Printed Circuit Board and Boards: Data package problems?

Try these steps to get front-end engineering on the open road

These are exciting times for the printed circuit manufacturing industry. Fabricators are experiencing double-digit growth, maintaining staggering backlogs, and shipping product at an unprecedented rate. Unfortunately, this growth has also created significant bottlenecks in the front-end engineering process.

A hot market coupled with incoming data packages of varied quality, outdated CAM systems, and a shallow talent pool have many front-end engineering departments struggling with data package bottlenecks. How can fabricators best resolve these bottlenecks and get onto the open road?

For starters, front-end engineering must be sure the rest of the front-end process is as efficient as possible. To that end, here are suggestions and checklists that can assist fabricators in the review of their own front-end systems.

Data Format: 274D vs. 274X

Bottleneck:

274D Gerber is one of the oldest tools available, and many designers use this format as their sole output option. With 274D data, the front-end operator is left to deal with many issues, some even prior to data import:

- Locate the aperture wheel within the files.
- Is the wheel for one file only or for all files?
- What is the format of the wheel (inch, mm, hard or soft dimensions, trailing zeros omitted, etc.)?
- What is the format of the Gerber files?
- Are all Gerber files in the same format?
- Was the aperture wheel received in electronic data format or paper copy?
- If paper copy, then all apertures must be manually typed in.

Assume the operator must manually specify (type) any size, shape, height, width, length, break widths, inner diameters, outer diameters, and positive and negative output. With 274D data, these options are typically left to the operator's interpretation.

Unfortunately, such interpretations often lead to output irregularities. Features become the choice of the operator rather than the designer. A simple aspect such as the angle of a thermal relief break can cause the thermals to break into one another or even isolate, which may be desired.

If the thermal reliefs are surrounded by voids in the ground (clearances) and the clearances lack sufficient annular rings to ensure connections, the clearances must be enlarged to meet manufacturing capabilities, thus causing the thermal to be isolated from the rest of the ground plane.

This may be the case with internal layer files, but the external files can show the same data, such as the surface mount component's length and width.

The operator must determine whether the aperture wheel calls out X first and Y second or Y first and X second. The wrong choice may cause the surface mount land to short into another feature or become oversized or undersized.

A mistype of a circle instead of a square may deprive the SM component of a land that is sufficient to ensure the component is level during wave soldering.

When typed incorrectly, soldermask clearance sizes can result in mask on lands or enlarged clearances that expose traces.

Open Road:

When asking a fabricator to manually manipulate the aperture wheel, the final part design is left to the front-end operator. When a designer doesn't have the option to export files using 274X, it is always helpful to provide the plotted artwork to the manufacturer for verification.

274X provides a quick and efficient means of importing data with no need for operator interpretation. The data provided are the data tooled.

Drawn Planes and Pads

Bottleneck:

Drawn planes are multiple overlapping traces or vectors used to build a large copper feature. By

| MicroCirtec Micro Circuit Technology GmbH mit Sitz in 47877 Willich |
| Registergericht: Amtsgericht Krefeld | Reg.-Nr.: HRB 2703 | Ust-Id-Nr.: DE 120 151 127 |
| Geschäftsführer: Hildegard Völker |
| Anschrift: 47805 Krefeld, Oberdiessemer Strasse 15 | Telefon: ++49 (0) 21 51 825 1 | Telefax: ++49 (0) 932 450 |
Nach DIN EN ISO 9001 : 2000 Zertifikat-Nummer: 05124-1 aus 2005

drawing a plane rather than producing layers in a negative format, one reduces the file sizes. In many instances the use of drawn images is a necessity.

If the design remains functional after changing the data from positive to negative at output of the Gerber data, the file size becomes smaller. Large data files are due to mass amounts of trace data information.

When working with drawn images, CAM operators may encounter these issues:

- The CAM system slows during the import of Gerber files and even causes software crashes.
- The DRC checks reflect false errors such as line widths and air gaps.
- The DRC checks cannot be run or have to be run layer-by-layer (versus all layers at one time), slowing the software and hence the CAM process.
- The ground or embedded signals must be increased or decreased due to manufacturing capabilities, creating the possibility of shorting a land into the ground or causing a slight void between draws in a plane.
- Draws on grounds with the added equation of step-and-repeat into panel form change a file size from 5MB to 30MB. Moving these heavy data from the workstation to the photoplotter adds another step. Data interpretation is now left to the software rastorization process. This can bog down the plotting station, lead to the misplacement of the draw, crash the rastorization software, and crash the photoplotter.

Conversion issues:

Error in the conversion from draw to flash. This can happen when a land that is chosen for conversion has an extra length of vector to it, and the CAM software converts the others that are similar to it into the same flash land. It can convert lands designed to be a bit larger (0.002" to 0.003") and also convert them to a smaller size.

(a) Allowing the operator and CAM software to convert lands could lead to errors in the design and, later, undesirable output.
(b) Unintentional swaping of X and Y locations.
(c) Offset or shift of original placement due to the software errors with the datum for the feature.

- Conversion of draws is also necessary if the Gerber files used to plot the artwork are used to generate an electrical test netlist and fixturing files. All fixturing software must convert lands in order to produce a dependable netlist, that is, one that will test all points. If draw to flash is not completed when generating these files, the software will not place a test point for that feature, leaving that net untested.

Open Road:

Output plane layers in a negative format.

Thieving and Copper Balancing

Thieving and copper balancing is the process of adding a copper pattern to the outer layers of a board. Balanced copper helps to ensure even plating and etching.

Bottleneck:

PCB designers have started adding copper thieving and balancing to the copper layers. Unfortunately, if the design rules are shared, the CAM operator is challenged with a confusing design rule check. The computer has problems understanding these features and calls out many errors.

Open Road:

The designer should give design rules for thieving to the manufacturer, and the copper features should be added to the CAM systems. An example of thieving design rules: 0.040" squares on 0.070" centers and 0.100" spacing from any copper feature. All added squares are to be covered with soldermask.

Standardized Drawing Formats

Bottleneck:

Blueprints can be produced and exported in many different formats, including HPGL, DWG, PLT, DXF, GBR, BMP, GIF, etc.

To keep up with customer demands, fabricators must purchase and be proficient with many different types of software packages. Not only is all this software costly, but guessing which software to load becomes a cumbersome and time-consuming task.

Having multiple software applications open taxes computer memory, slowing the computer and the blueprint plotter.

Open Road:

During the initial business engagement with the customer, the fabricator should suggest the optimal fabrication drawing format. Early format collaboration provides front-end departments with a clear understanding of what's required to move a job to production.

| MicroCirtec Micro Circuit Technology GmbH mit Sitz in 47877 Willich |
| Registergericht: Amtsgericht Krefeld | Reg.-Nr.: HRB 2703 | Ust-Id-Nr.: DE 120 151 127 |
| Geschäftsführer: Hildegard Völker |
| Anschrift: 47805 Krefeld, Oberdiessemer Strasse 15 | Telefon: ++49 (0) 21 51 825 1 | Telefax: ++49 (0) 932 450 |

Standardized Drawing Notes

Bottleneck:

Incomplete or inconsistent notes on the fabrication drawing.

Open Road:

Fabrication drawings should include the board specification callout (or IPC-A-600); copper weight callout (1/2 oz., 1 oz., 2 oz., etc.); finish requirement (HASL, OSP, immersion gold, etc.); hole size tolerance; specify plated vs. non-plated holes; silkscreen color; board thickness and thickness tolerance; route tolerance; scoring specification; quality requirements such as cross section, ionic readings, certificate of compliance, impedance readings with coupons, solder sample, electrical test certificate, etc.; pack and ship instructions; and an impedance requirement that does not reflect line width or dielectric callout.

Non-functional Lands

Bottleneck:

Non-functional lands for the CAM engineer and the manufacturing process engineer are a menace. Not only must the CAM engineer engage in a secondary DfM to remove them, but this same process may also lead to the removal of lands that are part of the design. For instance, even if the trace is intended to fall just short of the land (0.001"), the software may consider the land non-functional and remove it.

Open Road:

Designers should remove all non-functional lands.

Surface Mount Spacing

Bottleneck:

As designs have become smaller and tighter, the applications to produce these designs have not changed much. Most surface mount parts are spaced at pitches of 0.025", 0.0197", or 0.016". A problem arises when the component width is large enough that the air gap is reduced to 0.007" or less. With a 0.007" or less air gap, soldermask adhesion becomes extremely difficult. Fabricators prefer to have a minimum of 0.003" covering the traces and at least of 0.002" clearances. This is still quite hard to ensure. In most cases, the customer or design requires that soldermask be applied between surface mount lands. If the air gap falls below the minimum, then the manufacturer will request a window void of the soldermask at that location.

(Also take into consideration that there will be some etch compensation enlargement prior to plot. This often times causes the CAM operator to shave the sides of the surface mount lands to ensure that there will be no lifting and redepositing of soldermask flakes onto other features of the part.)

Open Road:

The standard rule should permit the windowing and voiding of soldermask in the area. Otherwise, construct the land width small enough to permit a 0.007" air gap.

Soldermask Clearance

Bottleneck:

Many Gerber files for soldermask come with the clearance for the features as one-to-one. If the manufacturer left this artwork as is, the result will be end product with soldermask on lands. This is not acceptable in most cases, though IPC standards permit encroachment on one side of the feature (for surface mount). By providing the soldermask data this way, it requires the CAM operator to increase all features by the requirement of the manufacturer. Some potential errors:

- The manufacturer's requirement may be at a different specification than the designer's.
- Exposed trace features.
- Slivers of soldermask may be left behind after increasing and shaving and redeposit on other areas of the part.
- Increase of clearances designed to have soldermask on that feature.

Often, Gerber files for soldermask are so large that CAM software triggers an error in the design. This causes the artwork to be questioned and edited, slowing the process.

Open Road:

If all design software automatically increased the soldermask by 0.006", these questions are eliminated.

Silkscreen Size

Bottleneck:

Many silkscreen files are generated using an 0.008" to 0.010" draw. In most cases, this is acceptable if the height is proportional. On small-dimension parts, however, an 0.008" to 0.010" draw will blur during the application process. This will cause a reject from the customer or the quality department or both. The Gerber file aperture size then must be reduced and left to the specification of the fabricator.

Open Road:

| MicroCirtec Micro Circuit Technology GmbH mit Sitz in 47877 Willich |
| Registergericht: Amtsgericht Krefeld | Reg.-Nr.: HRB 2703 | Ust-Id-Nr.: DE 120 151 127 |
| Geschäftsführer: Hildegard Völker |
| Anschrift: 47805 Krefeld, Oberdiessemer Strasse 15 | Telefon: ++49 (0) 21 51 825 1 | Telefax: ++49 (0) 932 450 |

Fabricator and customer / designer collaboration resulting in a 0.006" to 0.007" draw for silk–screen applications will increase the efficiency of the process.

Front-end process

Efficiencies are the on-ramp to the open road. Always give the customer a single contact with a backup contact so that no information is misinterpreted. To know your customers and their requirements, it is critical that manufacturers have the following items:

- Contact name, phone number, and e-mail address. (For quick-turn parts, a 24-hr. phone number is imperative.)
- A read-me file that states the Gerber listing, format, identification, and blueprint.
- A customer PCB fabrication specification if noted on the blueprint. If the PCB fabrication specification has a callout that cannot be met by the manufacturer, a deviation request must be initiated immediately.
- A "frequently asked questions" checklist from the front-end department can help the front-end process run more efficiently. An FAQ checklist can include many different types of questions:
- May we remove non-functional lands?
- May we add copper thieving?
- Are there logo placement restrictions?
- May we "teardrop" lands at trace intersections?
- May we rim void if a land appears where a non-plated through-hole is placed?
- May we window/void the SM flatpack if spacing does not permit for adhesion of the soldermask?
- May we use single-ply construction?
- Are there dielectric requirements?
- If hole diameter tolerances are not called out, may we use plus/minus 0.003"?
- If part size tolerances are not called out, may we use plus/minus 10 percent?
- If line width and air gap tolerances are not called out, may we use plus/minus 10 percent?

After these initial decisions are made, it is hoped that they may be applied toward future designs.

Know Your Capabilities

Every manufacturing facility should have a capability listing for the front-end planners and CAM operators. Planners and CAM operators both need to understand the processes that are performed by the fabricator.

Generally, the most knowledgeable front-end employees are those with experience in the production environment and who taught front-end engineering. Through cross-training of this nature, front end becomes an efficient extension of the manufacturing process.

Knowledge of the following items is critical:

- Etch compensation (based on the copper weight requirement).
- Plating balance of parts on the panel.
- Hole size based on plating requirement.
- Drill hit optimization.
- Spacing for the shearing of parts that require gold tip plating.
- Thieving on part, if necessary.
- Panel border minimums.
- Piece size minimums if tested via flying probe tester (test in piece or in panel form).
- Where the cross-section process should take place or whether it should take place in multiple processes.
- Organization of any special tools (drill or router bits, for example) for "out of the ordinary" processes.
- Material availability, either in-house or on order (and the latter's delivery schedule).
- UL capability.

Tip:

Have a pre-order check, or educate the sales team about facility capabilities.

The planner must know the manufacturing facility and its capabilities. By giving the process engineering department an opportunity to evaluate the process traveler and artwork, the planner will not only have a better understanding of the process, but the process engineering department will have the opportunity to make changes prior to the release of the product to production.

Items to consider before the process traveler is generated:

- Material availability.
- Shrinkage compensation.
- Drill registration tolerance.
- Drill size availability.
- Etching compensations.
- Plating compensations.
- Routing compensations.
- Testing capabilities.
- Time studies (provided by process engineering).
- Daily capacity of the plant (provided by production control).

| MicroCirtec Micro Circuit Technology GmbH mit Sitz in 47877 Willich |
| Registergericht: Amtsgericht Krefeld | Reg.-Nr.: HRB 2703 | Ust-Id-Nr.: DE 120 151 127 |
| Geschäftsführer: Hildegard Völker |
| Anschrift: 47805 Krefeld, Oberdiessemer Strasse 15 | Telefon: ++49 (0) 21 51 825 1 | Telefax: ++49 (0) 932 450 |

Laminate Material Performance

Many manufacturing facilities have multiple laminate vendors, which in turn have materials with different characteristics. For example, the internal layer stretch compensation for one vendor's laminates may differ greatly from that of another's.

Most vendors provide this information at the outset as a guideline; however, process studies for each material are necessary based on the specific lamination press used. If this is not performed, problems due to material movement and shrinkage will cause nightmares for the drilling department. (Likewise, prepreg differs in thickness, flow, cure, and stability.)

These characteristics can be used as base guidelines for fabricators to consider when planning the construction of the part.

- Balance the construction (dielectric) to eliminate warpage as a factor.
- Price the core material and prepreg to ensure material cost does not exceed the panel pricing factor.
- If flex material is used, determine which surface finish is optimal (e.g., HASL or white tin).
- If the part has many fine-pitch SM components, white tin is preferred, as it eliminates solder shorts from the equation.)
- Carbon ink ohm requirements, application, and spacing width of the key lands at plot.
- Double process of dry film or image using the selective gold process.
- Laminar thickness for heavy copper requirement.
- Choose gloss- or matte-finished soldermask.

By streamlining the front-end process, fabricators can eliminate data package bottlenecks and other related front-end issues. Doing so improves quality, decreases cycle time, and increases profitability.

| MicroCirtec Micro Circuit Technology GmbH mit Sitz in 47877 Willich |
| Registergericht: Amtsgericht Krefeld | Reg.-Nr.: HRB 2703 | Ust-Id-Nr.: DE 120 151 127 |
| Geschäftsführer: Hildegard Völker |
| Anschrift: 47805 Krefeld, Oberdiessemer Strasse 15 | Telefon: ++49 (0) 21 51 825 1 | Telefax: ++49 (0) 932 450 |