Merlin PCB Designer – Printed circuit design using CorelDRAW

Shareware version 2.1 - for CorelDRAW 7 to 11

Introduction

Probably you are professionally involved with printed circuit board manufacturing, wondering where to find a commercial version of a printed circuit board (PCB) design program that is up to the job of PCB routing without becoming overly complicated and expensive. Or maybe you are just making PCBs in small numbers, for your job or at home, and even a non-commercial version would constitute a significant part of your monthly expenses.

In either case, you may have found that PCB layout programs tend to be rather expensive and complicated pieces of software. But you have a nice graphics software already on your computer, like CorelDRAW, and you wonder whether you could use it for designing PCBs. In fact you can. The full version of Merlin PCB designer is an advanced package that offers you the possibility to just that.

Merlin PCB designer comes as a downloadable zip-file containing some files to customize your CorelDRAW environment in such a way that it is suitable for PCB layouting. The same zip-file also contains a component library, where you can find almost any component you
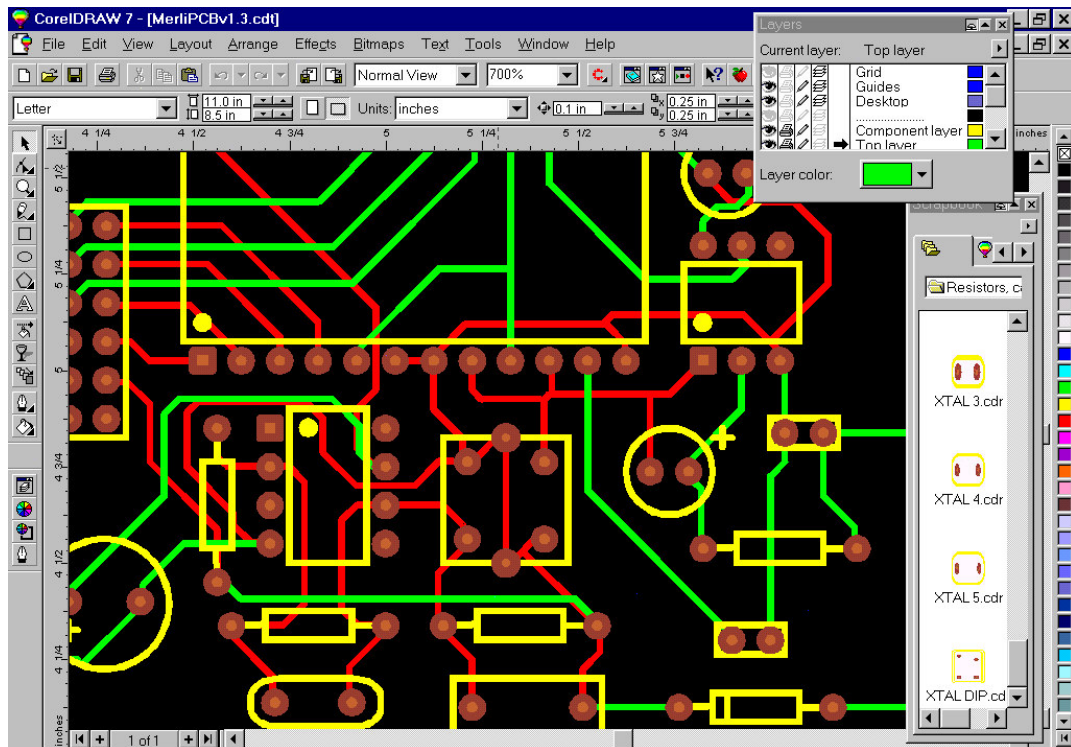

Fig. 1. Merlin PCB Designer

may ever need. The full version component library is much larger than the shareware library. This manual shows images of CorelDRAW 7. It is possible to use all versions from CorelDRAW 7 to 11. However, the windows described in this manual may look a bit different in other versions, and they may be found in a different menu than described here. Older versions of CorelDRAW than version 7 will not work with Merlin PCB Designer. The files to automatically generate plots and change layers are different for the different CorelDRAW versions. This will be discussed in the paragraph on automation.

The environment

In Fig. 1, we show how you will actually be routing PCBs. Components can be dragged from a library window at the right of your PCB layout. In addition there are two copper layers available, for designing one- and two-sided PCBs. The copper lines can be any thickness, but the default colors should not be altered unless you have experience with CorelSCRIPT. These colors are green for the top layer and red for the bottom layer. Both normal and surface mounted devices (SMD) layouts are available in the library for all common components. A grid is available and pressing the CTRL key will snap the copper lines to angles of multiples of 15°, for a neat layout. Rotating and duplicating are standard to the CorelDRAW package. The only thing not available is an auto-router!

After your layout is ready, you can change it to a mask ready for printing. If you use a laser printer and (semi-)transparent paper, you have a mask ready for the PCB production using photosensitive PCB material. In the rest of this manual, we will assume that you have reasonable experience with graphics software, like grouping objects or sending to front/back commands. If not, use the online help files of CorelDRAW if required.
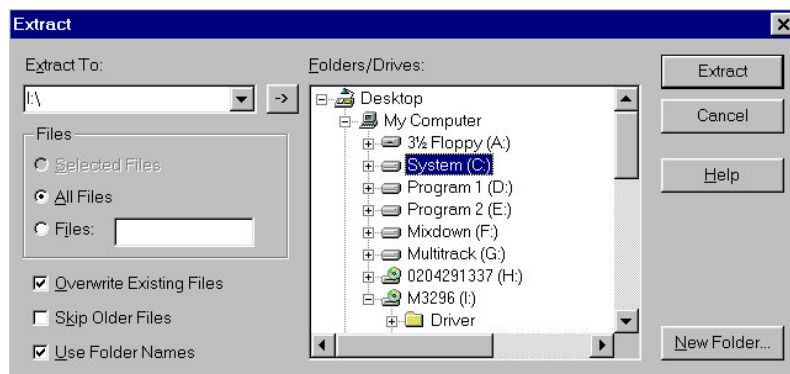
Installing the package and setting up

Fig. 2. Unzipping Merlin PCB Designer.

Unzip the file "MerliPCB.zip", and choose the root folder C:\ to unzip it in. Enter the password provided with your licence to unzip the file. It is not possible to use another location for the files, because part of the automation will not work if the merlipcb directory is not at

C:\. (Fig. 2). Check the box "overwrite existing files", and the "Use Folder Names" box. A folder "MerliPCB" is added to the root directory, containing the library, template and system files. Merlin PCB Designer 2.1 is now installed.

Getting started

When you start CorelDRAW after installing the package, you will not yet notice any changes. Open the file "MerlinPCB.cdt" in the MerliPCB\System\ folder (set the files that are displayed to *.cdt if you don't see it in the system folder). In this file the layer settings of the Merlin PCB environment are stored. A window containing questions about editing the template will appear. Unless you plan to change the appearance of the Merlin PCB environment, do not change the settings and press Ok (the working area turns black).

First zoom in to a scale appropriate for PCB layouting. This will usually be somewhere between 200 and 800%. If you like you can add a square with a white outline defining the board outline of the PCB (e.g. 100x160 mm).

Open the Layout≻Grid and ruler setup window. It will look like the one in Fig. 3. Set the horizontal and vertical spacing to 0.0125 inches and the "Snap to grid" on, like indicated in Fig. 3, and press ok.
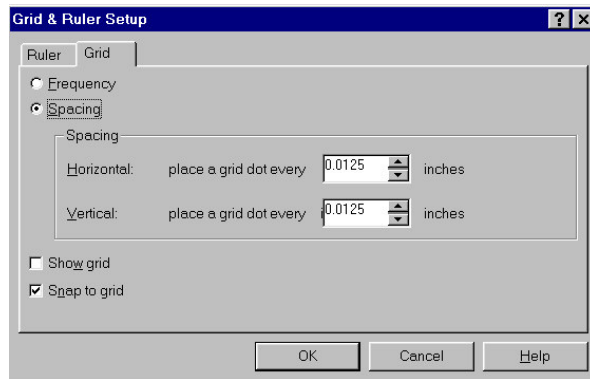

Fig. 3. Setting the grid

The component layer

Like dedicated PCB layout packages, Merlin PCB designer uses layers. These are defined in the layer roll-up window (Fig. 4). If it is not ready on top on your drawing area, you can find it in the menu View≻Roll-Ups≻Layer Manager. Here you find eight layers: The top three are used by CorelDRAW itself, the middle three by Merlin PCB for normal components and two copper layers. It is these you can use in a layout with through-hole components. The bottom two layers are used in conjuction with SMD devices. Drag and drop SMD components according to where they should appear in on the PCB, either on the SMD top layer or the SMD bottom layer.
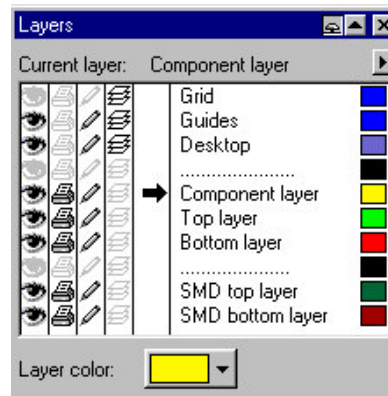

Fig. 4. Layer roll-up window

Layers can be made visible or invisible by clicking on the "eye" in the Layer Manager. We will refer to it as the "eye tool". During the routing process you may want to make layers temporarily invisible to get a better overview of the layer you are working on. If the Layer Manager window takes up to much space in your screen, you can press the little arrow at the top right and only a small part will remain visible.

The Merlin PCB library can be used easily with the scrapbook roll-up (Fig. 5). If not visible, open it by selecting View≻Roll-Ups≻Tools≻Scrapbook. Browse through the component library of MerliPCB until you have found the component you need (another paragraph later in this manual is dedicated to the library contents).

Pick the component with your left mouse button and drag it to the position you want to have it on your PCB layout (Fig. 6). Rotating can be done using the normal features of CorelDRAW in the top bar. A large number of SMD components is also available in the full version.

The full version contains some advanced features not available in the shareware version. One of them is the automation of changing layers. To be able to use them, the first time the full version is used, the scripts have to be copied to the scripts window. This is done as follows.

Open the Tools≻Scripts≻Script and Preset manager window (Fig. 7). The next time you open the window, they will appear automatically.

Fig. 5. Component library in scrapbook

In other versions of CorelDRAW than 7, it is advisable to put buttons next to the copy and paste buttons of CorelDRAW. This is done by going to the menu Tools≻Customization≻Commands≻window≻Corel scripts manager and drag the "scripts manager button to a place next to the copy button at the top left of your screen. The same can be done with the scrapbook button. Once they are there, you can always open them by clicking on them.

Open the "System Corel X" folder in the MerliPCB folder with the Windows Explorer or using "My Computer" on your desktop, with X the number of your CorelDRAW version. A number of files containing the extension *.csc are there, although less than in Fig. 7, because some of these are only available in the full version. Drag the files with this extension into the Scripts window like in Fig. 7 in the same way you drag a component to your drawing.

Fig. 6. Dragging a component

Put the black arrow near the middle of the Layer Manager window in the component layer if it is not already there. The Merlin PCB library can
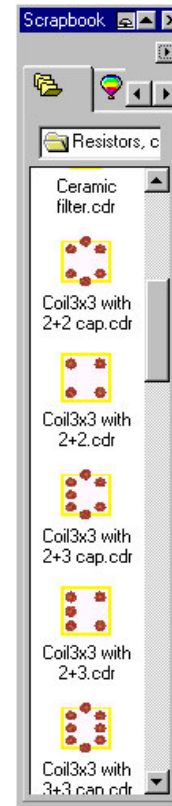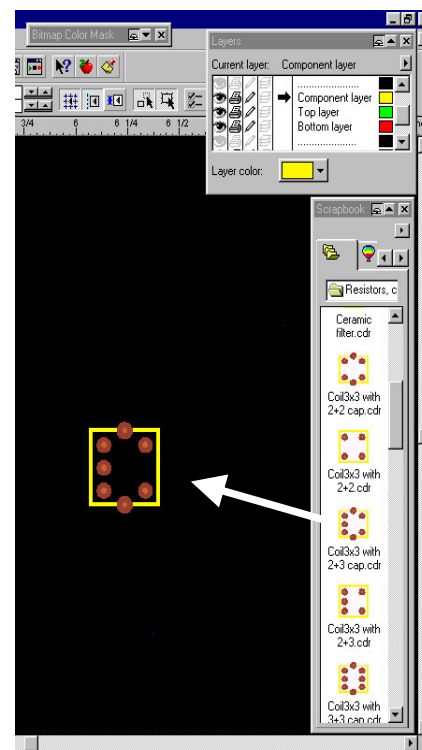
be used easily with the scrapbook roll-up (Fig. 5). If not visible, open it by selecting View➤Roll-Ups➤Tools➤Scrapbook. Browse through the component library of MerliPCB until you have found the component you need (another paragraph later in this manual is dedicated to the library contents). Pick the component with your left mouse button and drag it to the position you want to have it on your PCB layout (Fig. 6). Rotating can be done using the normal features of CorelDRAW in the top bar. SMD components are not covered in the shareware edition, but are available in the full version.

The top and bottom layers - drawing copper

Be sure that you have no object selected by the time you want to change from drawing in one layer to another. In case you are not sure, pick the arrow from the tools at the left of your workspace and click it somewhere in the page on nothing. Otherwise you run the risk of changing your components from the component to a copper layer. If you want to see the objects in a certain layer, you can make all other layers temporarily invisible in the Layer Manager.

Put the black arrow in the Layer Manager in the layer you want to draw. Then click on the pen-symbol from the tools at the left (Fig. 7). A window opens that lets you choose the color and line-width of the copper line you want to draw, among other things you may want to change, like roundings. Greenn is used for the component side (if required), and red for the copper on the solder side. To use the automatic generation option for the top and bottom layer to print the layout for making PCBs, you should stick to these colors unless you are experienced in editing Corel script files. If you are working on complex PCBs you may opt for some more colors to enhance clarity, e.g. different colors for power and data lines.

You can now draw any connections between the components, on two layers if required.

Remember to change layers both in the Layer Manger and using the right color every time you change layer, and make sure that nothing is selected at the moment you change your layer in the Layer Manager.

Note that in the full version, changing between layers and putting components is automated: only one click and you can draw a trace or put a component.

The library

There are two versions of the library; a shareware version containing only the most common components, and a full version containing a large amount of more specialized components including SMD. Here we will only cover the shareware version library. In the "MerliPCB" folder, there is a folder entitled "Components". The component folders usually appear automatically in your scrapbook window, but if not, you can browse within the Scrapbook window to find the Components folder.

You will find the following folders:
- Connectors
- DIP ICs
- Resistors, capacitors, coils and XTAL
- Transistors
- Other semiconductors
- Custom (empty)

These contain components often used. Of course it is possible to define your own components. If you take a component which looks a bit like the one you want to have and drag it to the workspace, you can edit it like any object in CorelDRAW after "ungrouping" it. Note that the grid will help you with the space between connecting pins. When your device is finished, save it in the "Custom" folder as a .cdr file. You can now access it like any other device using the Scrapbook window.

Merlin PCB Designer is very interested in layouts of uncommon components, which you can send as a .cdr attachment by email to merliPCB@hotmail.com.

Converting your PCB drawing to a printable version

If you want to print all layers, you can print them the normal way in CorelDRAW. The background is not printed, so if you like a black background you will have to define black square big enough to cover the whole layout, and "send it to back". Separate layers can be printed as well using the eye in the Layer Manager.

If the layout is to be used in a PCB production process involving photosensitive layers on the PCB material, a transformation is required. Because this transformation is rather elaborate, it has been automated using script files for easy, one-button operation. These files are "Generate top layer" and "Generate bottom layer"

Double clicking one of the two icons will start an automatic process that may take a while if you are working on a complex PCB. When the process is finished, you will find that a new drawing has been created. It contains the printable top or bottom layer for PCB production using the photosensitive layer method. The top layer is automatically mirrored for this purpose, while the bottom layer retains its original orientation. This only works with the default colors!

Fig. 7. Scripts for generating top and bottom layer prints

Printing for photosensitive PCBs is possible on overhead transparencies, chalk paper, or professional transparent paper especially meant for this purpose. Overhead transparencies tend to give lower resolution printouts than do the other two. A4 or letter size chalk paper is a cheap solution resulting in very high quality images.
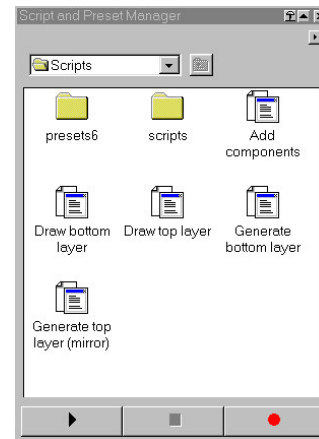
Comments on Gerber plots

Merlin PCB Designer is very well suited for PCB production using the photosensitive layer method. However, professional PCB manufacturers usually require the file to be in Gerber format, which is not supported by CorelDRAW, nor by Merlin PCB Designer. A lot of companies do exist that are able to do the conversion for you, but prices are generally rather high. Some shareware packages are available that seem to do the job. They do not accept .cdr files but AutoCAD .dxf files. This format can be generated by CorelDRAW (you will find the option under the types of files you can save your drawing in). You will also need a Gerber viewer.

Although these conversion programs probably work, Merlin PCB Designer does not guarantee that the conversion to Gerber plots using them is without problems. The safest way is to contact a company specialized in data format conversion. Many of them can be found on the internet.