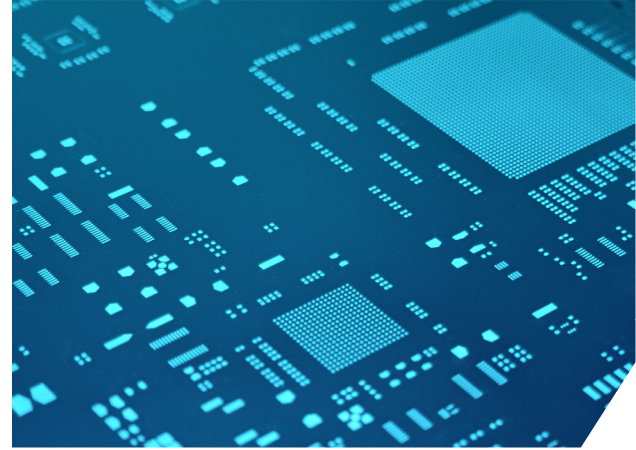


Considering the gerber plots when purchasing a solder paste stencil

By William Coleman, Photo Stencil and Travis Tanner, Plexus Corp. Manufacturing Solutions

When purchasing a stencil for printing solder paste on a printed circuit board (PCB), there are many things to consider, such as aperture size, area ratio, and the material the stencil is made of. One factor that is equally important, but often overlooked, is how the stencil manufacturer outputs the stencil given the Gerber files it is provided. This becomes extremely important, especially when it comes to programming a solder paste inspection machine. Plexus Corp., an electronics manufacturing and engineering services provider, was able to prove that a considerable amount of time and money can be saved when you work with a stencil manufacturer who is very exacting when it comes to providing plots for its stencils.



Plexus was engaged with a customer in its new facility in Guadalajara, Mexico. During training of how to program a solder paste inspection machine, a few new engineers received 274x Gerber plots from the customer's stencil vendor to use for internal engineering and qualification builds. It took hours to correct the errors in the files. The apertures were all made up of draws, nothing was flashed, and fiducials were made of hundreds of line segments.

Generally, Gerber files that are used to manufacture the PCB are given to the stencil manufacturer to produce a stencil. That Gerber plot helps the stencil manufacturer know and understand the customer's application. When all the draws are flashed and all the customer modifications made, a 274x check plot is output. Although a 274x Gerber plot is the most common format, other types of files can be exported, such as DXF and PDF.

To clean a stencil file, the manufacturer has to take the draws and translate them into apertures. A draw is a little line on the file. Draws are a problem because apertures should be one entity, or a flash. Draws, on the other hand, are comprised of multiple lines. In the case of the stencils being used by Plexus to program a solder paste inspection machine, there were a considerable amount of little lines that comprised the apertures. It is the job of the stencil manufacturer to clean up all those lines so only open apertures are demarcated and visible. Any lines inside the apertures prevent the creation of a good clean stencil.

“Using properly prepared stencils, like we receive from Photo Stencil, saves money and time. Having an actual comparison helped us to realize the importance of making sure best practices are used in preparing our stencils.”

-Travis Tanner, Plexus Corp.

The stencil manufacturer performs a task called flashing. Flashing converts multiple lines, or draws, into one entity. Fiducials should be made of a flash. It is easier to program a single flash than hundreds of lines. When the printer reads the stencil program, if there is just one entity, the printer can program it easily. It takes a considerable amount of time and printer memory if there are hundreds of lines that have to be read in order to create a program. Add to this employee time, the cost of running the printer, and down-time for preparation and programming and the costs quickly add up.

“I was training new engineers to program a paste inspection machine,” said Travis Tanner, Senior CIM/CAM Technician at Plexus. “For the first two programs I used their past vendor’s plots for internal engineering / qualification builds. It took hours to clean them up. Apertures were all made up of draws, nothing was flashed, and fiducials were made up of hundreds of line segments. The list goes on and on.”

“I requested that they use these two bad files to drive a point,” continued Tanner. “And the people I taught were reluctant. Their old behavior of using stencils was that they were ‘good enough’ and they were hesitant to talk to me about using anybody else. So I made them a bet. I gave them two of their vendor files and I took 12 Photo Stencil 274X Gerber plots. My time spent cleaning was zero because of the process Photo Stencil uses. It was a big eye opener for my team here. We were done within minutes rather than hours.”

To do the clean-up, Photo Stencil uses specialized stencil creation software that has a lot of latitude to enable Photo Stencil to clean up the files they receive and get them in a style that allows the stencil to be manufactured, and also gives the customer a format that fits their requirements in production. Programming on the paste inspection tool takes the Gerber file and then does scans of the printed board to understand placement and paste volume. In the case of Plexus, Photo Stencil provided them with “cleansed” data, which makes the programming much simpler than it would be otherwise. Any draws that are present are fixed manually. The program can then be uploaded into the paste inspection machine.

“Using properly prepared stencils, like we receive from Photo Stencil, saves money and time,” said Tanner. “Having an actual comparison helped us to realize the importance of making sure best practices are used in preparing our stencils.”