DESIGN TOOLS
PCB DESIGN
& MAKE

CAD/CAM Software for
Designing and Making PCBs

TECHSOFT

USER GUIDE

CONTENTS

TechSoft UK Ltd.,
The Grange,
Eryrys,
Mold,
Denbighshire, CH7 4DB
U.K.

Tel : +44 (0)1824 780318
Fax : +44 (0)1824 780564
Email : email@techsoftuk.co.uk
Web site : www.techsoftuk.co.uk

The program described in this manual is subject to continuous development and improvement. All information of a technical nature and particulars of the program and its use (including the information and particulars in this manual) are given by TechSoft UK Limited in good faith. However, TechSoft UK Limited cannot accept any liability for any loss or damage arising from the use of any information or particulars in this manual.

N.B. PCB Design & Make can be used with a computer running any screen resolution. However, at low resolutions (eg., 640 x 480), the drawing screen may feel cramped, and pads, tracks, etc., will seem coarse. TechSoft recommend a minimun screen display of 800 x 600 x 256 colours. (Higher screen resolutions will also improve the appearance, and enhance the use of many other software packages such as word processors, spreadsheets, etc.)

Printed: May 2000

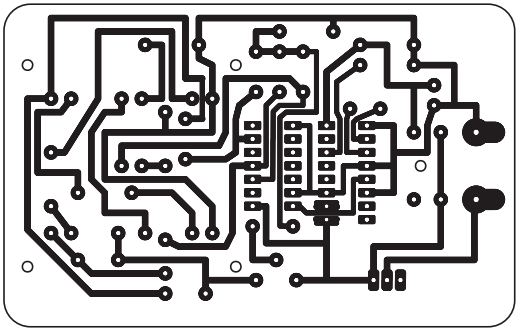
1 INTRODUCTION

Software Overview

Design Tools consists of a suite of integrated software packages developed to provide solutions for designers in many different areas. The PCB elements of the Design Tools Suite are PCB Design and PCB Make. These may be supplied separately as individual packages, though most users will have them supplied together as one integrated package - PCB Design and Make.

PCB Design

This is an easy to use PCB draughting layout and editing program. Its clear and logical interface make it ideal for the inexperienced user, whilst its advanced editing facilities give it the power to tackle very sophisticated circuit board layouts.

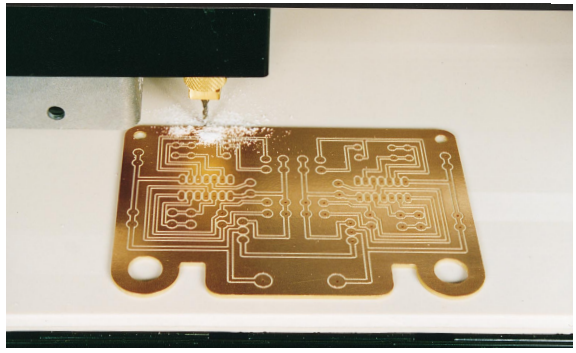


PCB Make

This is dedicated PCB manufacturing software. It can produce manufactured outputs to two distinct types of machines, vinyl cutting machines and engraving/milling machines.

The output to vinyl cutting machines (eg., Roland CAMM 1, STIKA, etc.), allows the circuit to be “cut” out of Cutronics self-adhesive copper foil. The foil circuits can then easily be transferred to any suitable substrate, eg., card, wood, plastic, etc. This is an ideal quick and cheap way to make simple circuits with relatively large pads and tracks. It also creates interesting new design opportunities. Projects can be created that require no separate circuit board - the circuit could be applied directly to the case, for example. The creation of ‘flexible’ circuits or circuit elements, is now possible.

The output to engraving/milling machines (eg., Roland CAMM 2, CAMM 3, MODELA, etc.), creates a circuit by engraving around each circuit path with a fine point tool onto conventional copper clad PCB board. If suitable tools are available on the machine, the software will also allow the holes to be drilled and the circuit board profile to be milled out. Thus it is possible to take a fully finished board from the machine, drilled and ready to solder up.



2 INSTALLATION

The software can be installed on any computer running Windows 95, 98, NT4, 2000, etc. The installation set supplied consists of a CD ROM and a floppy disk. The floppy disk contains your registration details and will be required during installation. **Keep this disk safe, as it may be required for future upgrades.**

The release CD ROM allows direct installation onto your target computer. If your target computer does not have a CD ROM drive fitted, the CD may be used with any computer that does have a suitable drive, to create an installation set of floppy disks. These floppy disks can then be used to install the software on the target computer.

On insertion of the CD into the drive, the installation program will normally auto-start. If the installation disk fails to start automatically, choose *Start > Run*, then type d:\setup (adapt this as appropriate if using a CD ROM drive other than d) and press ENTER.

When the install program starts, a splash window similar to that below will be displayed.

If installing directly onto the computer in use, choose the *Install NOW from CD* button, then follow the on-screen prompts.

If you wish to create a floppy disk installation set, choose the *Create floppy install disks* button then follow the on-screen prompts. When the floppy disk set has been created, install the software as follows:

1. Insert disk 1 in a floppy drive.
2. Choose *Start > Run*.
3. Type a:\setup (adapt this as appropriate if using a floppy drive other than a) and press ENTER.
4. Follow the on-screen instructions.

3 STARTING THE PROGRAM

Choose *Start > Programs > TechSoft Design Tools > PCB Design and Make (or Make or Design)*, or use the desktop shortcut icon.

Users of PCB Design can omit chapter ***7 - PCB Make.***
Users of PCB Make can omit chapters ***5 - PCB Design Tutorials*** and ***6 - PCB Design Notes.***
Users of PCB Design & Make should read all chapters.

4 DESIGN TOOLS HELP

Help is a standard Windows feature supported by a wide range of software packages. Design Tools Help gives you easy access to detailed information on every menu item, tool, dialog box, button and feature in PCB Design & Make.

You can access *Help* in the following ways:

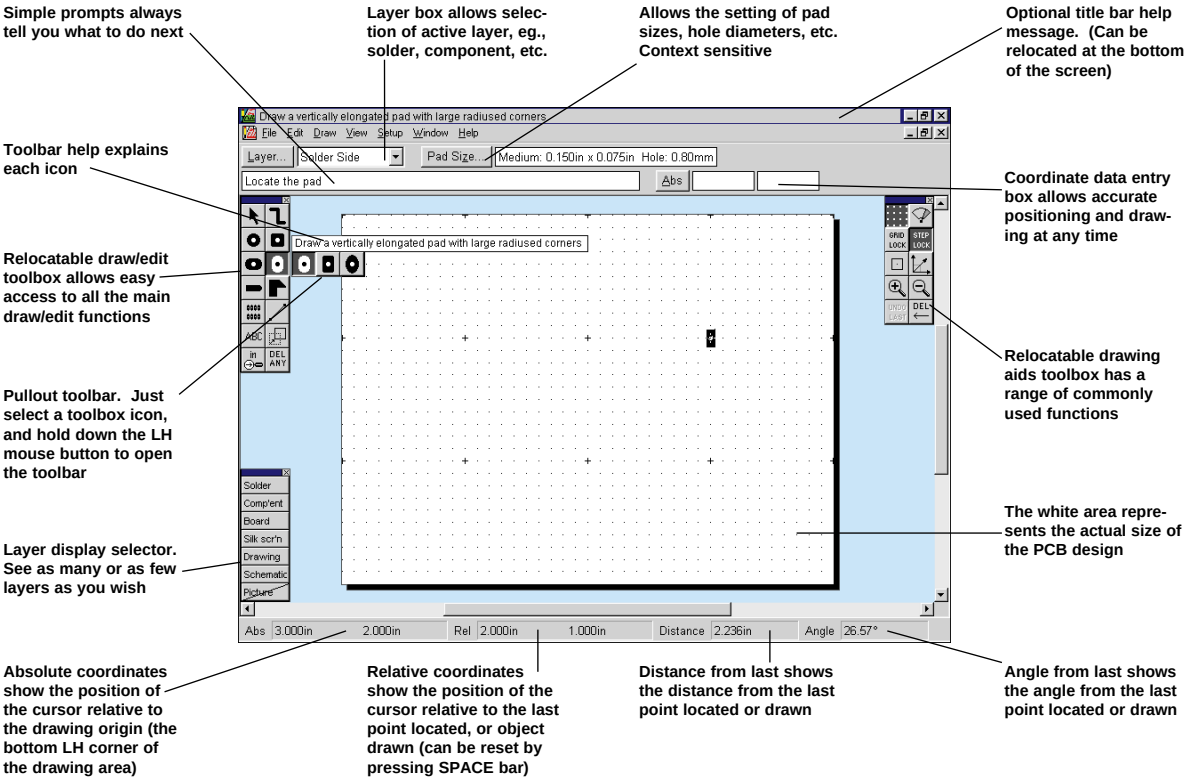
1. Use *Help > Help Topics*.
2. Click on the *Help* button in a dialog box for help on that dialog box.
3. With any menu item highlighted, press the *F1* key to see a Help window describing that menu item.
4. Press *Shift + F1* to display the Help cursor. Then click on any tool to see a Help window describing it.

5 PCB DESIGN TUTORIALS

TUTORIAL 1 - Screen Layout, Menu Selection, etc.

The PCB Design program window provides a clear view of the current layout, various information areas, and a selection of icons with tools to cover the most common design, editing and display functions. The main features of the program window are shown below. Although most users will stay with the standard layout, it is possible to use *Setup > Customise* to change not only the appearance, but also the operation of many of the software's features. For example, it is possible to have larger icons, help messages and prompts, user definable screen colours, etc. We do however recommend using the standard layout for the rest of these tutorials!

1. Start up PCB Design (see ***Starting the Program*** page 5) and familiarise yourself with the screen layout as shown below.



2. Menu items are normally chosen from the menu bar or the toolbox, using the mouse (although keyboard alternatives are available). To choose an item from the toolbox, position the pointer

over the appropriate icon and click the LH mouse button. Some items in the RH toolbox (the Drawing Aids toolbox), such as *Grid*, will cause the icon to stay selected (on) until it is chosen again.

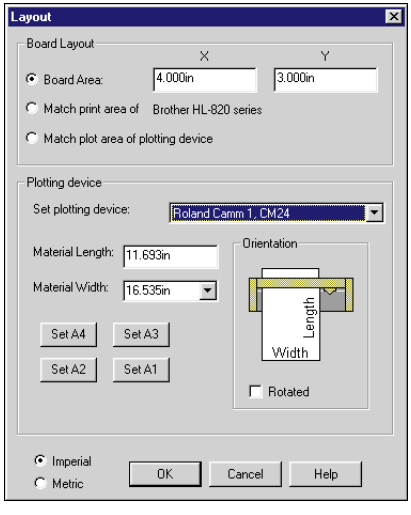
3. Many items in the LH toolbox (the Draw/Edit toolbox) have pullout toolbars. These are activated by positioning the pointer on the icon then pressing and holding the LH mouse button for a short while. The pointer can then be dragged along the icon bar until the required icon is highlighted, then the mouse button released. This selects the item, and changes the icon in the toolbox to that chosen.

4. Most menu items, eg., grid, pads, etc., have an associated dialog box for settings. To access these dialog boxes, double click on the appropriate icon with the LH mouse button, or click on the icon with the RH mouse button. (The associated dialog box for pads and tracks can also be accessed by clicking on the *Pad Size*/*Track Width* buttons on the attributes bar at the top.)

TUTORIAL 2 - Setting Up the Software to Suit Your System

It is possible to customise the look, feel and functionality of the software in many ways. For most users, however, the first priority is to set the PCB design size (the *Board Area*) and to set the correct plotting device. This is what this tutorial deals with. Once you have set these to suit your normal requirements, you will then save these values as the default.

1. Start up PCB Design. Select *Setup* > *Drawing* > *Layout*. A dialog box similar to the one shown below will open.



2. Use the dialog box to set both the design size of the board (the *Board Area*), and the plotter/cutter/miller you will be using most often. (The plotter/cutter/miller is only relevant to PCB Make users.)

   Most users will want to work on a "sensible" sized board, 4 x 3 inches is the default value. It is, however, possible to set the *Board Area* to match the printer or plotter to be used. This is particularly useful if making a lot of small circuits so that they may be "fitted onto" the maximum size available. For tutorial purposes, ensure that the *Board Area* is set to 4 x 3 inches.

   When using *Set plotting device* to set your particular plotter/cutter/miller, the dialog box will change to reflect the plotting device chosen.

3. You may wish to investigate some of the other options under *Setup > Drawing* or *Setup > Customise.* Any of the options set may be saved as default values in the next step. At this stage however, it is probably better to leave these options, and to come back to them after working through the other tutorials.

4. Select *Setup > Set As Default*. Click *OK* on the warning dialog box. Close down the software completely, then restart it. You will see that it starts up with the *Drawing Layout* settings you have made. (Any other settings made from the *Setup* menu, eg., the type of grid, will also have been saved.)

You may wish to use more than one setup, eg., if you have two or more different output devices. For example if using both a CAMM 1 and a CAMM 2, you could set up for your CAMM 1, then instead of using *Set As Default,* you could choose *Setup > Save Setup,* and give the setup a name such as "Setup for CAMM 1". You could then save a suitable setup for the CAMM 2 in the same way. On subsequently loading the software, the first action would be to load the correct setup for the work in hand using *Setup > Load Setup*.

N.B. If you ever wish to return to the factory default settings, select *Help > Restore Factory Defaults*, then *Setup > Set As Default*.

TUTORIAL 3 - Basic Circuit Drawing

1. Choose the *Round Pad* icon.

Move the cursor around, over the white board guide. Notice the changing coordinates at the bottom of the screen. Move the cursor to a position where you wish to position a pad, and click the LH mouse button. Repeat, dropping a few more pads here and there.

2. Choose the *Grid Lock* icon.


Now draw a few more round pads. The pads will now be fixed to grid positions (the small crosses on screen). As the default grid spacing is 1inch, the movements will be in very large steps.

3. Choose the *Step Lock* icon.


Now draw a few more round pads. The pads will now be fixed to step positions (the small dots on screen). These are set by default to 0.1inch. For most PCB design work it is suggested that the *Step Lock* is permanently set on. Double click on the *Step Lock* icon with the LH mouse button and a dialog box will appear. This allows the grid and step spacing to be changed. Many other icons have dialog boxes accessed in the same way. (Alternatively, dialog boxes may be accessed by clicking the icon with the RH mouse button.)

4. Choose the *Square Pad* icon and position a few more pads.

Note that when any pad icon is chosen, the pad size is displayed at the top of the screen. Click on the *Pad Size* button to open the *Pad Size* dialog box. In this dialog box you may select pre-set sizes or choose *Custom* and set your own size. (You may also access the *Pad Size* dialog box by double clicking on the pad icon with the LH mouse button, or by single clicking with the RH mouse button.)

5. Move the pointer over the *Horizontally Elongated Pad* icon.

Press and hold the LH mouse button. This will activate the pullout toolbar. The toolbar contains a number of similar pads. Slide the pointer along the toolbar to highlight each pad type in turn. The toolbar help above the toolbar describes each pad type. Choose a pad type by highlighting its icon, then releasing the mouse button. The chosen pad type will now be active in the main toolbox, and may be chosen as normal. Draw a few horizontally elongated pads of different types.

   There are other pad shapes namely *Vertically Elongated Pads* and *Edge Connectors* in the main toolbox - you might also like to experiment drawing some of these.

6. Choose the *Linear Track* icon.

Click the LH mouse button to start a track. Move the mouse to a new position and click

again. Move the mouse again and click. To complete a track, double click the LH mouse button (to fix the moving track) or click the RH mouse button (this will finish at the last fixed track). Note that when any track icon is chosen, the track width is displayed at the top of the screen. Click on the *Track Width* button to open the *Track Width* dialog box. You may select a standard width or enter any width you wish. (You may also access the *Track Width* dialog box by double clicking on the track icon with the LH mouse button, or by single clicking with the RH mouse button.)

7. Choose the *Zoom In* icon. 

Note the prompt reading *Locate one corner of zoom box*. Move the cursor approx. 0.5inch above and to the left of any "corner" that you have drawn, then click the LH mouse button. The prompt will change to *Locate opposite corner of zoom box*. Move the mouse to pull out a box down and to the right, about 1inch square, then click the LH mouse button. That area will now be redrawn to fill the screen. Try drawing some more tracks. Working "zoomed in" can be very helpful when working on small details.

8. Move the pointer over the *Zoom* icon. 

Press and hold the LH mouse button to activate the pullout toolbar. Slide the pointer along the toolbar to the *Zoom Last* icon then release it. This will restore the previous zoom level, in this case the full drawing screen. (You might like to try the effects of the other items in the *Zoom* toolbar, and also the *Zoom+* and *Zoom-* icons in the RH toolbox at this point.)

9. Choose the *Radial Lock* icon. 
Draw some more tracks. It will be seen that the track being drawn is constrained to 45° steps (by default). Choose the icon again to turn *Radial Lock* off.

10. Move the pointer over the *Linear Track* icon. 
Press and hold the LH mouse button to activate the pullout toolbar. Slide the pointer along the toolbar to the *Compound Track* icon then release it.

When you choose this icon a new toolbox will appear in the bottom right of your screen.  The first icon in this toolbox functions in a similar way to the *Linear track* tool. The next two icons allow curved tracks and arcs to be drawn. It is possible to change the style mid track by simply choosing the appropriate icon. Experiment to see how versatile these track drawing tools are.

11. Choose the *Linear Track Area* icon. 
Draw a simple shape such as a rectangle. Click the LH mouse button to fix each corner and either double click the LH mouse button or single click the RH mouse button to finish.

12. Choose the *Compound Track Area* icon from the pullout toolbar.

This will activate a similar toolbox to *Compound Tracks*, facilitating complex shapes.

If you draw a shape, then you choose the right most icon in this toolbox, by continuing drawing you can create an 'island', ie., an empty space within the shape. Have a try.

13. Choose the *Component* icon.

The *Load Component* dialog box will open. Load the component "Dil08" from the "Dils" library directory. Dil08 is the pin layout and schematic for an 8 pin dual in line chip (eg., 555, 741, etc.). The DIL will initially be coloured pink, and will be surounded by a dotted box with several small yellow handles (a marquee box). The fact that it is pink shows that it is "selected". Click on the centre handle. As the mouse is moved the DIL will move with it. Move the DIL to a clear part of the screen, then click the LH mouse button to "drop" the DIL at its current position. To "de-select" the DIL move the cursor to a clear area of the screen away from the DIL and click the LH mouse button. (More details of the functions of the marquee box, including simple moving, mirroring and rotating, are given in ***Tutorial 6**.)* Once deselected you will note that the pads are coloured blue, whereas all pads drawn before are red. These colours denote whether the pads (or tracks) are drawn under or on top of the board (see ***Tutorial 4*** for further details).

14. By now your screen may be rather cluttered so this is a good time to look at deleting. Simple mistakes may be undone by clicking on the *Undo Last* icon in the RH toolbox. This undoes the last operation, either drawing, transforming or deleting. Once undone, the icon becomes *Redo Last*. Choosing this effectively undoes the last undo. *Redo Last* is cleared by any further drawing or deleting operation. Also in the RH toolbox is *Delete Last*. This deletes back one object each time it is selected. *Undo Last* restores a deleted object, but only the last one deleted! Most other delete functions are in the LH toolbox, where there is *Delete Any*, *Delete Part*, and *Delete Inside Box*. *Delete Any* allows you to point to an object and delete it. *Delete Part* deletes any part of a track, line, arc, etc., back to the closest intersection points to the locating point. Finally, pressing *Ctrl + Delete* on the keyboard deletes the current selection (selection will be dealt with in ***Tutorial 6***), pressing *ALT + Delete* on the keyboard deletes the whole of the current drawing. Try some of the *Delete* options now.

TUTORIAL 4 - Layers

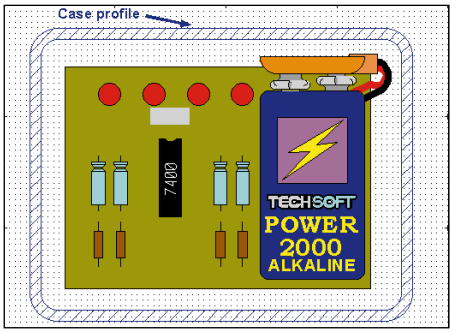
So far, most of the work done will consist of red pads and tracks. Red is used to denote objects which are on the underside of the board, referred to as the '*Solder Side*'. In order to design more complex layouts and include information other than pads and tracks, you will have to work with layers. Remember, whatever layer you are working on, your viewpoint is un-changed. You are always seeing the board from above, referred to as the '*Component Side*'. It may help to think of the board as a piece of glass and the layers as acetate sheets either side of it. Double sided boards have tracks and pads on both *Solder Side* and *Component Side* layers. The example below will illustrate the use of different layers.

1. Choose *File* > *Close* to clear any work you have done so far, then choose *File* > *New* to start a new drawing.

2. Choose any *Pad* icon then click on the downward arrow on the 

attributes bar and select a layer. You can opt to place pads on either side of the board, or both. Pads on the component side of the board are shown green and those on both sides, blue. You can also place tracks on either side, (but you cannot draw tracks on both layers at the same time), these too are coloured red or green accordingly. Try drawing some pads and tracks on different layers.

3. Choose *File* > *Close* to clear any work you have done so far, then choose *File* > *Open* and load the file "PCBTutor1.dtb" from the Tutorial directory.



4. This drawing is made up of several layers. You will see that initially all the layers are visible. Use the *Visible Layers* toolbox in the bottom LH corner to turn them on and off individually to get a clearer view of each. Any layer with a line through, like the *Picture* layer shown opposite, will be invisible. (N.B. Not all layers have any drawing on them.)

As you can see with this sample file, there are other details you might wish to include in a PCB Design in addition to pads and tracks. These can be placed on a selection of other layers:

The *Board Profile* layer is used to show the outline of the PCB and details things such as slots, mounting holes, etc. When using PCB Make to manufacture the board, drawing elements on this layer are used as the tool path for cutting out the board.

The *Silk Screen* layer shows the position of the components on the board. Silk screen printing is often used in industrial practice to show assembly workers where components should be positioned and to add company identifiers, serial numbers, product codes, etc. In school project work the *Silk Screen* layer can be printed onto paper to act as a guide to assembly.

The *Drawing* layer can be used to show other details related to the PCB, details of the enclosure, or adjacent objects for example. It can also be used to show drawings imported from 2D Design, etc.

The *Schematic* layer can be used to display the schematic diagram of the circuit alongside the PCB to facilitate checking, fault finding, etc.

The *Picture* layer can be used to show realistic illustrations.

If all the information is displayed at the same time the screen can be very confusing. The great advantage of using layers is that they can be turned on and off at will so that you only see what you need to see.

TUTORIAL 5 - Drawing Tools

Drawing tools can be used to create the board outline, show the size, shape and position of components or add other details to your PCB such as mounting holes. They can also be used to show related objects such as enclosures and 'off board' components.

1. Choose *File* > *Close* to clear any work you have done so far, then choose *File* > *New* to start a new drawing.

2. Choose the *Single Line* icon. 
   Note that the *Layer* has changed automatically to *Silk Screen*. If you wish to draw on a different layer, you may change it, but note that you cannot draw on either the *Solder Side* or *Component Side* layers. Try drawing a few lines.

3. Position the pointer over the *Single Line* icon then press and hold the LH mouse button to activate the pull out toolbar.

   Experiment with the other drawing tools. The drawing tool functionality should be self explanatory if you follow the prompt messages.

The tools here are actually some of the more commonly used drawing tools from Design Tools - 2D Design. If you need more elaborate drawing facilities then remember you can use 2D Design to create your drawing then bring it into PCB Design. See ***Importing Drawings*** page 41.

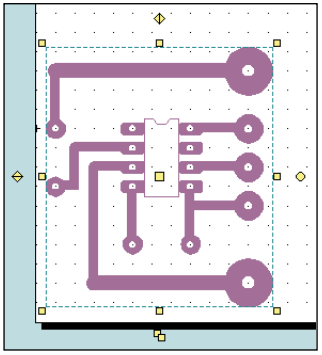
TUTORIAL 6 - Object Selection and the Marquee Box

As you will see in this tutorial, selection is carried out using normal Windows methods. Selected objects may be dragged, rotated, flipped, resized or copied quickly and easily.

1. Choose *File* > *Close* to clear any work you have done so far, then choose *File* > *Open* and load the file "PCBTutor2.dtb" from the Tutorial directory.

2. Choose the *Select* icon.

Move the cursor above and to the left of the circuit. Press the LH mouse button, drag a box around the whole circuit, then release (alternatively *Select All* using the keyboard shortcut *Crtl + A*). The circuit will be redrawn in pink, and surrounded by a dotted box (a marquee box) with yellow handles.



3. Ensure that *Step Lock* is on, then click on the yellow handle in the centre of the marquee box with the LH mouse button. As the mouse is now moved, the circuit will be moved around the screen. To "drop" the circuit, click the LH mouse button again.

4. Familiarise yourself with the marquee box functions, as described below.

   Clicking on the top diamond handle will "mirror" the circuit left right. Clicking on the left diamond handle will "mirror" the circuit up down.

   Clicking on the bottom "overlapping sheets" handle will produce a quick copy of the circuit.

   Clicking on the circular handle to the right will rotate the circuit. With *Radial Lock* on, this will be in fixed steps (45 deg. by default).

   Clicking on any of the edge or corner square icons will stretch, expand, or contract the circuit as appropriate. (**Warning** - moving these handles may alter the aspect ratio of the circuit and

will alter the spacing of pads (including items such as Dils). These functions should only be used with great caution. If they are used accidentally, *Undo Last* will restore the drawing.)

5. An individual object (track, pad, etc.,) can be selected by moving the pointer near it and clicking the LH mouse button. Clicking on another object in the same way will select the new object (de-selecting the first object). Clicking on a second object with the RH mouse button (or SHIFT +LH mouse button) will "add" or "remove" the object from the selection by toggling its select state.

   Multiple objects can be selected by dragging a select box around them with the LH mouse button. Dragging another select box in the same way will create a new selection (cancelling the first). Dragging a select box with the RH mouse button (or SHIFT + LH mouse button), will toggle the select state of the objects.

   To de-select objects, move the pointer outside the marquee box and click the LH mouse button. Individual objects may be de-selected using the RH mouse button (or SHIFT + LH mouse button) as described above.

   Try all of the above options to get the feel of the selection process.

   N.B. You will notice that when any part of the DIL is selected, the whole thing is automatically selected. This is because the DIL is grouped. This means that it will be treated as a single object. This makes such items much easier to move around, etc. To group objects, first select every object to be grouped, then use *Edit > Group* (or *Ctrl+G*). To ungroup objects first select them, then use *Edit > Ungroup* (or *Ctrl+U*).

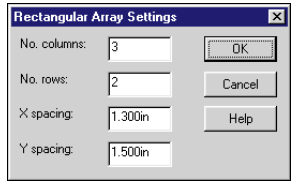
TUTORIAL 7 - Transformations

You have already tried simple transformations using the marquee box in ***Tutorial 6***. There are other transformations available in the LH toolbox, *Move/Copy, Mirror Image, Rotate, Rectangular Array* and *Circular Array*. All these operate in a similar way. First you must select the objects to be transformed using the *Select* option. Then choose the appropriate transform option (the objects will stay selected but the marquee box will disappear). Set/check the variables in the dialog box that appears, then follow the screen instructions. The following example illustrates the general principles of their use.

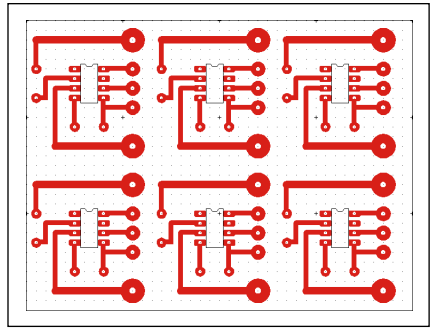
1. Choose *File* > *Close* to clear any work you have done so far, then choose *File* > *Open* and re-load the file "PCBTutor2.dtb" from the Tutorial directory. Choose *Edit* > *Select All* to select the whole circuit.

2. Choose the *Rectangular Array* icon from the *Transform* toolbar.

The following dialog box will open.



Ensure that *No. columns* is set to 3, *No. rows* to 2, *X spacing* to 1.3in and *Y spacing* to 1.5in. Click *OK* on the dialog box. The circuit will be repeated as shown below.



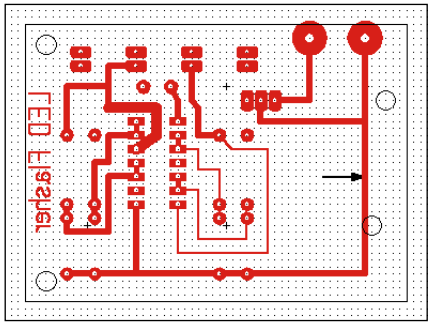
Now take a few minutes to investigate the operation of the other transformation options.

TUTORIAL 8 - Advanced Editing of Tracks

PCB Design has some very sophisticated (but simple to use) editing functions. These can best be demonstrated by working on an existing design.

1. Choose *File* > *Close* to clear any work you have done so far, then choose *File* > *Open* and load the file "PCBTutor3.dtb" from the Tutorial directory.

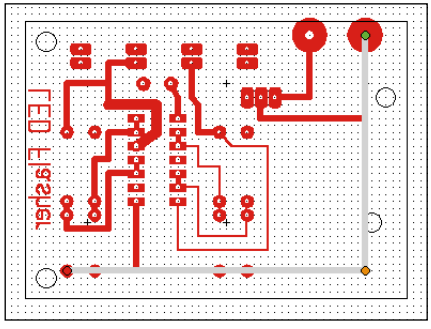
2. There are some 'deliberate mistakes' in this design, let's look at how to correct them. You can see that one of the tracks is cut by a mounting hole. As the hole cannot be moved, the track layout will have to be changed.



3. Choose the *Select* icon.

Point to the track to be edited (see arrow in the diagram above) and click the LH mouse button to select it. Note that the bottom part of the track is also selected. This is because the track was originally drawn in one operation.

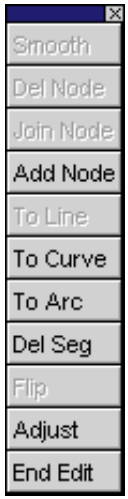
4. Notice the *Property/Start Edit* toolbox in the bottom RH corner of the screen. Choose *Start Edit*. The track will turn grey and 'nodes' (coloured circles) appear. The green node indicates the start of the track, the orange node(s) the intermediate point(s) and the red node the end of the track.

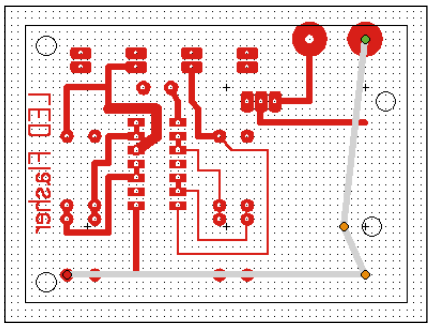


You should also notice that the *Property/Start Edit* toolbox has been replaced by the longer *Edit Mode* toolbox (initially all buttons will be greyed out). This will be used later.

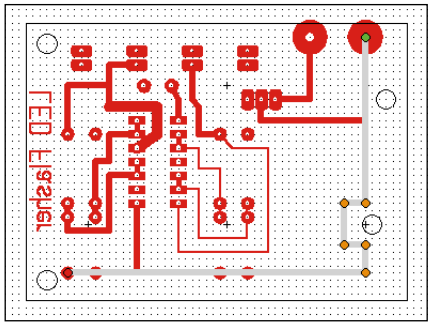
Move the cursor over the orange node, click the LH mouse button, and move the node to a new position. To fix it in its new position click the LH mouse button again, or to return to its original position click the RH mouse button. (If multiple objects are selected and if two or more nodes are coincident (eg., two tracks meet at a pad), as you move coincident nodes, all connected objects will move together.)

5. In this case moving the node will avoid the hole, but will produce a 'messy' circuit board. A better plan is to add more nodes to the track. First click on the *Undo Last* icon to restore the track to its original position. Then click on the track between nodes with the LH mouse button to sub-select it (the original locating point will be fine). At this point the appropriate buttons in the *Edit Mode* toolbox will become active. Choose *Add Node*. A new node will appear mid way along the selected track. Click off the drawing once so that the tracks revert to grey, then click on the new node and position it as shown below.

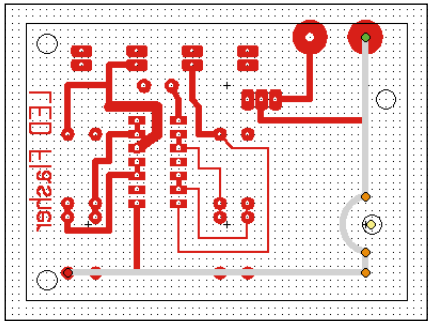




6. This is better but not very elegant. To improve things further, sub-select another part of the track and add another node. Repeat the process until the circuit looks like the layout below. (N.B. If you move a node which connects a sub-selected (pink) section and a normal grey section, they will separate. This can sometimes be used to good effect to 'break' tracks. However if it is done by mistake, *Undo Last* will re-connect the sections.)



7. For an even more elegant solution, delete the two LH nodes by sub-selecting them (drag a box around them), and clicking on the *Del Node* icon. Click on the remaining section of track between the nodes to sub-select it and click on the *To Arc* icon. This will enable you to curve the track neatly round the hole as shown below.



8. To remove the nodes, click “off” the drawing (twice, if objects are sub-selected), or click on *End Edit*. This will return the marquee box. Click off again to de-select.

N.B. All the same editing routines can be used on all types of drawing objects, lines, arcs, etc.

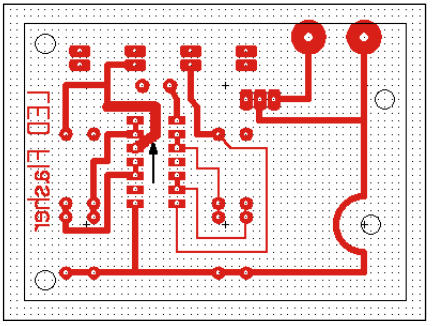
TUTORIAL 9 - Changing Track Width, Pad Properties and Layers

PCB Design allows very simple editing of track and pad properties. This is best demonstrated by working on an existing design.

1. Choose *File* > *Close* to clear any work you have done so far, then choose *File* > *Open* and load the file "PCBTutor4.dtb" from the Tutorial directory.

2. Look at the track connecting to pin 3 on the integrated circuit (see arrow in the diagram below). This track has been drawn too wide and is touching pin 2. Select the offending track, then click on the *Change Tracks* button at the top of the screen.

Reset the track width to 0.03in from the drop down list, (or type in this value), then click *OK*.

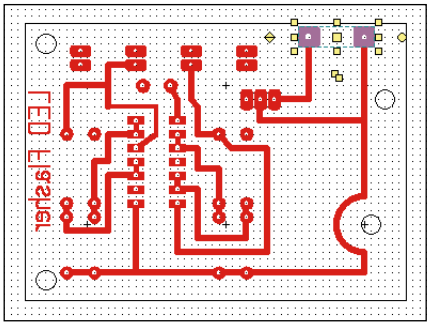


3. It is possible to change any number of tracks simultaneously. Look at the tracks in the middle of the board. You will see several that are unnecessarily fine. Select the fine tracks. If you do this by dragging a box around them, the four pads in the middle of the tracks will also be selected. This is fine, as changing the track width will not affect the pads.

Once you have selected the tracks click on the *Change Tracks* button. Set the track width to 0.05in, then click *OK*.

4. Selected pads may also easily be edited. Look at the battery connection pads (top right). The pads are unnecessarily large, so large that they extend beyond the edge of the board. Select both pads. Click on the *Change Pads* button.

Set the *Pad Type* to Square and the *Pad Size* to Medium, then click on *OK*.



Changing the pad properties will change all the selected pads, but notice that you can choose the type of pads to change. Thus, if you wish to change **all** the square pads for example, rather than try to select each individually, drag a select box across the whole board, click on the *Change Pads* button and untick *All Pads*, and tick *Square Pads*. When you now set a new pad parameter, only the selected square pads will be changed.

5. There are times when working with double sided boards that you may wish to move pads and/or tracks from the *Solder Side* to the *Component Side* layer or vice versa. To do this, select the pads/tracks as before and click on the *Change Layer* button. It is also possible to transfer drawing data from one layer to another, but it is not possible to transfer track data to a drawing layer or vice-versa.

6. Whenever a single object (pad, track, line, shape or piece of text) is selected the *Property/Start Edit* toolbox appears in the bottom right hand corner of the screen. We have already looked at the use of *Start Edit* to modify tracks. *Property* is an extremely useful tool, which PCB Design shares with 2D Design. Put simply, the *Properties* dialog boxes will tell you everything you might wish to know about an object, and give you the opportunity to change any of its parameters. There are too many variations to go into here, we suggest you experiment to find out just what can be done with this versatile tool.

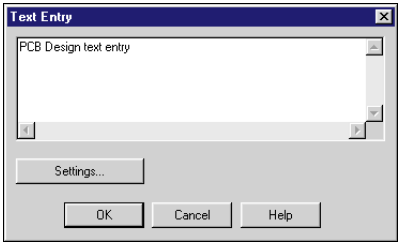
TUTORIAL 10 - Text

The PCB Design text facility can be used to add text to any layer. Text fonts used are Windows TrueType format.

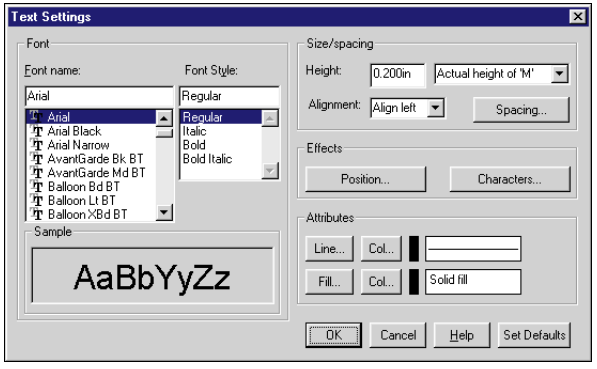
1. Choose *File* > *Close* to clear any work you have done so far then choose *File* > *New* to start a new drawing. For the purposes of this tutorial only, choose *Help* > *Restore Factory Defaults*.

2. Choose the *Text* icon.

Choose the layer you wish to work on from the drop down list in the attributes bar. Position the cursor and click where you intend the text to be. You need not be precise, the text can be moved later. The following dialog box will open.



Enter any short text phrase, then click on the *Settings* button. In the *Text Settings* dialog box (see below), you may select any True Type font installed on your computer, and apply a range of effects to the lettering.



There are too many options to go into in depth, so leave the default settings for now (remember to come back and experiment later!).

Click *OK* on this dialog box and also on the *Text Entry* dialog box to return to the design screen. Note, if you are putting text on the *Solder Side* layer it will appear mirrored. This is

correct, (remember you are looking at a transparent board with the design and text on the back).

Once entered, text may be treated like any other object and dragged around or transformed using any of the transformation tools. To quickly re-size a piece of text to fit in a space, select it then click on one of the corner handles of the marquee box **with the RH mouse button**. Once positioned click again. This will preserve the text aspect ratio and correct spacing, whilst changing the size. If you want to distort the text aspect ratio then click on a corner handle and move with the LH mouse button.

6 PCB DESIGN NOTES

Working with Component Libraries

Perhaps more than any other design activity, electronics is frequently a matter of piecing together building block circuits. Even when designing totally from scratch, the components used are usually standard items. It makes sense then to have these standard components and building blocks ready drawn so they can be dropped into your design, rather than drawing each one when you need it. With so many components and building blocks, a library system is needed to keep things where you can find them.

PCB Design uses the standard Windows filing system. All the basic components and building block circuits are kept in sub-directories of the 'PCBLib' directory. The names of most of the sub-directories are self-explanatory. Components are imported into the drawing using *File > Load Component*, and arrive 'grouped'. Grouping keeps the various pads, tracks, lines, etc., which make up a component, together so you can easily move them into place in your circuit. If you wish to adapt a component, first select it then choose *Edit > Ungroup*.

Creating your own libraries

You can add your own components or building block circuits to the library very easily. Once you have created the object, Choose *File > Save As Component.* The drawing will be saved as a component.

Although a range of standard components is supplied, it is likely that you will wish to amend or add to the components. Some items such as transistor packages, have been designed to manufacturer's specifications and therefore not all pins sit on a 0.1 inch grid (usually 0.05inch). All drilled holes in components are also to manufacturer's specifications, so, for example, DIL pads are drilled to 0.8mm. School users may wish to standardise on one drill size, eg., 1mm to simplify drilling operations.

You can modify components by choosing *File > Load Component*, making any alterations required, then saving again using *File > Save As Component.* It is probably best not to overwrite the supplied libraries but to create a new directory where all your own components are saved together.

Creating New Pads

There are a number of pad sizes pre-defined in PCB Design and at any time you may enter your own custom values for the pad you are working on. However there may be occasions when you need a special pad repeatedly and wish to have it added to the standard list. To do this choose *Setup > Drawing > Define Pads.* Choose the pad type you want then click on *Add new.*
Now simply enter the name for your pad and its parameters. The new pad will be saved with the current drawing, or may be saved as part of a setup. (See ***Setups and Customisation*** page 42.)

Printing

PCB Design will print a high quality image via any Windows inkjet or laser printer. First set the Windows printer driver up in the normal way then use *File > Print*. The *PCB Print Options* dialog box will open.

The *Print Graphically* option will print in full colour on a suitable printer, or greyscale on a mono-chrome printer.

The *Print solder layer photo mask* option will print the solder side layer only as a solid black image. This option should be used if a master image is required for a photo-reproduction process.

The *Print component layer photo mask* option will print the component side layer only as a solid black image. This option should be used if a master image is required for a photo-reproduction process.

The *Centre* option is ticked by default and the *Scale* is set to 100%. The default values ensure that the design is printed the correct size in the centre of the page.

When suitable parameters have been set, click on *OK* to proceed with printing.

N.B. By choosing *Setup > Drawing > Layout* then selecting *Match print area* of your printer, you can match the PCB board area to the maximum print area if desired. This may help when laying out a drawing to be printed.

7 PCB MAKE

This section applies equally to users who have created or modified a layout in PCB Design, and those who have imported a layout into PCB Make.

This manual is not intended to provide an in depth guide to the operation of individual machines. It assumes user familiarity with the output device. If this is not the case, this manual should be read in conjunction with the appropriate TechSoft or manufacturer's user guide for the machine.

PCB Make can be used to drive various pen plotters, Roland vinyl cutters (CAMM 1, Stika, etc.), and Roland milling/routing/engraving machines (CAMM 2, 3, Modela, etc.).

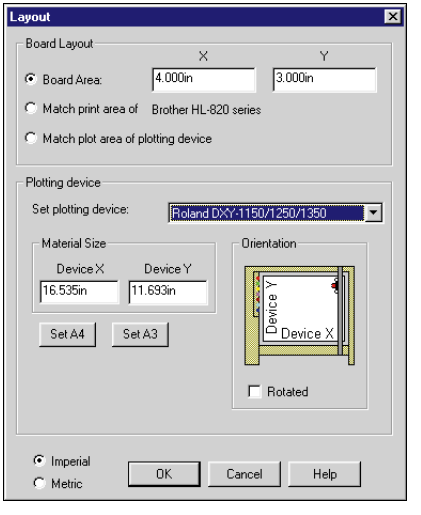
OUTPUTTING USING PEN PLOTTERS

Flat bed plotters can be used as an alternative to printers for preparing a conventional photo etch mask. There are built in drivers for the Roland and Graphtec ranges of plotters. If your plotter is not listed it may well work with the *General HPGL plotter* driver.

As an alternative to photo etching, many plotters can be fitted with oil based pens which are etch resistant. These can be used to plot the resist direct onto the copper clad board. TechSoft sell a special thin PCB board for this process which fits easily under the arm of most plotters.

Output - Step 1

Before outputting it is important to set up the correct driver for the machine to be used. This can be done "on the fly" at the time of output, but it is much better done even before any designing takes place using *Setup > Drawing > Layout*. A dialog box similar to the one shown below opens.



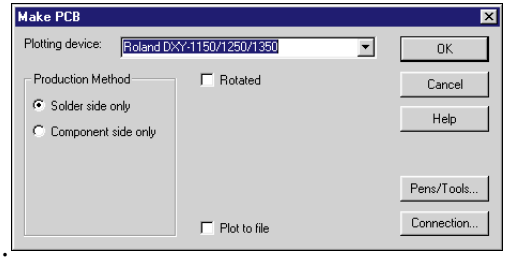
Board layout: This section of the dialog box allows users to set a suitable size for the *Board Area* displayed on screen (the white area). Most users will want to work on a "sensible" sized board (4 x 3 inches is the default value). It is, however, possible to set the *Board Area* to match the plotter to be used. This is particularly useful if making a lot of small circuits so that they may be "fitted onto" the maximum plotting area available.

Plotting device: This section allows the plotter type to be selected, and the actual material and its orientation, fitted to the plotter to be specified.

The layout and plotting device chosen can be saved in a setup file using *Setup > Save Setup* or *Setup > Set As Default*. (See ***Setups and Customisation*** page 42.)

Output - Step 2

When you are ready to output, choose *File > Make PCB.* The *Make PCB* dialog box will open. Check the parameters and options detailed below.



Production Method

Solder side only: This option allows only the solder side tracks to be drawn. It is the default option and should always be set for single sided boards.

Component side only: This option allows only the component side tracks to be drawn.

Pens/Tools

This is where parameters such as pen speed and diameter are set. The default values should work adequately. However the dialog box which opens allows full user control if required.

Connection

The *Connection* may be *Direct to port*, or *Via Generic / Text Only Printer driver*. By default, connection is set to *Direct to port*. In this mode all data is transmitted directly to the plotting device. This is the simplest option to set up, but will cause the computer to be tied up until output is completed. The *Via Generic / Text Only Printer driver* option makes use of the Windows print buffering capabilities. The *Generic / Text Only* printer driver is a neutral printer driver which allows plot-

ter commands to pass directly through it. It therefore allows direct control of the plotting device, whilst maintaining the advantages of the windows background printing and queueing facilities. Before using this option, however, the *Generic / Text Only* driver must be loaded and set as the current printer driver. The *Generic / Text Only* printer driver is installed from Windows using *Start > Settings > Printers > Add Printer... Manufacturer* "Generic". The driver itself is supplied on the Windows installation discs. **The Generic / Text Only printer driver's setup should have the appropriate output port set and also: *Paper size - A4, Paper feed - Continuous, No Page break*. If using a serial connection (eg.,. COM1, COM2), the printer driver's protocol should normally be set to 9600 baud, 8 data bits, 1 stop bit, No parity, Hardware flow control.**

N.B. If you wish to change the *Connection* permanently, the appropriate changes should be made, and then the setup saved using *Setup > Set As Default* or *Setup > Save Setup*.

Output - Step 3

When all parameters have been checked, click on *OK* in the *Make PCB* dialog box. There will be a pause whilst the computer calculates the tool path. This is normally just a few seconds, but with a slow computer and a complex circuit it could take some while - don't panic, be patient! (Once the tool path has been calculated it will be stored with the circuit, so repeat tool path generation will be instant.) Only objects on the *Solder Side*, *Component Side*, *Both Sides* and/or *Board Profile* layers will be included in the toolpath. (The *Board Profile* layer is normally used to show the outline of the PCB, slots, mounting holes, etc.)

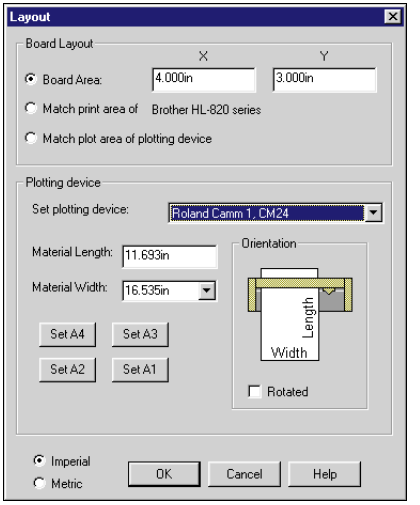
The drawing screen will change to show the toolpath and you will be asked '*Plot (whichever) side now?*' Check that the machine is ready to go, with material in place, then choose *Yes*. Follow the screen instructions carefully to completion.

OUTPUTTING USING VINYL CUTTERS

All Roland vinyl cutting machines (CAMM 1, ColorCAMM and STIKA, other than SCP-85) can be used to cut Cutronics copper foil to make simple 'chunky' circuits. Once cut, the circuit is 'weeded' and transferred to a suitable substrate using application tape. (The application tape can be the same as is used with self adhesive vinyl for sign writing.) Alternatively, complex circuits may be easier to weed if the application tape is applied before weeding, the full copper removed from the backing paper, and the excess copper weeded from the application tape.

Output - Step 1

Before outputting it is important to set up the correct driver for the machine to be used. This can be done "on the fly" at the time of output, but it is much better done even before any designing takes place using *Setup > Drawing > Layout*. A dialog box similar to the one shown below opens.



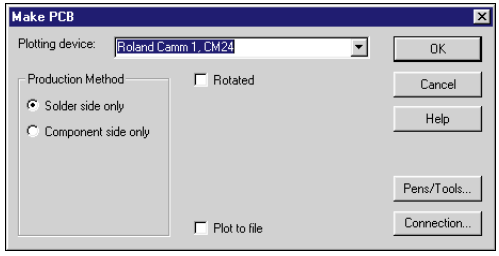
Board layout: This section of the dialog box allows users to set a suitable size for the *Board Area* displayed on screen (the white area). Most users will want to work on a "sensible" sized board (4 x 3 inches is the default value). It is, however, possible to set the *Board Area* to match the cutter to be used. This is particularly useful if making a lot of small circuits so that they may be "fitted onto" the maximum cutting area available.

Plotting device: This section allows the cutter type to be selected, and the actual material and its orientation, fitted to the cutter to be specified.

The layout and plotting device chosen can be saved in a setup file using *Setup > Save Setup* or *Setup > Set As Default*. (See ***Setups and Customisation*** page 42.)

Output - Step 2

When you are ready to output, choose *File > Make PCB.* The *Make PCB* dialog box will open. Check the parameters and options detailed below.



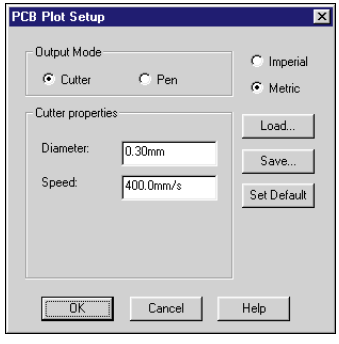
Production method

Solder side only: This option allows only the solder side tracks to be cut. It is the default option and should always be set for single sided boards.

Component side only: This option allows only the component side tracks to be cut.

Pens/Tools

This is where parameters such as cutter speed and diameter are set. The default values should work adequately. However the *PCB Plot Setup* dialog box allows full user control if required.



N.B. If *Pen* is chosen, rather than *Cutter*, the machine will be treated as a "Pen Plotter" (see ***Outputting Using Pen Plotters*** page 27.)

Connection

The *Connection* may be *Direct to port*, or *Via Generic / Text Only Printer driver*. By default, connection is set to *Direct to port*. In this mode all data is transmitted directly to the plotting device. This is the simplest option to set up, but will cause the computer to be tied up until output is completed. The *Via Generic / Text Only Printer driver* option makes use of the Windows print buffer-

ing capabilities. The *Generic / Text Only* printer driver is a neutral printer driver which allows plotter commands to pass directly through it. It therefore allows direct control of the plotting device, whilst maintaining the advantages of the windows background printing and queueing facilities. Before using this option, however, the *Generic / Text Only* driver must be loaded and set as the current printer driver. The *Generic / Text Only* printer driver is installed from Windows using *Start > Settings > Printers > Add Printer... Manufacturer* "Generic". The driver itself is supplied on the Windows installation discs. **The Generic / Text Only printer driver's setup should have the appropriate output port set and also: *Paper size - A4, Paper feed - Continuous, No Page break*. If using a serial connection (eg.,. COM1, COM2), the printer driver's protocol should normally be set to 9600 baud, 8 data bits, 1 stop bit, No parity, Hardware flow control.**

N.B. If you wish to change the *Connection* permanently, the appropriate changes should be made, and then the setup saved using *Setup > Set As Default* or *Setup > Save Setup*.

Output - Step 3

When all parameters have been checked, click on *OK* in the *Make PCB* dialog box. There will be a pause whilst the computer calculates the tool path. This is normally just a few seconds, but with a slow computer and a complex circuit it could take some while - don't panic, be patient! (Once the tool path has been calculated it will be stored with the circuit, so repeat tool path generation will be instant.) Only objects on the *Solder Side*, *Component Side*, *Both Sides* and/or *Board Profile* layers will be included in the toolpath. (The *Board Profile* layer is normally used to show the outline of the PCB, slots, mounting holes, etc.)

The drawing screen will change to show the toolpath and you will be asked '*Plot (whichever) side now?*' Check that the machine is ready to go, with material in place, then choose *Yes.* Follow the screen instructions carefully to completion.

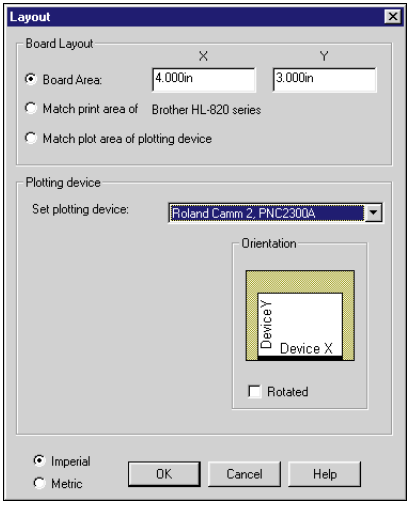
OUTPUTTING USING ENGRAVERS/MILLERS

Isolation engraving of PCBs is a technique used in the electronics industry for making prototypes and one-off boards. This method allows the board to be manufactured straight off the screen using just one machine. It is possible to just engrave the tracks and pads, centre dotting all holes using a vee cutter. Optionally a slot drill may be used to rout the board profile, and up to four PCB drills may be used for holes in the pads.

N.B. For best quality results, all users engraving tracks should refer to the section - ***Engraving Hints and Tips****, page 40.*

Output - Step 1

Before outputting it is important to set up the correct driver for the machine to be used. This can be done "on the fly" at the time of output, but it is much better done even before any designing takes place using *Setup > Drawing > Layout*. A dialog box similar to the one shown below opens.



Board layout: This section of the dialog box allows users to set a suitable size for the *Board Area* displayed on screen (the white area). Most users will want to work on a "sensible" sized board (4 x 3 inches is the default value). It is, however, possible to set the *Board Area* to match the engraver/miller to be used. This is particularly useful if making a lot of small circuits so that they may be "fitted onto" the maximum machining area available.

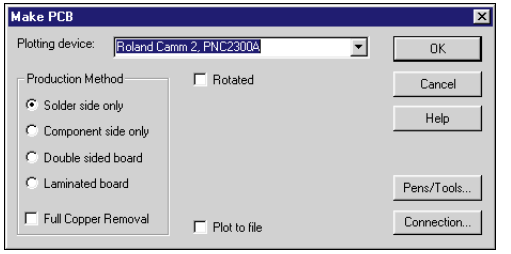
Plotting device: This section allows the engraver type to be selected.

The layout and plotting device chosen can be saved in a setup file using *Setup > Save Setup* or

Setup > Set As Default. (See ***Setups and Customisation*** page 42.)

Output - Step 2

When you are ready to output, choose *File > Make PCB.* The *Make PCB* dialog box will open. Check the parameters and options detailed below.



Production Method

Solder side only: This option allows only the solder side tracks to be machined. It is the default option and should always be set for single sided boards.

Component side only: This option allows only the component side tracks to be machined.

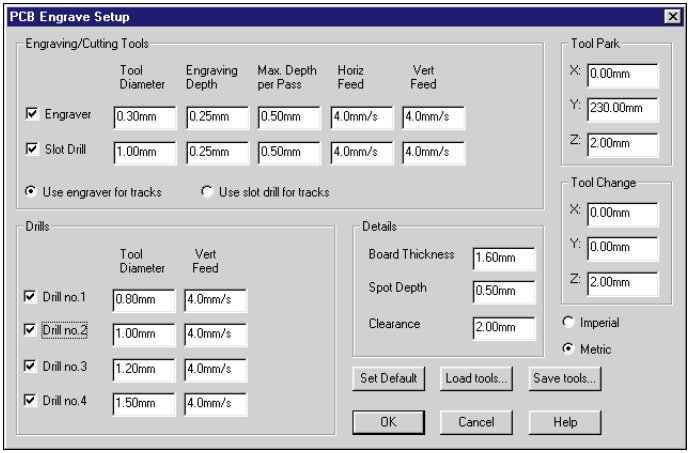
Double sided board: This is the standard option for double sided boards. The process initially machines the solder side of the circuit. User prompts then ask for the board to be turned over (flipped left/right). This must be done very accurately to be successful. The software assumes that the physical PCB is exactly the same size as the *Board Area*. (To ensure accurate flipping it may be convenient to fix a piece of waste material to the front left corner of the bed, and to machine an internal corner in it. Depending on the machine type, to work correctly, either the machine's X/Y origin (Home) is re-set to this corner, or the tool set point must be set to this point as appropriate.)

Laminated board: This is an alternative method of double sided board production. It involves machining two single sided boards then fitting them together back to back. (TechSoft's special thin (0.8mm) board is ideal for this purpose.) This sounds difficult, but is probably easier to achieve than the standard method as it does not require any accurate lining up when machining. Although the normal soldering operations should hold the two boards together, it may be a good idea to put in some dummy pads in the corners of the board and to solder some wire through these first. The board can then be handled more easily.

Full copper removal: This will engrave away all the copper except the tracks, rather than simply outlining the tracks. This is far slower and is rarely necessary from a practical point of view. However it may be useful for producing a 'cosmetically' correct board.

Pens/Tools

This is the most important of the output settings dialog boxes for day to day use. It is where most settings to do with control are found, such things as speeds, depths, offsets, etc.



The default setting uses an engraving tool to engrave the tracks and 'centre spot' the holes. If you wish to cut out the board (*Board Profile* layer) tick the *Slot Drill* box and check that the settings match your tool and board. If you wish to drill, tick the appropriate drill boxes. Note, the default drill sizes in this dialog box match the standard size pads in PCB Design. If you have used *Custom* sizes you may need to edit the drill table. Most of the settings in this box are self explanatory, with just a couple of points worthy of special mention.

Tool diameter: If the engraving tool followed the exact edge of, say, a track, then it would cut into the track by the radius of the cutter. If it was a thin track, this could leave no track at all! The software actually engraves around the track offset by half the tool diameter value. As a tapered cutter is used, the diameter will change with the depth. Thus, if engraving deeper than normal for some reason, the diameter value should be increased.

The default values should work fine, but bear in mind that they have been set to 'safe' values. With some experience you should be able to increase the feed rates significantly.

Connection

The *Connection* may be *Direct to port*, or *Via Generic / Text Only Printer driver*. By default, connection is set to *Direct to port*. In this mode all data is transmitted directly to the plotting device. This is the simplest option to set up, but will cause the computer to be tied up until output is completed. The *Via Generic / Text Only Printer driver* option makes use of the Windows print buffering capabilities. The *Generic / Text Only* printer driver is a neutral printer driver which allows plotter commands to pass directly through it. It therefore allows direct control of the plotting device,

whilst maintaining the advantages of the windows background printing and queueing facilities. Before using this option, however, the *Generic / Text Only* driver must be loaded and set as the current printer driver. The *Generic / Text Only* printer driver is installed from Windows using *Start > Settings > Printers > Add Printer... Manufacturer* "Generic". The driver itself is supplied on the Windows installation discs. **The Generic / Text Only printer driver's setup should have the appropriate output port set and also: *Paper size - A4, Paper feed - Continuous, No Page break*. If using a serial connection (eg.,. COM1, COM2), the printer driver's protocol should normally be set to 9600 baud, 8 data bits, 1 stop bit, No parity, Hardware flow control.**

N.B. If you wish to change the *Connection* permanently, the appropriate changes should be made, and then the setup saved using *Setup > Set As Default* or *Setup > Save Setup*.

Output - Step 3

When all parameters have been checked, click on *OK* in the *Make PCB* dialog box. There will be a pause whilst the computer calculates the tool path. This is normally just a few seconds, but with a slow computer and a complex circuit it could take some while - don't panic, be patient! (Once the tool path has been calculated it will be stored with the circuit, so repeat tool path generation will be instant.) Only objects on the *Solder Side, Component Side, Both Sides* and/or *Board Profile* layers will be included in the toolpath. (The *Board Profile* layer is normally used to show the outline of the PCB, slots, mounting holes, etc.)

The drawing screen will change to show the toolpath and you will be asked '*Plot (whichever) side now?*' Check that your machine, material and tools have been set ready to go then choose *Yes.*

Machine preparation and what happens next depends on the machine being used, for -

CAMM 2 refer to page 37.
CAMM 3 / MDX-500 refer to page 37.
MODELA MDX-3 refer to page 38.
MODELA MDX-15/20 refer to page 39.

CAMM 2

Ensure that each tool to be used has been set up and datumed (see below). Follow the on screen instructions carefully to completion of the machining process.

Tool setting and datuming

Before proceeding, ensure that you have sufficient top collets for all the tools you intend to use (Extra top collets are available directly from TechSoft). With the material stuck to the machine bed, move the cutter head over the material, then lower the cutting head to the Z0 position. Fit each tool to be used in turn, lowering it until it rests on the work before attaching a top collet. Remove each tool, with its top collet still attached so that each may be quickly re-fitted when needed. If you do a lot of PCB work the tools can be stored like this between jobs.

CAMM 3 / MDX-500

Ensure that each tool to be used has been set up and datumed (see below). Follow the on screen instructions carefully to completion of the machining process.

Tool setting and datuming

As the CAMM 3 and MDX-500 do not have an indexable collet system as standard, some thought needs to be put into the practicalities of tool changing, if multi-tool operations are to be carried out. TechSoft recommend the use of tools and collets as used with the Roland Modela MDX-15 (see the TechSoft Price List for further details). These collets have a 6mm shank and can easily be inserted into the standard CAMM 3/MDX-500 collet. The procedure is as follows:

Fit a Modela collet into the CAMM 3/MDX-500 and push it up tight to the bottom of the machine collet. Fit an engraving tool into the Modela collet with as little stickout as possible. Move the material under the tool and lower the cutter head to allow sufficient clearance between the bottom of the Modela collet and the workpiece for the stickout of the longest tool to be used (usually about 30mm). Set the Z0 position on the machine at this point, (key strokes vary on different models, see your machine's User Manual for details).

Loosen the screw in the Modela collet and lower the tool onto the surface of the material, then re-tighten the screw. Raise the cutter head and remove the Modela collet with the tool still fixed to it. Repeat this operation for all other tools to be used, **(but now lowering the head to the same Z0 each time).** If you do a lot of PCB work the tools can be stored like this between jobs.

Set the tool change position in the *PCB Engrave Setup* dialog box (see page 35), to be far enough above the board in Z (say 75-100mm) to ensure that tools can easily be removed and replaced. When subsequently changing tools ensure that the Modela collet is pushed up tight to the bottom of the machine collet as it is tightened.

MODELA MDX-3

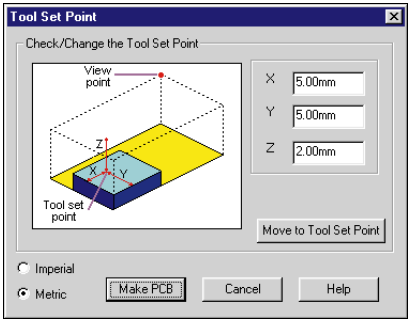
Ensure that each tool to be used has been setup and datumed (see below). Follow the on screen instructions carefully to completion of the machining process.

Tool setting and datuming

Before proceeding, ensure that you have sufficient top collets for all the tools you intend to use (Extra top collets are available directly from TechSoft). As the Modela has a large vertical (Z axis) travel, a built-up bed is necessary to avoid excessive tool stick out which will lead to vibration. A suitable bed can be made from layers of MDF, with a plastic top sheet (eg., ABS or acrylic). This can be assembled and stuck to the existing bed with double sided tape. The new bed should be about 30-35mm thick.

The standard method of zeroing tools with Modela (jogging the tool down to touch the work) is not sufficiently accurate for PCB work and does not allow for tool changing. TechSoft have therefore developed a more sophisticated method using software control.

Stick a piece of PCB material to the machine bed. Remove any tools from the Modela and switch it on. Press [VIEW] so that the machine head moves to the front LH corner and the VIEW light is off. With a PCB drawing on screen, proceed to maunfacture as normal. Before the actual machining takes place the following dialog box will open.



Click on *Move to tool set point*. Once the machine has finished moving, check that it is a suitable position for fitting tools, if not change the tool set point values accordingly and click *Move to tool set point* again. When you are happy with the position, fit each tool in turn lowering it until it rests on the work before fitting the collet. Remove each tool, with its collet still attached so that each may be quickly re-fitted when needed. If you do a lot of PCB work the tools can be stored like this between jobs. Once all the tools are set, return to the computer and either click *Make PCB*, or *Cancel* as appropriate. Once the tooling has been set successfully, use *Setup > Save Setup* or *Set As Default*. The *Tool Set Point* will then be saved with the setup and will in future only need to be checked on the way to outputting. (See ***Setups and Customisation*** page 42.)

MODELA MDX-15 / 20

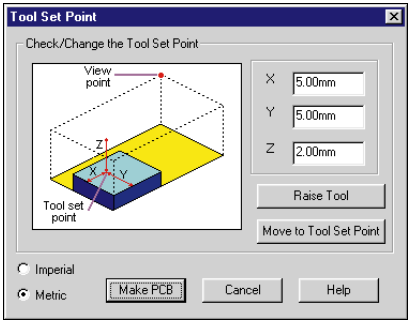
Ensure that each tool to be used has been setup and datumed (see below). Follow the on screen instructions carefully to completion of the machining process.

Tool setting and datuming

Before proceeding, ensure that you have sufficient top collets for all the tools you intend to use (Extra top collets are available directly from TechSoft). As the Modela has a large vertical (Z axis) travel, the cutter head must be set to the lowest position to avoid excessive tool stick out.

The standard method of zeroing tools with Modela (jogging the tool down to touch the work) is not sufficiently accurate for PCB work and does not allow for tool changing. TechSoft have therefore developed a more sophisticated method using software control.

Stick a piece of PCB material to the machine bed. Remove any tools from the Modela and switch it on. Fit the tool setup override, then press [VIEW] so that the machine head moves to the front LH corner and the VIEW light is **off**. With a PCB drawing on screen, proceed to manufacture as normal. Before the actual machining takes place the following dialog box will open.



Click on *Move to tool set point*. Check that it moves to a suitable position for fitting tools (the cutter head 25-30mm from the bed). If not, change the tool set point values accordingly and click *Move to tool set point* again. When you are happy with the position, click on the *Raise Tool* button. The cutter head will move to its maximum vertical position. Fit an engraving tool into a TechSoft collet, such that the stickout is minimal. Click on *Move to tool set point* again to move the cutter head back down. Loosen the collet screw to drop the tool onto the work surface, then re-tighten the screw. If multiple tools are to be used, raise the tool head and remove the tool with its collet attached, fit the next tool and repeat the procedure until all tools have been datumed on the work surface. If you do a lot of PCB work the tools should be stored like this between jobs. Once all the tools are set, remove the tool setup override, fit the cover, then return to the computer and either click *Make PCB*, or *Cancel* as appropriate. Once the tooling has been set successfully, use *Setup* > *Save Setup* or *Set As Default*. The *Tool Set Point* will then be saved with the setup and will in future only need to be checked on the way to outputting. (See ***Setups and Customisation*** page 42.)

ENGRAVING HINTS AND TIPS

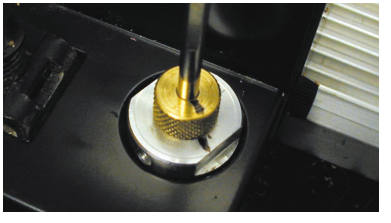
PCB board is normally available in two grades, FR2 and FR4. FR2 board uses a paper substrate, FR4 board uses a glass fibre substrate. In practice, for most circuits, there is little to choose between them, though the FR4 board is ultimately stronger and more durable. However, the FR2 board is far less abrasive when machined, and is strongly recommended to preserve tool life.

Engraving tools are available with H.S.S. (high speed steel), or carbide tips. Due to the abrasive nature of PCB board, TechSoft only recommend the use of carbide tools. Some Roland machines have an engraving tool supplied with them by the manufacturer. These tools are normally H.S.S.

TechSoft can supply both PCB materials and replacement cutters. The standard vee point engraving tool supplied by TechSoft has a 40 degree included angle with the tip ground to a flat, producing an effective tip radius of 0.1mm.

When an engraving tool is fitted to a machine, the normal manufacturing tolerances of both machines and tools, may cause the tool to rotate very slightly off centre (normally no more than a few hundredths of a millimetre). Because of the nature of the cutter geometry this may cause the cutter to rub, and rather than cut cleanly, a ragged cut or poor finish may result. This is normally remedied as follows:

When first using a new tool, arrange to take a small test cut (just a single short track should do). If the cut is not clean, turn the tool through 90 degrees in the machine spindle and try again. Repeat until the cleanest cut is obtained. Mark the position of the tool so that when it is re-inserted, it can be lined up correctly. A simple way of doing this is illustrated below for the CAMM 2. Users of other machines will be able to adapt this method to their own requirements.



A marker pen has been used to draw a line on the top of the machine spindle. A corresponding line has been drawn on the tool top collet. When the tool is inserted, the marks are simply lined up.

Soldering onto Engraved Tracks

This should not prove a problem as long as fine tipped pointed soldering irons are used with fine (22swg) solder. If this is the case bridging is unlikely to occur as the solder will not readily jump the vee groove created when engraving.

8 COMBINING DESIGNS

Drawings are normally loaded and saved using the standard Windows filing system options *File > Open/Save/Save As.* When a drawing is saved all the settings in *Setup > Drawing*, eg., layout, origin, etc., are saved with it. When a drawing is loaded via *File > Open*, its settings are loaded with it.

PCB designs are often quite small and considerable savings in time and materials can be made if several layouts are combined together to be printed/plotted/manufactured simultaneously. This can easily be achieved as follows.

Designs are created and saved as normal. Load one file as normal, or start a new PCB Design file. Set the appropriate *Board Area* using *Setup > Drawing > Layout*, then load each additional design in turn using *File > Add.* Once added, designs can be moved around the screen to obtain the 'best fit' prior to outputting. Files 'added' to a drawing do not change any drawing settings.

9 LINKS WITH OTHER SOFTWARE

For users who wish to autoroute circuits, or who wish to have the ability to produce PCB's from Crocodile Clips schematics, TechSoft have developed a link with PCB Wizard.

Having created your layout in PCB Wizard (Version 2.5 or later) choose *File > Make > Output to CAD/CAM.* This will transfer the data to the Windows clipboard. Start PCB Design/Make, and choose *Edit > Paste.* N.B. Only the *Solder Side, Component Side* and *Both Sides* layers are transferred.

Once transferred, PCB Wizard designs can be treated in exactly the same way as designs created in PCB Design.

Importing Drawings

Drawing data, including commercial clipart, can be imported into PCB Design. A variety of formats will be accepted, eg., *dtd (Design Tools drawings*), *wmf (Windows metafile), dxf (Data Exchange format),* etc. (N.B. PCB Design will only accept vector data, eg., lines, arcs, etc. It will not accept bitmap (raster) data. Formats such as WMF can hold both vector and bitmap data. When loading these type of files the bitmap data will be ignored.)

The data can be imported using *File > Import.* Alternatively, in the other application, *Copy* the drawing to the clipboard then in PCB design use *Edit > Paste* to import the drawing.

10 SETUPS AND CUSTOMISATION

It is easy to reconfigure the user interface to suit individual requirements or preferences. All the options under *Setup* are saved in a setup file. These options allow a great deal of choice over both the layout, and the look and feel of PCB Design.

Using *Setup > Drawing*, it is possible to set the output device type, the size of the drawing area, the units to be used, etc.

Using *Setup > Customise*, it is possible to remove items in the display, eg., the *Layer* button, individual coordinate displays, etc., thus simplifying it. It is possible to change the method of operation, eg., change the method of selecting the pull out toolbar from hold and slide, to a simple pick. It is also possible to customise the menu. Any menu item (and its associated icon) may be toggled on/off, etc.

Using *Setup > Plotting Devices* it is possible to setup the current tooling options, etc., for milling systems and vinyl cutters.

Thus, using setups, it is possible to tailor your software to your exact requirements as a 'turn on and go' system, with no need to make any further system changes as you work. Several named setup files may be saved so that individual users may have their own named setups, or setups may be created to suit different projects or machine output types.

To save a setup choose *Setup > Save Setup.* Removing the tick from the *Save Setup Options* dialog box will ensure that the *Save Setup* option is automatically removed from the saved setup. This will prevent users of such setups from saving new setups and possibly overwriting existing setups. This may be of particular interest to educational users.

It is, of course, possible to manually remove the *Setup > Load Setup* option using menu customisation. Again this may be of interest to educational users. However, if such a setup is saved as the default setup (see below), it will be impossible to change setups in the conventional way. The only way back to the standard configuration then is via *Help > Restore Factory Defaults.*

By selecting *Setup > Save as Default*, the current setup may be saved as the default, so that on subsequently re-loading the software, it will automatically come back set up exactly as you wish.

When a drawing is saved, all the settings available under *Setup > Drawing* (and the current plotting device settings chosen in *Setup > Drawing > Layout*), together with all the current settings such as pad size, track width, text height, etc., are saved with the drawing. When the drawing is loaded normally using *File > Open*, all the settings saved with the drawing are restored. If you wish to load a drawing without changing the current settings, use *File > Add*.

INDEX