PCB Creation Using ExpressPCB

As many of my long-time readers know, my preferred schematic drawing software is *ExpressSchematic*, and my preferred printed circuit board (PCB) design software is *ExpressPCB*. These two software packages are parts of a software suite known as *ExpressPCB*. They were produced as a means of encouraging users to purchase their manufactured PCB's through *ExpressPCB, LLC*, the parent company responsible for the software. The idea is that they give us the software free of charge. The software writes output files in proprietary formats that are not readily convertible to any industry-standard PCB manufacturing source files. As a result, users end up having to go back to the company to order the boards, at which time the company will also provide the user, upon request, with a copy of an industry-standard file set for the user's records. More about all of this later.

At present, the software is available in two different basic versions, *ExpressPCB Classic* and *ExpressPCB Plus*. The *Plus* version has some definite advantages in that there are some newer features that make the board design a bit more elegant than does the *Classic* version. Although I have both versions installed on my PC's, I tend to stick to the *Classic* version for several reasons:

1. I am intimately familiar with it and have developed work-arounds for many of the shortcomings of the earlier version;
2. I have an *extensive* library of custom component and device definitions and footprints that I have already developed; and
3. up to now, I have used an outside service to convert the *Express PCB .PCB* files to industry-standard board-production file sets, a service that is available only for older format of the files produced by the *Classic* version.

If you choose to use the *Classic* version of *ExpressPCB,* the problem of converting the *ExpressPCB .PCB (Classic)* data files to an industry-standard file set is no longer an obstacle. The industry-standard file format to which I refer here is the Gerber RS-274X format. This is a file format that is understood and accepted by virtually any PCB manufacturing house, and is actually a collection of individual files, each of which defines some specific aspect of the PCB design. Typically, the file set is uploaded to the board production house as a *.ZIP* file that contains all of the separate definition files.

Gerber filenames, by definition, are appended the filename extension *.GB* or *.GBR*. However, some other filename extensions are often used as a means of identifying the contents of the specific files in the file set. A complete Gerber RS-274X file set can include any and/or all of the following individual definition files:

- .APR – Aperture File;
- .APT – Aperture File;
- .EXTREP – Extension Report of Gerber Files;
- .REP – Report of Individual Layer Used Aperture List;
- .RUL – DRC Rules;
- .GKO – Keep-Out Layer;
- .GTO – Top Overlay;

- .GBO – Bottom Overlay;
- .GPT – Pad Master Top;
- .GPB – Pad Master Bottom;
- .GTS – Top Solder;
- .GBS – Bottom Solder;
- .GTL – Top Layer;
- .GBL – Bottom Layer;
- .GTP – Top Paste;
- .GBP – Bottom Paste;
- .G1, .G2, *etc.* – Mid-layer 1, 2, *etc.*;
- .GP1, .GP2, *etc.* – Internal Plane Layer 1, 2, *etc.*;
- .P01, .P02, *etc.* – Gerber Panels;
- .GM1, .GM2, *etc.* – Mechanical Layer 1, 2, *etc.*;
- .GD1, .GD2, *etc.* – Drill Drawing;
- .GG1, .GG2, *etc.* – Drill Guide;
- .DRL – Drill Data;
- .TXT – Drill Position;
- .DRR – Drill Tool Size; and
- .LDP – Layer Pairs Export File for PCB.

It should be noted that *most* Gerber file sets are not nearly so complete. A *typical* Gerber file set for one of my double-sided boards might include the **.GBL, .GBS, .GM1, .GTL, .GTO, .GTS, and .TXT** files.

I have recently been fooling around with ways and means of making valid Gerber file sets myself, without having to send the source **.PCB** file out to be converted. The long-term idea is to be able to make my own Gerber files for the newer **.RRB** file format from the *ExpressPCB Plus* version, as the day will be here sooner than I like when all support for the *Classic* version is dropped. *ExpressPCB LLC* has already announced that they will discontinue all *Classic* support and production of boards from the Classic file format when *Microsoft* drops support for the older application software running under the *Windows*® operating system. As a result, I decided that I will have to start using the *Plus* version and updating all of my component definition and footprints to the new format as well.

That brings me to the topic of this article… how to convert the *ExpressPCB* data files into a Gerber file set that will be accepted by the board production house of your choice. As luck would have it, I am not the only person dealing with this problem. However, I *am* more persistent than some others may be when it comes to seeking a solution. As such, I have found a great solution that seems to work very nicely. It involves installing a *perl* engine on the PC, and also installing a specific PDF print driver… but I am getting ahead of myself a little bit.

It turns out that there is a *perl* script written to parse the contents of **.PDF** files and then to convert those contents into a Gerber file set. The script is called **pdf2gerb.pl** and it is available for download at http://swannman.github.io/pdf2gerb/, where you should scroll down the page and click on the download link found there. The file retrieved will include the script and its necessary configuration file as well as some documentation and reference data and some sample files.

Unzip the retrieved file to a convenient folder, and then browse into the */doc* folder to find the **pdf2gerb.pdf** file, which will provide instructions and some resource links.

As already mentioned, a *perl* environment will be necessary in which to run the *perl* utility and script. A good choice for Windows® is *Strawberry Perl* (http://www.strawberryperl.com), which will install cleanly and easily into Windows® 10 or Windows® 11. Installing the *Strawberry Perl* package will append the *perl* folders to the default **PATH** environment variable, so you will be all set to run the app from within a command window in whatever folder you want.

Next up is the **.PDF** print driver. I have a licensed copy of Adobe Acrobat 11 on my PC, so naturally I tried using that as my print driver. I ended up with insurmountable errors in my output Gerber files. Next, I tried the Windows® native **.PDF** driver, with similar results. After that, I tried a stand-alone **.PDF** driver that I have used for years, called *PDF Factory.* Unfortunately, that also produced erroneous outputs. Finally, I followed the advice found in the **pdf2gerb.pdf** instructions and downloaded another driver called *PDF Creator.* The instructions reference version 1.3.2, but the most recent version is 5.1.2, which is what I downloaded. It can be found at https://download.pdfforge.org/download/pdfcreator/PDFCreator-stable. When you install this driver, decline all offers for additional software and add-ons – they are *not* needed.
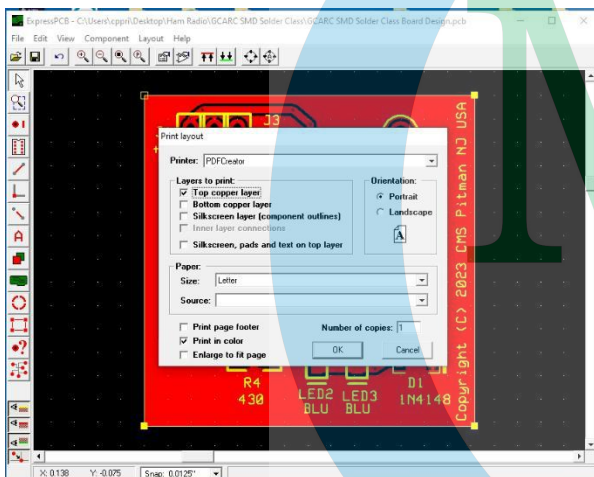

Figure 1 - ExpressPCB File > Print dialog

After installation of the *perl* software and the driver, the next step is to get your source **.PDF** files for conversion. This is done by opening up your PCB design in *ExpressPCB* and then printing three files, using the *PDF Creator* print driver. In the *ExpressPCB* **File > Print** dialog (Figure 1), first select the option to print only the top copper layer, naming that file **xxxx-top.pdf** where the *xxxx* is a text string of your choice. Next, repeat the print task, this time printing only the bottom copper layer and naming that file **xxxx-bottom.pdf**. Finally, print another file, this time selecting only the top silkscreen layer and naming the output file **xxxx-silk.pdf**. When printing these files, select only the layers instructed, and print the PCB at actual size, without any page footers or other details, using the default color set defined in *ExpressPCB.*

Create a folder off the root folder of the hard disk drive of the PC, giving that folder a name that you find convenient. Next, copy the following files into that folder:

- **pdf2gerb.pl**;
- **pdf2gerb_cfg.pm**;
- **xxxx-top.pdf**;
- **xxxx-bottom.pdf**; and
- **xxxx-silk.pdf**.

Remember that the *xxxx* part of those last three filenames are the text strings that you used in your **.PDF** file creations.

Conversion of the three *.PDF* files into a Gerber file set involves running the *perl* script, using the three filenames as command-line arguments. Note that the three filenames must be listed as arguments in the correct sequence of *top, bottom,* and then *silk.* Launch a command window by right-clicking the Windows® icon in the lower-left corner of the screen, selecting the *Run* command, and then typing in *CMD* in the *Run* dialog box. Click *OK* to launch the command window.

In that command window, use the *CD* command to change to the folder into which you copied the files earlier, *e.g.,* **"CD \myfolder"** followed by pressing the *Enter* key, where the quote marks are omitted and the *myfolder* string is replaced by the name of the folder that you created off the root. Your

Figure 2 - Perl script command line example

command line prompt will then change to read **C:\myfolder>**. You are now ready to run the *perl* script. That is accomplished by typing the following command at the command prompt and pressing *Enter*… **perl pdf2gerb.pl xxxx-top.pdf xxxx-bottom.pdf xxxx.silk.pdf** (Figure 2), of course replacing the **xxxx** with the text string that you used to name your *.PDF* files.
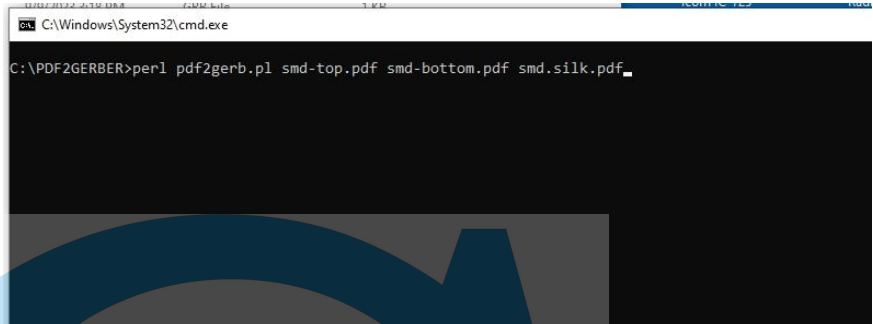

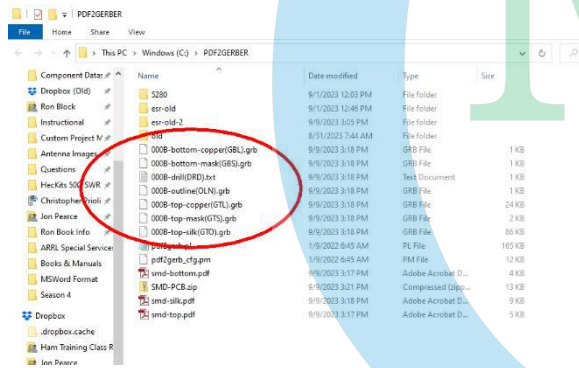
Figure 3 - Gerber files in working folder

If all was done correctly, the script will run, parsing the *.PDF* files and writing a series of *.GBR* files in that same folder (Figure 3). Each of the new files will have a string of text inside parentheses at the end of the filename, right before the extension. These text strings will coincide with some of the Gerber file types listed earlier.

Now it is time to check out your new files and make sure that they match up to your original PCB design file. To do this, you will need a Gerber file viewer utility. I recommend the *GERBV* utility, which can be downloaded from http://gerbv.gpleda.org/index.html. Scroll down the page to find the **GERBV** link, and then click the link. This Gerber viewer does not need to be installed. Simply copy the **GERBV.EXE** file to a convenient folder, *e.g.,* the same folder where the other files were copied, and then make a shortcut to that file in a convenient location.

After launching **GERBV**, click on **File > Open layers…** and browse to the location of the new Gerber files that were made. Select all of the new Gerber files and then click **Open**. A graphic representation (Figure 4) of the contents of the Gerber file set will be displayed on screen, with each layer in a different color. Select each layer individually and compare it to the original design, checking for accuracy. Once all of the layers have been proven to be accurate, combine the Gerber file set into a single *.ZIP* file, ready for submission to the board production house of your choice.
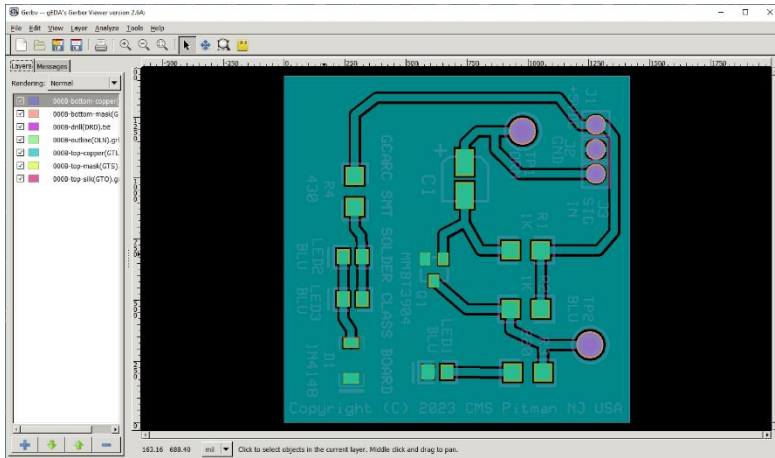
Figure 4 - GERBV board representation

I have now used this methodology to produce Gerber file sets from both versions of the *ExpressPCB* software, and have not had any errors apart from those encountered in the early developmental days. Remember that working at the command line requires precision, as any mistyped commands will result in an error. If errors are encountered, double check that you typed the command(s) properly. Syntax is everything when working at the command line. Don't worry about warnings returned during the parsing of the *.PDF* files – they are fairly common. Just examine the Gerber files with the viewer to validate that the files are accurate. Be sure to read all of the documentation for the *perl* script and its operation. Most of your questions and concerns will be answered in the documentation.

Unfortunately, the software gurus at *ExpressPCB, LLC* have thrown another monkey wrench into the works. They substantially changed the print output dialog in such a manner that in the Plus version, it is now no longer possible to print the three separate layers individually as is possible with the *Classic* version. This obviates the whole plan, as I still cannot generate clean Gerber file sets from the *ExpressPCB Plus* application, though I can do so nicely from *ExpressPCB Classic*. The problem is that when I use a filled plane, such as a ground plane, on some boards, the bottom copper layer does not render correctly. If I eliminate the filled plane, I can print and render the board. That is not acceptable to me. Maybe my long-term solution is not to use the most recent version of this software, but to switch off to some other software platform.

It would not be quite so onerous if it were not for the fact that the PCB's produced by *ExpressPCB, LLC* are prohibitively expensive. For a typical board design that I have produced, the boards cost me a total of about twenty-one (US) dollars delivered, for a minimum order of five copies of the board. This breaks down to two dollars for the boards and nineteen dollars for the DHL Express shipping of the package from China to my door. That is correct – extremely high-quality PCB's at a delivered price of forty cents per board plus shipping. This same order from ExpressPCB, LLC would come in at almost two hundred dollars. I cannot afford to buy my boards through their system. It is a shame that their software is so good. It ropes the user in, and then they set the hook when you try to order your boards.

A few years back, there was a piece of software on the market called *Copper Connection* that produced PCB designs in much the same manner as did the *ExpressPCB Classic* software, with the exception that this package would allow the user to generate the Gerber file set directly from the software interface. This software was very similar to the early *ExpressPCB Plus* version at the time. In what I see as an underhanded move, *ExpressPCB, LLC* bought out this software package and shut down any and all access to it, thus preventing users from generating their own Gerber file sets.

I am partially pleased with my solution to the file conversion problem, as I don't foresee the need to get much more complex in my PCB designs than the level at which I am already designing. However, as the documentation explains, some of the more advanced PCB design features are supported in this conversion script. I will continue to produce boards using the older software version, as it meets my current needs. Good luck, and happy converting!