

Gerber export from Easy-PC

When your PCB design is finished, you will need to export industry standard files readable by PCB manufacturers. To do this follow the steps below.

1. Open the “Output” menu and select “Plotting and Printing”
2. You will now be presented with the dialog shown in Figure 1. This dialog shows you the list of layers used in your design.

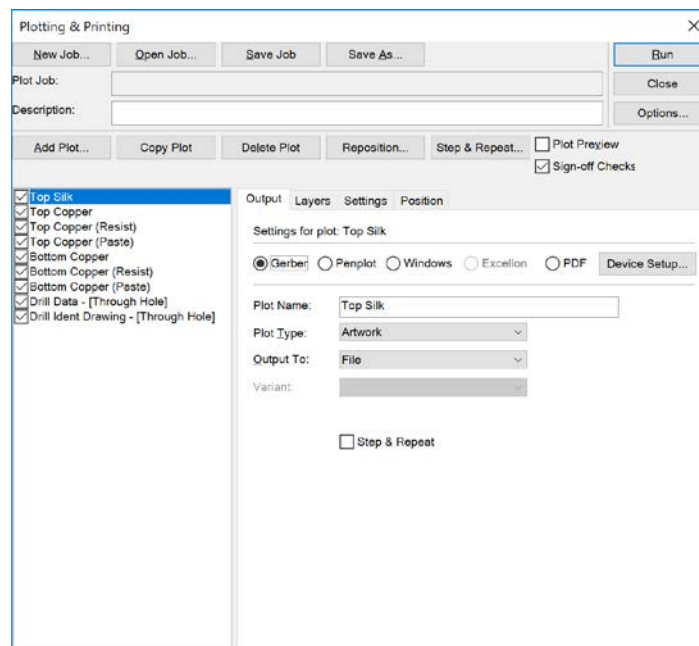


Figure 1 Plotting and Printing Dialog

We do not need to export all the layers; we only need the layers which contain information relevant to the design. This is a brief explanation of these layers:

- Top Silk – This layer contains the component outlines and component ids and will be printed in ink on the top of the finished PCB.
- Top Copper – This layer contains information about the shapes of all the pads and tracks that make up the electrical connections on the top surface of the PCB which will be chemically etched into the top copper layer of the PCB.
- Top Copper (Resist) – This layer contains information about the areas of the top surface of the PCB which will be covered with the green solder resist coating.
- Top Copper (Paste) – This layer contains information about any pads on the top copper surface which will need to be covered with solder paste to allow surface mounted components to be fitted and soldered in an oven.
- Bottom Copper – This is the bottom electrical layer of the PCB, similar to the top copper layer.
- Bottom Copper (Resist) – This is the green solder resist layer for the bottom of the PCB, similar to the top copper (resist) layer.
- Bottom Copper (Paste) – This is the solder paste layer for the bottom of the PCB, similar to the top copper (Paste) layer.

- Drill Data – [Through Hole] – This layer contains the sizes and locations of all the drill holes in the design and will be loaded into a drilling machine during manufacture.
- Drill Ident Drawing – [Through Hole] – This layer contains the same information as the other drill data layer, but in a slightly different format.

We only want the following layers will be exported: Top Silk, Top Copper, Top Copper (Resist), Bottom Copper, Bottom Copper (Resist), Drill Data – [Through Hole] and Drill Ident Drawing – [Through Hole]. Untick the layers that we don't want to export

3. Select the "Layers" tab. This shows which layers in the design will be used to make the output for the selected output layer (In the case shown in Figure 2, the Top Silk output layer will have information from the top silk layer in the design.)

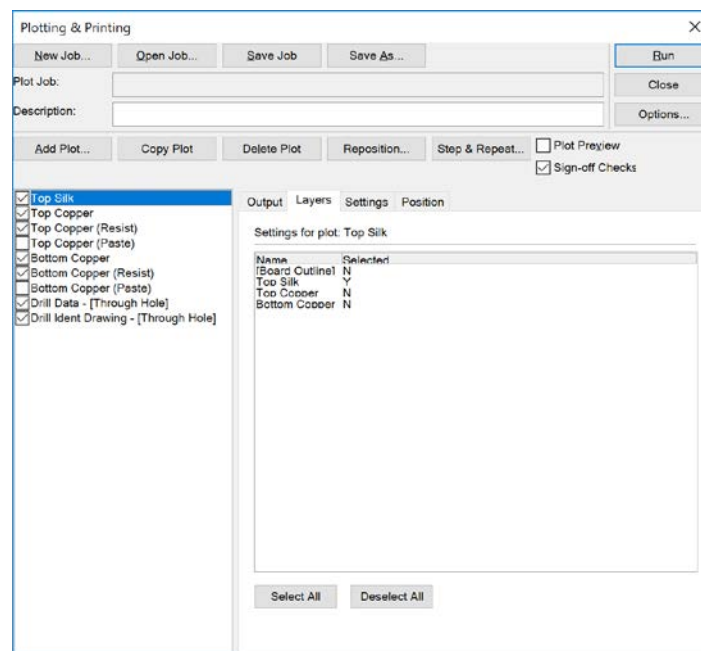


Figure 2 Layers Tab

You need to select each output layer in turn from the list on the left and make sure the [Board Outline] shows a "Y" next to it, so that the board outline is included in every output layer. This helps with locating and ensuring the correct size for the PCB and showing the manufacturer where to cut it out of the panel.

4. Select the "Settings" tab. This tab has various options, as can be seen in Figure 3, depending upon what kind of output layer you have selected. You must ensure that the check boxes marked "All Colours Black", "Fill Plated Drill Holes" and "Fill Un-plated Drill Holes" are both checked for all layers, and for the two resist layers, the "Pads-Only (Resist/Mask) Plot" check box should also be checked.

For the drill layers, this box looks slightly different. You should not need to change anything on the settings tab for drill layers.

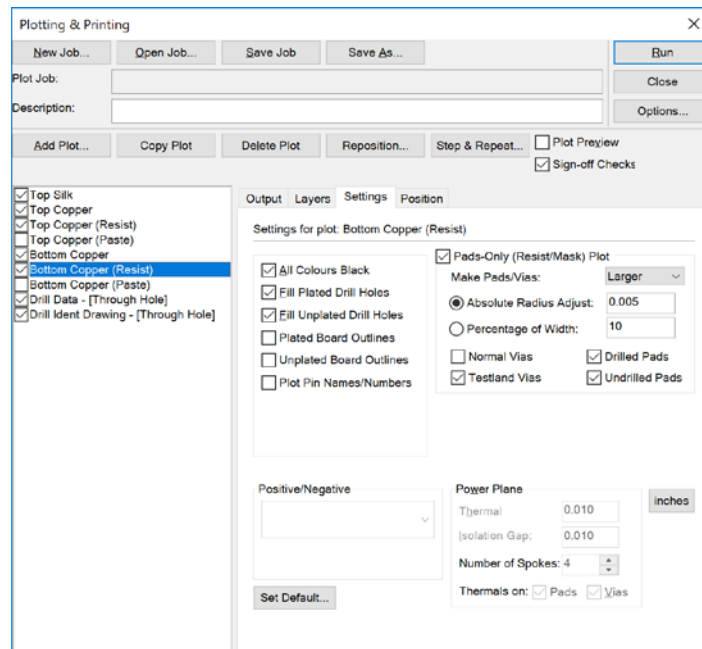


Figure 3 Settings Tab

5. On the “Position” tab, the only thing needed here is to make sure the “Scale” box contains “1.000” and the “Extents” group has “Whole Design” selected.
6. Click the “Options” button near the top right corner of the plotting & printing dialog box to display the “Options” dialog as shown in Figure 4.

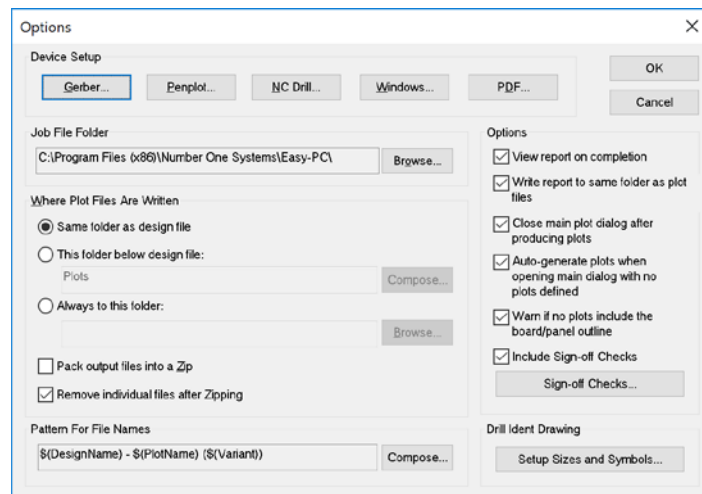


Figure 4 Plotting & Printing Options Dialog

This dialog allows you to select how your files are exported. When Gerber files are exported, one file is created for each layer, so we will have seven files. Carrying around seven files can get a bit tricky so we want to tick the “Pack output files into a Zip” which will compress all the files into a single file. We also want some kind of naming convention for the files so we know what they are. We can also change this here. Click the “Compose...” button to show the Compose File Name Pattern dialog box as shown in Figure 5.

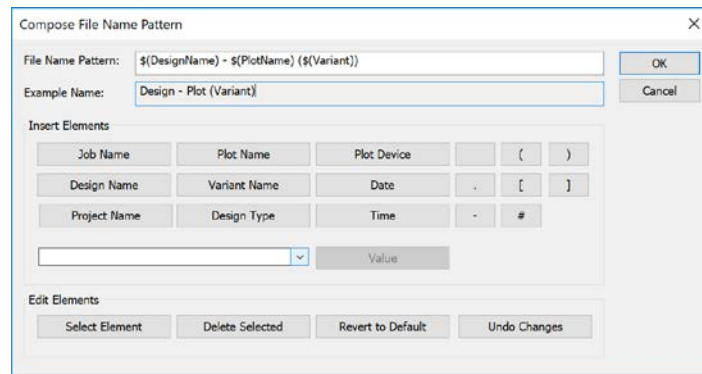


Figure 5 Compose File Name Pattern Dialog

In this dialog, you should change the File Name Pattern to include your name, as shown in Figure 6 and click “OK” to close the “Compose File Name Pattern” box and click “OK” to close the “Options” Dialog.

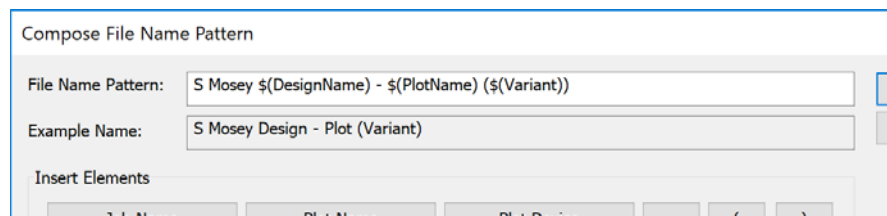


Figure 6 Example File Name Pattern

This will then automatically place your name at the start of each filename, this is so that I know whose files belong to who.

7. Click the “Run” button to export your design to a set of Gerber files. Several boxes will appear and disappear before you can read them – don’t worry this is normal. You will be left with two boxes, one of which will contain a plot report with lots of technical information relating to the Gerber files which you can ignore and close, and the other one asking if you want to “Save the current job first?” to which you can reply by clicking “No”.
8. Open windows explorer and browse to the folder containing your Easy-PC PCB design files. You should now see a new zip file there. Open the zip file and you should see something like the list shown in Figure 7.

Name	Type
Steve op-amp - Bottom Copper (Resist).gbr	GBR File
Steve op-amp - Bottom Copper.gbr	GBR File
Steve op-amp - Drill Data - [Through Hole] (Unplated).drl	DRL File
Steve op-amp - Drill Data - [Through Hole].drl	DRL File
Steve op-amp - Drill Ident Drawing - [Through Hole].gbr	GBR File
Steve op-amp - Top Copper (Resist).gbr	GBR File
Steve op-amp - Top Copper.gbr	GBR File
Steve op-amp - Top Silk.gbr	GBR File
Steve op-amp (Plot Report)	Chrome HTML Document

Figure 7 Example of files from Gerber output

9. You can test to see if the export has worked using the Ucamco online Gerber viewer to check the contents of your files. This can be found here: <https://gerber-viewer.ucamco.com/>. To use the viewer, simply click the green “Open Files” button in the corner and select the zip file with the Gerber files for your design. They will then be loaded into the viewer and you can check to see if the PCB is displayed correctly.