

# Gerber files

To have our design turned into a PCB we need to send it to a manufacturer. There are many choices, but for EE 333, our default vendor will be Oshpark. ([oshpark.com](http://oshpark.com)) Depending on board size and other factors, we may use Advanced Circuits ( [4pcb.com](http://4pcb.com)) or SEED Studio ([https://www.seeedstudio.com/fusion\\_pcb.html](https://www.seeedstudio.com/fusion_pcb.html)). There are many other possibilities.

All fab houses accept designs in the form of a set of Gerber files. The Gerber format defines the two-dimensional patterns of each of the layers in the PCB using a simple ASCII description. As a file format, it may be a bit old-fashioned, but it is still the standard method for transmitting PCB design information.

If you are interested in the details of Gerber files, here are some links:

- [https://en.wikipedia.org/wiki/Gerber\\_format](https://en.wikipedia.org/wiki/Gerber_format)
- <https://circuitpeople.com/Blog/WhatIsAGerberFile.aspx>
- [https://en.wikipedia.org/wiki/Joseph\\_Gerber](https://en.wikipedia.org/wiki/Joseph_Gerber) (about the inventor)

Note that Oshpark will accept designs in native Eagle format (.brd) and KiCad format (.kicad\_pcb), meaning that it is not necessary to create Gerbers if using Eagle or KiCad for the design and Oshpark for the fab.

Each of the basic layers that define our PCB requires one Gerber file. Again, the layers are

- top copper
- bottom copper
- top solder mask
- bottom solder mask
- top silkscreen
- bottom silkscreen (often, there will be no bottom graphics)

In addition, two more layers are needed, giving a total of eight

- board outline
- drill file (for through-holes and vias)

When designing a four-layer board, there will be two more files describing the copper patterns for the two internal layers.

More complex designs may include many more definition files. For almost all of our relatively simple EE 333 projects, files for the 8 layers described above will be adequate.

# Naming conventions

The default filename extension used by Ultiboard is “.gbr”. However, we will change to a file naming convention that is more widely recognized. Each file will use a unique extension that indicates clearly which layer information is contained in the file. If we have a design called “our\_design”, the file names are:

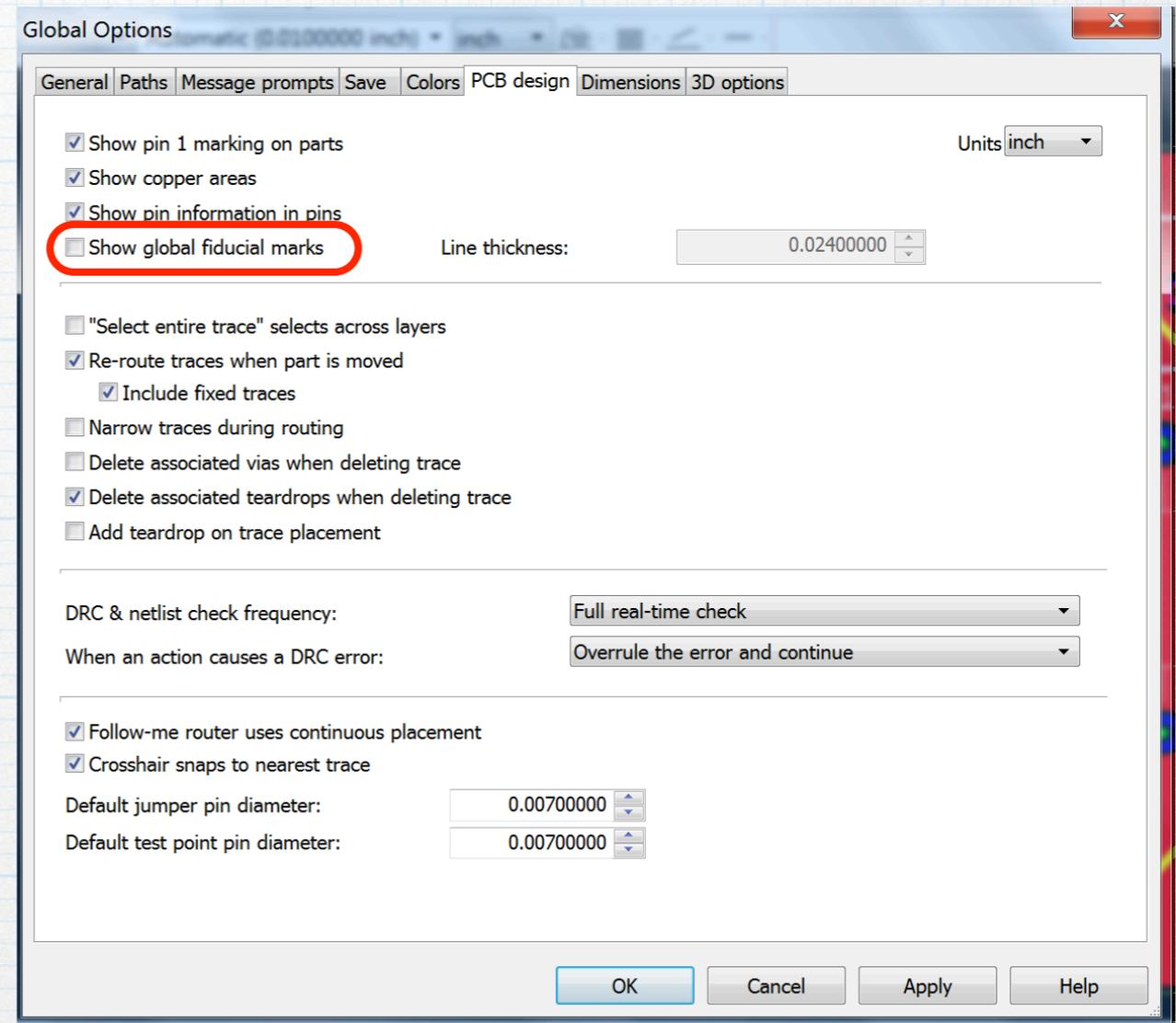
board outline	our_design.GKO (“oh” not zero)
top copper	our_design.GTL
bottom copper	our_design.GBL
top solder mask	our_design.GTS
bottom solder mask	our_design.GBS
top silkscreen	our_design.GTO
bottom silkscreen	our_design.GBO
drill file	our_design.XLN

.XLN stands for Excellon, which was an original brand of drilling machines. Note that the drill file is *not* a Gerber file — it uses a different format. If we have 4-layer design, the two additional copper layers would be “our\_design.G2L” (the one closer to the top) and “our\_design.G3L” (closer to the bottom).

# Generating Gerbers in Ultiboard

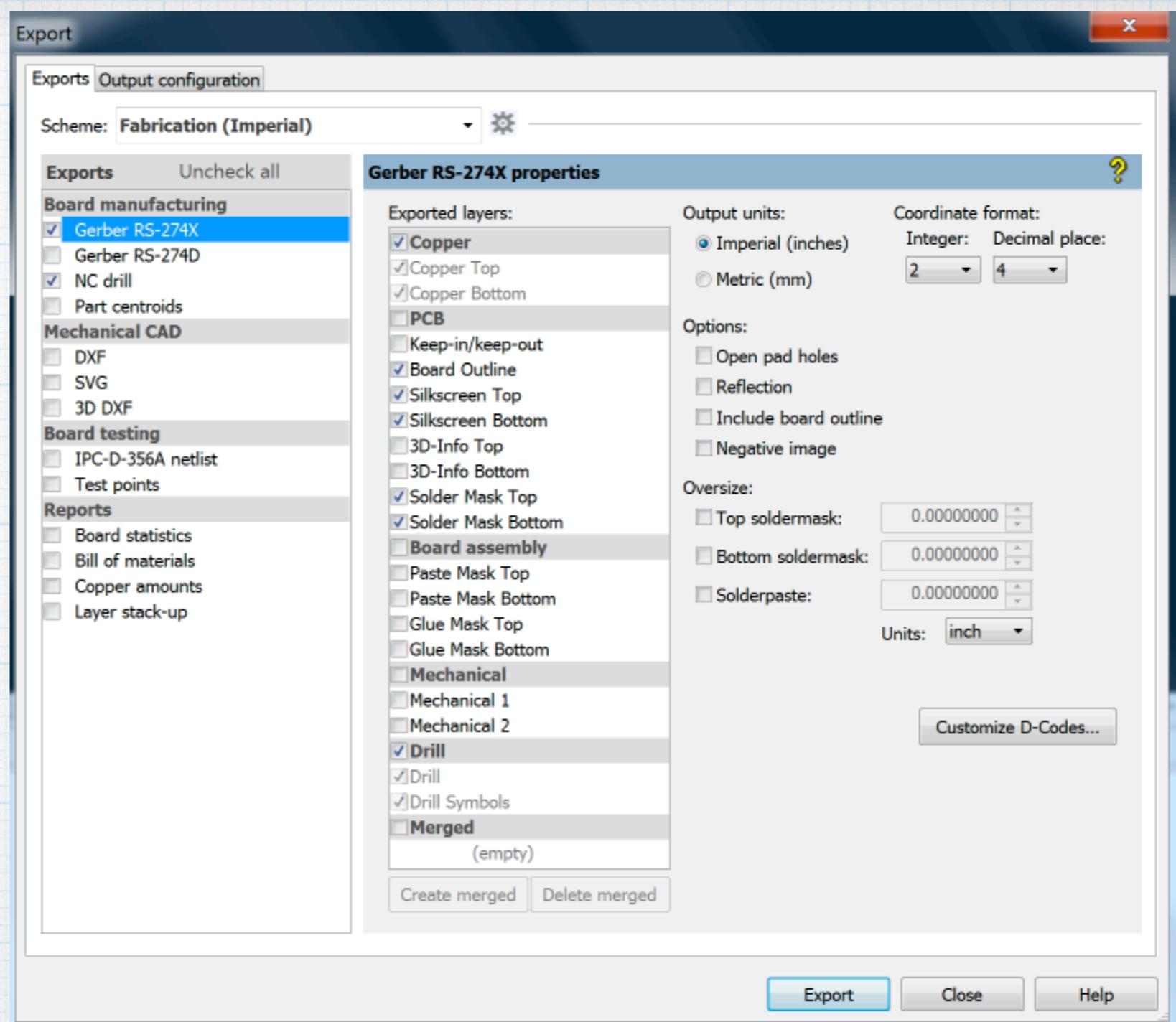
Once everything has been checked, double-checked, and triple-checked, from within Ultiboard, proceed as follows:

1. Choose the Options -> Global Options menu item. In the dialog that appears, uncheck the "Show global fiducial marks" item.



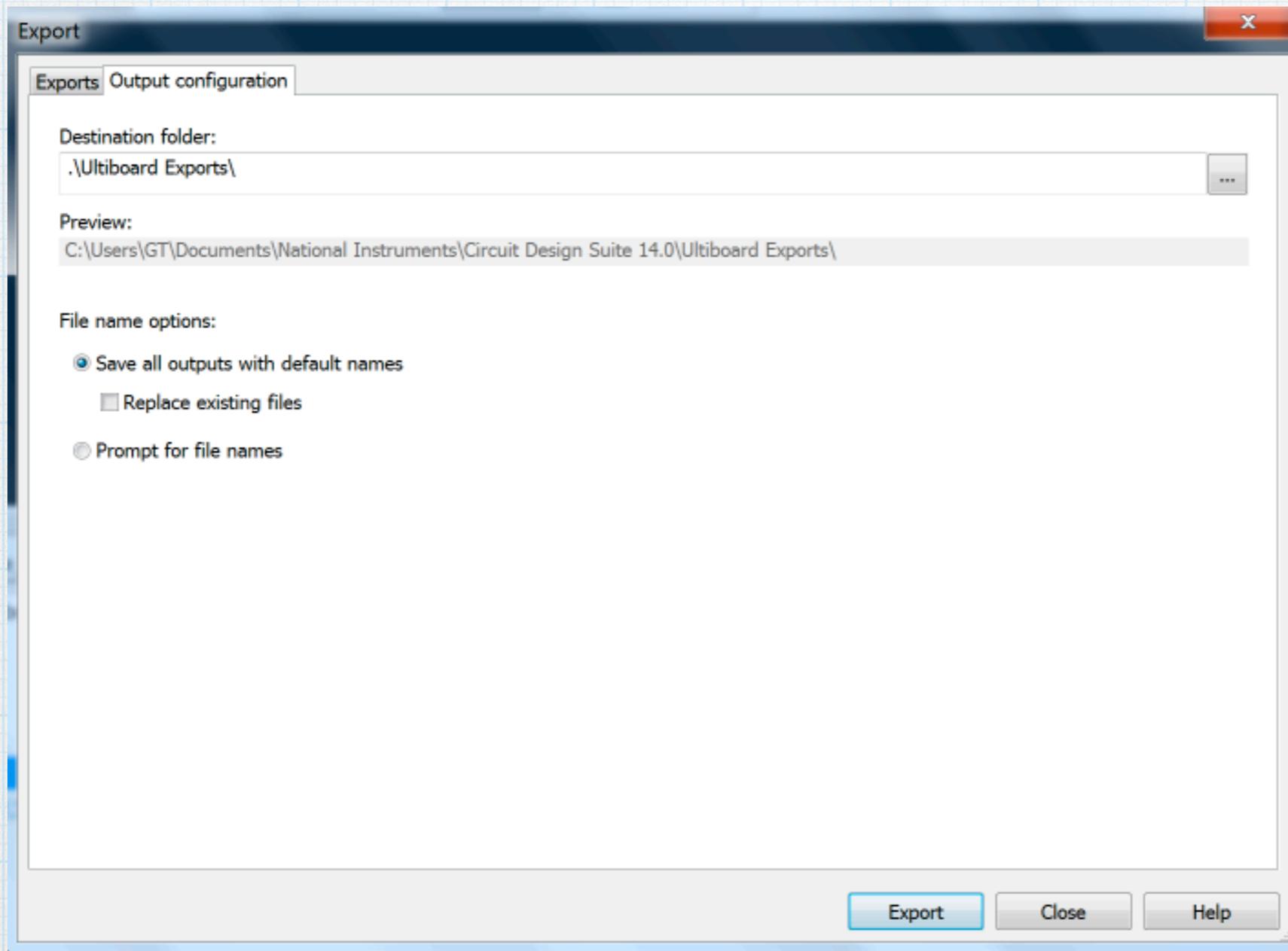
Fiducial marks are a set of reference points that can be added to help make visual alignments of the layers easier. However, they interfere with the algorithm that Oshpark uses to determine board size. If left in, the fiducials will make the board seem bigger than it really is and hence cost more than it should.

2. Choose the File → Export menu item.



Check for the following: The Scheme is “Fabrication (Imperial)”. Board manufacturing items should Gerber RS-274X and NC drill. (274D is an older format.) Make sure that the required 8 layers are checked in the export list as shown above. The units should be imperial (inches) and the coordinate format numbers should be 2 and 4.

### 3. Export the files



If you switch over to the Output configuration tab, you can select the path for storing the files. (The default path is fine.) If you select the “Save all outputs with default names” option and then click on the “Export” button, a total of 11 files will be created and stored in the selected folder.

The files will be fairly obvious, but the format does not match our recommended set of file names. Here is what is created:

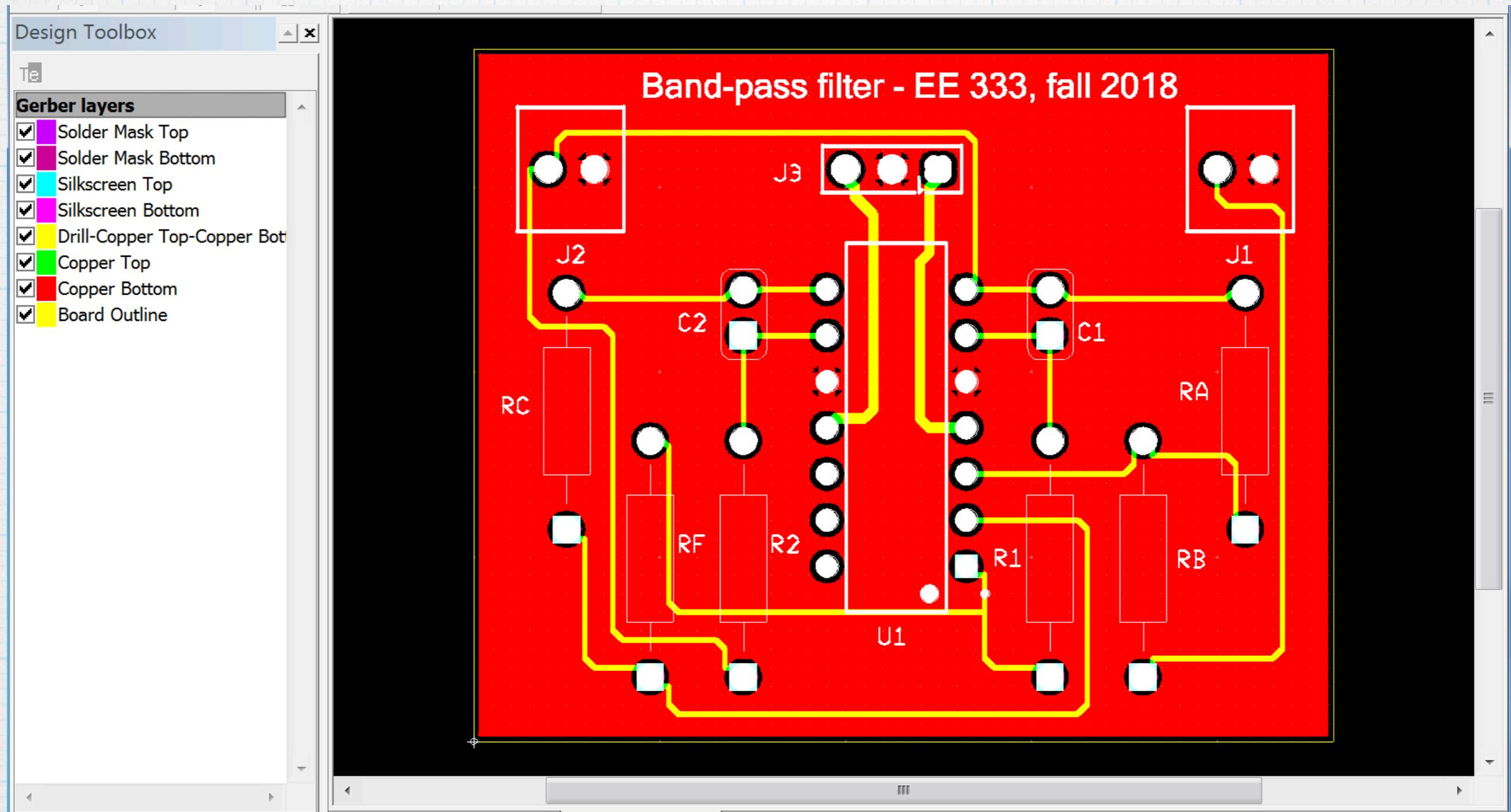
board outline	our_design - Board Outline.gbr
bottom copper	our_design - Copper Bottom.gbr
top copper	our_design - Copper Top.gbr
drill file	our_design - Copper Top-Copper Bottom.drl
	our_design - Drill Info.rep
	our_design - Drill Symbols-Copper Top-Copper Bottom.gbr
	our_design - Drill-Copper Top-Copper Bottom.gbr
bottom silkscreen	our_design - Silkscreen Bottom.gbr
top silkscreen	our_design - Silkscreen Top.gbr
bottom solder mask	our_design - Solder Mask Bottom.gbr
top solder mask	our_design - Solder Mask Top.gbr

# Check the Gerbers

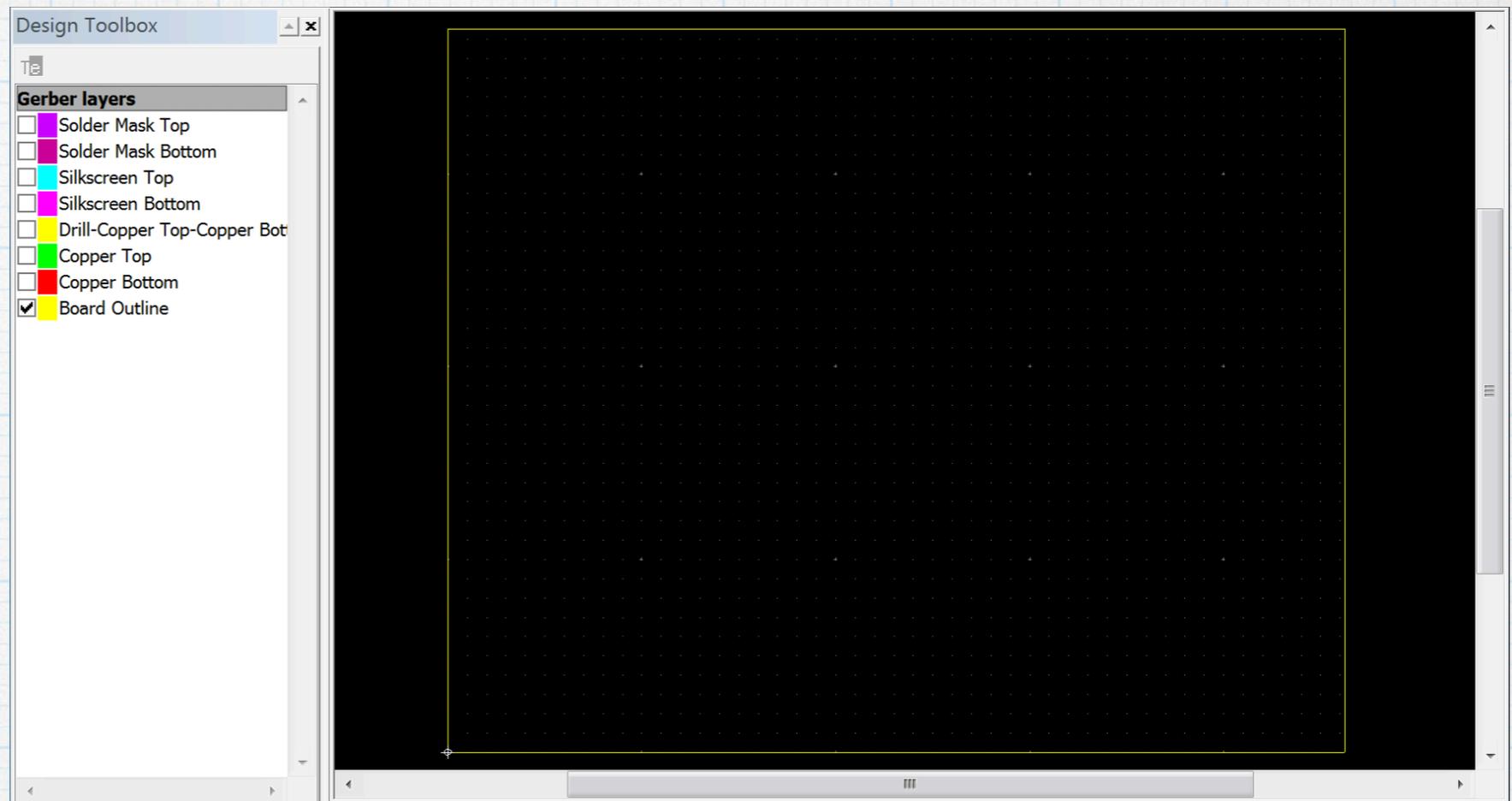
Before sending them off, we should check the Gerber files to make sure that they “look right”. Ultiboard has an option for viewing the Gerbers after they have been created, but there are a couple of oddities in how it works.

1. Back in the main Ultiboard window, choose “Open” from the File menu. In the dialog, move to the folder where the newly created Gerbers are stored — probably in “Ultiboard Exports”.
2. Nine of the “our\_design” Gerbers are displayed. Missing are the drill file (.drl extension), which is important, and the drill info file (.rep extension), which is not important. These can’t be opened by the Gerber viewer because they are not Gerber files!
3. However, drill information is contained in the “Drill-Copper Top-Copper Bottom” file, which is a bona-fide Gerber file.
4. Open all of the listed files, *except for* the “Drill Symbols-...” file. (It is a table that lists all of the drill sizes. It is redundant.) The files can be opened one at a time or all at once by control clicking.

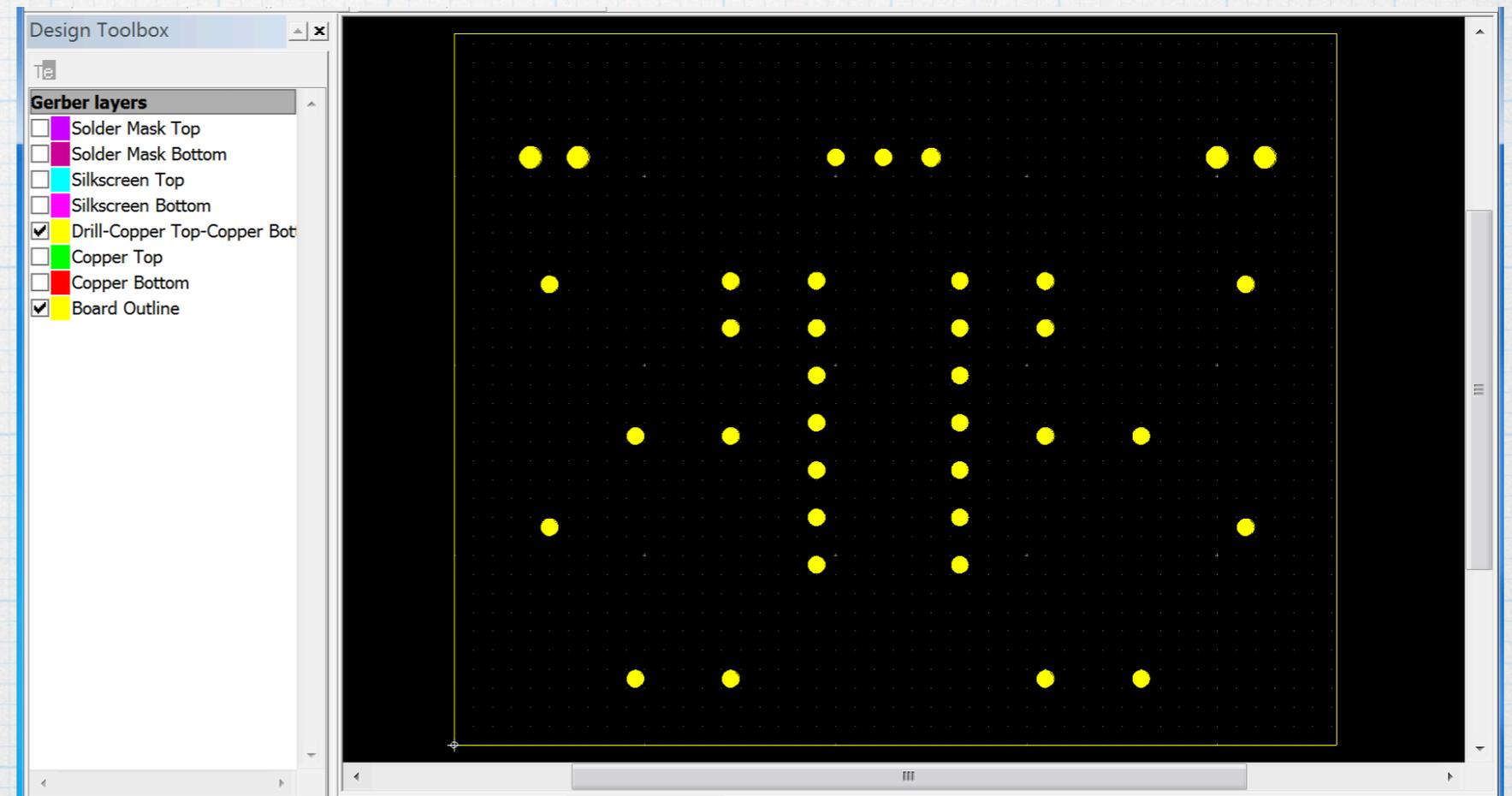
5. A window opens with a view containing all the layers of the board. It looks just like the view in the Ultiboard drawing window (or at least it should), but this view is generated using the Gerber files.



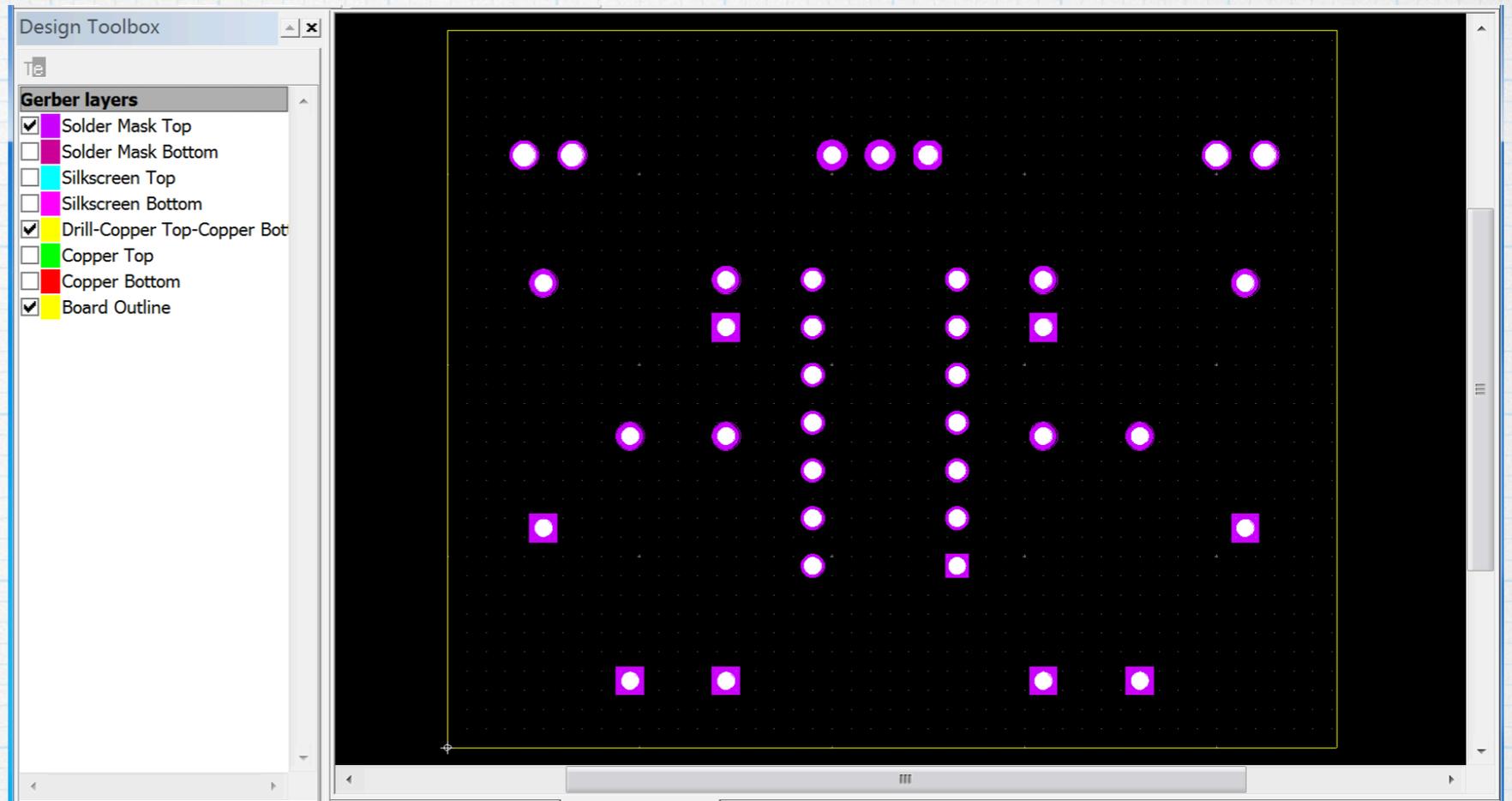
6. To check the Gerbers, it is probably better to examine the layers one or two at a time. Start by de-selecting all the layers except for the board outline. (Recall that you must click multiple times to “uncheck” the boxes.)



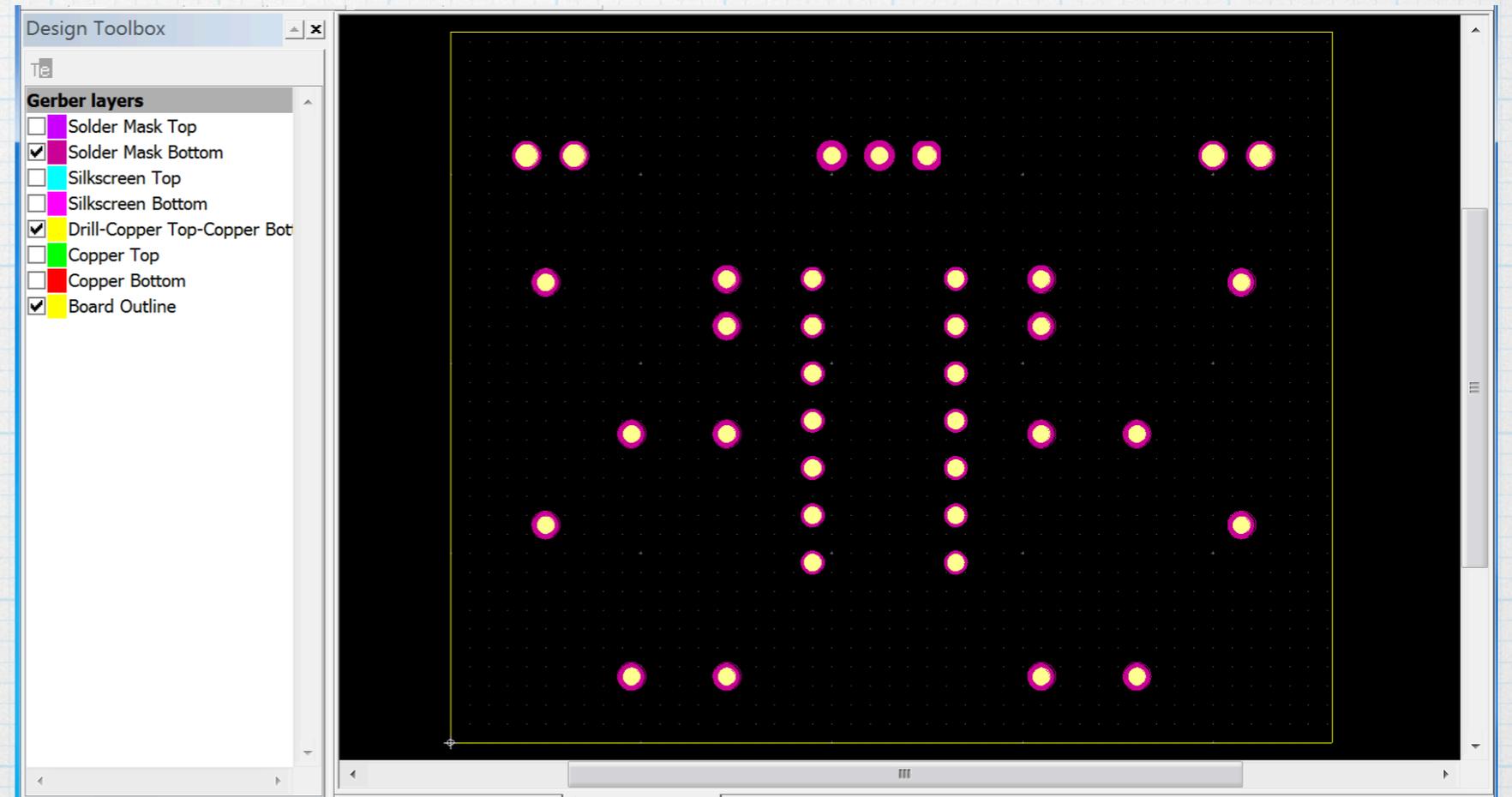
7. Add in the drill information. Make sure that all drill hits are inside the board outline. (A common mistake: using the wrong precision when exporting, causing the drills to be outside the boundary.)



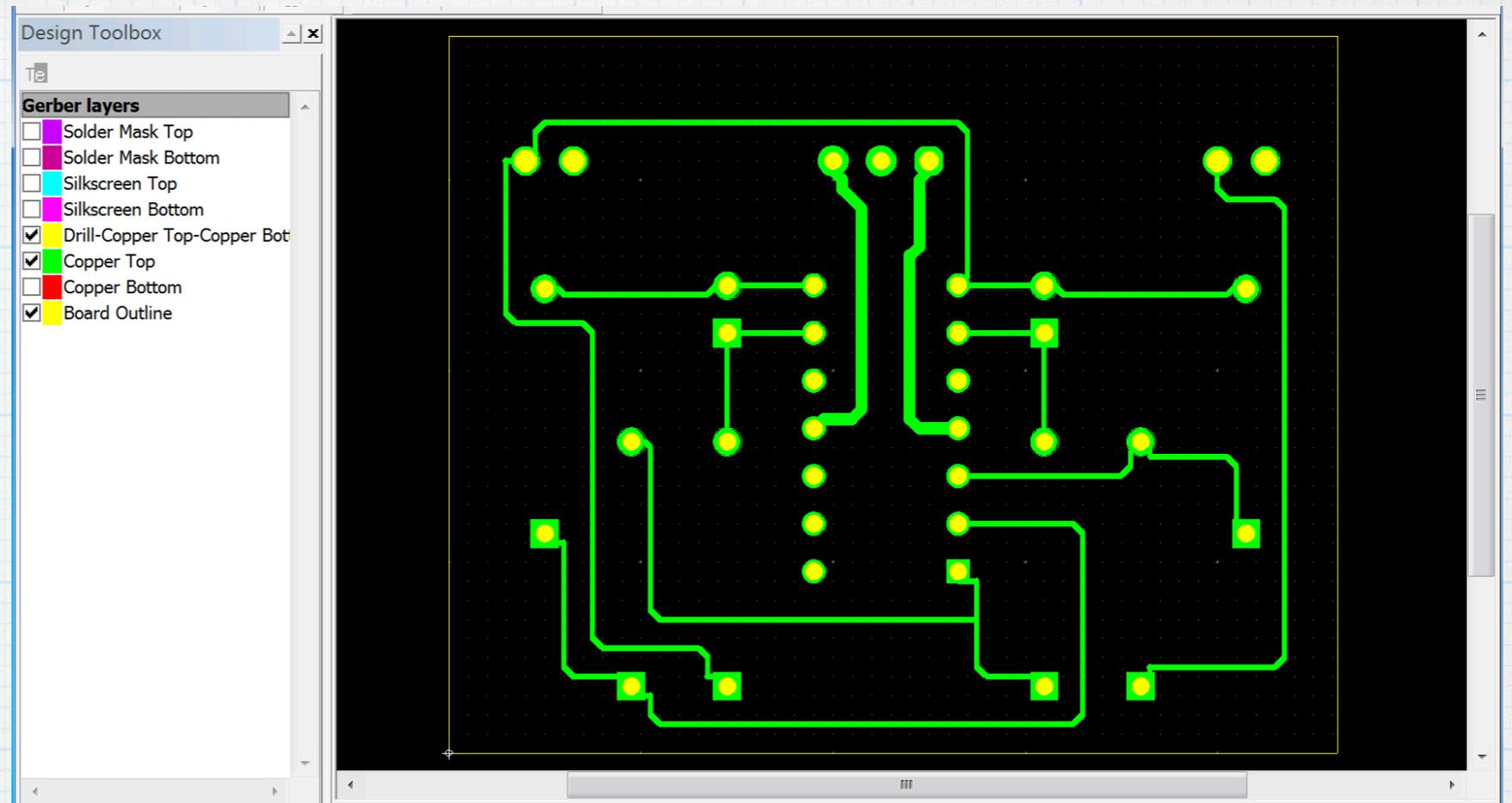
8. Add the top solder mask. Make sure that these align with the drill marks. (If doing surface mount, solder mask openings for contact pads will not be aligned any drill hits.)



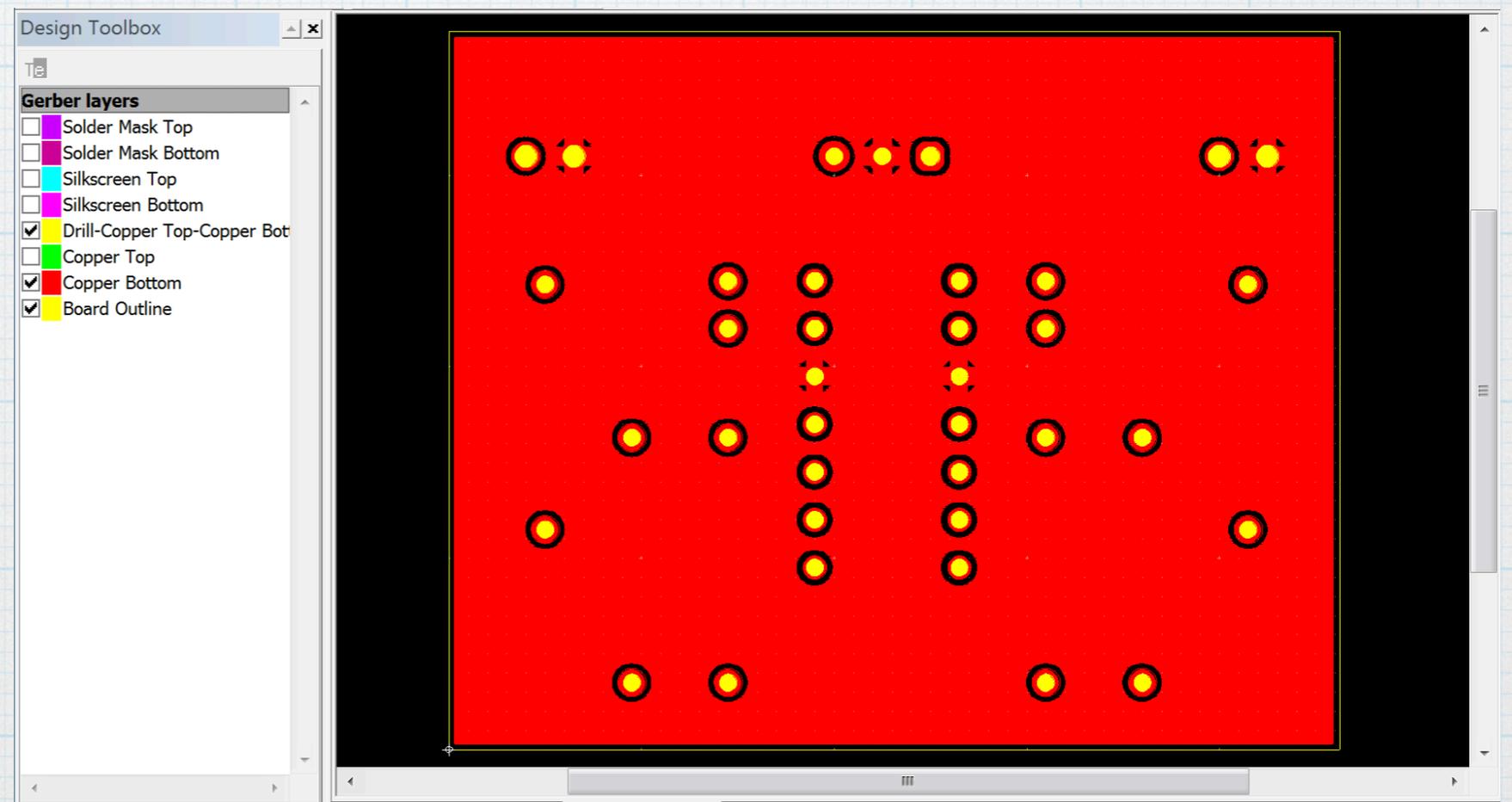
9. Do the same for the bottom solder mask.



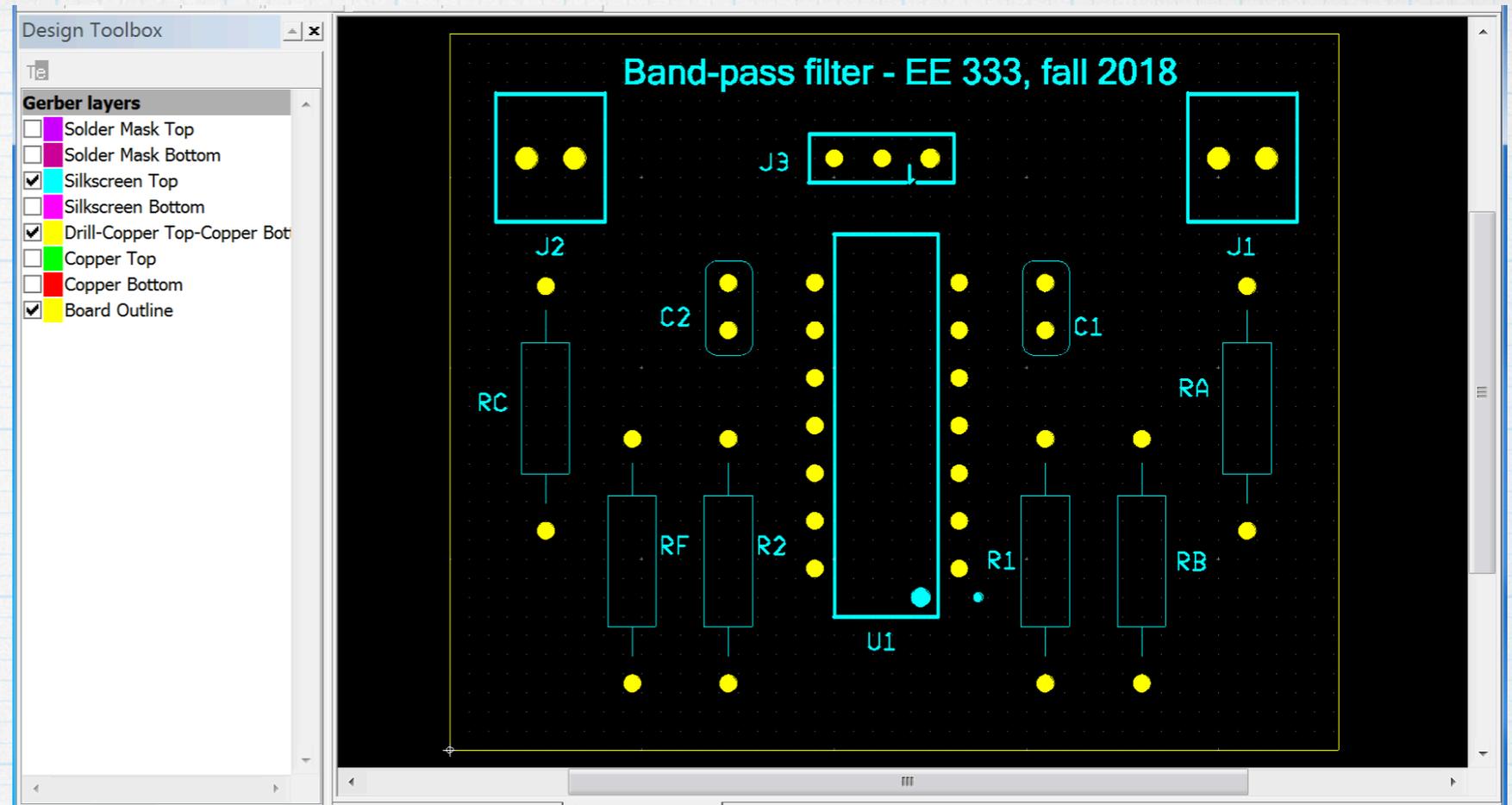
10. Check the top copper. Recall that we did not have a ground plane here.



11. Do the same for the bottom copper. We did use a ground plane on the bottom.



12. Finally, check the top and bottom silkscreens. (There were no graphics on the bottom in this design.)



It is apparent that Ultiboard is creating two files with drill information: an Excellon formatted file (...-Copper Top-Copper Bottom.drl) and a Gerber formatted file (...-Drill-Copper Top-Copper Bottom.gbr). A bit of experimentation has shown that Oshpark's auto-read system can correctly interpret either of these files. (The same for Seeed Studio.)

Obviously, some fabricators can use the Gerber description for the drills, but there is no guarantee every fabricator can do that. Therefore, we we will follow convention and use the Excellon file to describe the drills — it is the traditional standard and every fabricator should be able to use it. However, it is a bit odd that Ultiboard's own Gerber viewer cannot display the Excellon file.

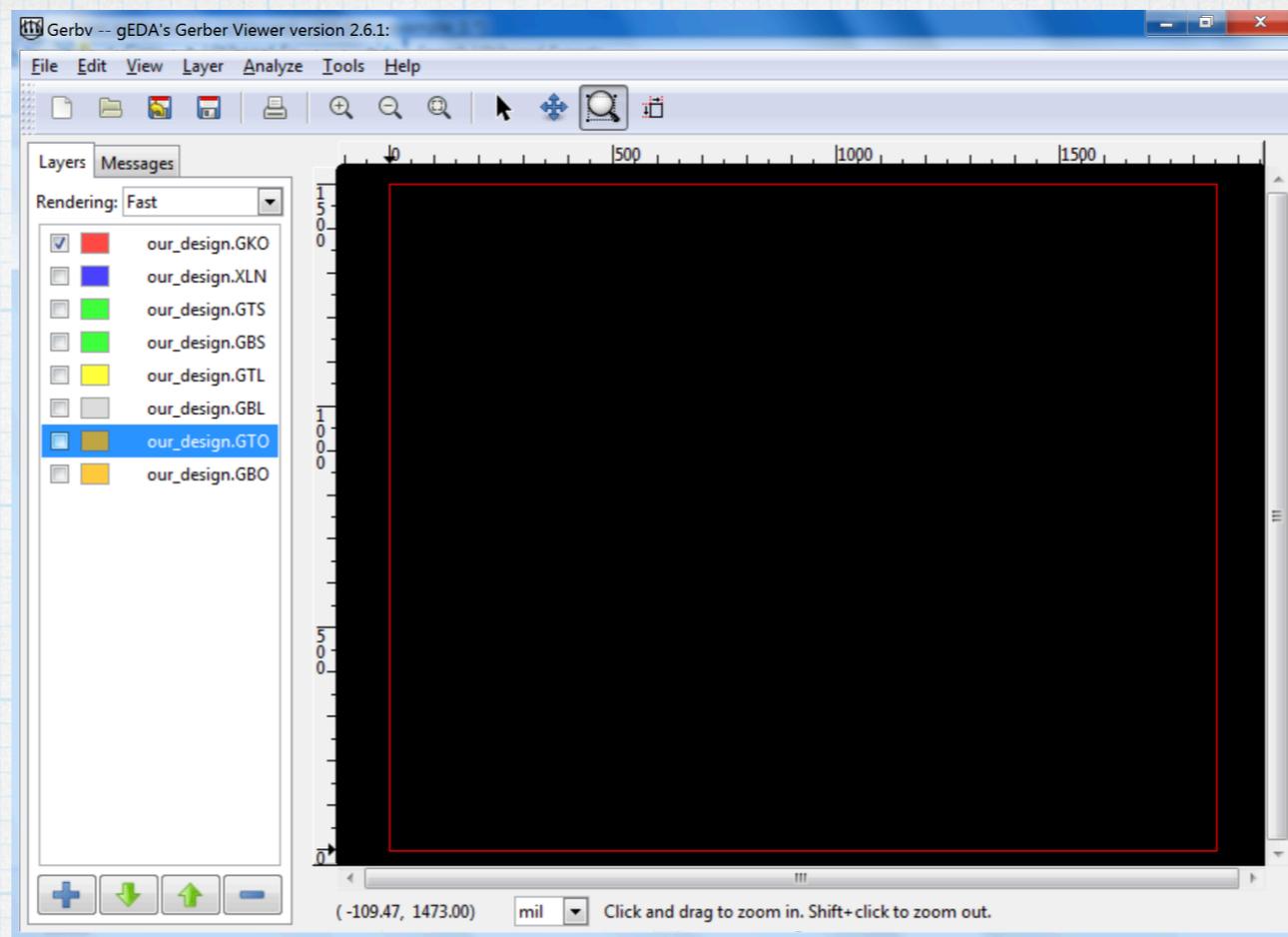
If we simply packaged up this set of files, as is, and shipped them to Oshpark, they would probably be fine. However, it is better to use conventional names to minimize possible hangups. This is especially true if we are sending the design to a different fab facility. So go to the folder where the files are stored and change the names.

our_design - Board Outline.gbr	our_design.GKO
our_design - Copper Top.gbr	our_design.GTL
our_design - Copper Bottom.gbr	our_design.GBL
our_design - Solder Mask Top.gbr	our_design.GTS
our_design - Solder Mask Bottom.gbr	our_design.GBS
our_design - Silkscreen Top.gbr	our_design.GTO
our_design - Silkscreen Bottom.gbr	our_design.GBO
our_design - Copper Top-Copper Bottom.drl	our_design.XLN
our_design - Drill-Copper Top-Copper Bottom.gbr	not used
our_design - Drill Symbols-Copper Top-Copper Bottom.gbr	not used
our_design - Drill Info.rep	not used

# gerbv

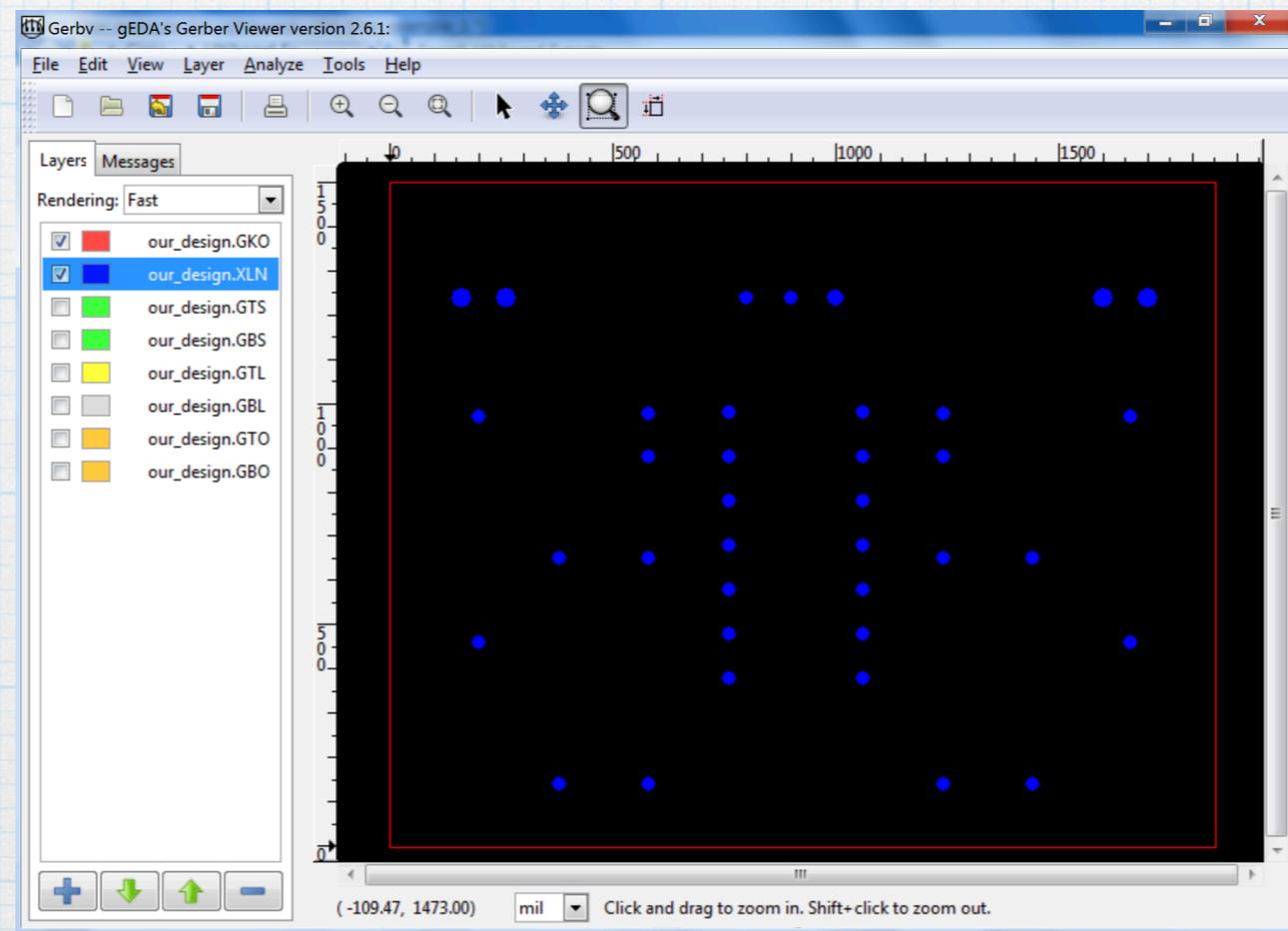
- We can also check the Gerbers independently of Ultiboard, using a separate viewer program called gerbv.
- Gerbv was originally developed to run on Unix workstations, hence the cryptic name and slightly antiquated look and feel.
- It is a free program. If it is not already on the computer, then search for it on the internet, download it, and install it. There are Windows, Mac, and Linux versions, although installing on a Mac is a bit strange because it runs inside an X-Windows shell. (A program called XQuartz must be installed first, and then gerbv runs from within that. Look for instructions on the internet and follow those.)
- Gerbv does work with the file extensions that we have used and it works properly with Excellon drill files — so none of the weirdness of the Ultiboard viewer.
- Once gerbv is running, load the Gerber files using the file menu or the big “+” icon in the lower left of the window.
- The different layers can be turned on and off using the check boxes.
- Again: Start with the board outline. Check that the drill hits are within the boundary. Check that solder mask openings align with drill marks. Finally, check the top and bottom copper and the silkscreens. Zoom in as needed to see details.
- It is possible to edit the Gerber files directly within gerbv, but that is probably not a good idea unless we become much more familiar with the details of Gerber files.

Board outline. (It's rather faint here.)

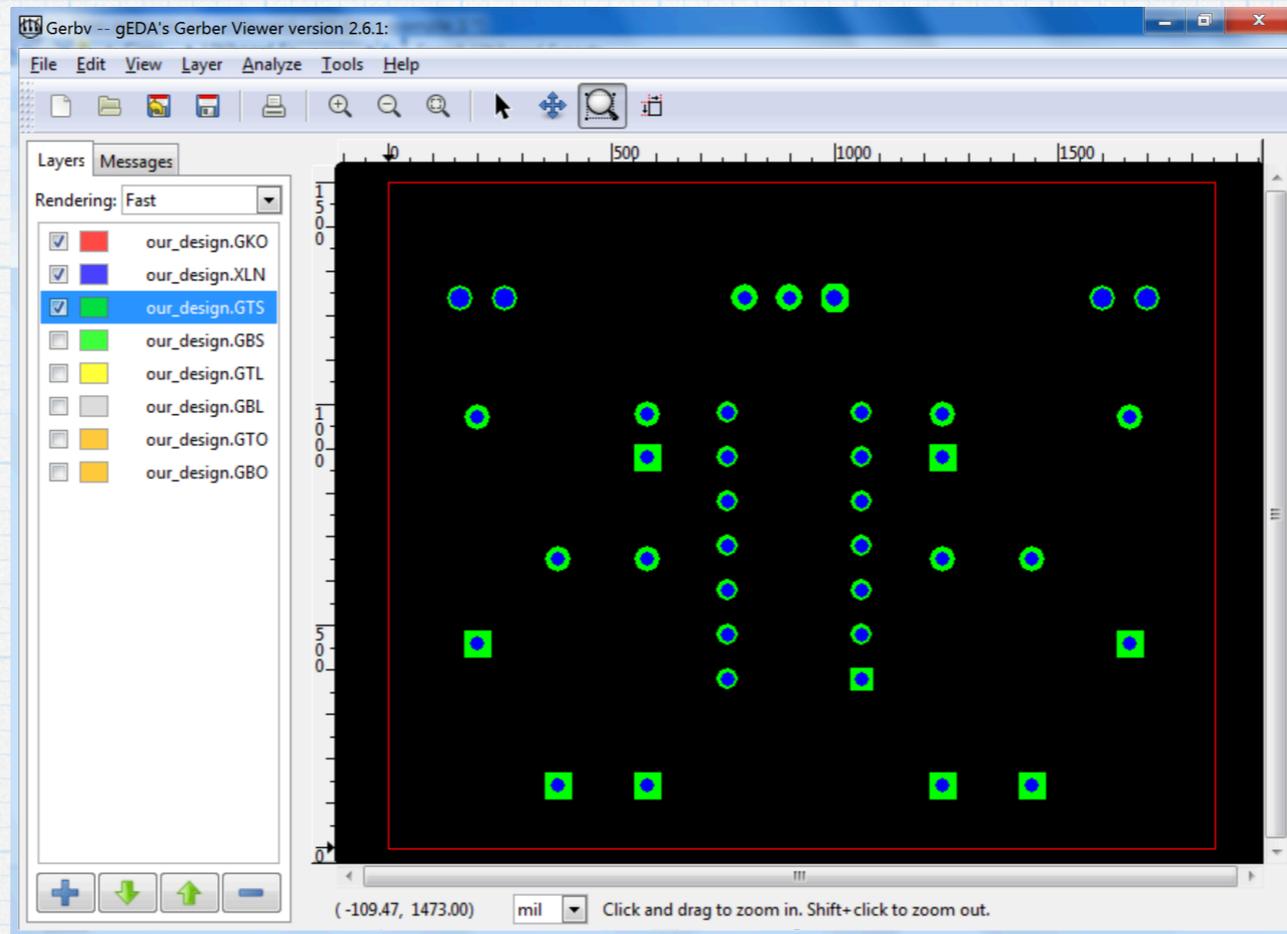


Layer colors have been changed from the default settings to make them more apparent.

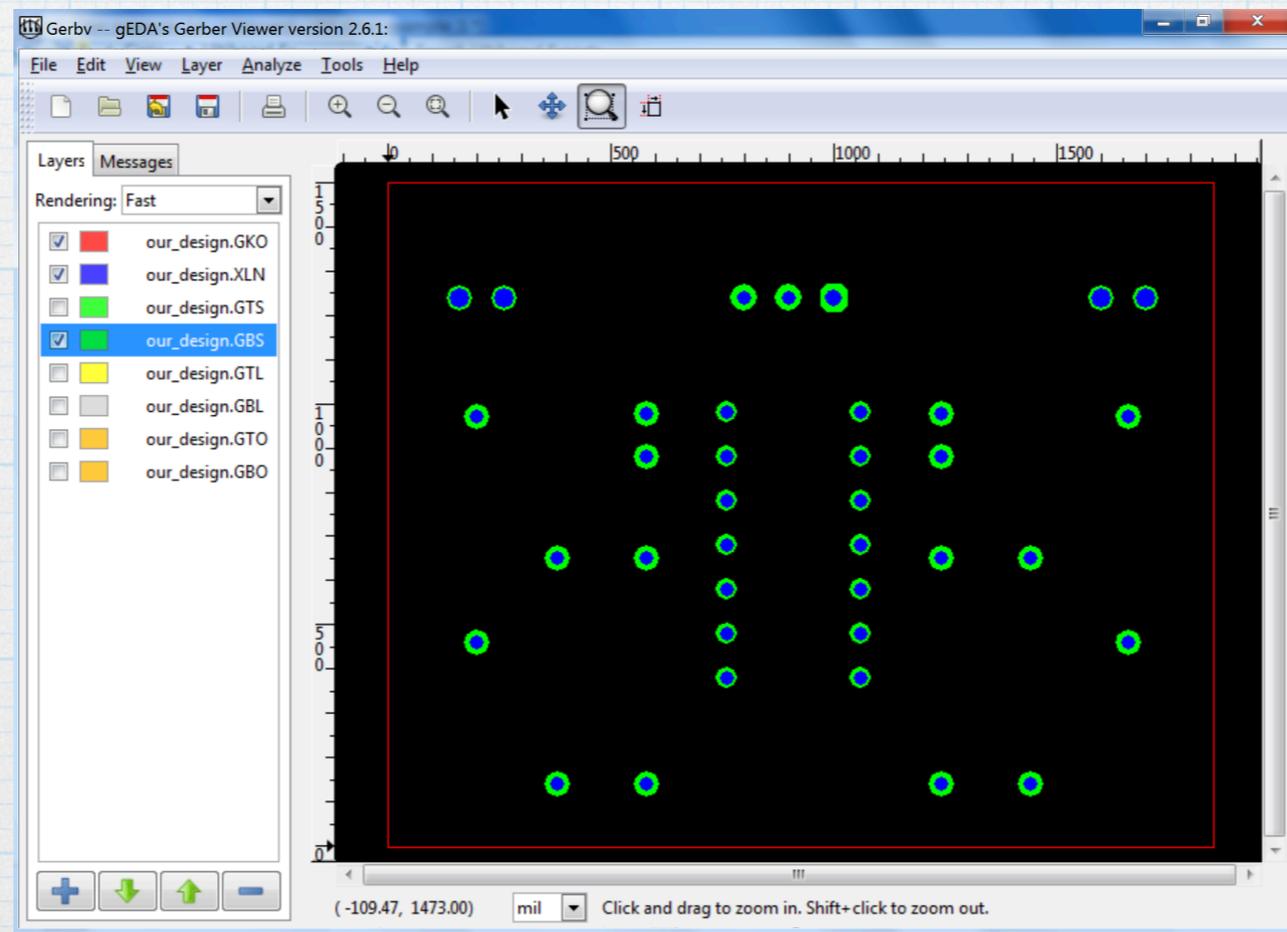
Drills.



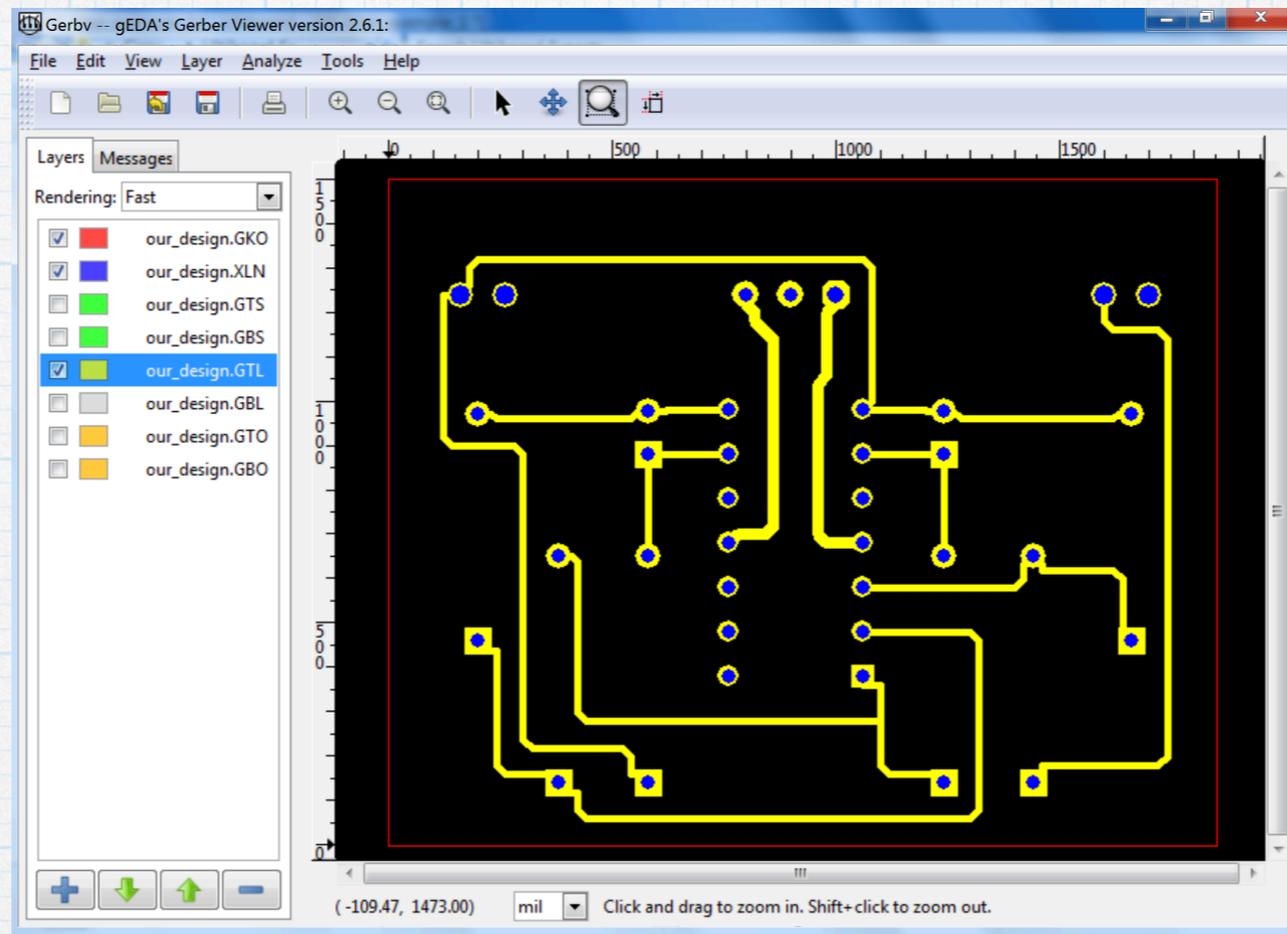
Top solder mask.



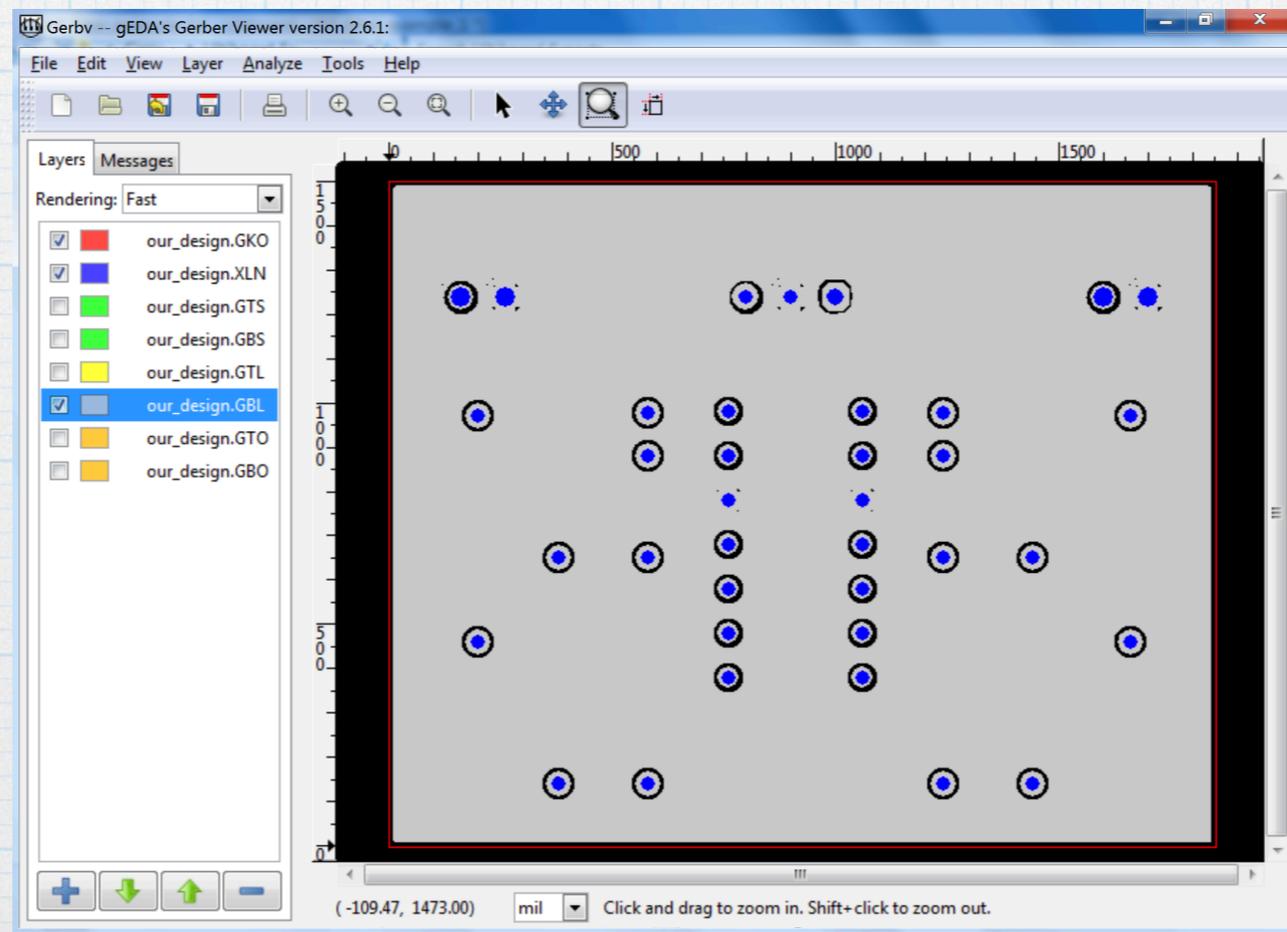
Bottom solder mask.



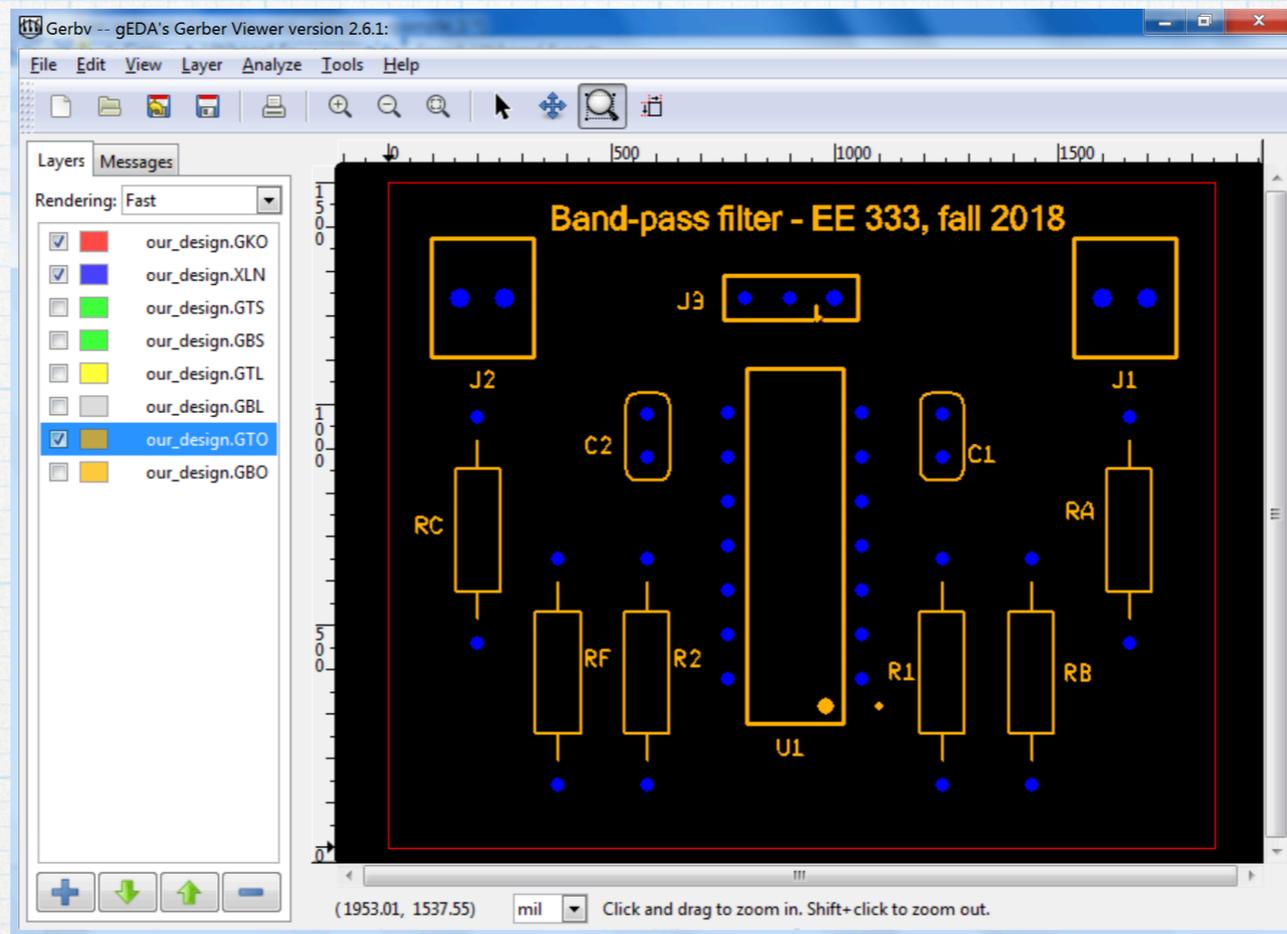
Top copper. (No ground plane.)



Bottom copper. (With ground plane.)



Top silkscreen.  
(There are no graphics on the bottom.)



# Zip it all up. Do a test upload.

- Put the 8 files in files into a zip folder. (On Windows, select all 8 files, then right click and select “Send to —> compressed (zipped) folder”. If you have transferred the files to a Mac, first select the 8 files, right click, and select “Compress 8 items”).
- Send the files to GT. He will double-check everything with gerbv and then send the order to the appropriate vendor.
- It is a good idea to test the upload process yourself. Both Oshpark and Seeed Studio allow files to be uploaded and checked for free. Go to either web site and upload the zip file. On Oshpark, various views of the board are generated automatically — follow the instructions to see how the files have been rendered. On Seeed Studio, there is a “Gerber Viewer” option that appears once the file is uploaded — click it to see various views of the board. If the files render properly at Oshpark or SeeedStudio, they will probably work anywhere.
- <https://oshpark.com>  
[https://www.seeedstudio.com/fusion\\_pcb.html](https://www.seeedstudio.com/fusion_pcb.html)