How to create 274X Gerbers from Protel AutoTrax DOS and EasyTrax DOS Files
Note this also works for CircuitMaker, TraxMaker, and Protel for Windows PCB and Gerber Files

Bonus: How to import 274-X files into SprintLayout 6.0 to create an editable PCB file

Two Methods to convert the files: The first (and preferred) method converts from the Protel PCB file, the second from the Protel 274-D Gerber and Drill files. They also work with TraxMaker Files

What you will need:

The Cadcentric.exe file (I used V4.1) from Les Hildenbrandt's site here: THANKS LES!

https://www.lh1.org/AUTOTRAX/autotrax.html#CCC

The file can also be found in the files section of https://groups.io/g/SprintLayout
along with test files and a SprintLayout .lay6 file of the demo board used for this tutorial.

The PCB file (Autotrax/EasyTrax/TraxMaker/Protel for Windows) or the complete set of Gerber files.

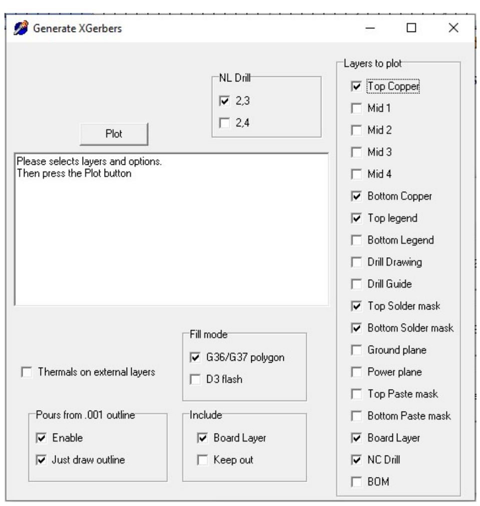
Note, that if you are using a PCB file, the PCB file must be in ASCII (Text) format as the Cadcentric conversion program depends on that format. If not, you will have to use the program that generated the file to create an ASCII one or if you also have Gerbers use the 274-D to X conversion method below.

First Method: Protel AutoTrax/EasyTrax/Traxmaker PCB file to 274-X Gerbers

1. Place the Cadcentric.exe file and the PCB file into an empty folder.
2. Launch the Cadcentric file by double clicking on the exe file.

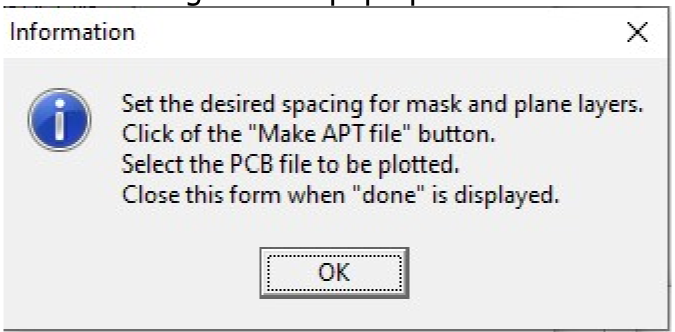
3. Click on: 

The following window will open:



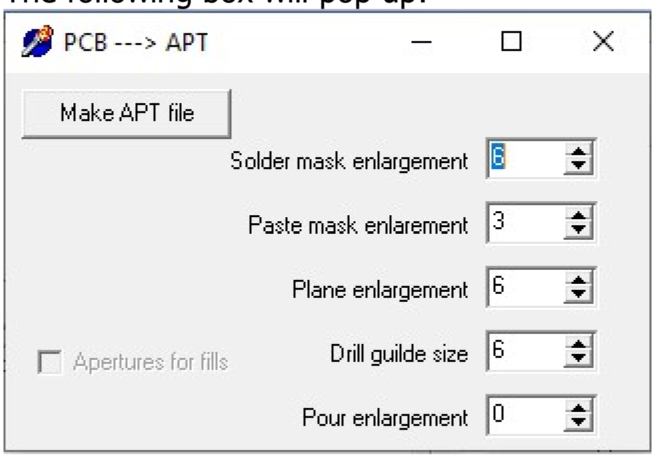
4. Select the layers you want to be included in the 274-X Gerbers. Leave the other selections as shown.

5. Click on the Plot button.
The following box will pop up:



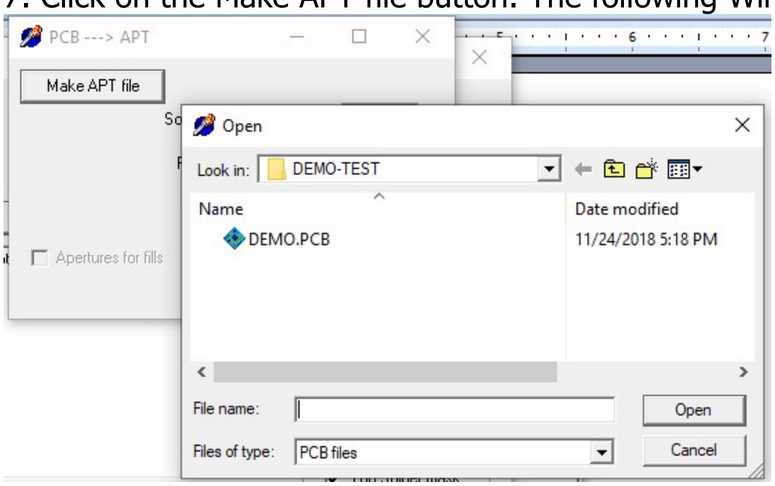
6. Click OK.

The following box will pop up:

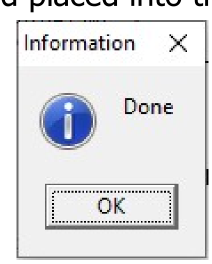


These are the suggested settings; you may change them as you wish.

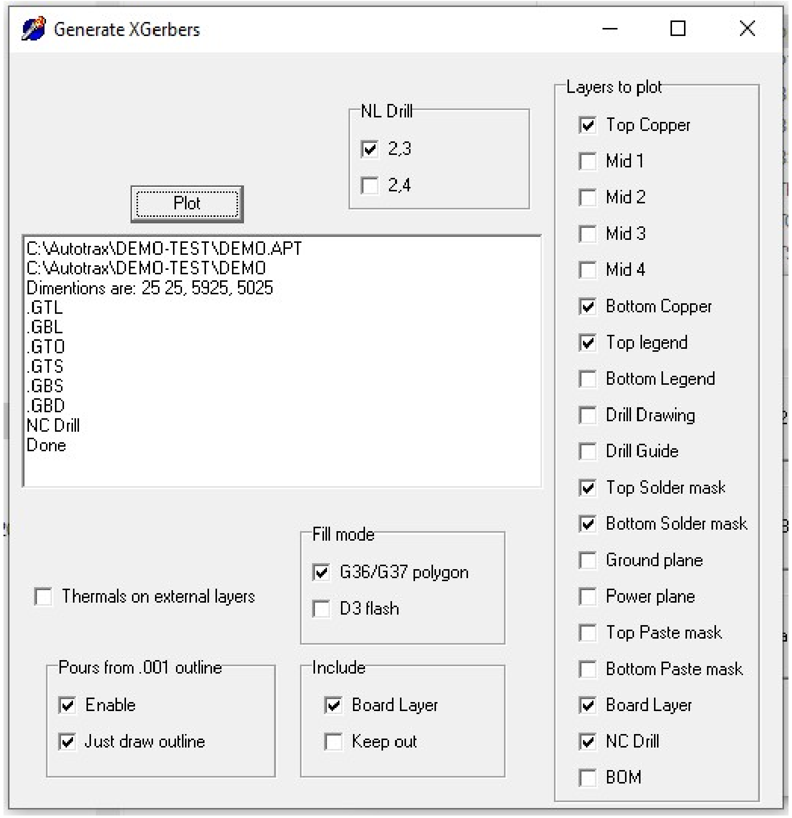
7. Click on the Make APT file button. The following Window will pop up:



8. Select the file you wish to convert and click the Open button.
The files will be generated and placed into the same folder as the PCB file.



The results will show up in the text box area as below.



9. Copy all of the new files to an empty folder and rename the file extensions according to the recommended list. See section below labeled **"Renaming the Gerber and Drill Files"**.

10. Using GerbV (or your favorite Gerber Viewer) confirm that the files are correct. If not, experiment with the Fill mode and re-generate the Gerbers.

Second Method: When you only have Gerber and Drill Files:

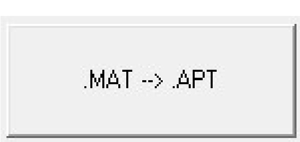
Creating 274-X Gerbers from 274-D and merge the TOL and TXT files into a single Excelon Drill file

Note: You must have an APT file or a MAT file to convert to an APT file to use the Cadcentric Utility to do a Gerber 274-D to 274-X conversion. If in the rare instance you have neither of these, you will need to have the native *.PCB file to do the conversion as above, or you will have to request that the originator of the Gerbers supply you with a complete set of Gerbers with APT or MAT file.

1. Place the Gerber and Drill (*.TXT) and Tool (*.TOL) files into a folder with the Cadcentric File

2. Launch the Cadcentric file.

3. If you have an APT file, skip to step 4. If you have a MAT file click the button labeled:



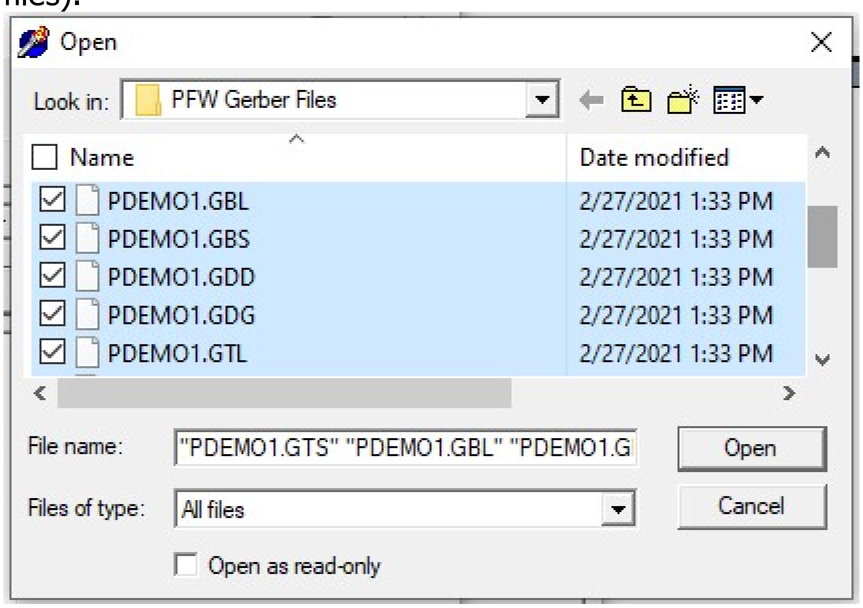
4. You have an APT file or just completed step 3, click the button labeled:

5. Click the button labeled:

The APT file should show up in the box as:

6. Click the button labeled:

A window will pop up where you can select the Gerber files (do not select the MAT or APT or DRL files).



Hold down the Shift key when left clicking the files to select more than one file. When you have completed selecting all of the files click the “Open” button.

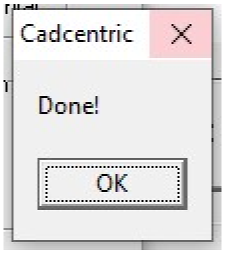
Note:

If you are converting the files for import to SprintLayout you will need files for a minimum of:

Copper Layers (Max 4), Top Silk Screen Layer, Bottom SilkScreen Layer. A board outline is desired but not necessary as you can fix later. If you do not know which layer is for a particular board layer, convert them all, and later figure out the layer assignment to when importing into SprintLayout 6.0.

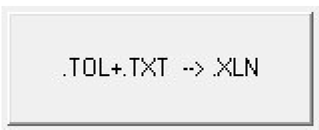
7. Click on the button labeled:

The 274-X files will be generated. Click on the button labeled "OK".

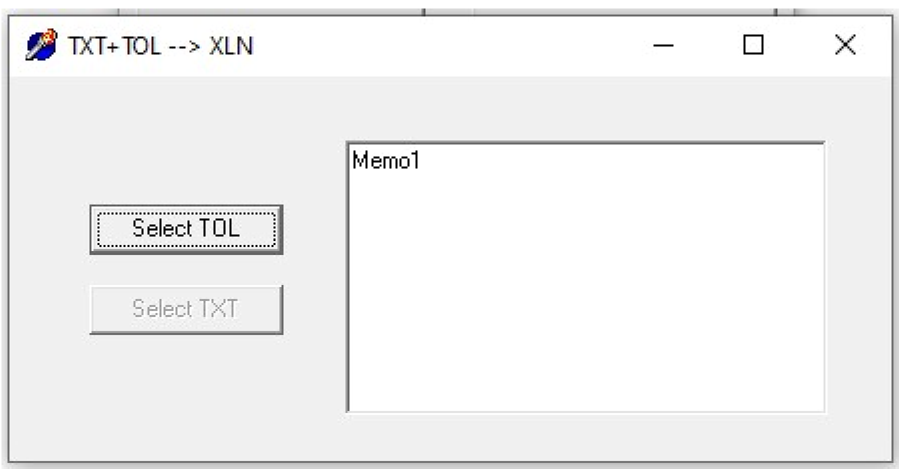


8. Close the Window leaving the Cadcentric Protel Autotrax Helper program window open, as you need to prepare the single drill file.

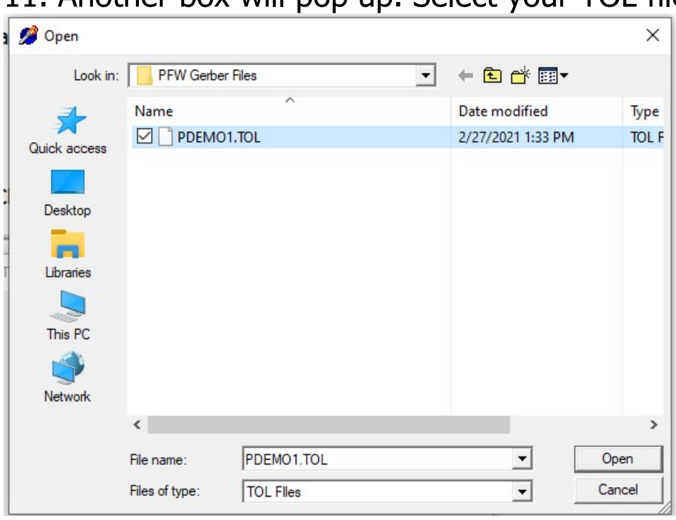
9. Click on the button labeled as below:



10. A box will pop up. Click on the button labeled "Select TOL" as below:

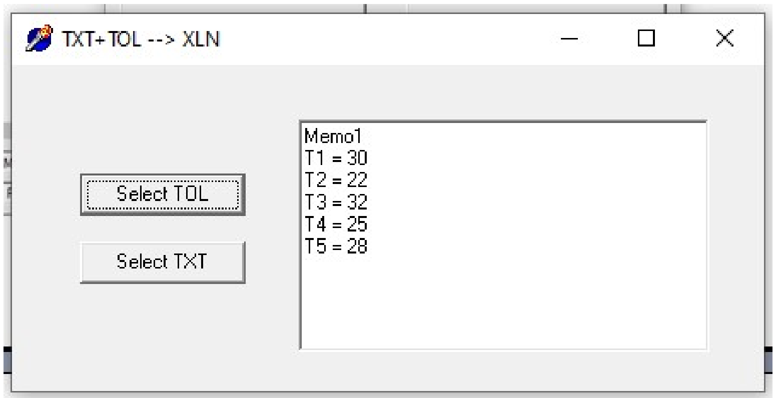


11. Another box will pop up. Select your TOL file then click "Open".

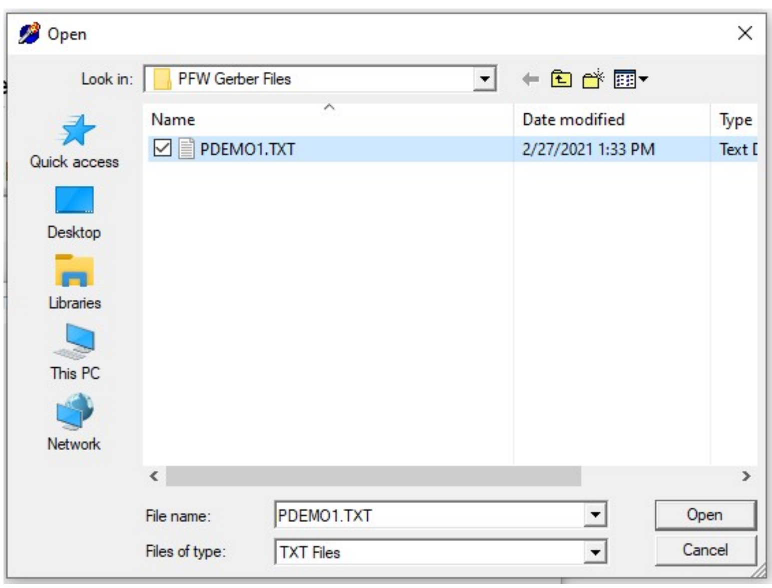


12. The tool data will be displayed in the text area and a new button "Select TXT" appears. This is the X-Y Drill locations file.

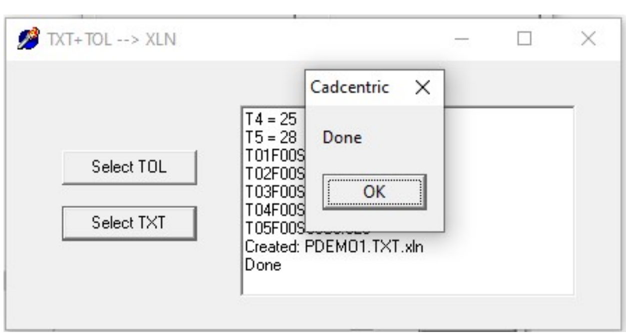
Click on the button labeled "Select TXT"



13. A new Open Window box will pop up. Select your TXT file and click "Open".



14. The TXT file will be converted and the following box will pop up. Click on the "OK" button.

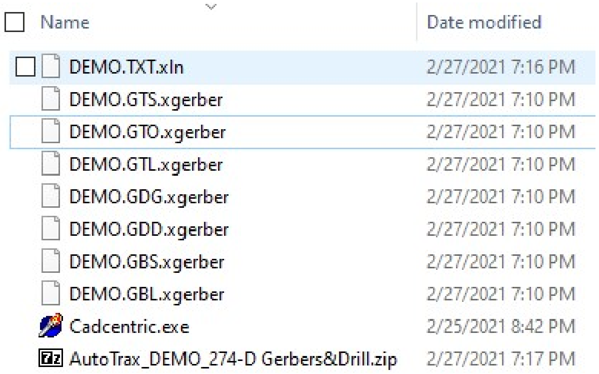


15. The file conversion is complete. You may now close all of the windows associated with the Cadcentric program.

Renaming the Gerber and Drill Files

Navigate to the folder where you placed the PCB or 274-X files. There you will find your original files and the converted files. It is suggested to select all of the original files and place them in a ZIP file and then remove them from the folder and move the ZIP file for safekeeping.

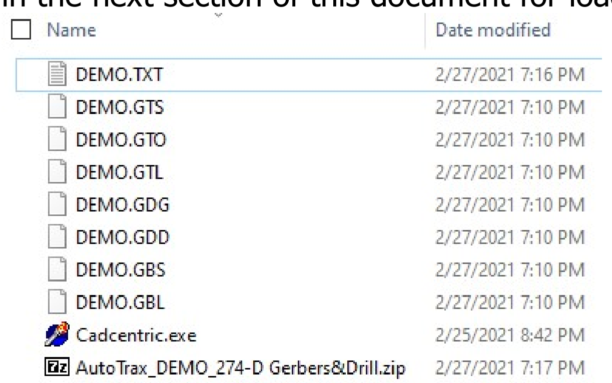
Here you can see a Windows Explorer view of a folder where the 274-D file have been Zipped and the converted to 274-X files before renaming:



Name	Date modified
DEMO.TXT.xln	2/27/2021 7:16 PM
DEMO.GTS.xgerber	2/27/2021 7:10 PM
DEMO.GTO.xgerber	2/27/2021 7:10 PM
DEMO.GTL.xgerber	2/27/2021 7:10 PM
DEMO.GDG.xgerber	2/27/2021 7:10 PM
DEMO.GDD.xgerber	2/27/2021 7:10 PM
DEMO.GBS.xgerber	2/27/2021 7:10 PM
DEMO.GBL.xgerber	2/27/2021 7:10 PM
Cadcentric.exe	2/25/2021 8:42 PM
AutoTrax_DEMO_274-D Gerbers&Drill.zip	2/27/2021 7:17 PM

Note that these files already have acceptable names, just the .xgerber needs to be stripped from the Gerber file names and the file extension for the .TXT.xln will need to be edited to either .TXT or .XLN. The .XLN file extension is recognized by JLCPCB.COM as a drill file extension.

The same files after the renaming: (The .XLN file is simply a copy of the TXT Drill file I made for use in the next section of this document for loading into SprintLayout 6.0).



Name	Date modified
DEMO.TXT	2/27/2021 7:16 PM
DEMO.GTS	2/27/2021 7:10 PM
DEMO.GTO	2/27/2021 7:10 PM
DEMO.GTL	2/27/2021 7:10 PM
DEMO.GDG	2/27/2021 7:10 PM
DEMO.GDD	2/27/2021 7:10 PM
DEMO.GBS	2/27/2021 7:10 PM
DEMO.GBL	2/27/2021 7:10 PM
Cadcentric.exe	2/25/2021 8:42 PM
AutoTrax_DEMO_274-D Gerbers&Drill.zip	2/27/2021 7:17 PM

Below are industry recognized naming structures for Gerber Files and the layer association:

Top Overlay (Silkscreen)	.GTO	Top Copper Layer	.GTL
Inner Layer 2,3 etc.	.G2L	Top Solder Mask	.GTS
Bottom Copper Layer	.GBL	Bottom Solder Mask	.GBS
Bottom Overlay (Silkscreen)	.GBO	Drill Drawing	.GDD
Drill Guide	.GDG	Pad Master	.GPM
Mechanical Layer	.GF1	Power Plane	.GP1
Ground Plane	.GD1	Keep Out/Board Outline	.GKO

Note that there are probably more layer names than are listed here. To remove any confusion as to the order of the layers when the board house builds the boards, a layer stack-up drawing should be included on one of the layers as this information needs to be communicated to the board house.

Note that in most cases, with modern board houses, the Drill Drawing (.GDD) and the Drill Guide (.GDG) and the Pad Master (.GPM) are no longer necessary as no one bombsights pads and drill holes anymore. It does not hurt to include them but in most cases they are not needed and may add confusion.

Special items such as internal cutouts need to be specified as per your board house recommendations. In the case of JLCPCB they want these and non plated holes on the .GKO layer.

Here is a table taken from JLCPCB for their recommended Gerber file naming. Your board house’s requirements may vary. It’s best to confirm this before committing your Gerber files for production.

Filename	Corresponding Layer
boardname.GTL	Top Layer
boardname.GBL	Bottom Layer
boardname.GTS	Top Soldermask
boardname.GBS	Bottom Soldermask
boardname.GTO	Top Silkscreen
boardname.GBO	Bottom Silkscreen
boardname.GKO	Board Outline
boardname.G2L	only if you’ re uploading a four layer board
boardname.G3L	only if you’ re uploading a four layer board
boardname.XLN	Drills

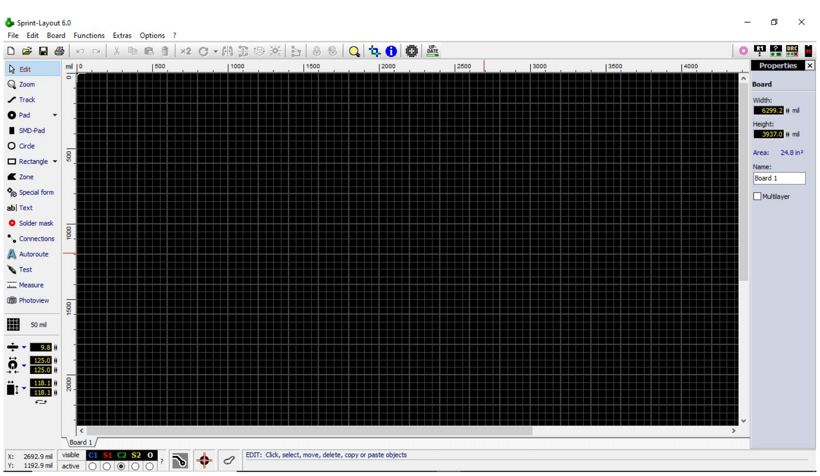
Importing the Converted 274-D to 274-X and Drill Files into SprintLayout 6.0

Note that we will be using the Autotrax Demo of files of a large demo board that I converted in the above procedure. The files are named as in the table above.

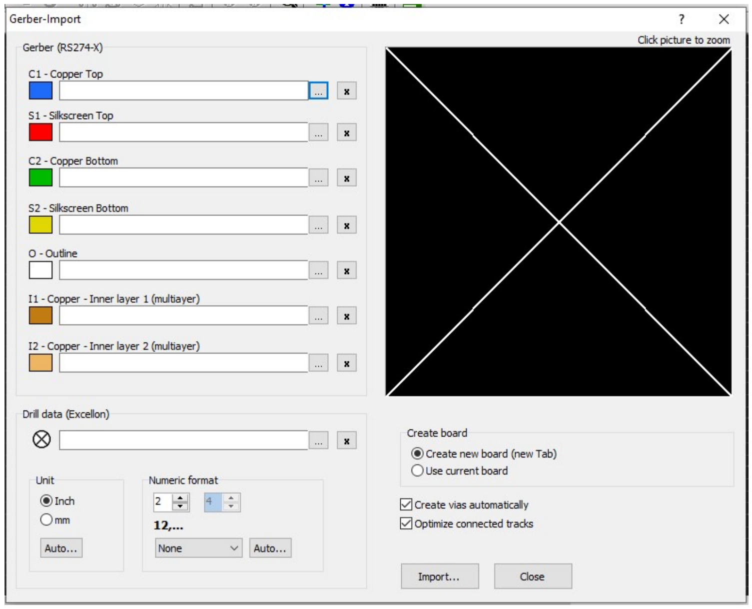
SprintLayout 6.0 can be found here:

https://www.electronic-software-shop.com/lng/en/electronic-software/sprint-layout-60.html?language=en

A demo version can be downloaded to use with this procedure and to learn the program.
1. Launch SprintLayout 6.0. The opening screen will look like this:

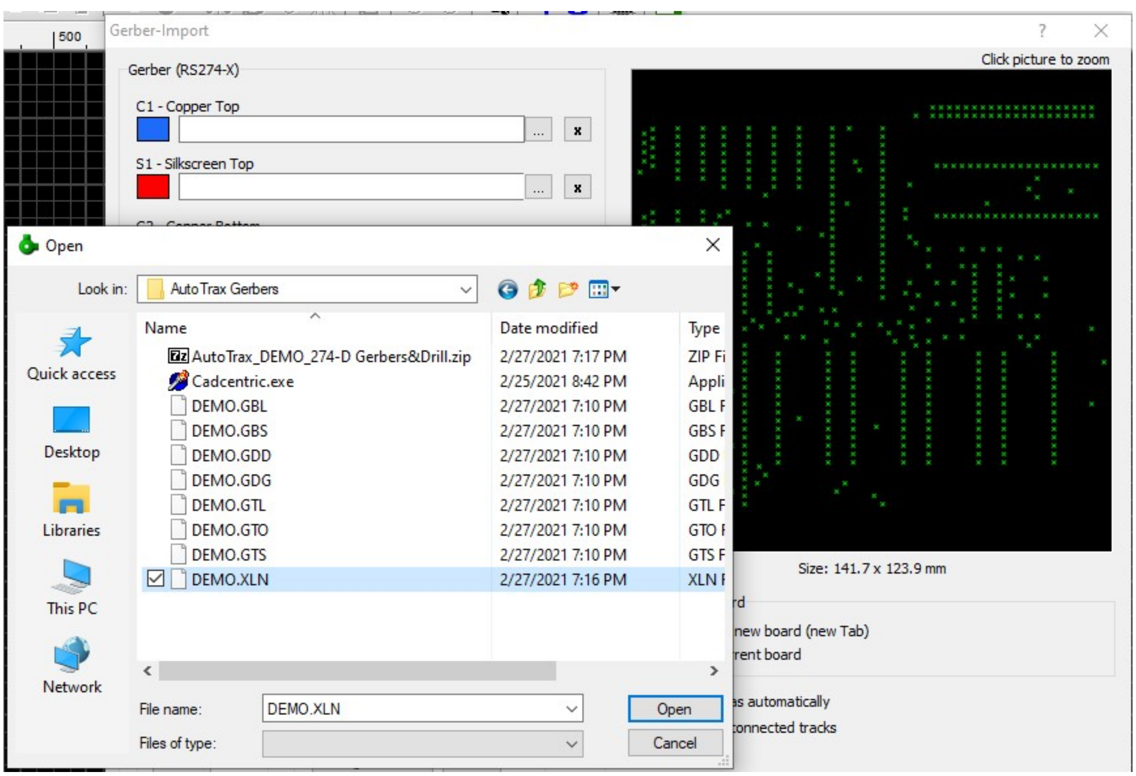


2. From the top left of the Sprint Layout screen select File then click on Gerber Import. The following screen will appear:

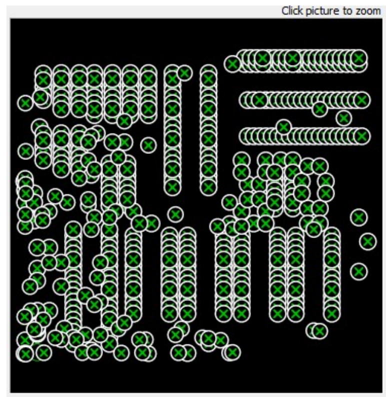


3. Since getting the drill data correct, is very critical, we will import the drill file first. Click on the button just to the left of the button with the X with the three dots. An Open window will pop up. Navigate to the location where you have the 274-X Gerber and drill files and select the .XLN file.

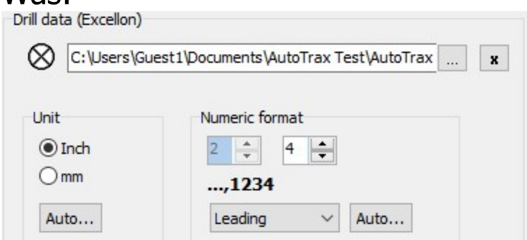
You will notice if the drill options are set correctly, the drill locations will show up in the Gerber/Drill preview pane displayed on the right in the background as below. (If it does not look like this it can be corrected.) Click the "Open" button.



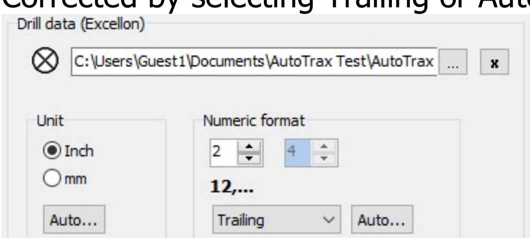
4. The file will be loaded. Note that in the area below the Drill Data (Excellon) the Auto mode is selected. In all cases for imported Protel converted drill formats Auto should work. If you are importing drill files from other sources you may have to experiment to find the correct manual setting. Below is an example of what an incorrect drill import setting looks like:
Preview pane view of an incorrect setting for the Excellon Drill Data Import:



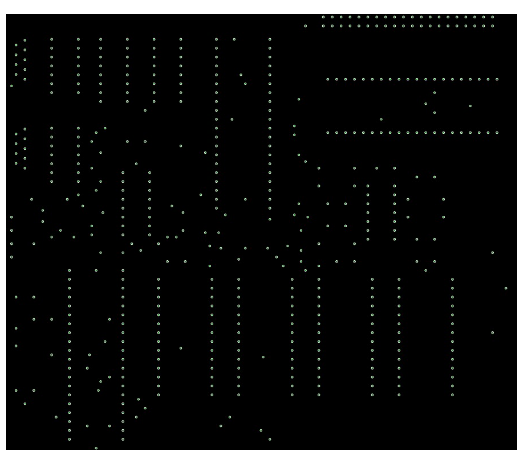
Was:



Corrected by selecting Trailing or Auto:

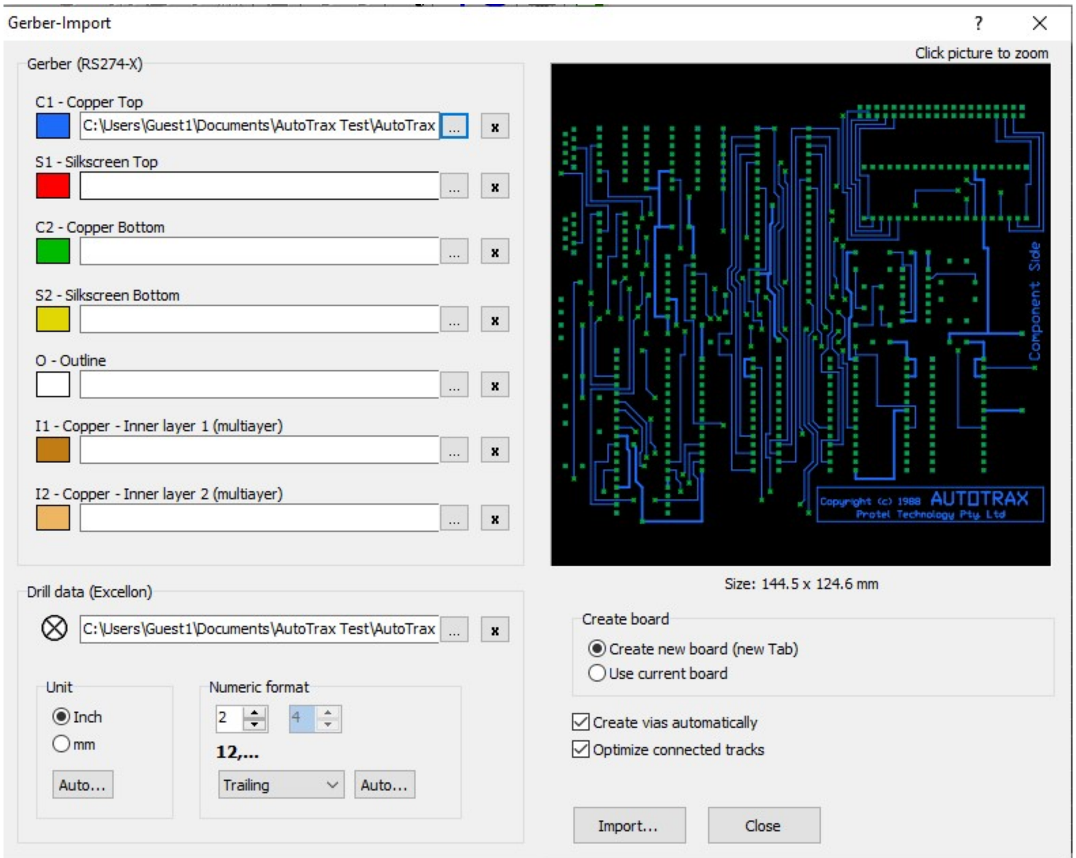


If you click on the preview picture you will see a zoom of the drill placement and notice that the drill placement looks correct. If you made a mistake and clicked on the wrong file just select the X and select another file.



Now that we have the drill files properly imported we can proceed with the Gerber file import.

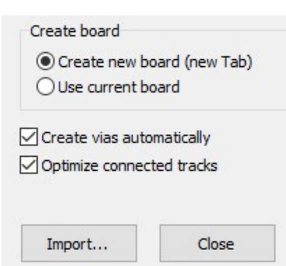
5. Importing the Top Copper Layer. In the same manner as above, click the button with the three dots to the left of the X for C1 – Copper Top. An open window will pop up. Select the .GTL file. The preview pane will show the file selected. If this looks correct then click “Open”.



6. In the same manner as above go ahead and import the C2 – Bottom Copper selecting the .GBL file and the S1 – Top Silk Screen selecting the .GTO file. If you have an outline layer .GKO you may import that as well. If not, do not worry as you can create one after importing the files in SprintLayout and then generate a new set of 274-X Gerber and drill files from Sprint Layout easily.

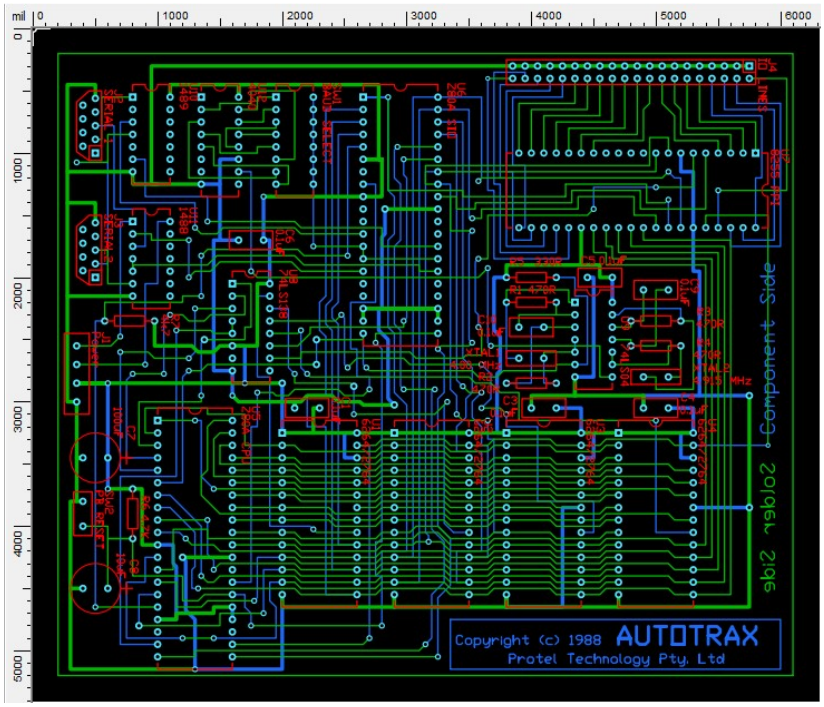
New Gerbers must be created to generate new solder mask layers for the converted PCB if you are going to manufacture the board from using Sprint Files. No matter what always check the files with a Gerber viewer like GerbV before sending them to a board house. Trust but verify!

7. After all of the files have been imported, check to see that the settings under the preview pane are set as:

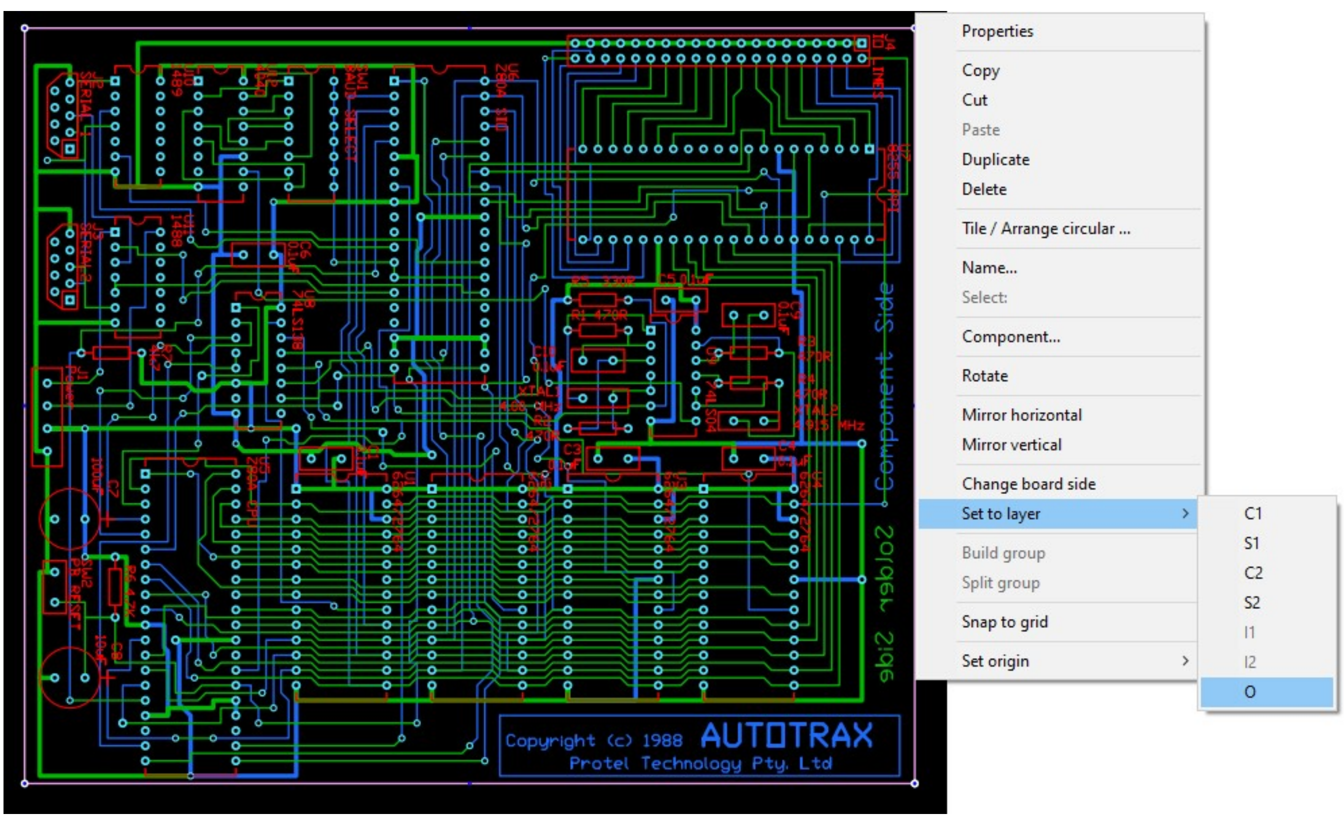


8. Click the Import button under the preview pane. The Gerbers and Drill Data will be imported and displayed on the main Sprint Layout screen.

Here is a screen clip from the board as imported. You can now edit the board as required.



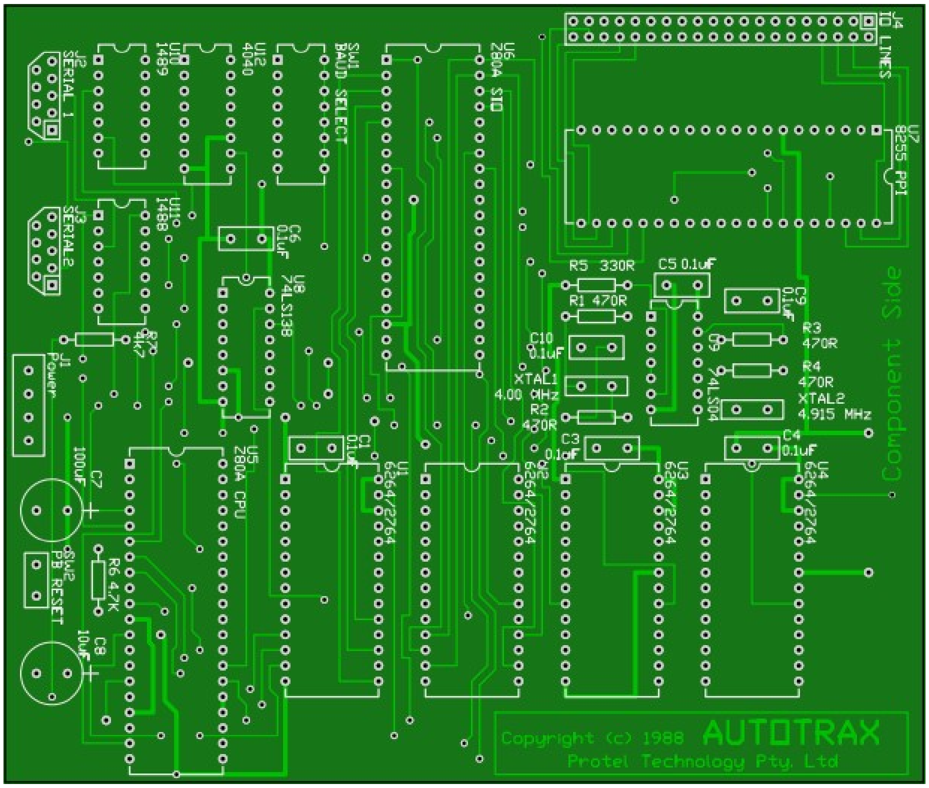
Note that there is a border around the board on the C2 copper layer. This can be selected and moved to the O – Outline layer. Click on the outline, right click and select “Set to layer” then click “O”
After you click the magenta highlighted trace will turn white when it is moved to the outline layer.



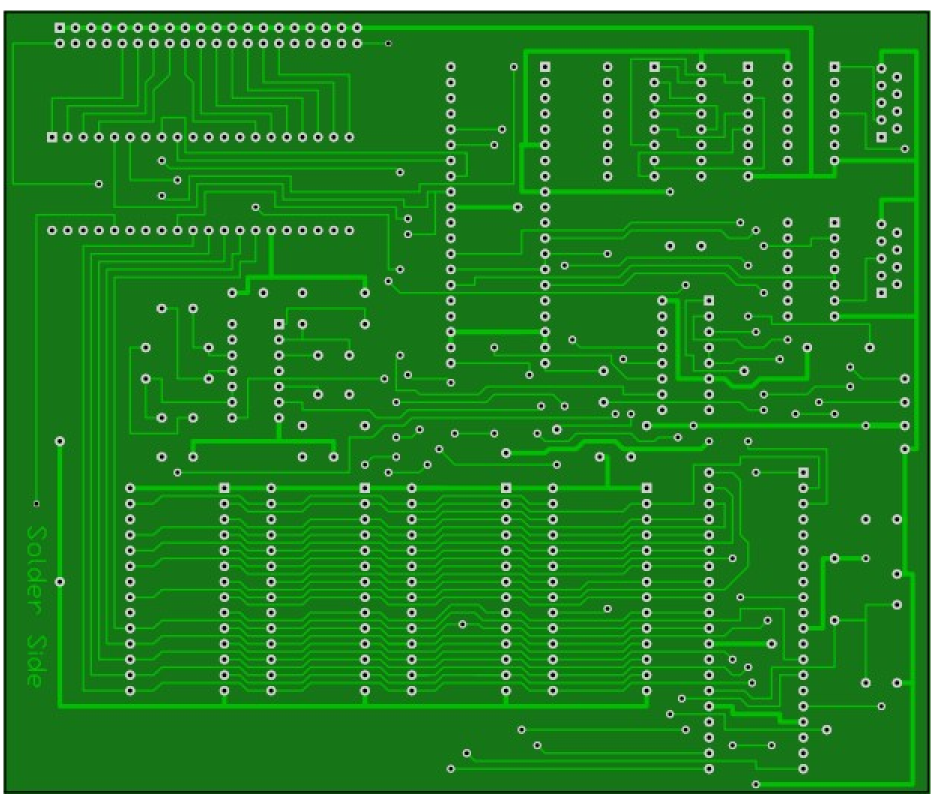
You can now save the file as a SprintLayout 6.0 file by selecting File then save. Once you have the file saved you can use this converted file to create 274-X Gerbers and Drill files of your edited board.

Here is a snap shot of the above board in SprintLayout Photo view mode (both board sides):
This is exactly what the board will look like when it is manufactured.

Top Side



Bottom Side



Some important notes on editing 274-X converted Gerbers in SprintLayout

In general the converted file can be used as is to generate production Gerbers (I'll leave that as an exercise for the reader, perhaps to be covered in a separate document). However for editing, several things need to be kept in mind:

1. Gerber files have no intelligence as they do not inherit information from the CAD system that was used to create them. They are simply flat files that contain simple directions.
2. As such when the files are imported into SprintLayout, the PCB file that is created has been created with simple tracks and pads with holes from the drill data. Tracks are created from the X/Y coordinates. Text is drawn as the movement of a draw aperture. The text is not editable as text, likewise what looks like components are not editable or movable as components as they just primitives like pads and tracks as well.
3. For a good primer on Gerber files and what they are can be found on Les Hildenbrandt's excellent website here: https://www.lh1.org/cadc3dir/gerbertut.html
4. **274-X files that have SMT components, the pads will not be clear of solder mask** so you have to select the top and bottom layer as required and use the Solder mask function to clear the SMT pads of solder mask one by one. This process is quick and painless with Sprint.

So you say, well I went through all of that work for nothing? Well yes and no. Since most of the work in creating a PCB is in the routing and component placement by using the 274-D to X conversion method most of the hard work is done and if your aim was just to create 274-X Gerbers, you are good to go after a final check with a Gerber viewer.

However, you can go well beyond that by using the power of SprintLayout and replace the primitives either with components from the SprintLayout component libraries or by using the simple SprintLayout component creation.

You can make components by copying and pasting the primitives from the converted PCB and pasting them into a blank board workspace by creating a new blank board, which will show up in new tab, (you can copy and paste between tabs) and then save the components to a new user library.

Once new components are created and saved, the original primitives and silk screen can be deleted and the new component put in its place. The details are beyond the scope of this article but it actually takes way less time to do what I have described here than to read this paragraph.

Other SprintLayout Features

The tab feature is very useful. There is a function that allows you to make a copy of a board in a new tab. This allows you to work on a copy of the board without touching the original. A new feature has just been added that allows you to save the board in a Tab as a single Sprint file.

The Tab feature is really useful when you have a group of boards as part of a project. Each board in the project can have its own tab. When you save the file, all of the boards are saved in a single file. When you have a file open you can also import boards to a tab.

In addition you can also save a board as a Macro in a library which can be very useful for making panels of boards, even different ones as you can cut and paste at will.

Here are some of the features I have used in over 10+ years since starting with SprintLayout 5:

1. Making editable PCB artwork from scanned or converted BMP’s using a BMP for each layer. The program has a mode that allows you to float a BMP file in the background and scale it to fit a grid. You then place pads, tracks and components on each layer to create the PCB. I have used this to convert boards from magazines by scanning the page. You can also use a screen capture program like Snagit® to capture the view of magazine artwork and then save the capture off the screen as a BMP.

2. Create panelized artwork with several board designs (or duplicates of the same board) in the same panel. Normally I let the PCB board house do the panels except when I had special requirement so I used the save board as a Macro feature and then pasted the different boards into the panel.

3. Normally Sprint is limited to 4 copper layers and one outline layer. Sometimes when you have a board where you need an additional layer to do for example, special fabrication notes. The easy way to do this: Make sure that the origin crosshair is set to either the bottom left or to the center of a mounting hole in the bottom left corner. Use the board copy function to copy your completed board to a tab. Change to the copy of the board. Delete everything on the board except the board outline on the O layer. Place the information that you want to appear on this layer. When you generate the Gerber file for the board copy make sure that the origin cross has not been moved, and change the file extension to one that is different than any of the files for the main board. Leave the base name the same. After you generate the Gerber files for the main finished board, copy the single Gerber file that you created from the board copy to the full set of Gerber files. As a last step, verify with a Gerber viewer that all of the files line up and that all is as you desire. You could also do things like create power and ground planes separately if you wanted to create a 6 layer board with 4 copper layers and 2 copper layers with power and ground planes.

4. Prototype production using the Toner Transfer method. Make a copy of the board and change all of the pads holes to 0.010. Vias may have to be enlarged for hand drilling with a drill press. You may have to experiment with the hole sizes to make sure that centers of the holes etch through and are visible for drilling as the toner may spread slightly when heated in the modified laminator.

5. Prototype production using an inexpensive CNC machine. Sprint has a feature that creates files that can be used with a third party free program called SL2M3 by Bernard Pahl (Google to find). This program converts the output to G-code files so that a CNC machine can be used to mill each side of the board and to do the drilling. The program can be used in conjunction with the paid for Windows CNC control program MACH3 or the free CNC control program LinuxCNC (formerly known as EMC2). The SL2M3 program also inserts pauses into the generated G-code for drill changes. This link up between the two programs works really well and I have used them to make many prototype PCB’s. One caution: To be able to make double sided boards the CNC frame must be absolutely square so that when you flip the board to the other side that the drilled holes will line up. Additionally you may need to run the G-Code files through an auto-leveling program to insure that the copper is milled precisely and that the milling cutter does not cut into the base material too much. You can find a free auto leveling program on the internet.

6. Auto plane pour for copper layers with a single click. You can set the air gap for all board items and even select a single item can be set to a unique air gap. Pads can be set to thermal reliefs or be connected directly to the ground plane. For thermal relief's they can be set to connect to one plane or both. This allows one plane to be power and the other to be ground.

There are probably a dozen or more features I could cover here. I have personally used in the last 40+ years just about every major and minor PCB CAD program on the planet and was a beta tester for some. When I could not get my personal copy of Tango PCB Series II+ running on modern operating systems with high resolution video I had to find a solution. Recently I had to resurrect my old PC and run TangoPCB. Although it was an excellent program, it made me realize that I really don't miss the old program all that much!

The main points on this software that made it very attractive to me:

- Very easy to use with a really short learning curve.
- Very inexpensive 49.90 euro (19% less if not in the EU) (cost for dinner of 6 at McDonalds)
- Very powerful, fast and low footprint
- Anti-aliased high screen resolution screen video (even Altium is not as good!)
- Can do PCB's as large as 500mm X 500mm (>19 X 19in) at 0.01mm (0.397mil) resolution
- Instant switching between inch and mm and quick change grids
- Part rotation adjustable to extremely low increments
- Mirror components or text vertical or horizontal
- Make arrays of pads/components, tile or circular as easy as falling off a log
- Making components is a breeze. Lay down primitives, draw a border around them, save!
- Can also capture components from a layout and save them!
- 274-X import to PCB (but you already know that now)
- Many methods to make prototype boards.
- Printer calibration feature both X and Y (Perfect for Toner Transfer PCB's)
- The Photo view and Gerber handling alone is worth the cost of entry
- No weird key entry or dongles. It comes embedded with your name, easy to move to a new PC. At the cost every Ham and hobbyist and engineer should have a legal copy.
- Updates within a major version are free, with a minimal cost to upgrade to the next version.
- Product support is outstanding on the very rare instance when you might need it.
- Runs on Win7 thru 10, 32 or 64 bit. Will run in WineHQ in Linux and runs at full screen resolution without any hitches. The installed program takes well under 10MB (yes megabytes)

I must apologize if my presentation sounds like a commercial, I don't mean it to be, and I do not have any financial interest in Abacom, I just really like their products as they are a real bang for the buck and have served me well even with very complex mixed signal boards.

I have probably made several hundred flawless layouts with the product both for commercial and hobby use. This is truly an outstanding program and I am sure that anyone that makes PCB's should give it a look, certainly anyone using old DOS software that has become difficult to maintain and install.

Please send Les Hildenbrant a big thank you email (les@hildenbrandt.com) for modifying his excellent Cadcentric program so it can easily be used with SprintLayout. Please sign up for membership to SprintLayout on groups.io. Email comments to SprintLayout@groups.io

Addendum 1 – Creating Gerber Files from Sprint Layout

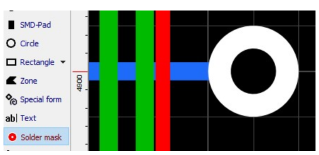
Reminder – If you used the procedure above to convert Protel based 274-D files for import into Sprint or if you imported 274-X files from another source, you will need to regenerate the 274-X and drill files by exporting them from Sprint to create a complete set including new solder mask files.

If you imported 274-X files that have surface mount devices make sure that you have used the Solder Mask mode and have selected all of the surface mount pads so that they are to be clear of solder mask.

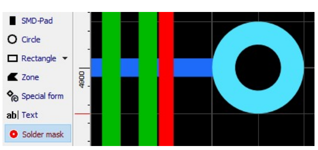
You must be sure that this is correct in the layout before proceeding. You may also use the Solder Mask editing button to cover Vias with solder mask, if you would like your board produced in that fashion. Above all, make sure that you check your generated Gerbers with a viewer.

Solder Mask Editing: In SprintLayout use the Solder Mask Button on the left 

Solder Mask Selection. Note that the Solder Mask Selection mode on the left has been selected and highlighted in light blue. Clicking an item (Pad or Track) will switch between mask and no mask.
Example of a pad that will NOT have solder mask applied. Note that the pad appears white.

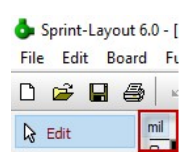


Here is the same pad WITH solder mask. Note that the change in color is when you are in the Solder Mask selection mode only. This is not to be confused when you are in the standard view mode in EDIT and though hole pads are shown in a similar blue color.

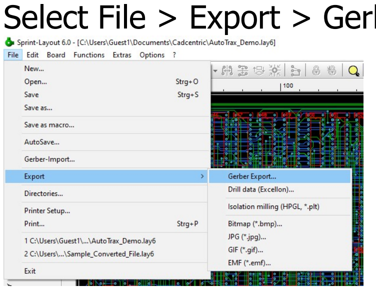


Prerequisite: Create an empty folder for the Gerber files. I chose C:\Gerber Files for this example.

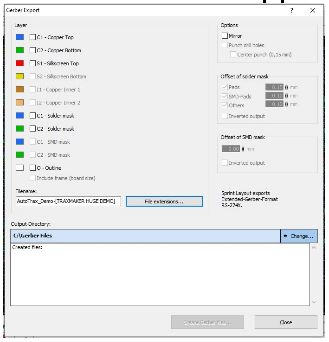
1. Open your converted .lay6 file in Sprint. If you have multiple boards in your file select the tab at the bottom and make sure that the layout file that you want to export to Gerber and Drill Files is displayed. Make sure that you are in mil mode not mm mode. Click in the area shown on the main Sprint screen in the left corner until mil appears.



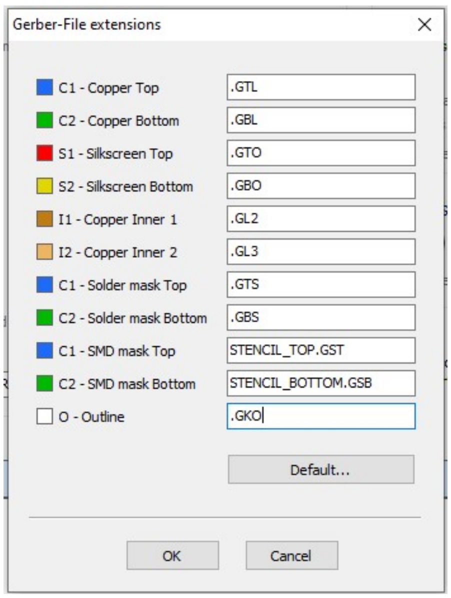
2. Select File > Export > Gerber Export



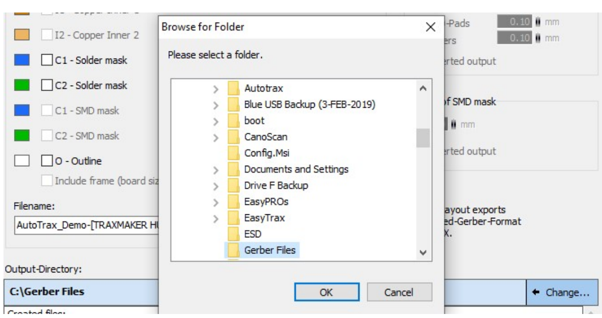
3. This window will appear:



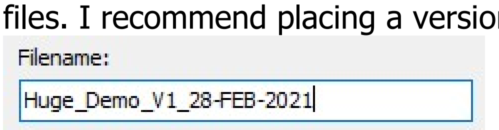
4. Click the "File Extensions" button. The window below will appear.
5. Edit the file extension to the standard below. Then click the "OK" button.



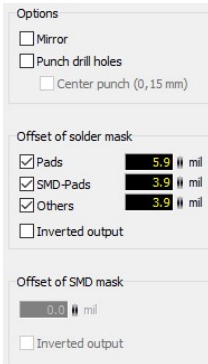
6. Click the "Change Button" and navigate to the empty folder you created for the Gerber/Drill files, and click "OK".



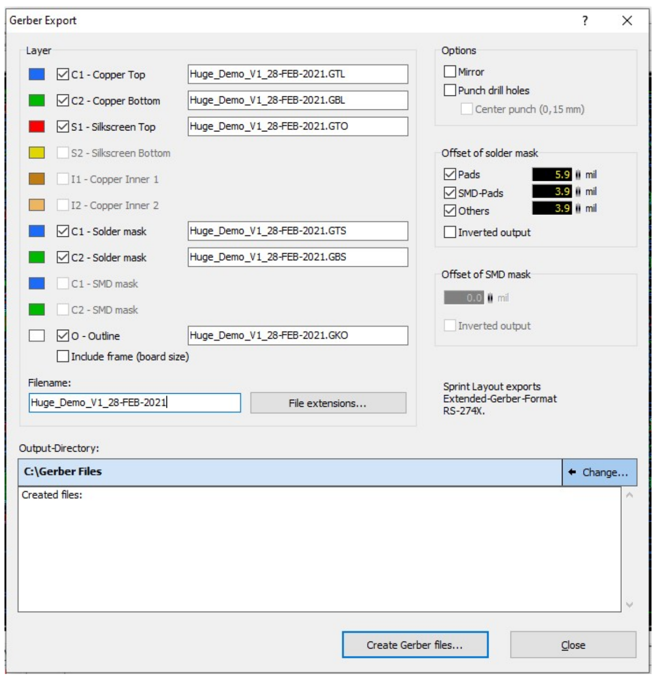
7. Click your mouse inside the Filename: box and edit the filename. This will be the base filename for all of the Gerber files. Remember this name as you will need it when generating the drill files. I recommend placing a version number and a date in the filename for reference.



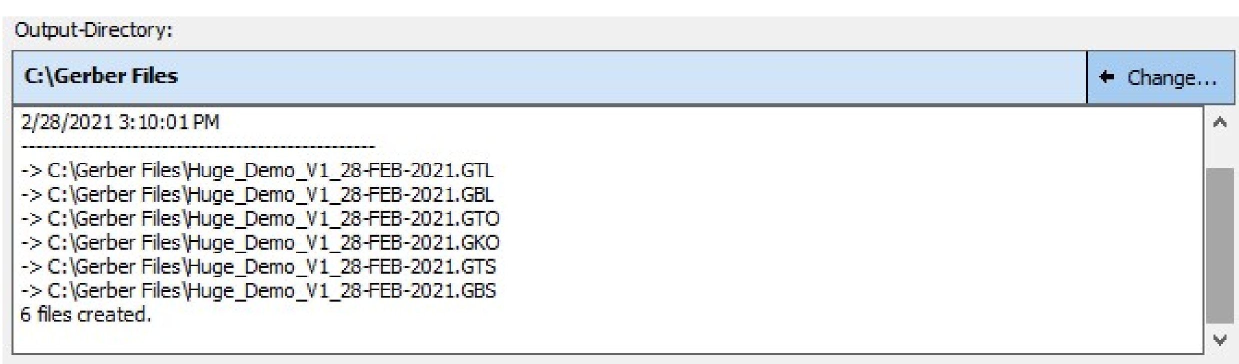
8. Check on the right side of the window and verify that the options are set as shown below. Normally the defaults should work fine, feel free to change them to meet your needs. The SMD Mask will not be grayed out if there are SMD pads on the layout.



9. The Gerber Export Window should look as shown below. You are ready to generate the files. Click the "Create Gerber files…" Button at the bottom of the window.



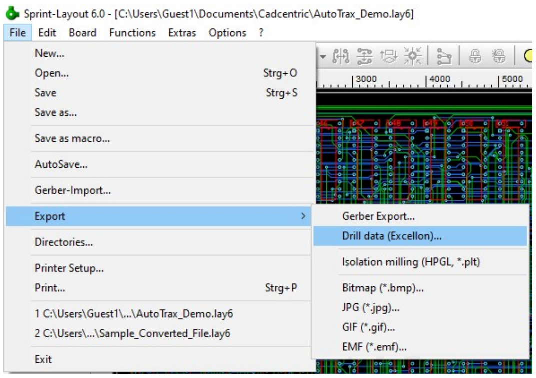
10. The files have been created and the Output Directory section of the window displays:



11. You may close this window and proceed to the Drill File creation in the next section.

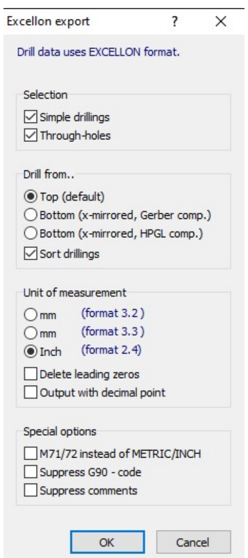
Creating the Drill Files

1. Verify that you are still set to the mil mode as in step 1 in the preceding section.
2. From the main SprintLayout screen select: File > Export > Drill data (Excellon)

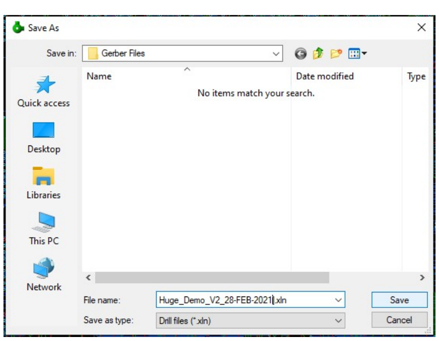


3. The following Window will appear.

Make sure that the defaults are set as shown and click "OK".



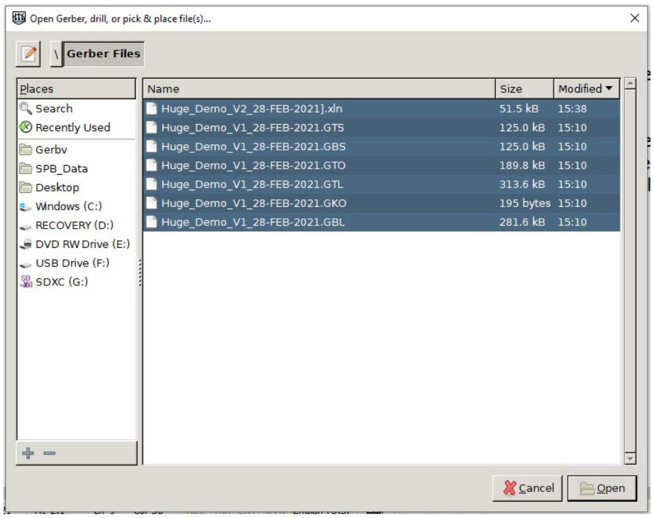
4. The following Window appears. Edit the filename to agree with the filename of the Gerber files leaving the file extension as XLN (or change it to .TXT if desired) Then click on "Save". The Gerber and Drill files are now ready for you to do a final inspection with a Gerber Viewer. In the next section the Gerbers will be loaded into GerbV (a separate program) and inspected for proper content.



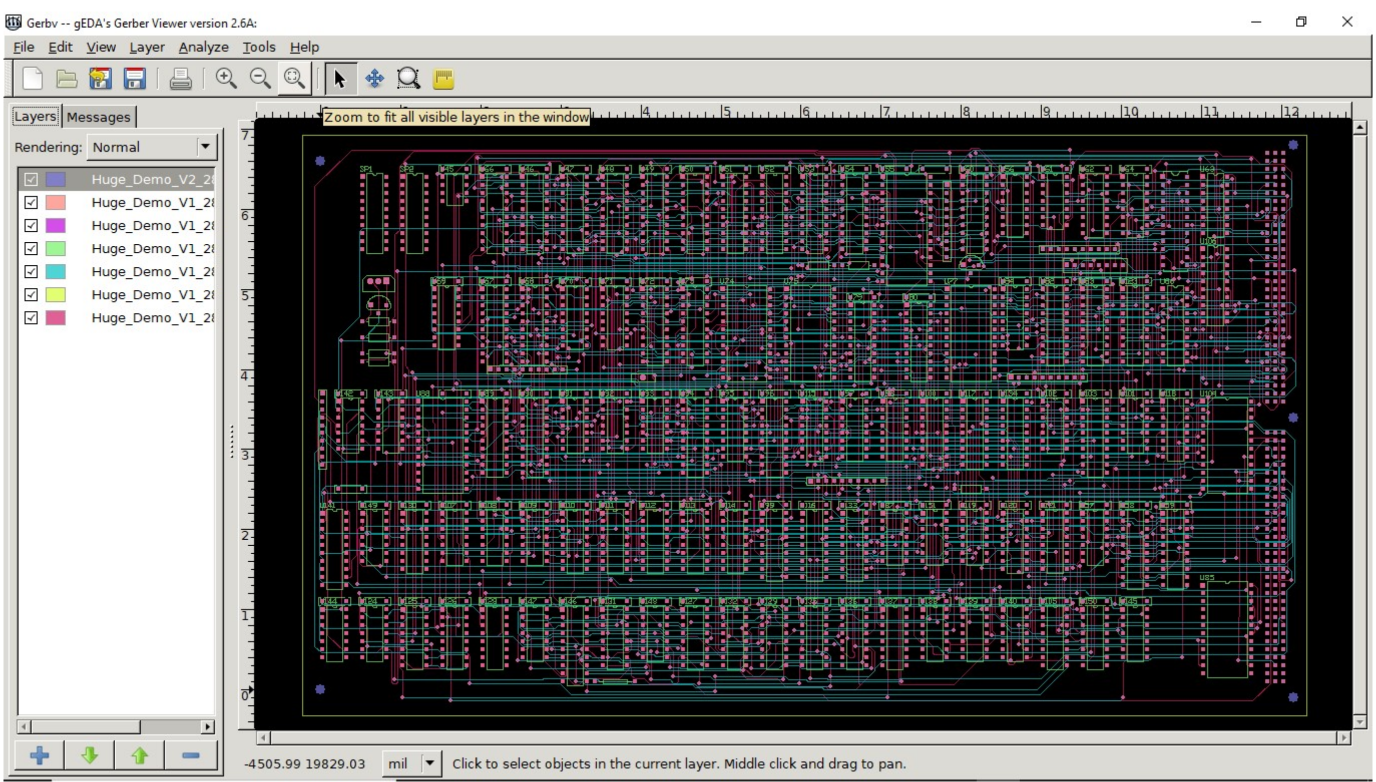
Inspecting the Gerber and Drill Files with a Gerber Viewer

In the example below we will be using **GerbV** a free and excellent Gerber Viewer. GerbV may be downloaded from: https://sourceforge.net/projects/gerbv/

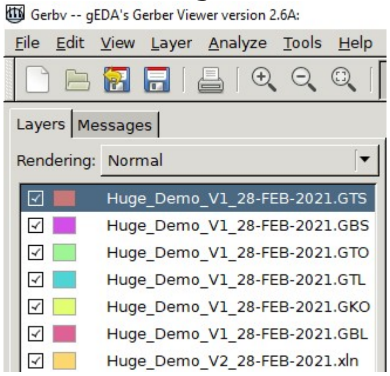
1. Launch GerbV and choose File > Open Layers(s) and navigate to the drive and folder where you placed the Gerber files (In my case C:\Gerber Files). Double Click on the folder to open it. Once open, hold down the left Shift key and move your mouse cursor to the last file in the list and left click to tag all of the files. The GerbV screen should look like this at this point:



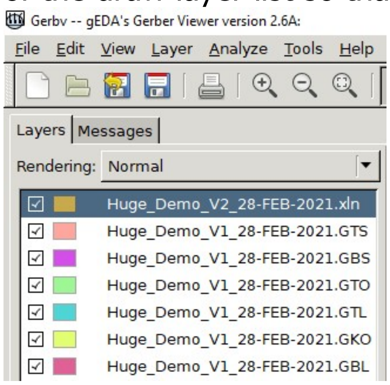
2. Click "Open"
3. The screen should look like this. It looks a lot like the Sprint Layout view of the PCB, as it should. Click the Zoom button to fit the board to the window.



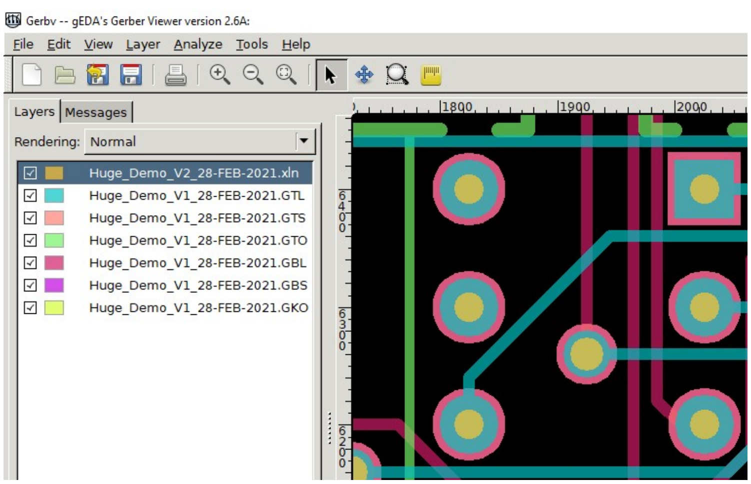
4. Look at the left pane where you can see the layer color assignment; you can change the colors by clicking in the color box. Note that the next time you open a PCB a different set of colors will be used. I believe that there is a way to make the settings permanent, but I have not found it yet. You can also turn the layer visibility on and off with the checkbox. This is handy when checking the drill file or removing clutter from the screen.



5. Use the mouse cursor and left click and hold to highlight the .XLN layer and drag it to the top of the draw layer list so that the list appears as:

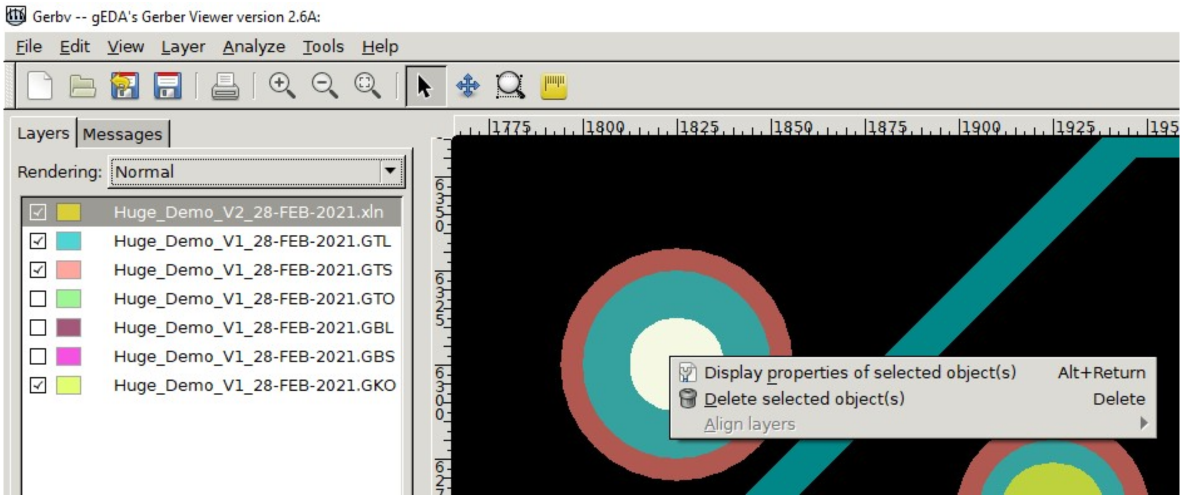


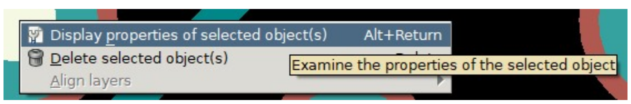
6. You can also rearrange the layers so that they are in the normal layer stack up order with the drill layer at the top:



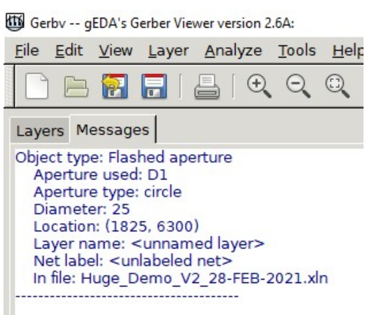
This will change the display so that the drill holes will be plainly visible in the layout display. Here is a clip where you can see the drill holes (25mils and 28mils) and also the pads, vias and the solder mask. Note that this is in composite color. To view one layer simply turnoff the others.

7. Here I have modified the colors and turned off the bottom layers so the top and drill layers can be examined in detail. Note when the active layer has been selected, (the .XLN) the center of the drill can be highlighted with a left mouse click. A right mouse click brings up a selection menu. Selecting "Display properties of selected object(s) with a left mouse click, will display the data for the selected object (in this case the 25mil drill size as detailed below).



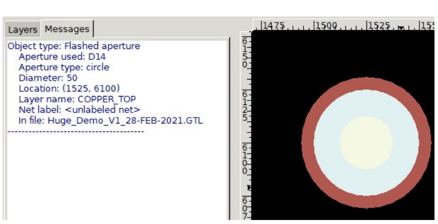


8. The diameter of the drill for this location is 25mils which is correct.

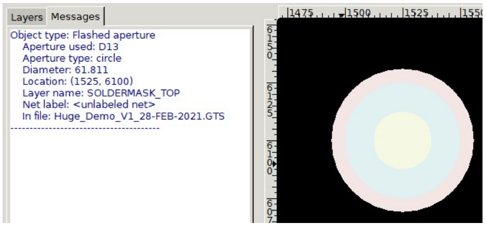


The main take away from this exercise: The drills are aligned with the pad layers, there is an adequate annular ring of copper for the pad after drilling, and the solder mask swell is adequate. After you have seen a few dozen boards in the viewer, you will be able to quickly determine if all is as it should be. All of the details can be checked by using the display properties as shown above.

9. Now that we know how to use GerbV basics, let's look at more details of a the Copper_Top 50mil pad. This looks to be in specification. We verified the drill size in the previous example.



10. Now let's inspect and measure the solder mask swell for the same pad. It measures 61.8mils which is correct for a 50 mil pad (50mils + 11.8 mil solder mask swell (~=5.9mil X 2). This insures that the solder mask will not encroach on the pads when the board is manufactured.



11. Here is a line on the .GBL (Bottom_Copper) layer. This line is plotted with a 10mil aperture which is exactly correct for a 10 mil track.

12. I will leave the rest of the board inspection as an exercise for the reader as the basics have been covered in this tutorial. In all respects the drill files and the Gerber files should mimic the CAD data generated by SprintLayout, or any other PCB CAD package for that matter.

I hope that this tutorial will be useful to you in your endeavor to create PCB's. If you have any comments or constructive criticism of anything I have presented here, I am open to hearing from you. Please put Sprint Layout in the subject line to avoid the spam filter.

Sam Reaves
ARS W3OHM
sam.reaves@gmail.com
SprintLayout@groups.io

Short author bio:

I have worked in electronics design since the middle 1970's, starting with Pulsecom in 1976 designing telecom devices. During this time, the PCB industry was still using tape and Bishop Graphics puppets for PCB artwork. In the 1977 thru 1988, I worked on UNIVAC mainframe computer systems for Western Union and first saw WinTEK smARTWORK, the first PC based PCB CAD. In the late 1980's until 2004 I designed electronics for the automotive industry for then VDO, a well known gauge and instrument cluster manufacturer, using TangoPCB, OrCAD VeriBest and a couple I can't remember. From 2004 through 2019 I was a staff scientist with Stowe Woodward/Xerium/Andritz and worked in their global research and development group, designing custom electronics for the paper industry including the electronics for their flagship product SMARTRoll®. Most of the special PCB's for these products were done in SprintLayout or earlier in P-CAD 2006. I have design expertise in circuit design from DC to daylight. I hold 10+ patents in electronics and related fields. I retired from the corporate world in late 2019. I still consult and design PCB's and electronics for fun, and sometimes profit!