

Power Electronics Lab

Printed Circuit Board Design

02/16/1999

Figure 1 summarizes the process of fabricating a printed circuit board using the LPKF milling machine in the Power Electronics Laboratory. PCB design starts with circuit design and schematic capture using CircuitMaker. CircuitMaker has a large library of component symbols and models. New symbols and models can be easily added. A Spice simulator within CircuitMaker can be used to test circuit operation, provided that Spice models for all components are available. CircuitMaker output is a netlist file that describes component packages and interconnects. The netlist created by CircuitMaker is the input for TraxMaker, a PCB layout design tool. TraxMaker has a large library of standard component footprints. New footprints can also be added. Tools for automatic or manual component placement and routing of interconnect traces are available. TraxMaker outputs layout design in industry-standard format (Gerber layer and Excellon drill files). These files are inputs for LPKF CircuitCAM, a tool that prepares the fabrication job for the milling machine. In the final step, the mill/drill file prepared by CircuitCAM is used by LPKF BoardMaster, which is a tool that controls the milling machine and executes the milling/drilling job.


In the PCB design process, a large number of intermediate files are created. It is a good idea to frequently save your work, and to backup the files using ftp, a floppy or a zip disk.

PCB design steps are described below in more detail. Examples are available on the Web.

1. Schematic Capture Using CircuitMaker

CircuitMaker is a simple tool for editing circuit diagrams. Each device symbol entered in a circuit diagram includes several data entries. The device data can be edited on the **Edit Device Data** window that pops up when you double-click a device and select **Netlist**, or if you right-click on the device and select **Edit Device Data**. It is important to understand the following entries:

- **Label-Value** shows device type, such as IRF640 for a power MOSFET
- **Designation** is the device label, such as Q1 for the power MOSFET
- **Package** is the name of the TraxMaker footprint, such as TO220H, which is a footprint for a power MOSFET in TO220 package mounted on the lab-kit heatsink.
- **Bus Data** is usually empty. This field can be used to indicate that a node is connected to other nodes with the same **Bus Data** label. This is particularly useful for ground and power connections. For example, VCC; entry in the **Bus Data** field indicates that the device node is connected to the VCC supply.
- **Spice Data** is a field that shows how the device is described in a Spice netlist for simulation. If the field is empty, the device Spice model is not available, and the circuit simulation is not possible.
- **Exclude from PCB** box should be checked for devices that should not be included on the printed circuit board. For example, a DC supply voltage source can be included on the circuit diagram, but since the voltage source is not on the PCB itself, **Exclude from PCB** box should be checked. All devices that should be placed on the PCB (including plugs, connectors, jumper switches, etc.) should be included on the circuit diagram with **Exclude from PCB** box unchecked.

Save your circuit diagram as *filename.CKT*. To generate the PCB netlist, click , accept all defaults in the **Export PCB Netlist** window, and save the netlist as *filename.NET*. TraxMaker starts, and the **Netlist Load** window should show 0 missing items. Check **All Nets** in the **Show Nets** section and click **OK**. Missing items are reported if any of the footprints you specified in CircuitMaker are not available in the libraries loaded by TraxMaker. You may need to add a library or edit and add a new footprint in TraxMaker, and then reload the netlist file.

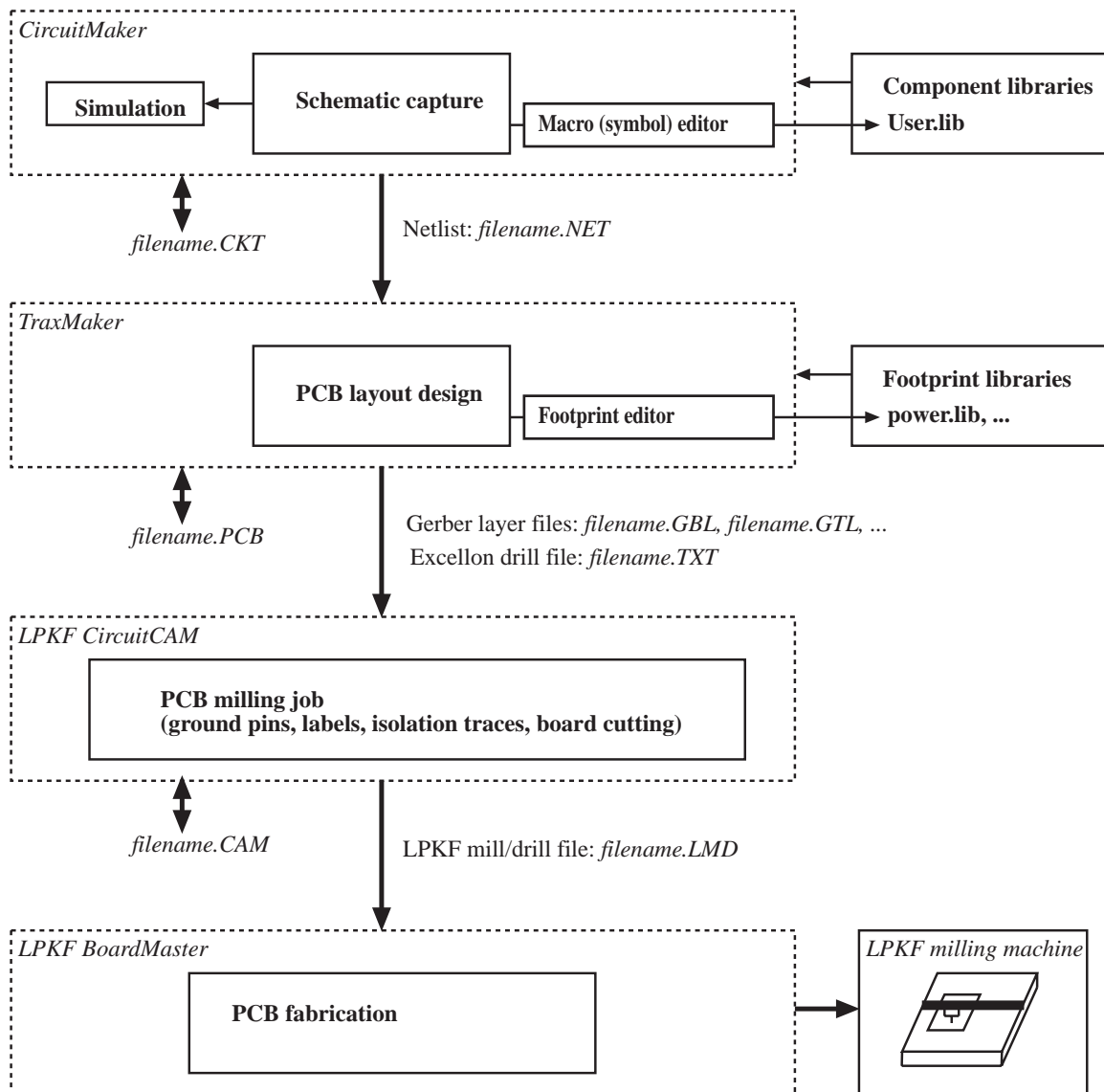


Fig.1 Printed-circuit board design steps

2. PCB Layout Design Using TraxMaker

2.1 Component placement

The first step in the PCB layout design is placement of components. Initial auto-placement of components rarely gives satisfactory results. It is recommended that you manually place components to minimize the board size, simplify routing of traces on the board, have all plugs, connectors etc. grouped and placed appropriately, etc.

2.2 Check ground connections

In TraxMaker, a connection to ground should show up as a green cross on a pad, with no net connections between this pad and other pads. Using the PCB milling machine to manufacture the board, we can leave a ground plane on both top and bottom sides of the board, and therefore ground connections require no routing. If TraxMaker does not show green crosses at the ground nodes, if it shows net connections between ground nodes, or if green crosses are shown on nodes that are not ground connections, then: (1) check SMD to Pwr/Gnd box in **Setup > Router** window, and/or (2) identify the ground net in **Netlist > Pwr/Gnd Nets**. You may then also need to discard the design and reload the netlist. You should

not proceed before all ground connections show up correctly, with green crosses and no nets connected to the ground pads.

2.3 Routing


The main part of PCB layout design is routing of interconnections (nets). TraxMaker includes an auto-router that can simplify the routing task. With a through-hole (as opposed to surface-mount) PCB fabricated using the milling machine, it is most convenient to place most if not all of the routing traces on the bottom side (bottom layer) of the board. The default router setup (in **Setup > Router**) should be with the bottom layer as the single layer. Set the default track width to 40 mils (**Setup > Tracks**) and select **Route > Board > OK** to start the auto-router. The auto-router should be able to complete most of the routing, but further manual or semi-manual adjustments are most often needed. You may need to do manual routing of individual nets, complete routing on the top layer, retry auto-routing on both layers, reduce the default track width (but do not use less than 20 mils), edit auto-routed nets, etc. TraxMaker has many options available to complete the routing task, and the best approach is to simply experiment a bit.

In designing layout for power electronics circuits, it is necessary to make sure that track widths are adequate for the current levels expected in the power stage. As a general rule, using a standard “1oz” copper-foil board, the track width should be greater than or equal to about 25 mils per Amp of current.

2.4 Design Rule Checking

Once you completed routing of the board, it is a good idea to run a design rule checker (**Netlist > Design Rule Check**), to verify that the connections you made match the nets loaded from your circuit diagram. The DRC function can also be customized to check for a number of other rule violations.

2.5 Board outline, spacer holes, etc.

With placement and routing completed, you should minimize the board outline, i.e. the box drawn on the “keep out layer” around the board, and add any mechanical components that were not included on the circuit diagram. For example, to add holes for the board spacers, you can place ROUND250 pads (click on  to add a pad) on the corners of the board. Double-click on the pad to check **Relief to Ground** box, so that you can later in CircuitCAM remove the pad and leave only the screw hole for the spacer. Also set the hole size to 150 mils.

2.6 Export Gerber and Drill Files

In the final step of the PCB layout design, it is necessary to export Gerber layout files and drill files. Save your work in *filename.PCB*. Do the following:

- **File > Create Gerber File**
- Keep all defaults (0 all offsets, 0 all Aperture Matching, check Auto Generate Aperture File, check Embedded Aperture RS274X)
- Specify *filename.Apt* and *filename* (for Gerber files)
- **Click More Options**
 - Check Batch Print
 - Check Top Layer, Bottom Layer, Top Overlay, Keep Out Layer, and Ground Plane
 - Check Pads, Vias, Text, Single-Layer Pads
 - Uncheck Title Block
 - Keep all other defaults
 - **OK**
- **OK**

This will create a set of Gerber files that describe various layers of the PCB layout design.

Next, create drill files:

- **File > Create N/C Drill File**
- Specify N/C Drill File name: *filename.DRL*

This will create a set of drill files that describe location and types of drill holes on the board.

3 Milling Job Preparation Using LPKF CircuitCAM

CircuitCAM is a tool that prepares the design for manufacturing.

Start LPKF Prototyping > CircuitCAM, and start a new design: File > New > Default. If this is the first time you started CircuitCAM, it will work as a demo version. Go to Config > General Settings > User, and enter the serial number and the enabling number (available on the Web page with PCB design instructions). Then shut down and restart CircuitCAM.

3.1 Import layer and drill files

3.1.1 Import definition of drill tools:

File > Import > *filename.TOL*

In the Import window,

check File Type: Aperture/Tool select

Aperture/Tool List: enter *traxdrl*

Aperture/Tool Template: select *TraxTools.txt*

OK

3.1.2 Import definition of apertures used to define shape, size and position of layers:

File > Import > *filename.Apt*

In the Import window,

check File Type: Aperture/Tool select

Aperture/Tool List: enter *trax*

Aperture/Tool Template: select *TraxApe.txt*

OK

3.1.3 Import Gerber-x layer files using *trax* Aperture/Tool List:

File > Import > *filename.GBL* as BottomLayer

File > Import > *filename.GGD* as SolderMaskTop

File > Import > *filename.GKO* as BoardOutline

File > Import > *filename.GTL* as TopLayer

File > Import > *filename.GTO* as SilkScreenTop


3.1.4 Import drill (Excellon) file using *traxdrl* Aperture/Tool List:

File > Import > *filename.TXT* as DrillPlated


At this point, layout in CircuitCAM should look like the layout in TraxMaker. If any errors were reported during the import steps 3.1.1-3.1.4 you should stop and consult with the instructor.

3.2 Ground connections

On the PCBs fabricated using the milling machine, all unmilled copper can be used as ground (or power) plane on both sides of the board. Therefore, no pads or traces are needed to make ground connections. Since the layout prepared in TraxMaker includes BottomLayer and TopLayer pads around ground connections, these pads should be removed manually in CircuitCAM. To help identify the pads to be removed, you can look at the SolderMaskTop layer which was used to import the ground layer from TraxMaker. Ground connections pads are only outlined as opposed to other pads which are filled in this

layer. To remove the ground pads, click Layers button , and uncheck Selectable box for all layers except the TopLayer and the BottomLayer. Close, return to board editing, select and delete TopLayer and BottomLayer pads from all ground connections. Only a drill hole is needed to connect to the ground plane on either side of the board.


3.3 Text Labels


To place text labels, first make only the TextTop (or TextBottom) layer selectable. Then, choose TextTop (or TextBottom) from the pull-down menu. Click on the text button  to place text labels on the board. Text size should be at least 2mm. Finally, select all text labels on the board, and choose LpkmMillingTools and UniversalCutter 0.2mm from the pull-down menus as shown here:




Click on the arrow button to unselect all text labels.

3.4 Board cutting

The milling machine can cut the PCB out of a larger board. With nothing selected on the board, click  button to open the window used to define the board cutting trace. Accept all defaults (Layer > BoardOutline, Cutting Outside, LpkmCuttingTools, ContourRouter 2mm long, GapWidth 1mm (or 39.3 mils)), and click Run. This will create a CuttingOutside trace around the BoardOutline trace which was imported from TraxMaker.

Make only the CuttingOutside layer selectable and return to board editing. Select a segment of the CuttingOutside trace. Use + or - to position the cross X in the middle of the segment. Click  to insert a gap in the segment. Insert one gap in the middle of each of the four sides of the outline box. The gaps are necessary so that the board stays in place as the machine cuts along the CuttingOutside trace. Finally, once the cutting traces are generated, delete BoardOutline on the board.

3.5 Generate Isolation Layers

The main feature of CircuitCAM is the ability to calculate isolation traces for a given board layout. Simply click  and, after some processing, IsolateTop and IsolateBottom traces along which milling will occur are generated. There should be no errors reported. This step completes preparation of the board for fabrication.

3.6 Export drill/mill file

Save your work in *filename.CAM*. Generate milling/drilling file: File > Export > LPKF CircuitBoardPlotter. File *filename.LMD* will be generated. If errors are reported, the report file should be inspected (Window > Report). In most cases, the errors can be ignored because they relate to manufacturing phases that are not used on the milling machine in the lab.

4 PCB Fabrication Using LPKF BoardMaster

Copy *filename.LMD* to a floppy. This file is used by LPKF BoardMaster which runs on the PC attached to the milling machine. An instructor should be present to help with manufacturing the board.