FABRICATION DATA & DOCUMENTATION

The Complete PCB Data Package

SEGMENTS OF SANITY FROM THE GUIDE

EMA Design Automation®

Fabrication Data & Documentation: The Complete PCB Data Package

CAD/CAM data has taken over the world of electronics. Since design and manufacturing is driven by data, much of what is accomplished on the PCB fabrication PCBA manufacturing floors has been standardized. However, the electronic manufacturing service provider's challenge has not only been to create the machinery and processing; but also to keep up with installing the new generation processors which are running at incredible speeds and require special considerations. Along with the 'traditional' CAD/CAM data, there is still a need for textual information through notes and special detailing to provide a complete engineering design package and help ensure a smooth manufacturing process.

You have spent a lot of time designing your board and getting it 'just right'. It is critical you provide your manufacturing partner the information they need to meet your design intent. Without guidance from you, the designer, the fabricator will either be forced to make assumptions or delay manufacture.

Let us look at what composes a complete PCB data package and examine the notation and graphic documentation which must accompany it.

Fabrication Data
File Naming Convention
Gerber Data Files
Design Continuity Verification File
Drill File
Neutral Database File

File Naming Convention

Avoid using non-descript or vague names like "TOP.GBR" for files. Without specificity a data file name will surely be confused as to its purpose. As shown in the examples below, incorporating a part number and revision into the file name links the file to a document control system. Adding a clear functional description to the filename will surely help anyone using he data to realize what is its purpose without having to view it. Part Number, revision, and function. Examples:

- 424242_B_LAYER_1.PHO
- 424242_B_DRILL.DRL
- 424242_B_FABDWG.DXF

Gerber Data Files for All Required PCB Artwork Layers

This is the starting point for graphic CAM tooling. Referred to as "artwork" the supplier modifies this data to conform to their manufacturing capabilities to meet finished feature requirements specified in the fabrication drawing. After the layout is complete, the software can output these files, each of which will become the basis for the etched copper pattern for each PCB layer.

When outputting Gerber file data, use the very common RS-274X output. This selection uses special technology referred to as *embedded apertures* to create the artwork shapes.

Industry Standard Design Continuity Verification Output

IPC-2581 or ODB++ design output and an IPC-D-356 format netlist will help the PCB supplier check to make certain the artwork matches the design intent electrically before and after fabrication. The more intelligent and comprehensive the data, the better.

Industry Standard Drill File
A common output format is Excellon. Drill files define the nominal X, Y position of the holes, but keep in mind, the numeric values output from CAD software define the finished hole size, not the drill size. Drills must be larger than plated holes specified to compensate for plating thickness; the supplier will select the appropriate drill size.

Neutral Database File
Providing the source file for the complete design database allows a PCB supplier to access the source data needed to determine design intent. Unlike "dumb" graphic Gerber data, source data provides intelligent data, net names, and connectivity information which can be helpful in determining artwork anomalies and moving the design through the CAM process quickly. Intelligent source database formats are best output in IPC-2581 or ODB++, but some systems accept ASCII output of the original layout database.

PCB Documentation

There are other aspects of the design which the CAD data cannot address alone. For instance, CAD data defines the shape of an exposed component land, but not the metalized finish. It is critical to understand graphic CAD data defines mostly geometric, nominal values for the features which are shown. There are no perfect manufacturing operations– every bit of CAD data will be subject to manufacturing tolerances.

How will the effect of manufacturing variance and so many other aspects of the design be addressed? Data and machinery run the PCB process while graphic PCBA documentation is required to start the process. Graphic PCB documentation also helps to close out and finalize the PCB process by giving the inspection stakeholders a graphic example of what the finished PCB will look like and provides notes and specification for its final electro-mechanical configuration.

The Fabrication Drawing

There are several basic documentation elements which will need to be included in the fabrication drawing:

The Fabrication Drawing
Pictorial
Drill Symbols
Drill Chart
Holes Drilled and Routed
X0Y0 Datum Feature
Dimensions and Tolerance
PCB Stack-up Detail
V-Score Detail
Tab-Route Detail
Intentional Shorts Table
Board Outline and Thickness
Impedance Specification
Fabrication Notes

Pictorial
CAD data must be accompanied by textual fabrication specifications which are best documented in an engineering drawing. he PCB fabrication drawing serves less to indicate how a PCB is to be manufactured and more to specify how the PCB shall be laid up. CAD data can define nominal values for features such as drill geometry and the PCB outline.

Drill Symbols
Drill symbols are shapes or textual characters generated by the PCB layout software to graphically show a hole's position on the PCB. When selecting symbols for drill sizing, always use a unique symbol for each type of hole containing the same attributes. For instance, there may be several .023 [0.58] diameter holes required on a PCB. If the holes are

Fabrication Data & Documentation: The Complete PCB Data Package

alike with regards to plating requirements and hole diameter tolerance, they may be assigned the same symbol. However, if there is a different attribute, such as a non-plated condition or a different tolerance requirement, the hole of the same size will require a different symbol which will need to be reflected on the drawing accordingly.

X0Y0 Datum Feature

The X0Y0 Datum is a hole on a PCB which is preferably located in the lower-left corner of the design.

Dimensions and Tolerances

Once designation of the origin point (X0Y0) for the design database are relative to practical interface points on the PCB. Dimensions and tolerances shall originate from X0Y0 on the PCB.

Fabrication drawing dimensions do little to communicate to the supplier how to fabricate the PCB – the PCB design tool output (CAM) data does this. Fabrication drawing dimensions are used by manufacturing and inspection personnel to gauge the accuracy of the process and machinery. Accuracy, expressed by the definition of tolerance values, must be documented on the fabrication drawing for the supplier to measure their success and adjust the process as required. CAM data does not include the establishment of tolerance zones and therefore, has no bearing on whether a snap-fit feature will work at next assembly.

PCB Stack-Up Detail

A stack-up detail provides imperative, up-front, cost-related information to your purchasing department and the supplier's quoting department before the fabrication process ever begins. This detail documents a generic view of the vertical design intent– the layer count, the copper layer thickness, the dielectric materials in between, and the presumed thickness and dielectric properties of the solder mask material covering the outer-layer surface copper. For controlled impedance, adopting a stack-up philosophy which allows the board supplier to vary the dielectric constant of materials, trace widths, and spacing to achieve the specified impedance is recommended. This will allow usage of materials with wider availability, and is critical to improving the supplier's ability to deliver the highest performance at the best cost.

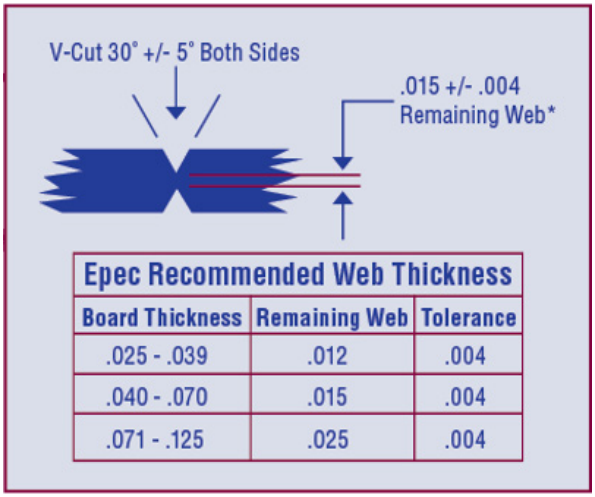
Layer Name	Type	Material	Thickness (mm)	Dielectric Material	Dielectric Constant
Top Silk	Overlay				
Top Solder Resist	Solder Mask	Surface Material	0.01016	Solder Resist	3.5
1	Signal	Copper	0		
Dielectric1	Dielectric	None	0.32004	FR-4	4.8
AGND	Signal	Copper	0		
Dielectric2	Dielectric	None	0.32004	FR-4	4.8
3	Signal	Copper	0		
Dielectric3	Dielectric	None	0.32004	FR-4	4.8
4	Signal	Copper	0		
Bottom Solder Mask	Solder Mask	Surface Material	0.01016	Solder Resist	3.5
Bottom Silk	Overlay				

Panel Processing

When the design is set up for panel processing, the PCB image is multiplied several times to process many PCB's on the same manufacturing panel to maximize yield of the panel. If the PCB you are designing will be fabricated and assembled outside of your facility, you will not have to utilize a V-Score or a tab-route detail. These will be added by the suppliers.

V-Score Detail

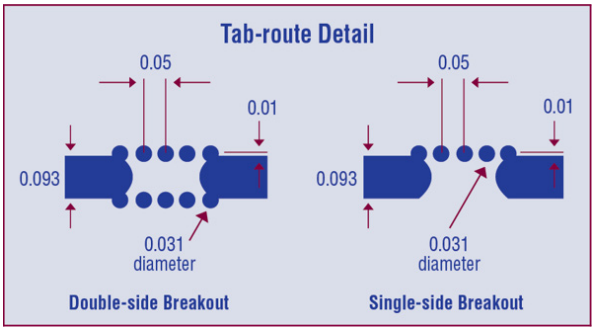
Typically, it is uncommon to provide a V-score detail on a fabrication drawing. A fabrication drawing should describe the finished part and therefore should not show extra processing material unless required. If required, it is because the project stakeholders are intending that the PCB will be processed in-house. These assembly stakeholders will need manufacturing assembly arrays with V-Score cuts between the strait edges of the PCB to provide rigidity during the assembly operations. It's best to always check-in with both the assembly house and the fabricator for optimal v-score parameters.



Epec Recommended Web Thickness		
Board Thickness	Remaining Web	Tolerance
.025 - .039	.012	.004
.040 - .070	.015	.004
.071 - .125	.025	.004

Tab-Route Detail

Like the V-Score detail, the tab-route detail is only provided if there is in-house assembly processing. Tab-routing is used to support the PCB in the assembly array with short break-away tabs when straight edges are not available on the PCB or a smooth (routed) machine finish is required. The best source for tab-route detail info is the assembly supplier, because they control the end purpose process of excising (de-paneling) the individually assembled PCBA's.



Intentional Shorts Table

Sometimes it is necessary to connect multiple signals together in a design at a specific point. When a net with a significant name needs to connect with another net having a different name, a designer will often use a 'bond' part to connect the nets. This bond is a copper shape that makes the connection and can trick the design rules checker. However, when the connected copper shapes are compared with the IPC-D-356 netlist, a red flag is raised at the PCB supplier's CAM department. The very common practice of net joining will almost always cause a stop order to a job, unless it is described on the drawing in detail. The supplier does not know if the 'short' between multiple nets is intentional or not. An Intentional Shorts Table can be used to effectively communicate the important information of bond reference designator, net-names, XY location, and layers to the supplier's CAM department up front; reducing the need for stop orders.

Intentional Shorts at Following Locations:				
Ref Des	X Loc	Y Loc	Layer	Signals
BND1	4.400	0.200	1	3V3OUT/+3.3V
BND2	9.175	0.425	1	+14.3V, 14V3OUT
BND3	6.490	0.025	1	2V5OUT/+2.5V
BND4	9.250	1.500	1	VLEDOUT/VLED
BND5	7.125	0.050	1	1V4OUT/+1.4V

Board Outline and Thickness

Overall PCB length and width may be shown as reference on the fabrication drawing if the features need not be inspected. A note for minimum radius for internal corners and overall profile tolerance may be specified in the fabrication notes.

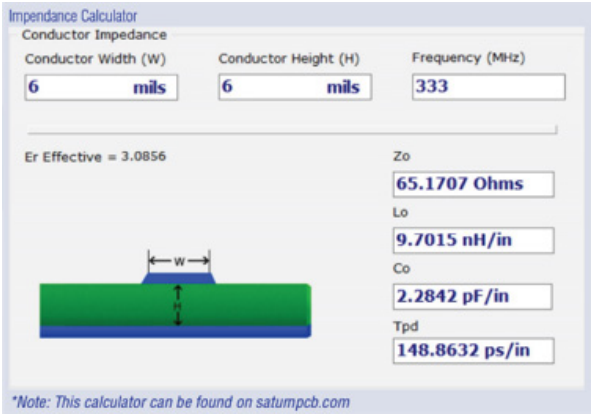
Impedance Specification

Impedance-controlled designs require control of four physical variables in the stack-up:

- Trace thickness
- Trace width
- Trace distance from reference plane(s)
- Dielectric value of the material

Fabrication Data & Documentation: The Complete PCB Data Package

A designer may use a basic field solver or impedance calculator to determine trace widths and spacing to be used for impedance requirements. At the start of the layout, the designer should do a feasibility check to select the optimal trace widths for impedance-controlled lines; be sure to validate the values in the stack-up with the PCB supplier. Use care when adding information to the stack-up detail regarding three of the four variables mentioned above. Finished copper thickness is an important consideration regarding the various conductors' current-carrying capacity. However, the values of trace width, distance from reference plane, dielectric constant of the material (Er), and material sources should be allowed to be adjusted by the supplier for best DFM.



Conclusion

CAD/CAM data has replaced the need for human contact with the features comprising the finished PCB during manufacturing cycles. Manufacturing stakeholders rely on data automation and machinery to create some fine lines and complicated mechanical PCB features. A common assumption is the manufacturing processes and machinery of today create hardware that perfectly matches the nominal layout. However, it is a sad fact in the PCB design industry that PCB designs continue to be revised due to lack of understanding in design specification, namely textual and graphic documentation.

The data which drives the high-tech machinery is strictly nominal. All machinery and processes have a tolerance which will yield parts that will rarely, if ever be exact. In design, the awareness of how much manufacturing variance can be leveraged by adding the variance factor into the layout to prevent assembly conditions which are too tight or too loose.

Data has other shortcomings: CAD data will not automatically color the solder mask to be blue or the legend ink to be yellow. Embedded CAD data will not automatically select RoHS materials or automatically pull high temperature grade material into a stack-up during the lamination phase. Completing the fabrication cycle still relies heavily on textual and graphically detailed information provided on a comprehensive fabrication drawing.

A sufficiently embellished fab drawing including the information provided herein will fill in the blanks embedded data leaves out. And while CAD/CAM data tells the supplier's machinery what to do, graphic and textual information provided on the fab drawing guide the inspection department in checking how well it did.

For more, download The Hitchhikers Guide to PCB Design.