayed, which will form the clearance areas around holes. The round pads are "anti-pads". They are larger than the drilled hole they have been placed over. This means that the hole will not be connected to the power plane. The other pads are heat-relief pads. They maintain a connection between the hole and the copper area, to ensure that heat will not be dissipated through the plane when soldering to the pin that passes through the hole.
- The size of these pads is controlled by the **Edit ⇒ Sizes Table**.
- Anything that is added to a power plane layer forms a clearance area in the copper plane. For instance, a track can be added around the edge of the board (using the **Enter** menu) to stop the copper plane reaching to the edges of the board, or text (in relief) could be added for reference purposes on the films that are produced.
- Note: when the power planes are created, everything on the layer is deleted first to avoid errors occurring.
- When outputting the power plane layers, they should be output alone, layer 0 should **not** be output at the same time. The data that is output has to be photographically reversed in order to produce the required artwork.
- If the artwork is modified, the power planes **MUST** be created again.

### Connecting SMD's to the plane.

Surface mounted pads that should be connected to the plane are not automatically connected because there is not a hole through which the connection can be made.

There are two ways to make the connection, as follows:

#### Method 1:

- In the artwork editor (using **Enter**), add a track from the surface mounted pad on the relevant layer, to a conventional pad on the same power rail.  
The surface mounted pad will be connected to the plane via the piece of track and the heat-relief pad that is automatically added to the conventional pad. (The checking routines will flag any errors that are introduced.)
- Create the power planes once all the routing is complete.

#### Method 2:

- In the artwork editor (using **Enter**), add a track from the surface mounted pad on the appropriate layer, to an entered via hole.
- Complete all the routing.
- Create all the power planes.



If the artwork is viewed at this stage the added vias will have anti-pads around them, so the surface mounted pads will not be connected to the plane. (This is because the entered tracks and vias are not included in the artwork netcode file.) Check the artwork.

Connectivity errors should occur, but they should correspond to all the surface mounted pins that are not as yet connected to the plane.

- If any other errors are indicated, correct them and re-run the checking routines to ensure further errors were not introduced. Proceed only, when no other errors are reported.
- Create a new artwork netcode file (**File** ⇒ **Maintenance** ⇒ **Reconstruct netcodes**).
- Now generate the power planes again. The vias will now have heat-relief pads around them on the relevant layers.
- Re-check the artwork. There should be no errors.

### Reconstructing the netcodes

When the tracks are digitised, they have a unique internal *netcode* assigned to them. This allows features such as track hi-lite and copper fill to operate correctly.

If tracks are routed by the Spectra autorouter or entered in the artwork editor then a *null* netcode is assigned - Ranger2 recognises the piece of track but not which net it belongs to, for instance, VCC, CLOCK, GND, etc. If track hi-lite is used, the tracks are ignored, or if copper fill is used, the tracks are avoided even if they are attached to the selected datum point. Amended tracks can also lose their internal netcode, or you can mix them up (for instance moving a GND track onto a VCC pin).

A routine called *Reconstruct netcodes* is supplied in order to assign the correct internal netcode to any pieces of track that do not have one, or that have become incorrect. Before it is run, ensure that there are no short circuits on the design. Any short circuits will cause an invalid netcode to be assigned and therefore the hi-lite, fill, etc. routines will not operate correctly.

So, after correcting any short circuits, select **File** ⇒ **Maintenance** ⇒ **Reconstruct Netcodes**. From the window that appears as shown in Figure 15. 5, select **Start**.

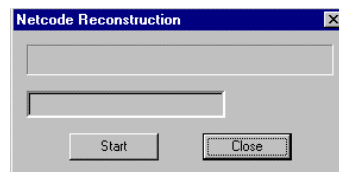


Figure 15. 5

When the message *Netcode reconstruction complete* appears in the window, select **Close**. All the tracks will now have valid internal netcodes assigned.

### Window Move, Rotate, Copy, Delete

The *Window* commands in the artwork editor can be used to move, rotate, copy and delete rectangular areas of the artwork. Individual layers or a selection of layers can be included in the window operations. If parts are moved, their new positions are automatically back-annotated to the part placement and parts list editors.

#### Defining the area and layers within the window

Select **Window** ⇒ **Define**, a green rectangle appears that represents the area to be defined. Select each corner of the rectangle to move it into the position required, so that the rectangle surrounds the area to be operated on.

#### Select the layers to be included

Select **Window** ⇒ **Rules**, a layer selection palette appears on the right-side of the display. Select the layers that are required to be included in the window operation, the \* alongside the layer indicates that layer will be included in the window. The words **All** or **None** can be selected to quickly select or deselect all the layers. Parts are included in the window operation if the layer they are on, is included. For example layer 0 will include parts on layer 0, layer 1 and/or 2 will include parts (SMD's) on layers 1 and/or layer 2.

#### Performing window move, rotate, copy

Select the window command required (**Window** ⇒ **Move/Rotate/Copy**) then select the new position required. When you are happy with the new position, select **Window** ⇒ **Go** to implement the change.

#### Performing window delete

Select the **Window** ⇒ **Delete**, followed by **Window** ⇒ **Go**, the contents of the window is removed.

## ADDITIONAL INFORMATION

---

### Changing the board profile

The board profile may be changed at any time by returning to the profile editor and making the changes required. The position of part pads can be displayed to assist in the modification (**View ⇌ Part Pads**).

If the profile is made smaller or moved, then the artwork should be updated to ensure all the parts, tracks, silk-screen, etc. remain within the profile. The *Window* commands in the artwork editor may prove helpful when moving areas of the artwork.

## Appendix

### MASTER DEVICE LIBRARY LISTING

#### ■ Volumes 0 - 19: TTL, CMOS

Volume 0	74XX	(IEC)
Volume 1	741XX	(IEC)
Volume 2	74XXX	(IEC)
Volume 5	CMOS	40XX & 40XXX
Volume 6	CMOS	45
Volume 7	CMOS	140XX & 141XX
Volume 8	CMOS	145XX
Volume 10	74XX	
Volume 11	741XX	
Volume 12	742XX & 743XX	
Volume 13	744XX & 745XX	
Volume 14	746XX & 747XX	
Volume 15	748XX	
Volume 16	741XXX & 742XXX & 748XXX	

#### ■ Volumes 20 - 29: Linear

Volume 20	Timers
Volume 21	Operational Amplifiers
Volume 25	A-D & D-A Converters
Volume 29	Special Function IC's

#### ■ Volumes 30 - 39: Semi-conductors

Volume 30	Bipolar Transistors
Volume 31	Field Effect Transistors
Volume 32	Diodes
Volume 35	Crystals

#### ■ Volumes 40 - 49: Passive Devices

Volume 40	Resistors
Volume 41	Variable Resistors
Volume 45	Capacitors

#### ■ Volumes 50 - 51: Electromechanical

Volume 50	Connectors
Volume 51	Test Points

#### ■ Volumes 60 - 69: Micros & related devices

Volume 60	Motorola 68XX & 68XXX
Volume 61	Intel 80X86
Volume 62	Zilog 8000
Volume 63	National 320XX
Volume 64	Rockwell 65XX
Volume 67	Peripheral ICs
Volume 68	PALS
Volume 69	Memory Devices



**A**

algorithms. *See* Auto-route  
 alignment markers, 64  
 allocate, 21  
 ambiguous bus spurs, 24  
 angle  
   45 degree, 30, 51  
   checking, 55  
   connections, 22  
 anti-pad, 27, 65, 73  
 aperture, 63  
 arcs. *See* Curves  
 artwork, 49  
 artwork netcodes. *See* Netcodes  
 artwork, printing. *See* Outputs  
 auto-pitch. *See* Grid  
 auto-route, 46, 49, 51, 52  
   costs, 49  
   digitising for, 46  
   final optimise, 53  
   interim optimise, 53  
   layers, 49, 72  
   memory, 52  
   minimum clearance, 49  
   orthog1, 52  
   orthog2, 49, 52  
   rip-retry router, 53, 72  
   routing algorithms, 49, 50, 52, 53  
   routing resolution, 49  
   setup, 49  
   time limit, 49  
   track sizes, 47  
 auto-save, 3

**B**

back annotation  
   after gate/pin swapping, 44  
   after part renumbering, 72  
   from artwork editor, 70, 75  
 backing up, 33  
 board profile, 30  
 bottom side. *See* Layer  
 bus name, 22  
 buses, 22

**C**

changes. *See* Modifications and Tips  
 checking, 24, 28, 52, 55, 59  
 circles, 11  
 circuit. *See* Schematic  
 clearance. *See* Checking  
 code field, parts list, 34  
 colours, 17, 58  
 component outlines, 38  
   adding pins/pads, 38  
   datum, 40  
   field in parts list, 24, 34  
   free text, 41  
   in schematic parts, 10  
   names, 38  
   pin numbers, 39  
   silk-screen, 39  
   surface mounted, 40

viewing, 6  
 x/y readout, 39  
 component side. *See* Layer  
 connection, 21, 43. *See* Signal and Power  
   length, 43  
   trace, 56  
 connection list. *See* Parts/wiring list  
 connectivity, 52  
 connectivity errors, 56  
 Cooper & Chyan. *See* Spectra  
 co-ordinates. *See* X/Y readout  
 copper areas, 59  
 copy  
   devices from master library, 17  
   devices to master library, 24  
   job, 15  
   outlines to/from master library, 41  
 costs. *See* Auto-route  
 creating a new job/design. *See* Job creation  
 curves, 11  
   tracks, 52, 55  
 customer name, 2  
 cutter. *See* NC routing tool

**D**

datum, 12, 30, 38, 40, 41, 42  
 dcodes  
   description, 63  
   table, 65  
 defaults  
   design, 26  
   master, 26  
 delete  
   artwork, 53  
   parts/connections/tracks, 71  
   router data, 53  
 description, parts list, 34  
 design creation. *See* Job creation  
 design rule checking. *See* Checking  
 device  
   component outline name, 10  
   creation, 9, 13  
   equivalent pins, 15  
   equivalent symbols, 13, 14  
   name, 9  
   pin numbers, 12, 15  
   power pins, 13, 14  
   properties, 9, 10, 13  
   rotating, 13, 15  
   saving, 13, 15  
   terminals/pins, 12, 15  
   text, 12, 14  
   text height, 10  
   value, 10  
 digitise, 41, 46, 52  
   for auto-routing, 46  
   for manual routing, 47  
   pads, 46, 47  
   power planes, 74  
   remaining, 56  
   tracks, 46, 47  
 disk space, 1  
 double-sided. *See* Layer and Auto-route, layers

draw, 10, 63  
 drill drawing. *See* Outputs  
 drill hole, 27, 28, 40, 62  
 duplicate connections, 47

**E**

edge connectors, 40  
 emulsion up/down, 65  
 equivalent pins, 15  
 equivalent symbols, 13  
 errors. *See* Checking  
 Excellon, 68

**F**

filled areas, 59  
 filter, 18  
 final optimise. *See* Auto-route  
 find part in parts list, 25  
 fixing holes, 71  
 fixing/unfixing  
   parts, 45  
   tracks, 53  
 flags, 55, 56  
 flashing, 63  
 flipping. *See* Parts  
 footprints. *See* Component outlines  
 function keys, 5

**G**

gap. *See* Checking  
 gate swapping. *See* Swapping  
 gerber. *See* Outputs  
 gold plated edge connectors, 40  
 grid, 11, 18, 30, 31  
   auto-pitch, 11, 18  
   non-standard, 40  
   snapping, 11

**H**

hardware, 1  
 heat-relief pad, 27, 65, 73  
 hole. *See* Drill hole  
 HP-GL, 61

**I**

ID, 34  
 ident position, 12, 41, 57  
 inch, 26  
 inner layers. *See* Layers  
 installing, 2  
 interim optimise. *See* Auto-route  
 internal netcodes. *See* Netcodes  
 invalid entries. *See* Parts/wiring list

**J**

job, 4  
 job creation, 8  
 job index (job.idx), 2  
 junction blobs, 22

**K**

keepout, 33, 59, 62, 64

**L**

labels, 41, 56

layer

- 0, 1 and 2, 28, 39
- adding pads to, 38
- assignments, 28
- auto-routing, 53
- bottom side, 28
- component side, 28
- double-sided, 26, 49, 50
- inner copper, 51, 72
- inner power plane, 73
- multi-layer, 26, 51, 72
- output, 62
- properties, 58
- silk-screen/text, 56, 57
- single-sided, 26, 49, 50
- solder side, 28
- swapping, 51, 72
- top side, 28

library

- job device, 9
- job outline, 38, 41
- master device, 9, 17
- master outline, 38, 41
- masters, 8

license, 1, 2

line/pad digitising. *See* Digitise

lite, 1

load a job, 4

lock, 22

**M**

macros, 24

manual routing, 50

- digitising for, 46, 47
- layers, 50, 72
- multi-layer, 72
- track sizes, 47, 51, 59

master library. *See* Library

master part code. *See* Part code

master sizes table. *See* Sizes

maximum copper. *See* Filled areas and Power (plane)

memory. *See* Auto-route

metric, 40

minimum clearance. *See* Filled areas.

*See* Checking and Auto-route

mirror, 20, 44, 65, 67

mm, 26

modifications, 70. *See* Tips

- adding parts and/or connections, 70
- moving parts, 70
- outlines, 41
- removing parts and/or connections, 71
- renumbering, 71
- Specctra, 54

mounting holes, 71

multi-layer. *See* Layer and Auto-route, layers

multiple connections, 47

**N**

NC drill, 60, 68

NC routing tool, 33

necking, 51

net lists. *See* Parts/wiring list

netcodes, 54, 75

no. of pins on package, 10

no. of power pins, 10

no. of symbols for device, 10

node, 25

non-electrical, 23

numbers

- pin in schematic, 21
- pin in schematic part, 12
- pin, in outline, 39

**O**

on-line help, 1

open job, 4

optimise. *See* Auto-route

orthog1. *See* Auto-route

orthog2. *See* Auto-route

orthogonal connections, 22

outputs

HP-GL, 60

outlines. *See* Component outlines

outputs

artwork pen plotting, 61

artwork photoplot, 64

artwork printing, 67

dcodes, 63

drill drawing, 62

general, 60

Gerber, 63

Houston Instruments, 60

layer selection, 62, 64

layer selection, soldermask, 66

mirror, 65, 67

NC drill, 68

penplotter configuration, 60

penplotters, 60

photoplotter (Gerber), 63

power plane layers, 62, 64

printers, 66

scale, 61, 62, 63, 67

schematic penplotting, 61

schematic printing, 66

soldermask, photoplot, 65

soldermask, printing, 68

target markers, 64

**P**

package outline name, 10

pad

deleting, 39, 41

digitising. *See* Digitise

display, 43, 58

hole, 27, 39, 40

replacing, 41

sizes, 27. *See* Sizes, table

smd, 28

swell. *See* Outputs, soldermask

page

multiple, 23

selection for printing, 61

size, 17

panning, 11

part code

master table, 29

prefix, 10, 29

table, 29

part placement, 43

parts

adding, 17, 70

fixing/unfixing parts, 45

flipping, 20, 44

labels. *See* Labels and Silk-screen

position, 42

removing, 44, 71

renumbering, 71

rotation (device), 13, 20

rotation (outline), 43

parts/wiring list, 34

back-annotation, 44, 71

compiling from schematic, 24

find part, 25

gate/pin swapping, 34

invalid entries, 24

power rails, 36

repeating parts, 35

saving, 36

viewing, 24

pen plotter. *See* Outputs

photoplotters. *See* Outputs

pin swapping. *See* Swapping

pins

adding to outline, 38

adding to schematic part, 12

assigning number in device, 12, 15

assigning numbers in schematic, 21

identifying number, 39

modifying, 39

number in outline, 39

number positions, 12

plane assignments. *See* Layers

points, 58, 73

power

auto-routing connections, 50, 53

connections, digitising, 46

in wiring list, 36

names, 27

pins in devices, 14, 25

plane, 28, 47, 54

plane output, 62

rail connections, 25

rail definition, 27

track size, 26

prefix. *See* Part code

primary routing layer, 51

printing. *See* Outputs

product license. *See* License

profile, 30

PSpice. *See* Spice

**R**

RAM, 1

Ranger1, 2

Ranger2 for DOS, 2

reconnect  
   power, 44  
   signals, 43  
 red, 25, 40  
 reference designators, 21, 41  
 registration. *See* License  
 renumbering, 71  
 repeat parts, 35  
 resolution, 26  
 rip-up/retry autorouter. *See* Auto-route  
 router (NC)  
   definition, 33  
   output, 68  
 routing  
   algorithms. *See* Auto-route  
   automatic. *See* Auto-route  
   manual, 50  
 rwidth, 65

## S

saving, 1, 2, 13, 15, 21, 33  
 scale, 12, 61  
 schematic, 17  
   adding connections, 21  
   adding parts, 17  
   assigning part/pin numbers, 21  
   buses, 22  
   compiling parts/wiring list, 24  
   device symbol, 10  
   devices, creation, 9  
   devices, multi-symbol, 9  
   devices, one symbol, 9  
   gate/pin swapping. *See* Swapping  
   macros, 24  
   modifying connections, 22  
   multi-sheet, 17, 23  
   non-electrical, 23  
   part values, 21  
   printing. *See* Outputs  
   rotating, flipping, and moving parts, 20  
   search volumes, 18  
   sheet size, changing, 17  
   signal names, 23  
   unallocated parts, 24  
   window operations, 23  
 secondary routing layer, 51  
 short circuits, 56  
 sides. *See* Layer  
 Sieb & Meyer, 68  
 signal  
   connections routing, 53  
   connections visibility, 43  
   name, 22, 23, 25  
   track size, 26  
 silk-screen  
   checking, 28  
   generation, 56  
   ink, 38  
   labels, 41  
   layer assignment, 28  
   outline, 38  
   outline definition, 39  
 single-sided. *See* Layer and Auto-route, layers  
 sizes  
   changing track size, 51  
   in outline, 41  
   master table, 27  
   modifications, 59  
   pads/tracks in use, 59  
   table, 26, 41, 46  
   table changes, 42  
 SMD's, 39, 40, 51, 54, 62, 74  
 solder side. *See* Layer  
 soldermask. *See* Outputs  
 Spectra, 1, 2, 53  
 Spice, 10  
 subnet, 56  
 surface mounted component. *See* SMD's  
 swap group. *See* Swapping  
 swapping  
   back-annotation, 44  
   define equivalent pins, 15  
   defining equivalent symbols, 14  
   enabling, 10, 24  
   gate/pin on layout, 44  
   links, 34  
 swidth, 65

## T

target markers, 64  
 terminals. *See* Pins  
 text  
   free in schematic parts, 12  
   height, schematic parts, 10  
   in outlines, 41  
   non-electrical, 23  
   on copper layers, 57  
   schematic devices, 12  
   silk-screen, 56  
 through plated holes, 6

tips, 29, 41, 45, 47, 51  
 title bar, 8  
 tooling holes, 71  
 top side. *See* Layer  
 trace, 56  
 tracks  
   arcs. *See* Curves  
   digitising. *See* Digitise  
   fixed/unfixed, 53  
   routing/modifying, 50  
   sizes. *See* Sizes, table  
   un-routed, 56  
 tray, 17

## U

unallocated, 24  
 undrilled, 27  
 unfilter, 18  
 units, 26  
 unused pins, 25

## V

value  
   changing on schematic, 21  
   default, 10  
   position, 12  
 vias, 27, 28, 39, 51, 53, 66  
 viewed from the top/bottom, 38  
 visibility, 43  
 volumes, 17

## W

window operations  
   artwork, 75  
   schematic, 23  
 wire lock, 22  
 wiring list. *See* Parts/wiring list  
 worked example, 1

## X

x/y  
   position of parts, 35, 42  
   readout, 31, 39  
   shifts outputs, 64

## Z

zoom in/out, 4, 11

**Seetrax CAE Ltd, 28 Vine Farm Close, Poole, Dorset, BH12 5EJ, England**

**Telephone: +44 (0)1202 528686**

**Fax: +44 (0)1202 528848**

**Email: [sales@seetrax.com](mailto:sales@seetrax.com) or [support@seetrax.com](mailto:support@seetrax.com)**

**Web-site address: [www.seetrax.com](http://www.seetrax.com)**