

Fabricating Printed-Circuit Boards (PCBs) – Submission Procedure

Lucas Andrew Milner, *Student Member, IEEE*, and Gabriel A. Rincón-Mora, *Senior Member, IEEE*
Georgia Tech Analog and Power IC Design Lab

Given the difficult work required to design a PCB prototype, submitting the design for manufacturing should be painless and automatic. The process involves a few steps, which are relatively easy to learn and execute with a guide. Providing such guidance is the purpose of this document.

A sample PCB schematic and corresponding layout designs are shown below in Figure 1. The board design embodies the design and test plan teachings discussed in the following two documents:

www.rincon-mora.com/research/pcb.pdf

www.rincon-mora.com/research/pcb_plan.pdf.

Holes for banana jacks and probe hooks are combined with familiar surface-mount components to enable thorough measurements without requiring significant PCB real estate and/or exhaustive PCB layers. These features are specified and labeled within both the schematic and layout for reliability and debugging ease, which is especially important in the lab evaluation phase. All of these features can be implemented on a two-layer board, which is not only easier to handle and less expensive but also expeditiously delivered.

There are several other useful practices a designer should employ before submitting a design to make certain that the board returned from the manufacturer is easily assembled and tested. First, all parts needed in the design should be in hand before a design is submitted. Board layout programs allow designs to be printed to scale, and the designer should take advantage of this by placing all parts on a print out to verify that all footprints are accurate. This will also save time and money because the designer will not have to wait for parts to arrive after the board or pay to have them delivered more quickly. Also, since it may be desirable to examine chip features with a microscope or to photograph the populated board, the dimensions of the board should be chosen to fit the probe station, if possible. Then, holes for isolated posts should be added at the corners of the board so that it can stand level for soldering and testing and so that traces, jacks, or components are not short-circuited when rested on a conductive (e.g., metallic table or mat). Also, it will be helpful to connect ICs to the board through sockets because replacing damaged or unsatisfactory ICs is time-consuming and may damage the board, if repeated several times. This is especially important for the device under test because it will likely be replaced many times. Sockets are much cheaper for dual in-line (through hole) packages than for surface mount packages and significantly easier to solder, and for these reasons they should be preferred. Also, sockets do not have to fit an IC perfectly. Only the pin width and pitch (separation) and the IC width are critical – if the length of the IC is shorter than the socket, it may still work. Verify that an IC will fit into its socket before the board is ordered. All of these suggestions should help save time and effort during assembly and testing, and will therefore avoid roadblocks and delays, in the long run.

Two companies that specialize in manufacturing PCBs for prototyping and other low volume orders are Advanced Circuits (www.4pcb.com) and PCB Express (www.pcbexpress.com). Both companies fabricate custom two-layer boards and ship them the next business day, and offer a variety of other specials that vary from minimum order requirements, lead times, and number of layers to price. Advanced Circuits tends to be less expensive, unless the designer is willing to accept sponsorship from PCB Express and the obligations tied therein.

To order a simple two-layer board from either company, three or four files must be generated and compressed into a single zip file, which will be uploaded via the website. The essential files are:

- *.top – A Gerber file showing top level traces, pads, and vias.
- *.bot – A Gerber file showing bottom level traces, pads, and vias.
- *.drc – A text file giving the x and y coordinates of all holes.
- *.oln – A Gerber file showing the outline of the entire board.
- *.slk – A silkscreen or legend layer showing the outline of all components and their assigned names and values.

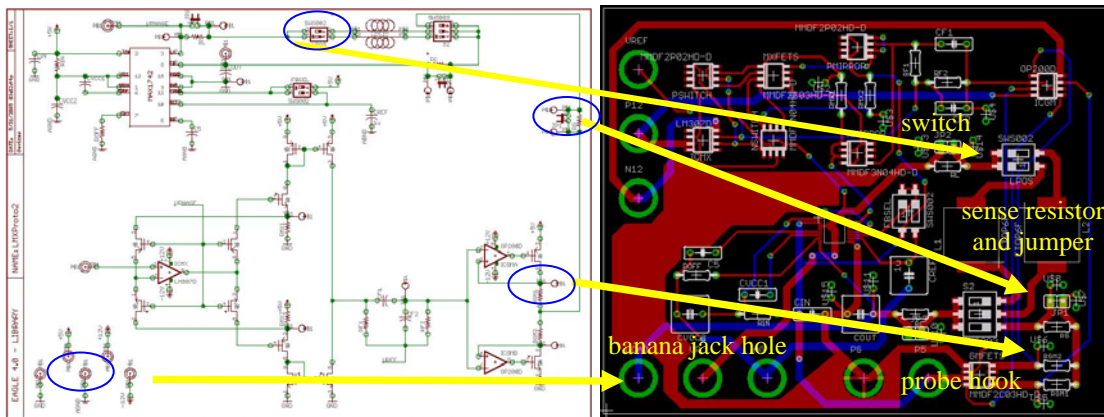


Figure 1. A sample PCB schematic and corresponding layout highlighting the use of connectors and test points.

If desired, the .oln file can be combined with the .top or the .bot file. The addition of a silkscreen or legend layer (.slk) is important for evaluation and debugging ease. Including the legend layer may incur additional cost and shipping delay in the order of a couple of days. These files are created using a computer-aided manufacturing (CAM) processor, which is built into the software Eagle. A free but limited version of the software is available at www.cadsoft.de/. Instructions for using the CAM processor are included below.

Using the PCB Software

1. The Correct Units

First, open the board file (*.brd) in Eagle. Type *RUN* in the command line, or select *Run...* from the File menu. The open file dialog window appears and shows the contents of a directory called *ulp*. Choose the file called *drillcfg.ulp*, which gives the option of choosing *mm* or *inches* as the units for the output files. Make sure to use *inches*, because AC and PCBX expect it. Eagle then shows a list of all the drill sizes that are used in the board design. After this list is accepted, Eagle asks for a place to save the list, which will have the file extension *.drl*. This information will be included in the *.drl file so the *.drl file is not really needed. The important thing here is to know that the output files are in inches.

2. Creating the Drill File

Click on the CAM button, which is in the action toolbar and looks like two turquoise frames on a roll of film – a window for the CAM processor opens. Choose *File*, then *Open*, then *Job...* from the menu. From the ensuing open file dialog, choose *excellon.cam*, which is in folder *cam*. Note that this changes field *Job Section* to “Generate drill data” and the *Output Device* to “EXCELLON.” Note also that the file to be created will have the extension *.drl* and only layers 44 and 45, corresponding to Drills and Holes, are selected in the list on the right. Do not change anything here, just click *Process Job*. The *.drl file is now in the project directory. This is a text file, unlike the others, which are all Gerber files. This file contains nothing but hole sizes and x and y coordinates in ASCII format.

3. Creating the Gerber Files

At this point, the designer can take a shortcut, courtesy of PCB Express. They offer CAM scripts to generate the files required by each level of their service (<http://www.pcbexpress.com/technical/tips.php#conv>). After downloading the CAM scripts, simply open the one that corresponds to the chosen PCB Express service in the CAM processor window and click *Process Job* to generate every file necessary. Of course, these files can be sent to any other company, if the designer chooses to do so because of better pricing options.

To begin managing the CAM processor, the CAM processor window must first be opened – note there is only one folder tab open. This folder tab must be changed, and new ones as well. This blank Job can be saved using any chosen name. The Output Device should be changed to “GERBER_RS274X,” making sure the offsets are zero and that only “pos. Coord,” “Optimize,” and “Fill Pads” are selected in the Style Box. For each Gerber file to be created (*.top, *.bot, *.oln, *.slk), only the *Job Section*, *Output File*, and selected layers will be changed. For example, to define a top layer, “Top Side” should be typed in *Job Section*, *Output File* should be changed to “.top,” and every layer, except 1, 17, and 18 (corresponding to Tops, Pads, and Vias), should be deselected. A new tab should be created by clicking *Add* for every Gerber file that the manufacturer requires. As mentioned above, most basic PCB manufacturing services require three Gerber files, in addition to the *.drc Excellon file (for drills and holes). The layers that must be selected in each case are summarized in Table 1. When done, the Job should be saved for future reference and executed by clicking *Process Job*. All the Gerbers files are generated exactly as they would have been had the scripts from PCB Express been used.

Table 1. The Most Common Gerber Files and the Layers Encoded in Each.

Job Section	Top Side	Bottom Side	Outline	Silkscreen or Legend
Output File	.top	.bot	.oln	.slk
Layers	1 Top 17 Pads 18 Vias (20 Dimension)	16 Bottom 17 Pads 18 Vias (20 Dimension)	20 Dimension	20 Dimension 21 tPlace 25 tNames 27 tValues

4. Checking the Files

After the drill file and all the Gerber files have been generated, they should be checked for accuracy. The *.drc file can be inspected through Notepad to verify that it contains drill sizes and hole coordinates. To check the Gerber files, one can use Viewmate (www.pentalogix.com/), which is free. All of the Gerber files should look as if viewed from above, looking down or through the top layer. The bottom layer should not be reversed or mirrored. An example board design with its corresponding Gerber files are shown below.

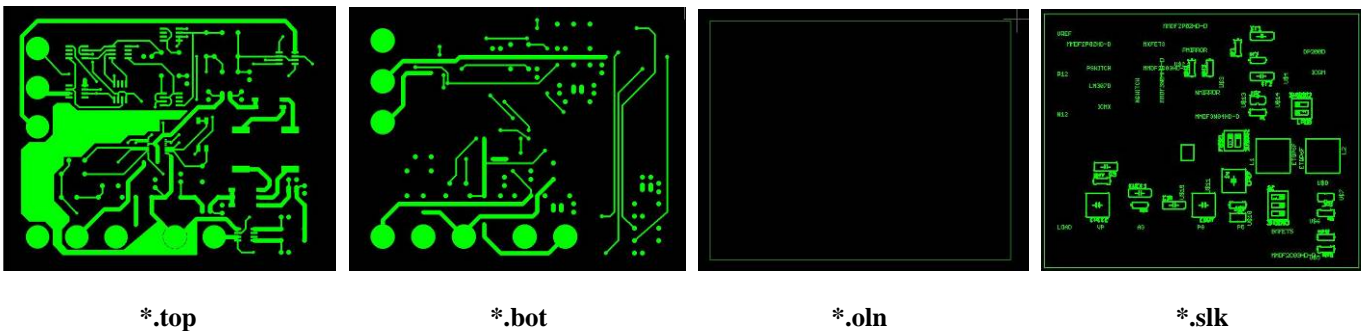


Figure 2. Gerber files corresponding to the sample PCB design in Figure 1, viewed in Viewmate.

5. Submitting the Files

If everything appears to be correct, the files should be compressed into a single zip file. The website should be checked for any extra instructions demanded by the manufacturing company. A text file that describes the contents of the zip file may be required and should be included within the zip file. Ignore requests for any Aperture files because Eagle has included that information in the Gerber files. Ordering from Advanced Circuits or PCB Express is like ordering from anyone else online – simply select the zip file when prompted. Figure 3 shows a populated board that was manufacture by Advanced Circuits.

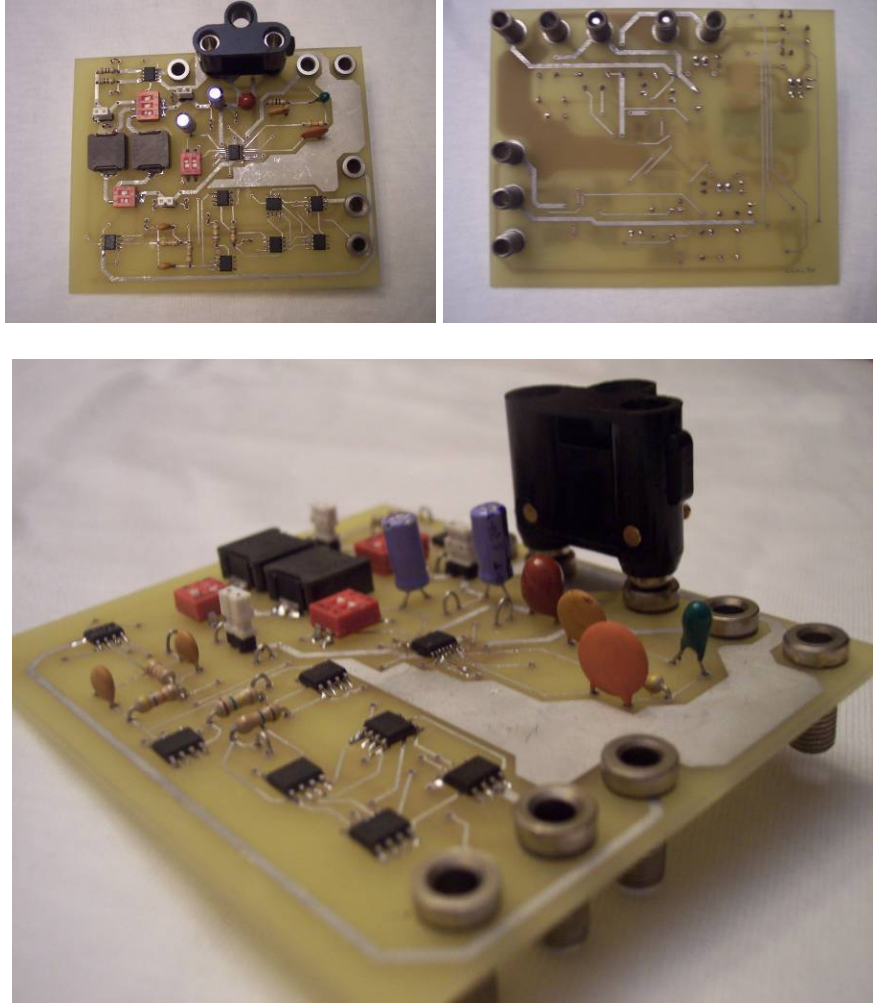


Figure 3. The manufactured and populated PCB.