



Software Tutorials

© Copyright TechSoft UK Limited 1995-2010

TechSoft UK Ltd.,
Falcon House,
Royal Welch Avenue,
Bodelwyddan,
Denbighshire, LL18 5TQ
U.K.

Tel : +44 (0)1745 535007
Fax : +44 (0)1745 535008
Email : email@techsoft.co.uk
Web site : www.techsoft.co.uk

All rights in this booklet and the program are reserved. Reproduction, adaptation, or translation, without the prior written permission of TechSoft UK Limited is prohibited, except as allowed under the copyright laws.

The program described in this booklet 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 booklet) 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 booklet.

All trademarks acknowledged.

Roland, CAMM1/2/3, ColorCAMM, MODELA, STIKA are trademarks of Roland Digital Group.

Microsoft®, Windows®, Windows NT®, Vista® and Windows 7® are U.S. registered trademarks of Microsoft Corporation.

PCB Wizard and Circuit Wizard are trademarks of New Wave Concepts Ltd.

CONTENTS

1	INTRODUCTION	4
	About this Booklet	4
	Software Overview	4
2	SYSTEM REQUIREMENTS	5
3	INSTALLATION	5
4	2D PCB HELP	6
5	TUTORIALS	7
	Tutorial 1 - Screen Layout, Menu Selection, etc.	7
	Tutorial 2 - Setting Up the Software to Suit Your System	8
	Tutorial 3 - Basic Circuit Drawing	9
	Tutorial 4 - Layers	12
	Tutorial 5 - Drawing Tools	14
	Tutorial 6 - Object Selection and the Marquee Box	15
	Tutorial 7 - Transformations	17
	Tutorial 8 - Advanced Editing of Tracks	18
	Tutorial 9 - Changing Track Width, Pad Properties and Layers	21
	Tutorial 10 - Text	23
	Tutorial 11 - PCB Manufacture by Photo-Etching	25
	Tutorial 12 - PCB Manufacture by Knife Cutting	26
	Tutorial 13 - PCB Manufacture by Engraving/Milling	28
	Tutorial 14 - Component Libraries and Creating New Pads	32
	Tutorial 15 - Links with Other Software	33

1 INTRODUCTION

About this Booklet

2D PCB can be used without reference to a printed manual. Most functions are self-explanatory and *Help* is always available. However, to achieve the best results in the shortest time, it is recommended that you carefully work through the 2D PCB tutorials in this booklet. The tutorials are intended as a practical guide to using the software on a day to day basis, not as a deep technical reference. For technical reference information use the *Help* facilities within the software.

Printing this booklet

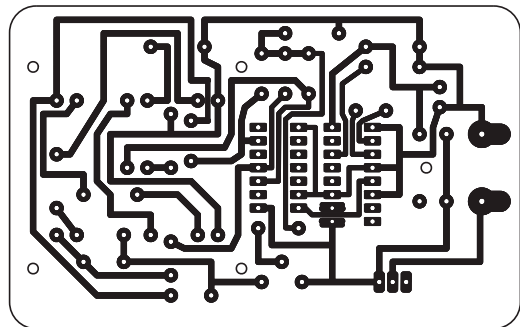
This booklet is primarily intended to be used as a printed reference, and has been formatted as such. Users wishing to follow the tutorials “heads up” on screen are advised to follow the tutorials from the *Help* facilities within the software.

Software Overview

The Design Tools software suite consists of a range of integrated packages developed to provide solutions for designers in many different areas. 2D PCB is the printed circuit board design and manufacture software element of the suite.

Designing

2D PCB provides an easy to use PCB draughting layout and editing program. Its clear and logical interface makes it ideal for the inexperienced user, whilst its advanced editing facilities give it the power to tackle sophisticated circuit board layouts.

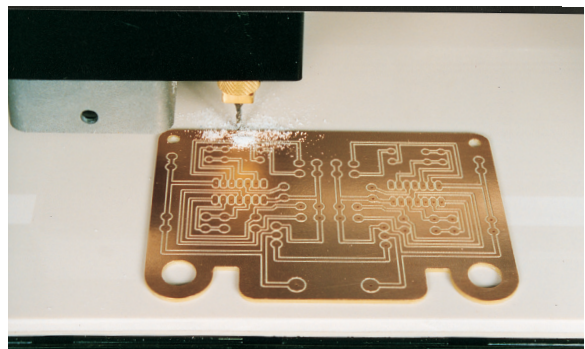


Manufacturing

2D PCB can produce outputs to manufacture PCBs on two distinct types of machine - engraving/milling machines and knife-cutting machines.

Isolation Engraving

The output to engraving/milling machines (eg., RotoCAMM, Modela, Modela Pro, CAMM 2, CAMM 3, etc.), creates a circuit by engraving around each circuit path with a fine point tool onto conventional copper clad PCB board. This process is called isolation engraving. While not electronically necessary, it is also possible to remove all unused copper to provide an excellent “prototype” of a conventional printed circuit 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.



Knife Cutting

The output to knife-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 easily achieved



2 SYSTEM REQUIREMENTS

1. 2D PCB will run on any system using Windows 2000, XP, 2003, Vista or Windows 7. Although the software should run on the minimum specification computer for any particular operating system, large drawings, particularly those with large or numerous bitmaps, may benefit from a higher specification machine to improve speed.
2. 2D PCB requires a **minimum** screen display of 800 x 600, though a screen resolution of 1024 x 768 or greater is strongly recommended. (To change the screen resolution, from Windows you normally choose *Start > Control Panel > Display > Settings.*)

3 INSTALLATION

2D PCB is a licensed product and must only be used on computers for which a licence is held. The software may be supplied on a CD or it may be downloaded. A separate licence file (.lic) is also required. This may be supplied either on a CD or it may be emailed to you. The licence file will be required at installation or when the software is first run (if the licence file is not available the software can only be run in Demo mode). **Keep your licence file safe, as it may be required for future upgrades.**

If installing from a CD, on insertion of the CD into the drive the installation program will normally auto-start. If for any reason 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 on the keyboard.

Follow screen prompts carefully through to completion of installation. Immediately after installation, run the software (choose *Start > All Programs > TechSoft Design Tools > 2D PCB*, or use the desktop shortcut icon), and follow the prompts to install your licence file if necessary.

Licence upgrades (eg., for new device extensions, or from single user to site), may often be done by simply installing a new licence file (obtainable directly from TechSoft). To install a new licence file, choose *Help > Software Licence* then click on *Update Licence*.

4 2D PCB HELP

Help is a standard Windows feature. *Help* gives you easy access to detailed information on every menu item, tool, dialog box, button and feature in 2D PCB.

You can access *Help* in the following ways:

1. Choose *Help > Help Contents*.
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. Choose *What's This?* from the upper icon toolbar or *Help > What's This* or press *Shift + F1* to display the Help cursor. Then click on any tool to see a *Help* window describing it.

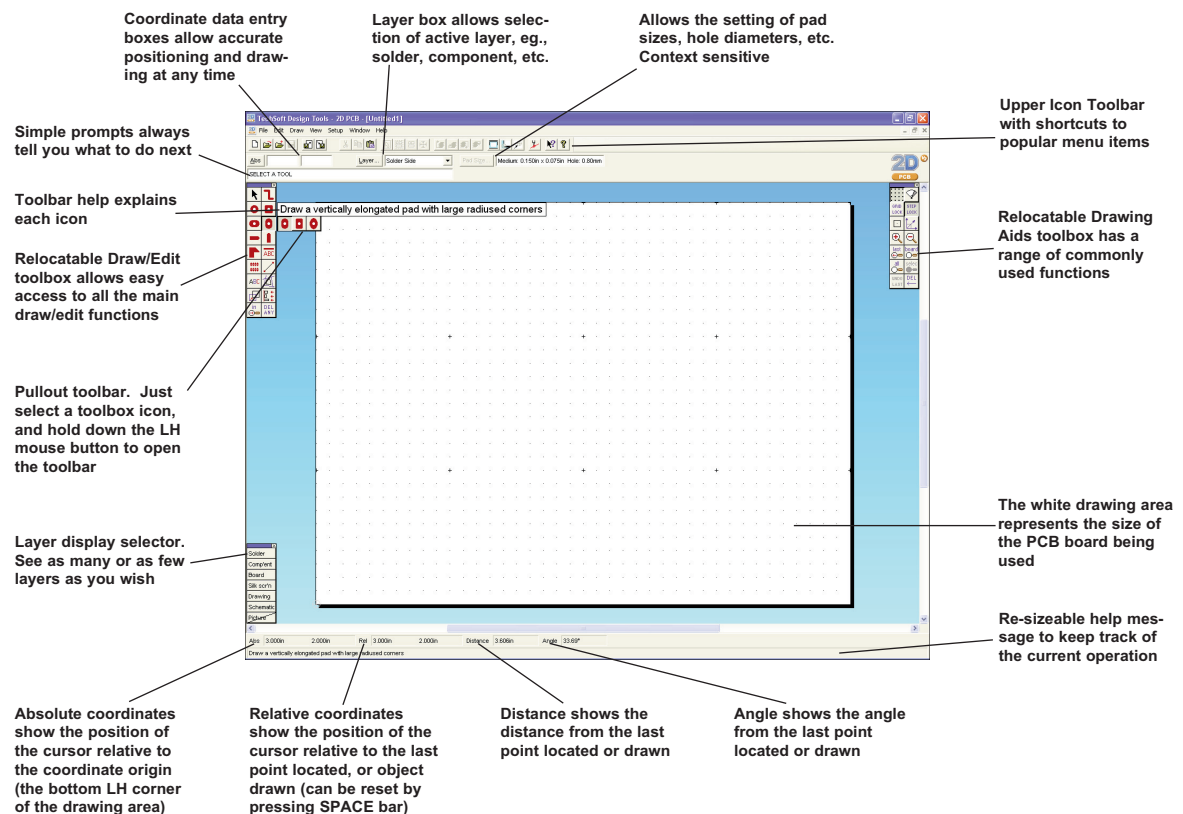


5 TUTORIALS

TUTORIAL 1 - Screen Layout, Menu Selection, etc.

The 2D PCB program window provides a clear view of the current drawing, various information areas, and a selection of icons with tools to cover the most common drawing, editing and display functions.

1. Start up 2D PCB (*Start > All Programs > TechSoft Design Tools > 2D PCB* or use the desktop shortcut icon) and familiarise yourself with the screen layout as shown below.

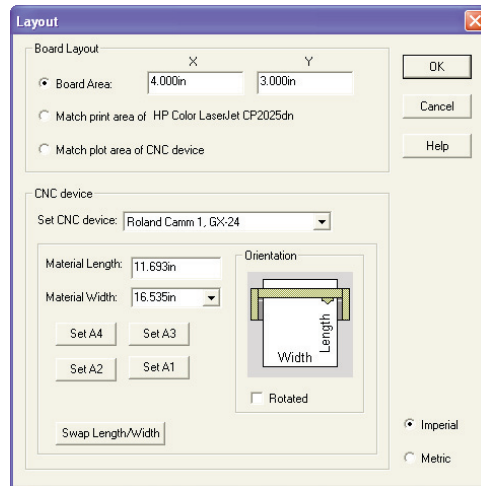


2. Menu items are normally chosen from the menu bar or the toolbox, using the mouse (although keyboard alternatives are available). To choose an item from the toolbox, position the pointer over the appropriate icon and click the LH (left-hand) mouse button. Some items in the RH (right-hand) toolbox (the *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 CNC 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 2D PCB. Choose the *Layout* icon from the upper icon toolbar (or *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 CNC device you will be using most often. (If you are not using a CNC device it does not matter which device is selected.)

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 CNC device 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. N.B. For tutorial purposes, ensure that the *Board Area* is set to 4 x 3 inches.

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 if, for example, you have two or more different output devices. If using both an engraving machine and a knife cutter you could set up for your engraving machine, then instead of using *Set As Default*, you could choose *Setup > Save Setup*, and give the setup a name such as “Setup for Engraver”. You could then save a suitable setup for the knife cutter 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

2D PCB can be used to draw PCB objects (parts of a circuit such as pads and tracks) and graphical objects (eg., lines, arcs). PCB objects are drawn in solid red if they are on the *Solder Side* of the board; solid green if they are on the *Component Side* of the board; or solid blue if they are on both sides. This tutorial deals with drawing PCB objects.

1. Choose the *Round Pad* icon (LH toolbox).

Move the cursor around, over the white board area. 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 (RH toolbox).

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 1 inch, the movements will be in very large steps.



3. Choose the *Step Lock* icon (RH toolbox).

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 0.1 inch. 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 (LH toolbox) 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 (LH toolbox).

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 (LH toolbox).

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 (LH toolbox).

Note the prompt reading *Locate one corner of zoom box*. Move the cursor approx. 0.5inch below 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 up 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. Choose the *Zoom Last* icon (RH toolbox).

This will restore the previous zoom level, in this case the full drawing screen. (You might also like to try the effects of the *Zoom+* and *Zoom-* icons in the RH toolbox at this point.)



9. Choose the *Radial Lock* icon (RH toolbox).

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.

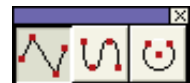


10. Move the pointer over the *Linear Track* icon.

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 (LH toolbox).

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 *Track Area* toolbar (LH toolbox, click and hold on the *Track Area* icon then drag and release).



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 (LH toolbox).

The *Load Component* dialog box will open. Load the component “Dil08” from the *Dils* library folder. Dil08 is the pin layout and schematic for an 8 pin dual in line chip (eg., 555, 741, etc.). The DIL will initially be highlighted pink, and will be surrounded 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 de-selected 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 try deleting and undoing as described below.

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*. Clicking on this effectively undoes the 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 chosen. (*Undo Last* restores a deleted object, but only the last one deleted!)



Most other delete functions are in the LH toolbox accessed from the *Delete* toolbar (click and hold on the *Delete* icon):

Delete Any allows you to point to an object and delete it.

Delete Part deletes any part of a line, arc, circle or bezier curve back to the closest intersection points from the locating point.

Delete by Box allows you to draw a simple rectangular box that will delete any items inside or crossing it.

Delete by Lasso allows you to draw a complex shape that will delete any items inside or crossing it.

Pressing *Delete* on the keyboard deletes the current selection (see **Tutorial 6** for more information about selection).

Pressing *ALT + Delete* on the keyboard deletes the whole of the current drawing.

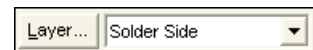
TUTORIAL 4 - Layers

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.

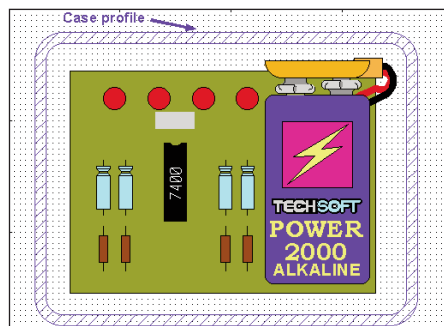
1. Start up 2D PCB, or close any open drawings (*File > Close*). Choose the *New* icon from the upper icon toolbar (or *File > New*). For the purposes of this tutorial only, choose *Help > Restore Factory Defaults*.



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 the *Open* icon from the upper icon toolbar (or *File > Open*), and load the drawing "PCBTutor1.dtb" from the *Tutorials* folder.



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 manufacturing 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.

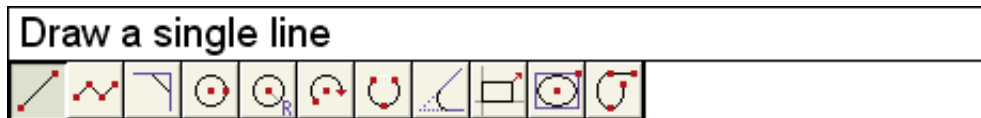
The *Picture* layer can be used to show realistic illustrations.

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. Start up 2D PCB, or close any open drawings (*File > Close*). Choose the *New* icon from the upper icon toolbar (or *File > New*). For the purposes of this tutorial only, choose *Help > Restore Factory Defaults*. 
2. Choose the *Single Line* icon (LH toolbox). 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 *Lines* pullout toolbar.



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

N.B. 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 which can then be used in 2D PCB. 2D Design files can be opened directly in 2D PCB (*File > Open*) though this will also "import" the drawing settings such as size, grid settings, etc. More usual ways of bringing 2D Design drawings into 2D PCB would be to choose the *Add* icon from the upper icon toolbar (or *File > Add*), or to copy and paste directly from 2D Design. 

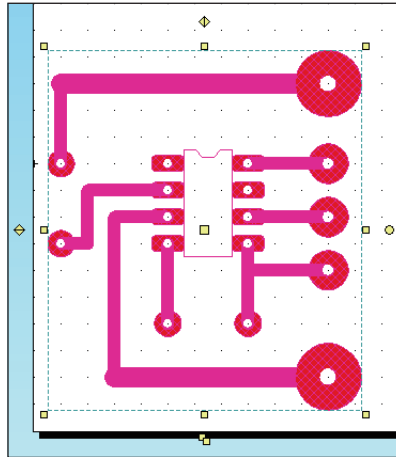
TUTORIAL 6 - Object Selection and the Marquee Box

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. Start up 2D PCB, or close any open drawings (*File > Close*). Choose the *Open* icon from the upper icon toolbar (or *File > Open*), and load the drawing "PCBTutor2.dtb" from the *Tutorials* folder. For the purposes of this tutorial only, choose *Help > Restore Factory Defaults*.



2. Choose the *Select* icon (LH toolbox). Move the pointer below 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 *Ctrl + A*). The circuit will be highlighted 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.

6. 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 choose the *Group* icon from the upper icon toolbar (or *Edit > Group*, or press *Ctrl+G* on the keyboard).






To ungroup objects, first select them, then choose the *Ungroup* icon from the upper icon toolbar (or *Edit > Ungroup*, or press *Ctrl+U* on the keyboard).



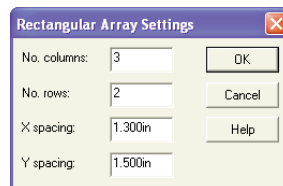
Try grouping and ungrouping some objects.

TUTORIAL 7 - Transformations

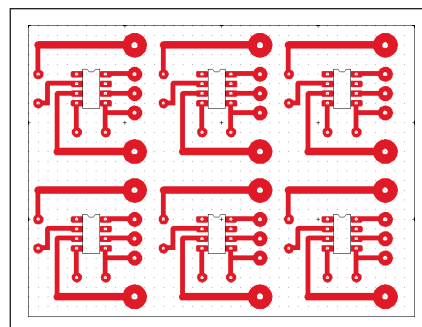
You have already tried simple transformations using the marquee box in **Tutorial 6**. There are other transformations available in the LH toolbox, *Move/Copy*, *Mirror Image*, *Rotate*, *Rectangular Array* and *Circular Array*. All these operate in a similar way. First you must select the objects to 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. Start up 2D PCB, or close any open drawings (*File > Close*). Choose the *Open* icon from the upper icon toolbar (or *File > Open*), and load the drawing "PCBTutor2.dtb" from the *Tutorials* folder. For the purposes of this tutorial only, choose *Help > Restore Factory Defaults*. 
2. Choose the *Select All* icon from the upper icon toolbar (or *Edit > Select > All*, or press *Ctrl+A* on the keyboard) to select the whole circuit. 
3. Choose the *Rectangular Array* icon from the *Transform* toolbar (LH toolbox, click and hold on the *Transform* icon then drag and release). 

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.

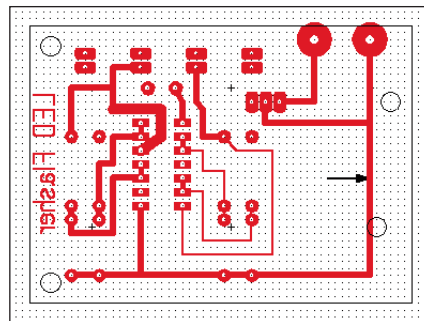



Now take a few minutes to investigate the operation of the other transformation options.

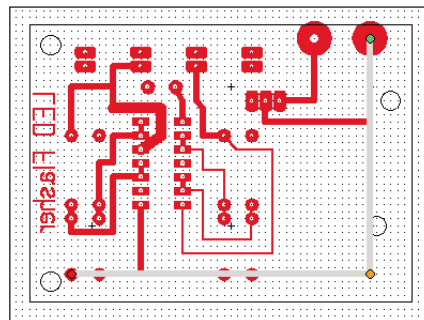
TUTORIAL 8 - Advanced Editing of Tracks

2D PCB has some sophisticated (but simple to use) editing functions. These can best be demonstrated by working on an existing design.

1. Start up 2D PCB, or close any open drawings (*File > Close*). Choose the *Open* icon from the upper icon toolbar (or *File > Open*), and load the drawing "PCBTutor3.dtb" from the *Tutorials* folder. 
2. There are some 'deliberate mistakes' in this design so 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.



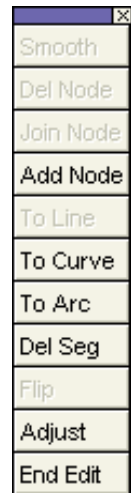
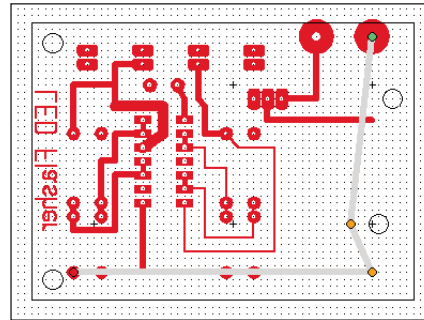
3. Choose the *Select* icon (LH toolbox). Point to the track to be edited (see arrow in the diagram above) and click the LH mouse button to select it. Note that the bottom part of the track is also selected. This is because the track was originally drawn in one operation. 
4. Notice the *Property/Start 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.



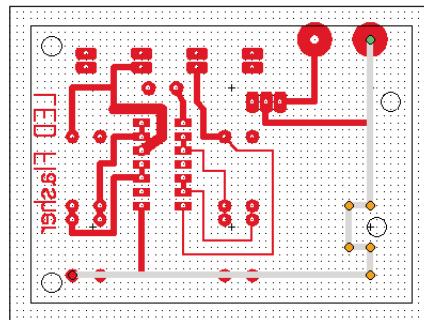
You should also notice that the *Property/Start Edit* toolbox has been replaced by the longer *Edit Mode* toolbox (initially all buttons will be greyed out). This will be used later.

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.)

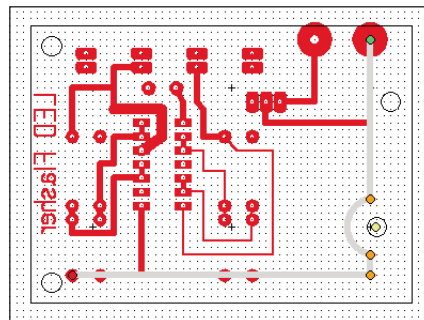
5. In this case moving the node will avoid the hole, but will produce a 'messy' circuit board. A better plan is to add more nodes to the track. First click on the *Undo Last* icon to restore the track to its original position. Then click on the track between nodes with the LH mouse button to sub-select 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.

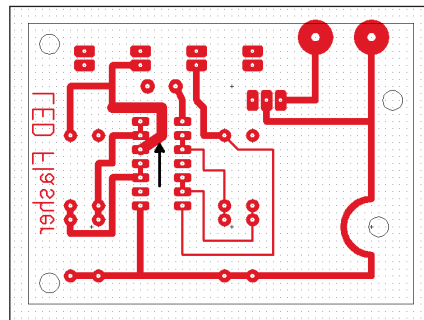
TUTORIAL 9 - Changing Track Width, Pad Properties and Layers

2D PCB allows very simple editing of track and pad properties. This is best demonstrated by working on an existing design.

1. Start up 2D PCB, or close any open drawings (*File > Close*). Choose the *Open* icon from the upper icon toolbar (or *File > Open*), and load the drawing "PCBTutor4.dtb" from the *Tutorials* folder.



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*.

Change Tracks...

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*.

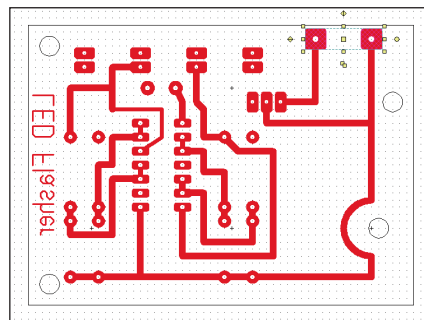
Change Tracks...

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*.


Change Pads...



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 graphical layer to another in a similar way. Note, however, that it is not possible to transfer PCB data to a graphical layer or drawing data to a PCB layer using this method. To do so you must use the special functions in the *Edit > PCB/Drawing Conversion* menu. 

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 2D PCB 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

2D PCB supports two types of font format - Windows Outline font format and TechSoft Font format.

Windows Outline Fonts are widely available in many different typefaces. By definition each character consists of a closed outline. Whilst these are fine for printed output, they are sometimes difficult to reproduce at small scale on a CNC machine such as an engraver.

TechSoft fonts can be created and edited in 2D Design (part of the Design Tools suite). As well as outline fonts they can also be stick fonts, ie., fonts with characters defined by line strokes. These fonts reproduce well on a CNC engraver. Several TechSoft stick fonts are supplied with the software.

There are two different text icons in the LH toolbar, the *PCB Text* icon and the *Graphical Text* icon.

When *PCB Text* is chosen, text can only be placed on the *Solder Side* layer or the *Component Side* layer, and will be red or green accordingly.



When *Graphical Text* is chosen, text can only be placed on the graphical layers.

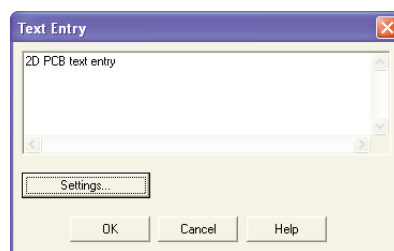


The general method of text input is the same for both types of text, but *Graphical Text* has a wider range of options.

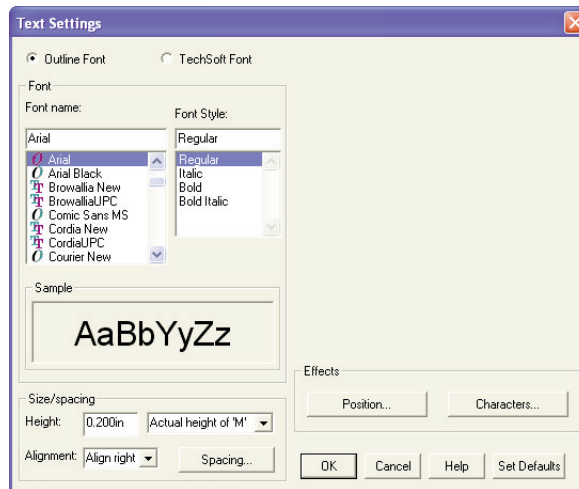
1. Start up 2D PCB, or close any open drawings (*File > Close*). Choose the *New* icon from the upper icon toolbar (or *File > New*). For the purposes of this tutorial only, choose *Help > Restore Factory Defaults*.



2. Choose the *PCB Text* icon (LH toolbox). Choose the layer you wish to work on from the drop down list in the attributes bar. Position the cursor at the bottom LH corner of where you intend the text to be, then click to locate. (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 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. (All of the options are self-explanatory, 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.

If you put text on the *Solder Side* layer it will appear mirrored. This is correct - imagine that you are looking at a transparent board with the PCB layout and text on the back. Text on the *Component Side* layer will not be mirrored.

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.

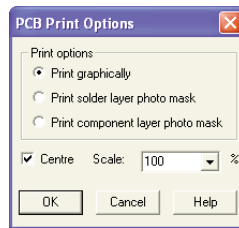
If a PCB board is manufactured on an engraving machine using the default settings, all text will be simply engraved out of the background copper. However, if *Full Copper Removal* is set, Windows outline font characters will be machined as islands, ie., the characters will be left as copper characters. If TechSoft fonts are machined with *Full Copper Removal*, a rectangle of copper around the text will be left and the stick font characters will be engraved into this.

TUTORIAL 11 - PCB Manufacture By Photo-Etching

This tutorial provides an overview of the process of outputting from 2D PCB for manufacture by photo-etching. Users who do not intend to use this process can skip this tutorial.

There are several photo-reproduction systems available that can produce a PCB from an image produced on a laser printer or an inkjet printer.

To print from 2D PCB first set the Windows printer driver up in the normal way then use *File > Print PCB*. 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 monochrome printer. This output is **not** intended for PCB manufacture, but for graphical, reference or other purpose.

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 the *Layout* icon from the upper icon toolbar (or *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.



TUTORIAL 12 - PCB Manufacture By Knife Cutting

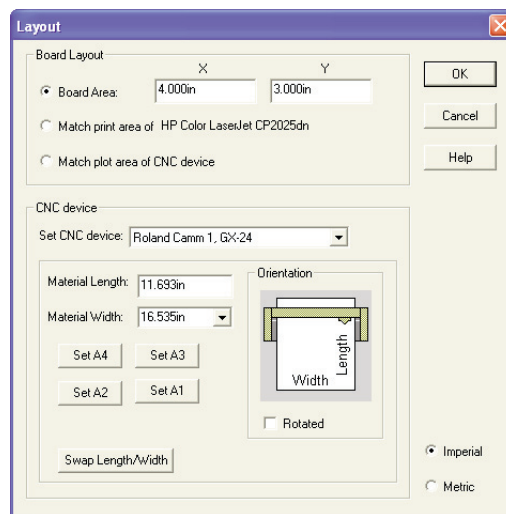
This tutorial provides an overview of the process of outputting from 2D PCB for manufacture by a knife cutting machine (eg., Roland Stika, CAMM 1, etc.) Users who do not intend to use this process can skip this tutorial. Most knife cutters can be used to cut Cutronics copper foil to make simple ‘chunky’ 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 by choosing the *Layout* icon from the upper icon toolbar (or *Setup>Drawing>Layout*). A dialog box similar to the one shown below opens.



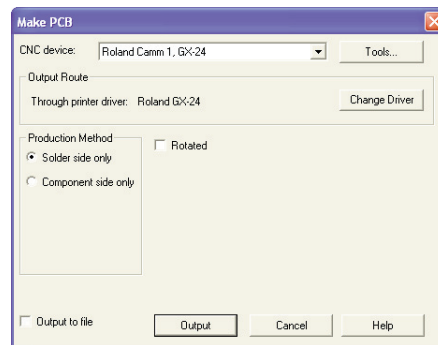
Board Layout: This section of the dialog box allows users to set a suitable size for the *Board Area* displayed on screen (the white area). Most users will want to work on a “sensible” sized board (4 x 3 inches is the default value). It is, however, possible to set the *Board Area* to match the CNC device 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.

CNC 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 CNC device chosen can be saved in a setup file using *Setup > Save Setup* or *Setup > Set As Default* (see **Tutorial 2**).

Output - Step 2

When you are ready to output, choose the *Make PCB* icon from the upper icon toolbar (or *File > Make PCB*). The *Make PCB* dialog box will open. Check the parameters and options detailed below.



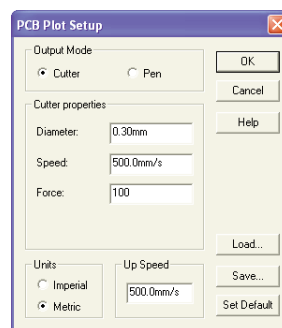
Production method

Solder side only: This option allows only the solder side objects 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 objects to be cut.

Tools

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



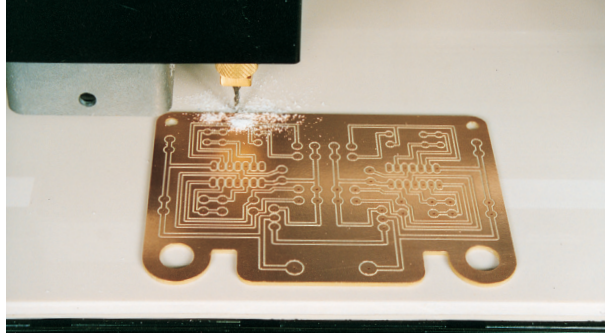
Output - Step 3

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 '*Output (whichever) side now?*' Check that the machine is ready to go, with material in place, then click *Yes*. Follow the screen instructions carefully to completion.

TUTORIAL 13 - PCB Manufacture By Engraving/Milling

This tutorial provides an overview of the process of outputting from 2D PCB for manufacture by an engraving/milling machine (eg., Roland MODELA, MODELA Pro, RotoCAMM, etc.) Users who do not intend to use this process can skip this tutorial.



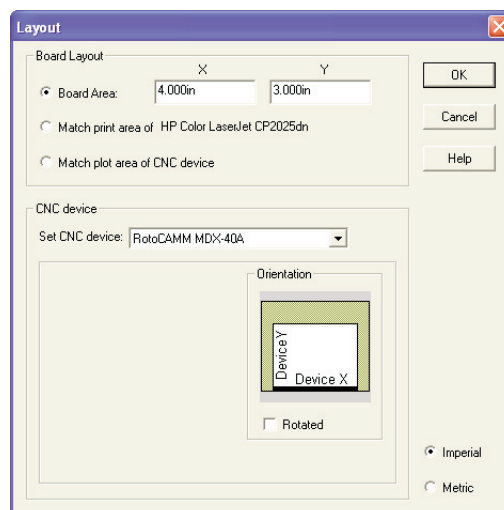
Isolation engraving of PCBs is a technique used in the electronics industry for making prototypes and one-off boards. Using this method a fine line is engraved around each circuit path, isolating it from the rest of the copper, thus forming “tracks”. This can be a very quick process, as only a small amount of copper is engraved away. From a functional point of view there is no need to remove the waste copper from the board, but users may occasionally wish to do this for “cosmetic” purposes when producing a “prototype” product. The removal of waste copper is called *Full Copper Removal*.

It is possible to produce a fully finished PCB straight off the machine using a vee cutter to engrave around the tracks, a slot drill to rout the board profile, and up to four different sized PCB drills to drill the holes in the pads. Optionally the routing out of the board may be missed out, or just a single vee cutter used to engrave the tracks and pads, centre dotting all holes ready for manual drilling later.

2D PCB supports many different machines and each will have its own detailed procedure. However, the principles for all such machines are outlined below. Further comprehensive details regarding outputting to individual machines, and how to get the best out of them, are detailed in the Inset Packs included with most CNC machines supplied by TechSoft.

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 by choosing the *Layout* icon from the upper icon toolbar (or *Setup>Drawing > Layout*). A dialog box similar to the one shown below opens.



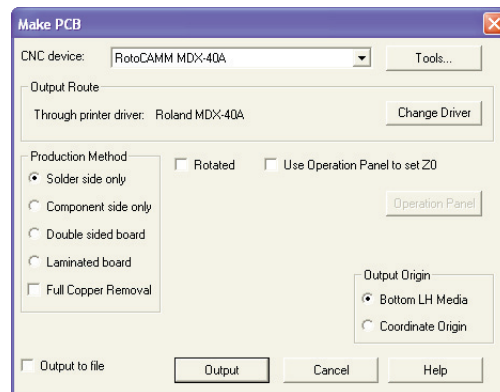
Board Layout: This section of the dialog box allows users to set a suitable size for the *Board Area* displayed on screen (the white area). Most users will want to work on a “sensible” sized board (4 x 3 inches is the default value). It is, however, possible to set the *Board Area* to match the engraver/miller 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.

CNC device: This section allows the engraver/miller to be selected.

The layout and CNC device chosen can be saved in a setup file using *Setup > Save Setup* or *Setup > Set As Default* (see **Tutorial 2**).

Output - Step 2

When you are ready to output, choose the *Make PCB* icon from the upper icon toolbar (or *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 objects 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 objects to be machined.

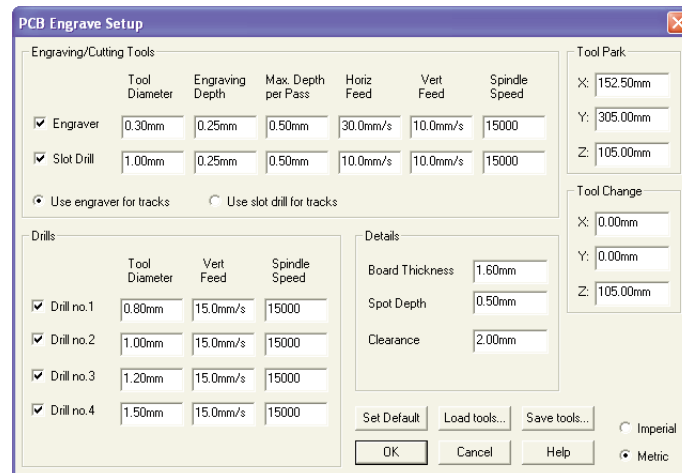
Double sided board: This is the standard option for double sided boards. The process initially machines the solder side of the circuit. User prompts then ask for the board to be turned over (flipped left/right). This must be done very accurately to be successful. The software assumes that the physical 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 1mm 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.

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 2D PCB. 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 effective 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 on most machines, but bear in mind that they have been set to ‘safe’ values. With some experience many users should be able to increase the feed rates significantly.

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 ‘*Output (whichever) side now?*’ Check that your machine, material and tools have been set ready to go then click *Yes*.

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.

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 can normally be accommodated by simple adjustment - see the TechSoft Inset Course manual supplied with your machine for details.

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.

TUTORIAL 14 - Component Libraries and Creating New Pads

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.

2D PCB uses the standard Windows filing system. All the basic components and building block circuits are kept in folders within the 'PCBLib' folder. The names of most of the folders are self-explanatory. Components are imported into the drawing by choosing the *Component* icon from the LH toolbox (or *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 the *Ungroup* icon from the upper icon toolbar (or *Edit > Ungroup*).



Creating Your Own Component 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 the *Component* icon from the LH toolbox (or *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 folder where all your own components are saved together.



Creating New Pads

There are a number of pad sizes pre-defined in 2D PCB 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 *Tutorial 2*).

TUTORIAL15 - Links With Other Software

For users who wish to autoroute circuits, or who wish to have the ability to produce PCBs from schematics, TechSoft have developed a link with Circuit Wizard (formerly PCB Wizard).

Having created your layout in Circuit Wizard choose *File > Make > Output to CAD/CAM*. This will transfer the data to the Windows clipboard. Start 2D PCB, and choose *Edit > Paste*. N.B. Only the *Solder Side*, *Component Side* and *Both Sides* layers are transferred.

Once transferred, Circuit Wizard designs can be treated in exactly the same way as designs created in 2D PCB.

Importing Drawings

Drawing data, including commercial clipart, can be imported into 2D PCB. A variety of formats will be accepted, eg., *wmf* (*Windows metafile*), *dxf* (*Data Exchange format*), etc.

The data can be imported by choosing the *Import File* icon from the upper icon toolbar (or *File > Import*). Alternatively, in the other application, *Copy* the drawing to the clipboard then in 2D PCB use *Edit > Paste* to import the drawing.



N.B. To convert drawing data to PCB objects (eg., pads and tracks) use the *Edit > PCB/Drawing Conversion* menu.

TECHSOFT

TechSoft UK Limited
Falcon House
Royal Welch Avenue
Bodelwyddan
Denbighshire
LL18 5TQ

Tel : 01745 535007
Fax : 01745 535008
Web site : www.techsoft.co.uk
Email : email@techsoft.co.uk