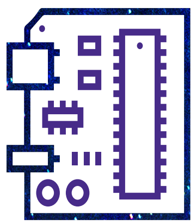
ASSEMBLY DATA & DOCUMENTATION

Process Overview: Part 1



SEGMENTS OF SANITY FROM THE GUIDE

EMA Design Automation®

Assembly Data & Documentation: Process Overview, Part 1

Data drives the assembly process—a complete package of data must be supplied to the assembly stakeholder. As in any other process without complete and accurate data, the process will stall or become confused. This results in time-consuming queries between the supplier and the customer. There are several items of a complete assembly data package that should be included:

Assembly Data Package
Bill of Materials File
Neutral Database File
X,Y Placement File
Solder Paste Stencil Artwork File
Test Point Location File

The Bill of Materials File (BOM)

The assembly process starts with the evaluation of all materials published in a BOM report which is exported as data output from the schematic end of the design and layout process. A BOM which lists all PCBA components, their manufacturing part numbers, descriptions, and corresponding reference designators is a powerful tool for use by a supplier management purchasing agent. The sooner the component information can be captured and sent to them, the better. Challenges will arise and can include item costliness, long lead time, discontinuation, and obsolescence. The ability to flag potential procurement problems requires visibility.

The exported BOM must list all the electronic and mechanical parts used in the assembly, including the bare PCB, all components, and even mechanical hardware, adhesives, coatings, and wire. The EMS planning department will start with this list to check for two very important attributes: availability and cost. If the EE has checked for these two important component virtues during the front-end design of the PCB layout, everything should go quite well and even better if the EMS supplier negotiates volume discounts for the components.

The Neutral Database File

Provision of a neutral CAD database is just as important to the assembly provider as to the PCB supplier. A neutral database such as ODB++ or IPC-2581 is leveraged differently by the assembly supplier. AMEs base their BOM checks on comparisons with intelligent manufacturing data provided in neutral formats such as IPC-2581 or the widely-used ODB++ format. Once all the design for assembly (DFA) checks are complete and the PCB assembly process begins, all the parts are kitted and moved forward to be assembled.

Source data provides intelligent data, net names, etc. These can be helpful in sorting out component placement, orientation anomalies, and moving the design through the placement CAM setup process quickly. Intelligent source database formats are best output in IPC-2581 or ODB++ because they provide the most usable information to the board fabricator and assembly processor.

The X,Y Placement File

The X,Y placement file, commonly referred to as a "pick and place" file, is the output data which will be used by the assembly stakeholder to program the placement machinery. The file output can be very simple ASCII text data, but must include enough information for the operation to run smoothly. As with all things, PCB and PCBA, check in with the assembly stakeholder regarding their preference. For each part to be placed, the X,Y placement file needs to specify a reference designator, side of placement, X,Y location and angle of orientation.

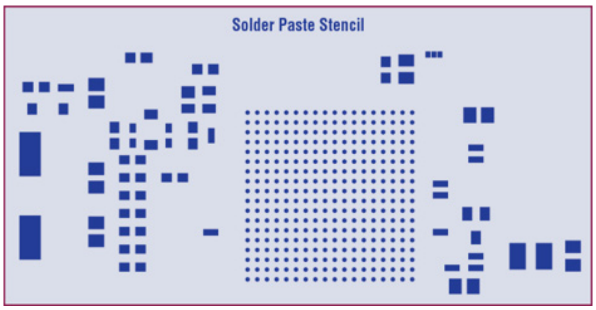
X,Y Placement File

File Edit Format View Help

```
Project file name: XXXXX_PLAMENT.ASC
Date:               12:00:00 AM Monday, December 00, 2018
Devname      Refds        Side   Loc                  Centroid             Angle
===========  ===========  ====== ==================== ==================== =====
NODEVNAME    J3           top    10.9250:7.0000       10.9250:6.9000         0
NODEVNAME    U4           top    10.7506:6.5861       10.6600:6.4700         0
NODEVNAME    U3           top    10.7506:5.9451       10.6600:5.8290         0
NODEVNAME    R15          top    11.2300:6.8950       11.2305:6.7950         0
NODEVNAME    VR1          top    11.5830:6.9115       11.5830:6.7710         0
NODEVNAME    C1           top    11.7720:6.6961       11.7720:6.6620         0
NODEVNAME    R14          top    10.5500:6.1210       10.6500:6.1205         0
NODEVNAME    LED1         top    11.2590:6.1500       11.2200:6.1500         0
NODEVNAME    LED2         top    12.0440:6.1500       12.0050:6.1500         0
NODEVNAME    LED3         top    12.4340:6.1500       12.3950:6.1500         0
NODEVNAME    LED4         top    13.2190:6.1500       13.1800:6.1500         0
NODEVNAME    LED5         top    13.6090:6.1500       13.5700:6.1500         0
NODEVNAME    LED6         top    14.3940:6.1500       14.3550:6.1500         0
```

Solder Paste Stencil Artwork File

Before any SMT parts can be placed, the PCB must run through a process which will deposit a thin layer of solder paste onto the component footprint lands which have been etched and plated onto the outer layer(s) of the PCB. A solder paste stencil artwork file is usually output in Gerber format from the source design layout and includes all the required openings to allow solder to be screened onto the SMT lands.



Test Point Location File

If testability will be implemented on the PCBA, a file is needed to define all the test point locations for the test fixture supplier to build the equipment. Like the placement file, the test point output file can be very simple ASCII text data. The file is output from the source design layout and derives its data from test point lands which have been specifically identified for test. The file needs to include net names, side of access X,Y locations, and test point names if applicable.

PCBA Documentation

PCB Documentation is critical to manufacturing success. Though data and machinery run the PCBA manufacturing assembly process, complete, graphic PCBA documentation begins the process. Tangible documentation reflecting what the data will produce is critical for quotation and setup reference. A complete PCBA document gives manufacturing stakeholders the ability to view the finished PCBA and start the assembly process with the end in sight. Additionally, the same helps to finalize the PCBA manufacturing process by giving the inspection stakeholders views and notation with which to evaluate the finished product.

There are three essential elements of PCBA documentation: the schematic, bill of materials (BOM) and Assembly Drawing.

Schematic

Documentation for the schematic is most likely the simplest act of PCB documentation. Since the schematic served as the source design file driving the layout of the PCB, the schematic becomes a “document” by simply placing a proper format around it. How this is provided, whether through a source file, paper copy, or PDF is usually dependent on company culture. There are typically no extra details added to the schematic at the time of formal documentation.

Bill of Materials (BOM)

The BOM documentation is an official, formatted document which may be embedded inside a complex, corporate data management system. Documentation for the BOM is always output from the source: the schematic document. The part line items are pulled into a paper or electronic environment which allows for signature approval and formal release for distribution to purchasing, manufacturing assembly, and inspection stakeholders.

Assembly Data & Documentation: Process Overview, Part 1

Assembly Drawing

Like the fabrication drawing, the PCB assembly drawing serves less to indicate how a PCBA is to be manufactured and more to specify how the assembly shall perform. The assembly drawing will show all the parts to be assembled onto its two outer surfaces. It is used as a guide for placement programmers and inspection personnel to measure how well the parts were placed on the board, how well the solder joints were formed and how clean they must be at the end of the process. How will our final inspection stakeholder know what is intended and what is expected without a clear, pictorially-rich document to specify the design intent?

To serve as a viable manufacturing reference, the complete assembly drawing needs to include at least three key items for the assembly manufacturing stakeholders:

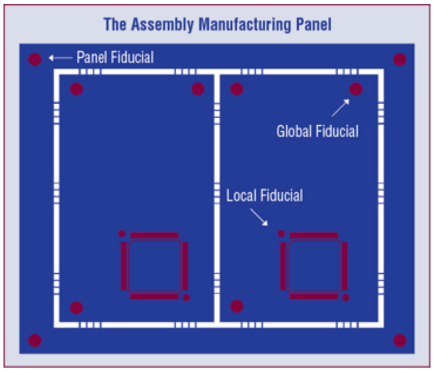
- Assembly Pictorial
- Assembly Details
- Assembly Notes

Fabrication vs. Assembly Documents

Compared to specifying the thousands of holes, trace layers, and processes involved with bare PCB documentation, the documentation for assembly appears simple. The people finishing and inspecting the PCBA will appreciate the time you spent on the front-end. If you have successfully completed and ensured the accuracy of your schematic and bill of materials, the efforts will have served to “pay it forward” as they are fed into the creation of a clearly illustrated, detail-rich, and simply-notated assembly documentation to serve the needs of the assembly, test, and field service stakeholders.

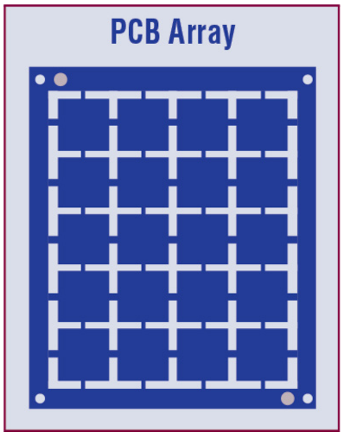
Manufacturing Panel

Be aware that the PCB on screen at the time of layout will need to be multiplied into an array form (or panel) for volume manufacturing. To reduce manufacturing time, the manufacturing engineer will work with the PCB supplier to order the PCBs in an array form. A manufacturing panel array will make it possible for many PCBs to be processed together on a single manufacturing panel which will carry all the PCBs down the manufacturing line together. An array of PCBs organized onto a panel in a 3-inch x 5-inch configuration could yield 15 completed PCBAs after a single pass through the assembly line, effectively reducing the manufacturing run time for the PCBA to 1/15 when compared to a single.



Besides a good overview of the PCB assembly process, what does a PCB designer need to do to create the best manufacturing panel array for the design? Surprisingly, very little. In fact, unless the design is going to be processed by the PCB designer’s own company, don’t do it. Assembly paneling is best determined by the assembly stakeholder, based upon their manufacturing requirements. If the PCB designer does not know where the PCB is destined to be built and knows nothing about the provider’s equipment or capability, it is best to leave the design as a one-up and let the assembly provider do the rest to fit their process.

The singulation method (sometimes referred to as excising or de-paneling) of the finished PCBA must be determined based upon the board outline shape. The process of singulation is accomplished using one of three common processes—V-scoring, tab-routing, and punching—each of which requires uniquely different panel design specifications.

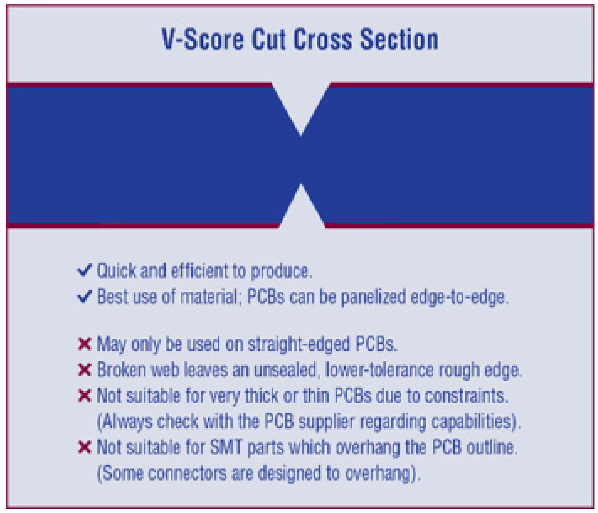


You may be wondering, if PCB designers have very little to do with the creation of the manufacturing panel array, why do I need to worry about it? Well, the fact is regardless of the excising method used, the PCB assembly stakeholder will need space inside the board edge to perform the operation. Therefore, it is important for the PCB designer to understand the processes and requirements needed to create a manufacturable design. For example, often overlooked is the IPC recommendation to provide approximately .020 [0.51] clearance between the nominal PCB outline and all copper on the PCB to allow for successful assembly processing.

IPC-2221 & IPC-2222 provides some excellent guidelines for helping your EMS provider create manufacturable, cost effective panels from your design. To help you have a basic understanding of the design process, below is an overview of common excising processes and what information need to be considered when designing.

V-Scoring

V-Scoring is accomplished by the machining of shallow, linear cuts on both sides of PCB panel, each defining the edge of the rectangular PCB. The depth of the cuts penetrates approximately 1/3 into the PCB from each side, leaving approximately 1/3 of the PCB material left to constrain the PCB in the panel during the assembly operations. After assembly, the panel is moved to a “pizza cutter” type device which utilizes a sharp, round blade to cut the remaining web material in the v-score and break the PCB free.

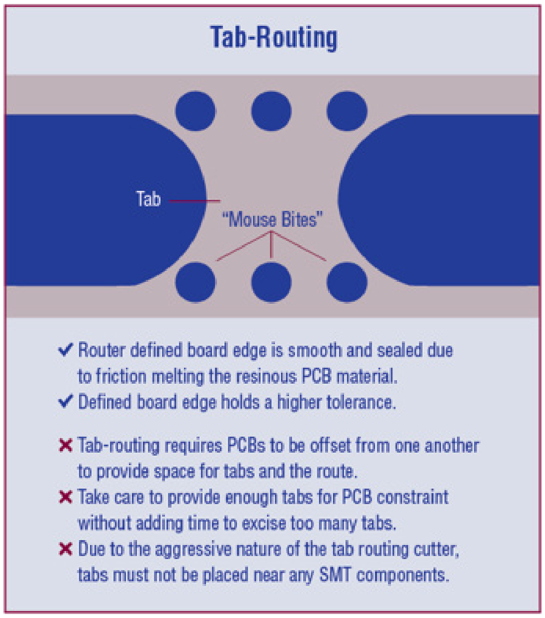


Tab-Routing

For PCB designs which cannot tolerate v-score, tab-routing is an excellent option for the assembly stakeholder. Tab-routing requires a certain amount of space around the PCB outline to allow for a routing operation to remove all the material between the board edge and the panel.

Without a web, the individual PCBs would simply drop out of the panel if it were not for a few keenly placed “tabs”. The panel designer strategically places them around the PCB perimeter to “tack” and constrain the PCB within the panel. The tabs are formed by the high-speed routing tool exiting the route path for a few millimeters and then plunging back in to continue cutting the outline.

Like the v-score webs, the tab-routed tabs remain in place throughout the assembly process. In the case of tab routing, the PCBAs are singulated using a pneumatic hook cutting-device, especially designed for removing tabs. The hook cutter enters from the bottom side of the PCB route and pulls downward on the tab, crushing the material away from the edge.



Punching

On very high volume PCBA runs, a punching method can be utilized to shear the PCB assemblies away from the panel. This process typically requires the use of non-glass and non-fibrous material such as the CEM family of laminates. There are very exacting design criteria which must be considered to provide allowances for the punch/die equipment to perform properly. It is highly recommended to contact the PCB supplier if there is any possibility for the punch excising method to be used.

Conclusion

If there is anything to learn from the disciplines of the manufacturing and assembly floors it is that a process of checks and verification must be in place at every step. While the designer does not have much involvement in the creation of the PCB array, it is important to understand what is involved so you can make sound design decisions. This way when the design is ready for production, you will avoid any manufacturing hiccups. Part two of this article will dive deeper into each step of the assembly process.

For more, download The Hitchhikers Guide to PCB Design.