

Avoid Seven Common Mistakes with Gerber Files

By **Tony Breglio**
Advanced Circuits

The Gerber format for photoplotters is the definitive industry standard for the creation of printed wiring boards (PWBs). Today, the most popular standards are the Gerber RS-274X (274X) and RS-274D (274D). Here are seven common mistakes we often find with Gerber files sent to Advanced Circuits to be used for PWB manufacture.

1. Missing aperture list

An aperture list does not need to be sent with 274X format files. If you send 274D format, we use your aperture list together with your Gerber files to create your artwork. Your Gerber files specify what to do and where. Your aperture list specifies what tool to use. Please output one comprehensive aperture list for all layers, rather than a separate aperture list for each layer.

Requirements

One comprehensive aperture list is required for all layers, English units. Please do not modify the aperture list that your software outputs.

Resolution

All layout packages that output 274D also output an aperture list. Common extensions include .rep, .apt and .apr. If you have difficulty outputting an aperture list, please send 274X format.

2. Missing Excellon drill

The standard for PWB drilling information is the Excellon drill list for equipment from Excellon Automation. We use your Excellon drill file to determine what size holes to drill and where. Plated and nonplated holes should be included in one drill file, with different tool numbers for plated and nonplated holes.

Requirements

Excellon format, ASCII Odd/None, 2.4 Trailing Zero Suppression, English units, no step and repeats.

Resolution

Nearly all layout packages will output an Excellon drill file. If you cannot generate one, we can, in most cases, create one from your fabrication drawing for an additional fee.

3. Missing tool list

We use your tool list in combination with your Excellon drill file to create your drill. Your drill file specifies where to place the holes. Your tool list specifies what tool to use. The tool list should be embedded in your Excellon drill file or sent as a separate text file. Using a tool list provided on a fabrication drawing is not useful because it eliminates many of our automatic verifications and makes data entry errors far more likely.

Requirements

Tool list embedded in Excellon drill file or sent as a separate text file.

Resolution

If your layout software will output an Excellon drill file, it will probably also output a tool list. Common extensions include .tol and .rep.

4. Missing Gerber files

We often receive orders submitted with no Gerber files whatsoever.

Requirements

We prefer Gerber 274X or 274D, English units.

Resolution

We have the capability to use ODB++ files but can also convert many other file formats to Gerber.

5. Insufficient annular ring

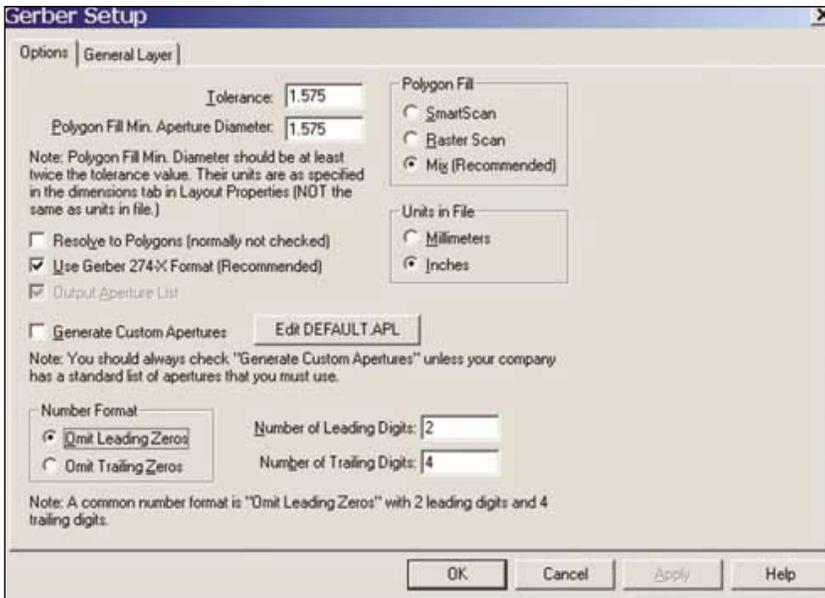
The annular ring is the donut (“annulus”) created when your drill pierces a copper layer. It is defined as the radius of this donut. For example, a 0.030-inch pad with a 0.020-inch hole would have a 0.005-inch annular ring. This is required for complete plating on vias and good solder joints on component holes.

Requirements

We require a minimum of 0.005-inch annular ring for vias and a minimum of 0.007-inch for component holes.

Resolution

Many layout packages provide this as a DFM check. Setting sufficient annular ring in your layout software is



▲ **Figure 1. Gerber setup screen from the GENESYS RF and microwave design software suite from Eagleware Corporation.**

the preferred method, because with this method, copper spacing will be maintained.

6. Copper trace width/spacing

Copper spacing is the minimum gap between any two adjacent copper features. Trace width is the minimum width of a copper feature, usually traces.

Requirements

We require a minimum of 0.005-inch trace width/spacing. A premium is charged for trace width/spacing less than 0.008 inch.

Resolution

All layout packages provide trace width/spacing as a design for manufacturability (DFM) check. Setting sufficient trace width/spacing in your layout software is the preferred method. Trace width and spacing push and

pull against one another and changing a problem area may require rerouting traces, adding vias, or moving components.

7. Insufficient inner clearances

Inner clearance is the minimum distance from the edge of a hole to any adjacent, unconnected, inner layer copper. Sufficient inner clearances help ensure that your drill does not cause shorts to your inner copper layers. This is important for both plated and nonplated holes because nonplated holes may either cut into an adjacent trace, or cause shorts during assembly.

Requirements

We require a minimum of 0.012-inch inner clearance, with 0.015-inch preferred.

Resolution

We can resolve most inner clearance issues if negative image inners are provided. Setting these clearances in your layout software is the preferred method because this will maintain intended connectivity. Although most layout packages provide this as a DFM check, not all do. Those that do not can usually be manipulated to check for this violation by setting spacing and annular ring higher.

A rule of thumb: spacing + annular ring = inner clearance. Another trick that can help resolve problem areas is to move the affected traces to outer copper layers where this is not an issue. ■

Author information

Tony Breglio is engineering manager with Advanced Circuits. He received a bachelor of science degree in industrial and manufacturing engineering from Western New England College in Springfield, MA. He may be reached via E-mail: tony@4pcb.com; Tel: 303-418-2337; or Fax: 720-489-3767.