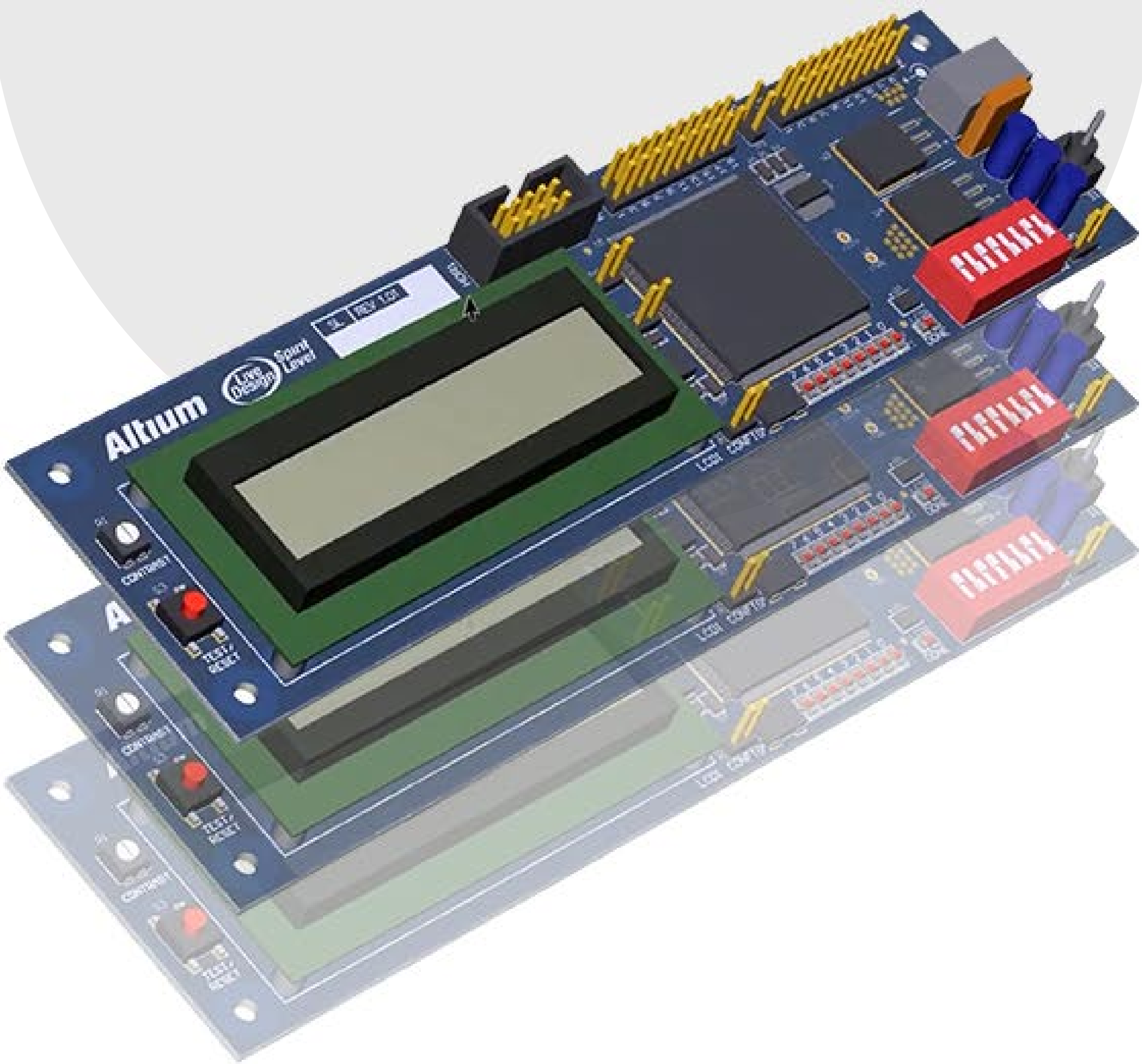


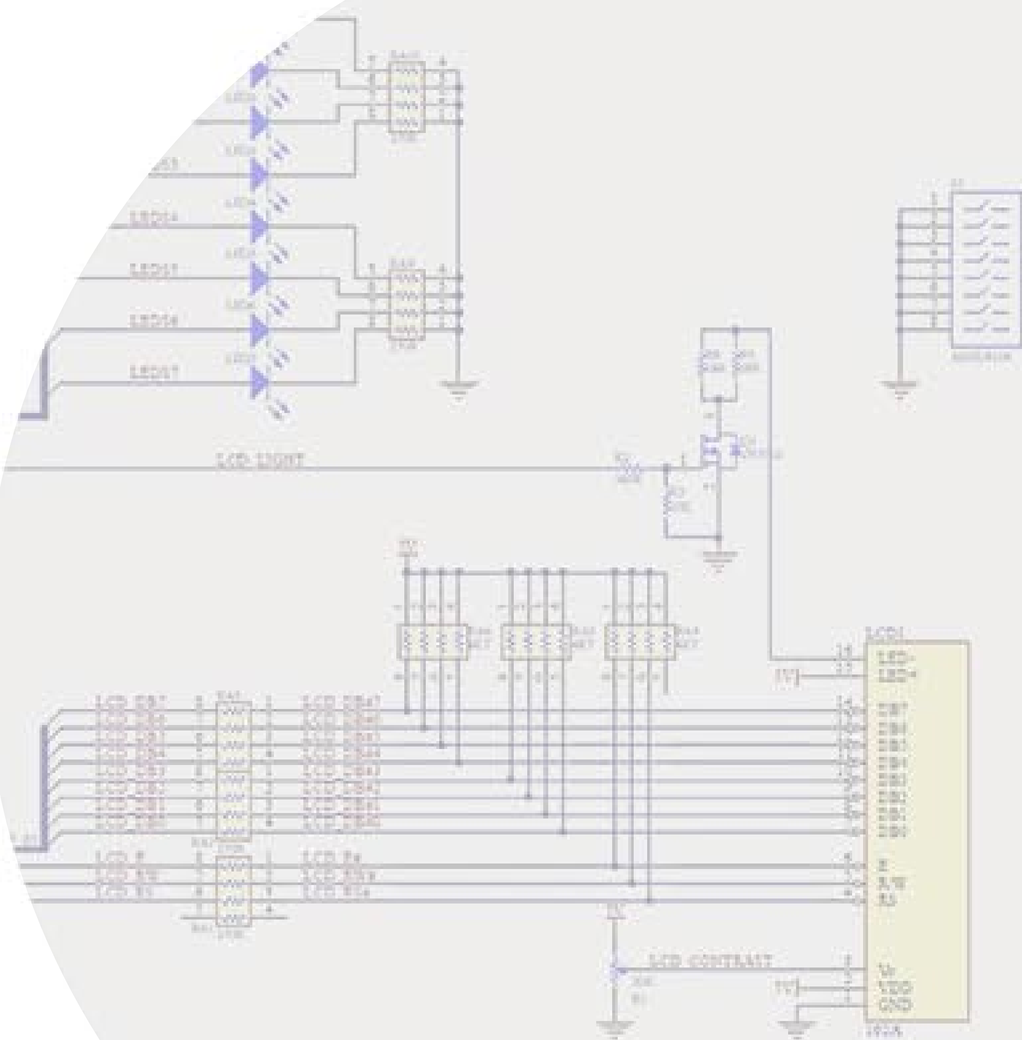
Navigating PCB Manufacturing Part 1

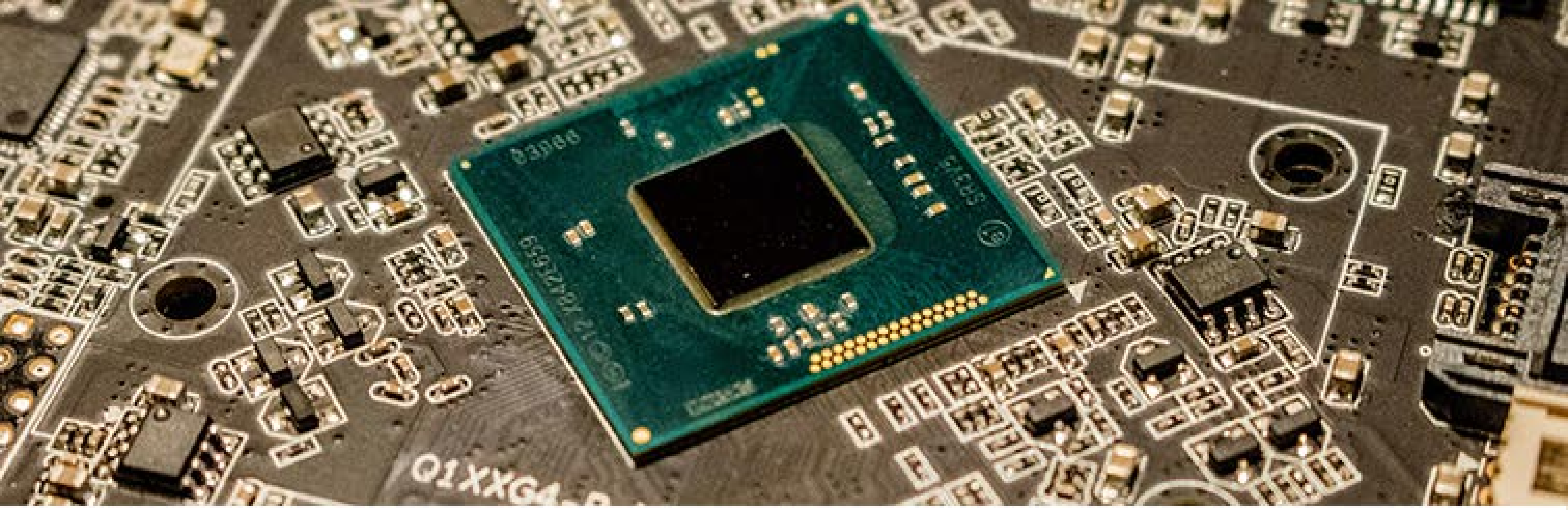
A deeper insight into the PCB manufacturing process.

Altium

Table of Contents

Part 1





Front-End Engineering a PCB Product

In the broadest application of the term, “front-end engineering” refers to an engineering design approach used to control project costs and thoroughly plan for a project. The PCB product development process refers to all the steps taken prior to the board being moved from the design to the fabrication process. This article will review those steps, define what happens in them and what the critical elements are for each of them. Subsequent articles will address the actual operations performed during the fabrication process.

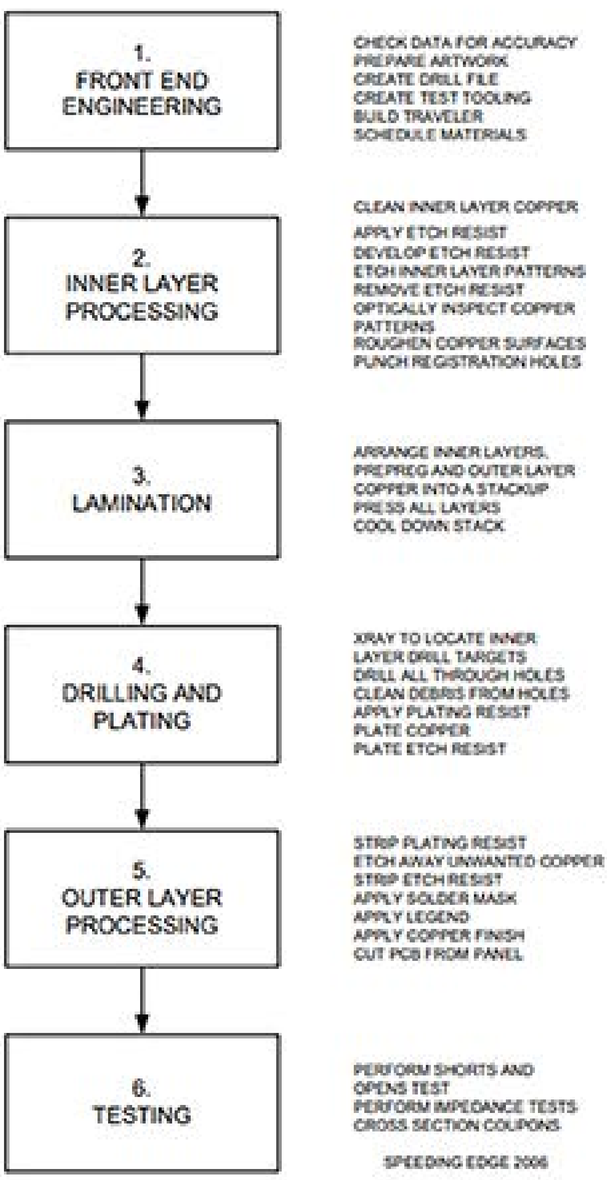


Figure 1. Multilayer PCB Fabrication Process Flow

What’s In the Files

As shown in Figure 1, front-end engineering is the first step in fabricating a PCB. It should be noted that the process shown in this figure is standard across the PCB fabrication industry for the manufacturing of multilayer PCBs.

Here, all of the PCB design data created during the layout process is provided to the fabricator so that the manufacturing process can begin. Computer-aided manufacturing (CAM) stations process the design data through a series of steps that result in the tooling needed during the PCB fab process. Figure 2 shows a typical front-end engineering station.

The information provided for this process includes:

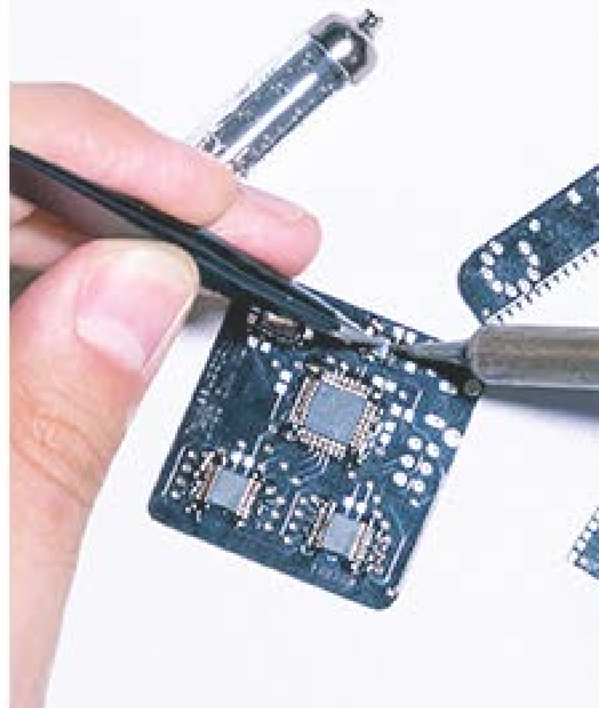
- The “Gerber” data or images of the layers of the PCB.
- The netlist that shows the connectivity of the PCB.
- The stackup information.
- The drill information.
- The fabrication specification.
- The materials specification.

Figure 2. Typical Front End Engineering Workstation

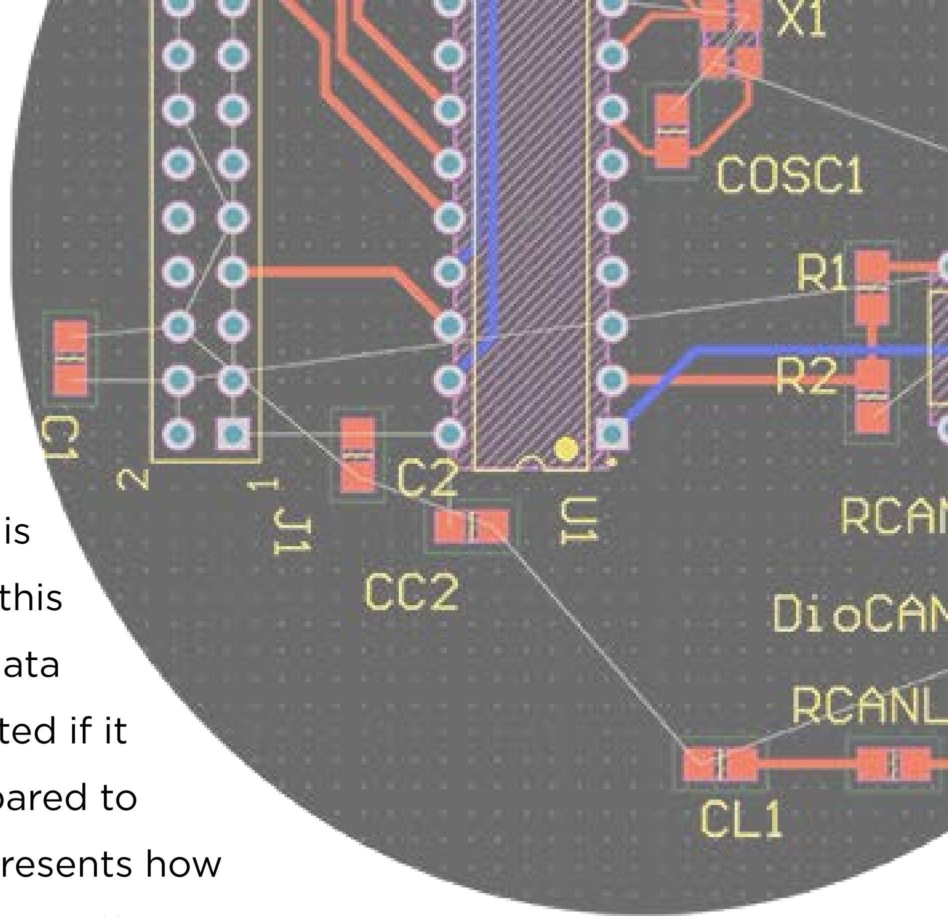
As can be seen, by the above, ensuring all the information delivered to the fabricator is complete and accurate will have a direct bearing on the outcome of the fabrication process. Table 1 is the list of the typical data files required by a fabricator. There are several formats used for this data, including GenCam, Gerber, and OCB++.

xxxx-yyy.aXX	Artwork layer 1 Thru XX (xxxx-yyyaXX is the part number of the design, where XX is the layer number)
xxxx-yyy.smt	Soldermask TOP side
xxxx-yyy.smb	Soldermask BOTTOM Side
xxxx-yyy.sst	Silkscreen TOP Side
xxxx-yyy.ssb	Silkscreen BOTTOM Side
list.apt	Standard Aperture List
xxxx-yyy.IPC	IPC-356 Data for Care Board Test
xxxx-yyy.rpp	Drill Report File for Plated Holes
xxxx-yyy.drp	N/C Drill File for Plated Holes
xxxx-yyy.rpn	Drill Report File for Non-Plated Holes
xxxx-yyy.drn	N/C Drill File for Non-Plated Holes
xxxx-yyy.fbd	Fabrication Drawing
xxxx-yyy.fyi	Engineering Contact Person

Table 1. Typical Data Files Used to Fabricate a PCB



Initializing the Front-end Engineering Effort



The first step in the front-end engineering process is to check the design for accuracy. A critical part of this process is to synthesize a netlist from the Gerber data or artwork that shows how the PCB will be connected if it is built to the artwork. This synthesized list is compared to the CAD netlist (provided as noted above) and represents how the PCB should be connected. This netlist comparison effort is the irst crucial step in the PCB tooling process and should never be omitted no matter how tight the schedule may be. The netlist information needs to be complete and accurate (thereby avoiding the old “garbage in/garbage out” scenario). If the Gerber data netlist and CAD netlist do not agree, then no further work should be done until the differences are resolved. This netlist compare operation is a vital safeguard against errors that may creep into the design data along the way. Failing to make this effort frequently results in PCBs that are wrong and unusable from the get-go. Once the netlist exercise has been successfully completed, the next step is to check the design artwork to ensure:

- Correct trace widths.
- Correct clearances.
- Proper registration between all the layers.
- The pad sizes are adequate for the drill sizes and tolerances.

If the PCB is fabricated, it is controlled impedance. The design engineering team has failed to provide the trace widths, laminate thicknesses, and styles required to arrive at the correct impedance in each layer. The front-end engineering group will use impedance predicting tools to arrive at this crucial information. From our experience, it is unwise to leave this critical step up to the fabricator alone because each one will do it to suit its standard processes. This can result in having a completely different PCB from two different fabricators using the same set of film. This is one of the key reasons we spend so much time on stackup design in our courses and our writing. Not getting it right can cost time, money, missed market windows, or all three.

Manufacturing Tooling

The next step following the preceding is the generation of manufacturing tooling. This consists of:

- Production artwork for each layer.
- Drill files.
- Test tooling.
- Routing profiles.
- Plating schedules.
- Etching schedules.
- Lay-up Instructions.
- Lamination schedules.
- Quality tests.

Each of the foregoing is described below.

Production artwork consists of a film for each PCB layer along with film pieces for the solder mask on each side and the legend or silkscreen for each side. This artwork is different than the design artwork in the following ways:

- The trace widths will be made wider to allow for the narrowing that occurs during the etching process.
- The actual size of the artwork will be slightly expanded to allow for the material shrinkage that takes place during the lamination process.
- Manufacturing tooling features will be added around the perimeter of the panel in which the PCB is being built. These features include:
    - Registration targets.
    - Test structures.
    - Resin dams to even the flow of resin in the prepreg as it softens during lamination.

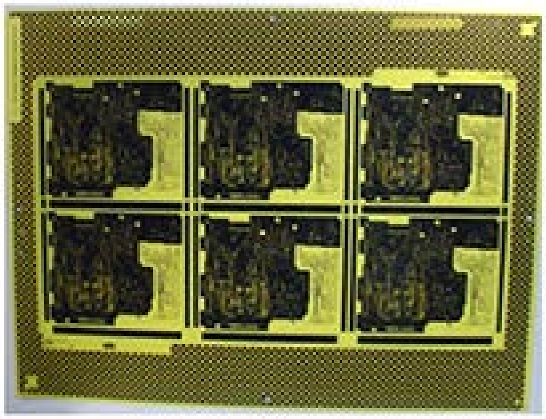


Figure 3. Inner layer detail shown after the application of black oxide.

Photo on the left shows the manufacturing tooling, resin dams (the pattern of black dots), and test structures are visible in the panel's boundary.

Drill files include the drill sizes and locations for all of the holes, both plated and non-plated. They are organized to provide the most efficient drill to travel from hole to hole. If the finished hole size has been specified, process engineering will calculate the drill size needed to arrive at the finished hole following plating. It is very useful to add specific notes regarding drill size. Traditionally, the finished hole size was specified, and the fabricator than chose a drill size that suited its process. With today's designs and the tight spacing of component pins, there is not a lot of margin for variations in hole size. That's we recommend choosing the drill size as part of the pad stack design process and then freezing it. This results in specifying the drill size in the drill chart as opposed to the finished hole size. If laser or controlled-depth drilling or backfilling is specified as part of the design, those files will also be created as part of the drill files.

Test tooling includes the information necessary to build the test fixture, the wiring rules for that text structure, and the netlist used by the tester to verify that the connectivity is correct.

Routing profiles include instructions for a machine (router) that cuts the PCB from the panel in which it was built. If the PCBs are connected in a sub-panel to facilitate assembly, the instructions will include creating the groove lines or lines of drilled holes that will be used to break the PCB from the sub-panel after assembly. Figure 4 shows a panelized PCB containing nine small PCBs within the panel. It has been designed to optimize the assembly process. The light areas represent the material to be removed around each PCB during the routing process. Following assembly, each PCB will be broken out of the panel.

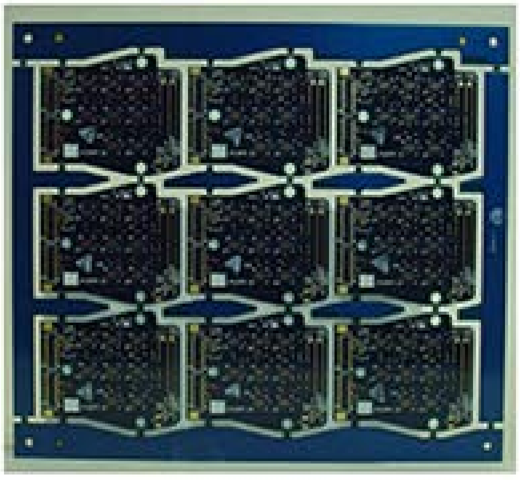


Figure 4. A Panelized PCB with 6 PCBs per Panel Ready for tAssembly

Plating Schedules define what kinds of metals will be plated onto the outer layers of the PCB and how long the panel will need to remain in each plating step to achieve the required metal thickness. It should be noted that creating inner layers does not involve plating.

Etching Schedules describe the etching steps required and how long the PCB or inner layers are to remain in each etching step.

Layup Instructions describe how the inner layers, subassemblies (or details), prepreg layers, and outer layer copper foils are arranged to arrive at the final stackup. This information includes how many PCBs will be included in a single press opening and how they are to be separated.

Lamination schedules include how much pressure is to be used during lamination, the temperature profile for the lamination step, the duration of the press cycle, and how the laminated PCB will be cooled.

In addition to the netlist compare tests described above, additional Quality tests check for shorts and open, trace-to-via clearances, and via-to planes-clearances.

Summary

Front-end engineering focuses on all of the steps necessary to move a PCB from the design to the fabrication process. This is a critical part of the manufacturing process, and the thoroughness of it will ensure that the manufactured PCB matches the as-designed PCB.

Would you like to find out more about how Altium can help you with your next PCB design? Talk to an expert at Altium or discover how to easily export native designs or other EDA file formats with Altium Designer's PCB-to-Gerber converter functionality.

References:
Ritchey, Lee W. and Zasio, John J., "Right The First Time, A Practical Handbook on High Speed PCB and System Design Volume 2.

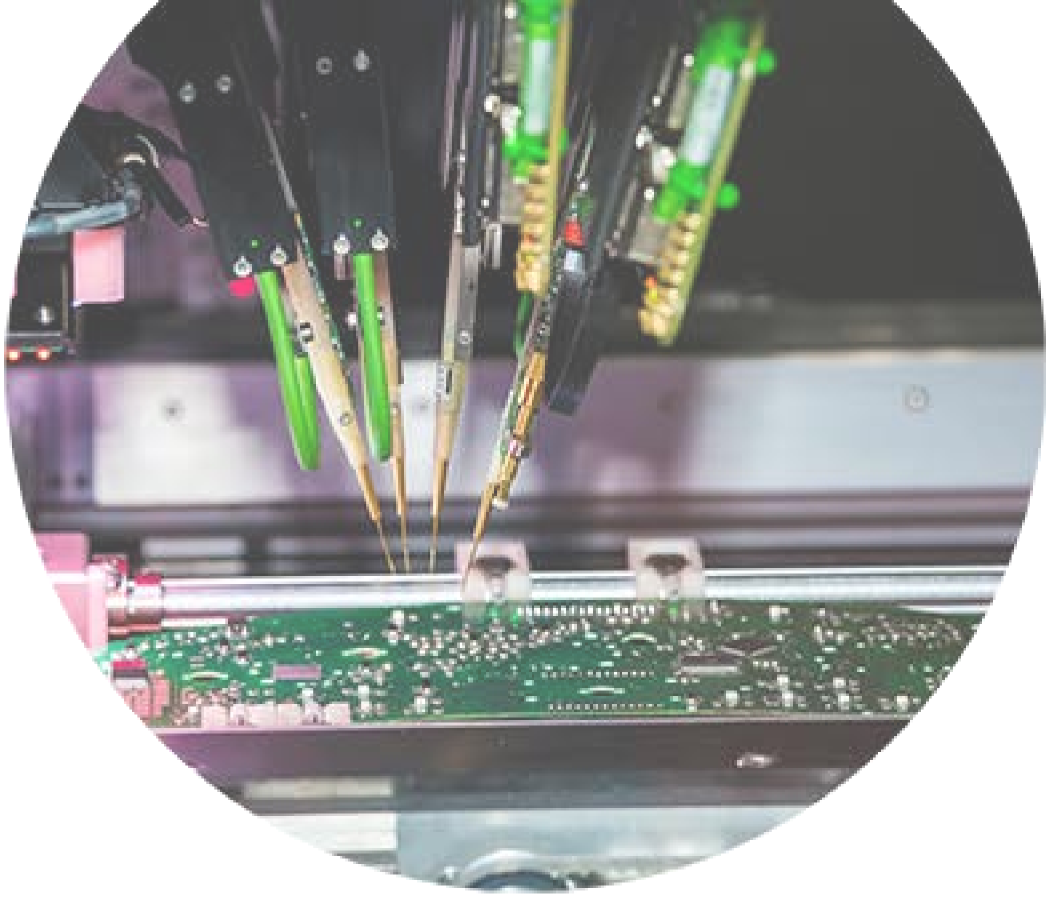
How to Compare PCB Manufacturing Services for Your Board

When you run a design firm, there's one type of email you're bound to receive more than any other: sales emails from PCB manufacturing services. I understand that companies need to get their names out there and want to win business, so this post isn't meant to denigrate those companies. There are plenty of PCB manufacturing services you can find online, and they can all start to blend together. If you're searching for a new service provider, it can be hard to compare all of them and find the best manufacturer that meets your needs.

Some companies will spam you with offers like "2 to 48 layer HDI RF metal core PCB" or something similarly absurd. While experienced designers can spot bogus manufacturers from afar, there is always a temptation to go with the lowest priced, supposedly fastest overseas company you can find. However, there is a lot more that should go into choosing a PCB manufacturing service than just price.

How to Choose Your PCB Manufacturing Service

When you're looking at engaging with a PCB manufacturing service, you'll find a lot of information on their website to use for comparison. There is a lot of overlap between different services in terms of price and lead times, but there are multiple metrics you should use to compare different services.



To start, look at the manufacturer's capability limits. Most manufacturers are pretty competitive in this area and can reliably fabricate down to something like 4 mil traces, 8 mil vias, and few mil spacing on standard stackups. After you've verified your board matches their capabilities, there are other metrics you should use for comparison:

Price vs. Lead Time

Price and lead time tend to follow an inverse relationship: one goes up while the other goes down. If your board house offers flexible lead time options, you can offset some of the fabrication costs by extending your lead time. This applies to both fabrication and assembly. Similarly, if you need your boards turned overnight, you'll have to pay extra for it. Make sure your PCB manufacturing service can accommodate your lead time targets within your budget.

One thing I did not account for in my first manufacturing run was the shipping time required to move from fabrication to assembly, and to deliver the finished PCBA to me. Also, I did not account for the time required to ship components to the manufacturer, which delayed the start of PCB assembly. Make sure you account for these other aspects of the PCB manufacturing process so you don't get surprised by any delivery delays.

Fabrication, Assembly, or Both?

Some manufacturers will provide fabrication and assembly in-house, while some will outsource assembly to a partner. Keep this in mind because you may need to have your abricated boards and your components sent over to an assembler, assuming you won't do assembly yourself. Some PCB manufacturing services will offer a turnkey service, where they handle everything related to your production run and you don't have to worry about much else. This is good for designers who may not be familiar with the procurement and consignment process or have time to manage procurement.

Production-Line Flashing

This is more for the embedded developer who wants to eventually produce at high volume, but it's still something to consider. Some PCB manufacturing services will offer production-line firmware flashing services, where your code is flashed into memory before the boards are sent out to you. This is not something you'll need to do during PCB prototyping, but you would need to consider it if you take your prototype into high-volume manufacturing.

Plating and Copper Weight Options

Look at any PCB manufacturing service website, and they all offer just about the same specifications on copper: 0.5 or 1 oz./sq. ft. copper with ENIG plating on the outer layers, and 1 oz./sq. Ft. copper on inner layers or planes. If you need different copper weight and/or plating, make sure to check on your fabricator's capabilities and stackup options.

Available Materials and Stackups

I've put this last because it's almost always the last thing new designers think of when planning a production run for a board. Instead, the very first thing you should do before designing your board is create a proposed stackup, and then check with a fabricator to make sure they can produce something at or close to those specifications. You can send a screenshot of your proposed stackup to the fabricator, and they can confirm whether their material stocks can match those requirements, or they can propose an alternative stackup.

Layer	Type	Thickness		Trace layer	Type	Value	Trace
L1		0.035mm		Outer	Single	50ohm	12mil
PP	7628*1	0.2mm		Outer	Diff	100ohm	8/8mil
L2		0.0175mm		Outer	Diff	90ohm	10/8mil
Core	FR4*1	0.265mm	0.3mm (inc copper)	Inner	Single	50ohm	10mil
L3		0.0175mm		Inner	Diff	100ohm	6/8mil
PP	7628*1	0.2mm		Inner	Diff	90ohm	8/8mil
L4		0.035mm					
	Finished	0.77mm					

Example stackup table from a PCB manufacturing service. Note that this board has non-standard thickness.

Make sure to let your fabricator know if you have an impedance requirement (usually 50 Ohms single-ended). If you've made a mistake when calculating the stackup and trace width, a good fabricator will tell you the correct stackup that they can produce. Remember, if it can't be manufactured, then you might as well not even design it.

The Takeaway: Talk to Your PCB Fabricator First

From the above list, there are many dimensions you'll need to think about to compare PCB manufacturers. It doesn't hurt to talk to your fabricator early to get an idea of their capabilities to ensure your run is successful and meets your expectations.

When you need to access an easy-to-use PCB layout tool that includes everything needed to build high-quality manufacturable circuit boards, look no further than CircuitMaker. When you're planning your next manufacturing run, you can instantly generate the output files needed to put your board into production with a PCB manufacturing service. All CircuitMaker users also have access to a personal workspace on the Altium 365 platform. You can upload and store your design data in the cloud, and you can easily view your projects via your web browser in a secure platform.

How to Create a PCB Manufacturing Cost Estimation

Unless I receive a layout job directly from a manufacturer, my clients will usually ask if I'm a manufacturer. My company doesn't manufacture boards, but I think it's important to keep clients informed of the manufacturing costs for new PCBs. Once I started gathering estimates for some of my first projects, I started to better understand the primary cost drivers of PCB manufacturing and how to properly structure a comprehensive estimate for my clients.

Unless I receive a layout job directly from a manufacturer, my clients will usually ask if I'm a manufacturer. My company doesn't manufacture boards, but I think it's important to keep clients informed of the manufacturing costs for new PCBs. Once I started gathering estimates for some of my first projects, I started to better understand the primary cost drivers of PCB tmanufacturing and how to properly struc-ture a comprehensive estimate for my cli-ents.

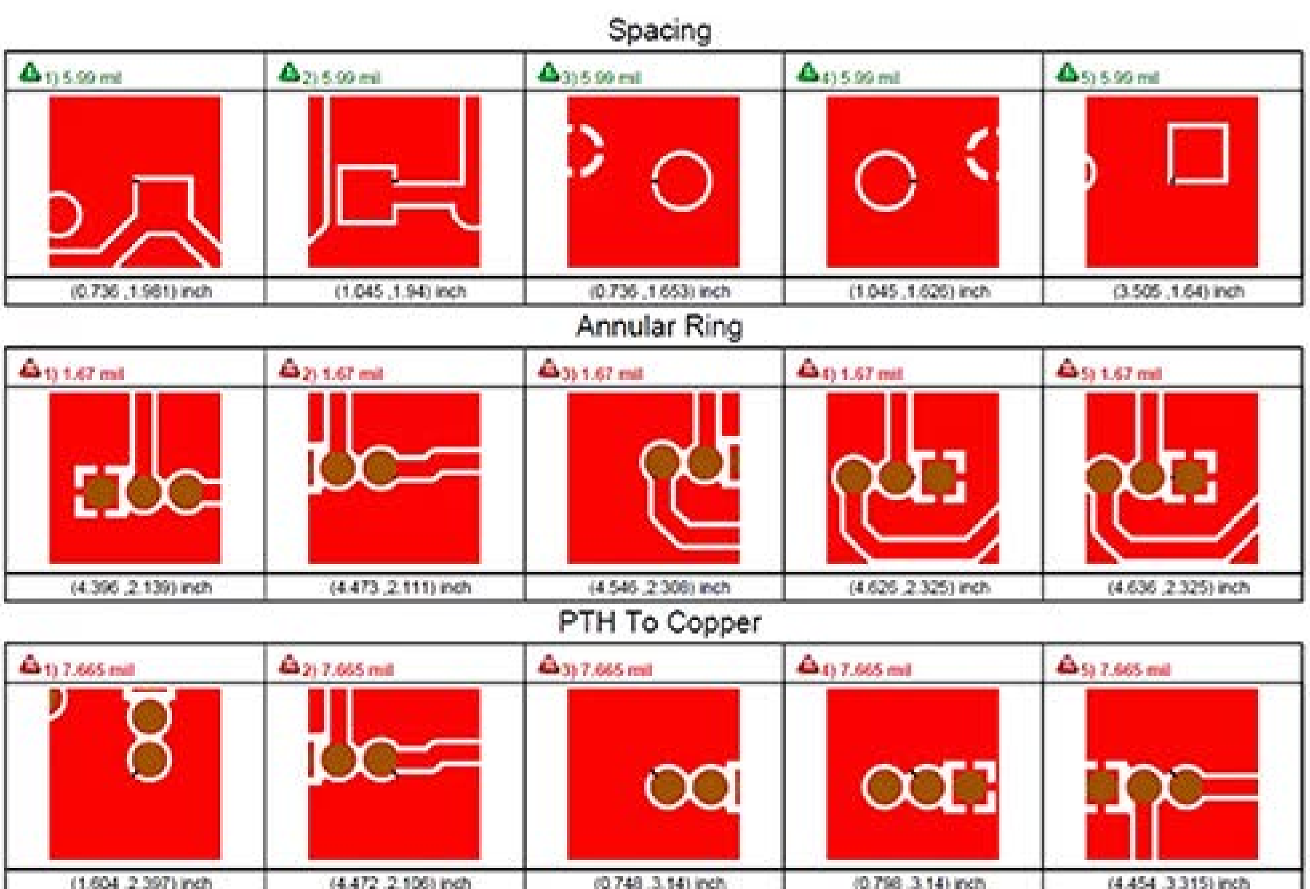
What Drives PCB Manufacturing Cost Estimation?

There are a number of factors that drive manufacturing costs. In general, producing more boards, using more vias per board, using smaller traces/vias/mounting holes, and using more components per board will all increase manufacturing costs. However, the costs involved are not linear. In other words, double the number of boards and components will not double the total cost of an order.

There are some important reasons for this. There are a number of fixed costs involved in any manufacturing run, including tooling costs, design review, stencil creation, and others, depending on your requirements. Other costs are variable and are determined on a per-board basis. The cost per board can decrease as order size decreases, which has less to do with panelization and more with amortizing fixed NRE and tooling costs across an entire fabrication run. The same idea applies to assembly (see below).

Pre-Production DFM Review and Inspection

Some PCB manufacturing houses will offer different levels of service as far as a pre-production inspection, design and engineering review, and DFM review are concerned. Taking advantage of a full engineering review can significantly increase the per-board cost at low volume as it carries an additional fixed cost. Note that, some level of PCB design review has to occur for the design to be quoted and to ensure it is even manufacturable. If you violate some basic design constraints (clearances being the most important), your project might get marked no-bid until you correct some basic errors.



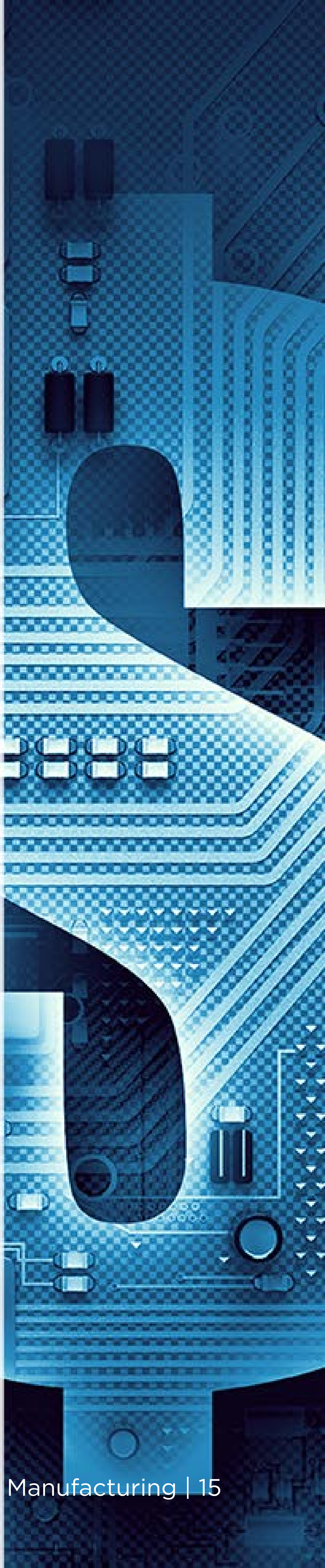
When your design goes through a thorough DFM review, your fabricator will check your files to ensure your design can be manufactured in their process.

This is where level of service and the extent of any review is very important. Sometimes, an electrically correct and manufacturable design choice will not trigger a DFM violation in terms of fabrication ability, but it may create risk of an assembly defect or a total failure of your product. A perfect example is pads that are too close together without sufficient solder mask definition on the pad. The reverse is possible as well, where an electrically correct design choice triggers a DFM violation, yet the design choice was intentional. My company does many designs involving RF structures and waveguides, and we sometimes get a notification of a short circuit during a PCB design review or when running DRCs. Be sure to inquire what is involved in this type of review and determine which design deliverables you need to provide before adding an engineering review service to your quote. It's always good if you can anticipate these intended or unintended DFM violations and to communicate them with the CAM engineer at your fabrication house.

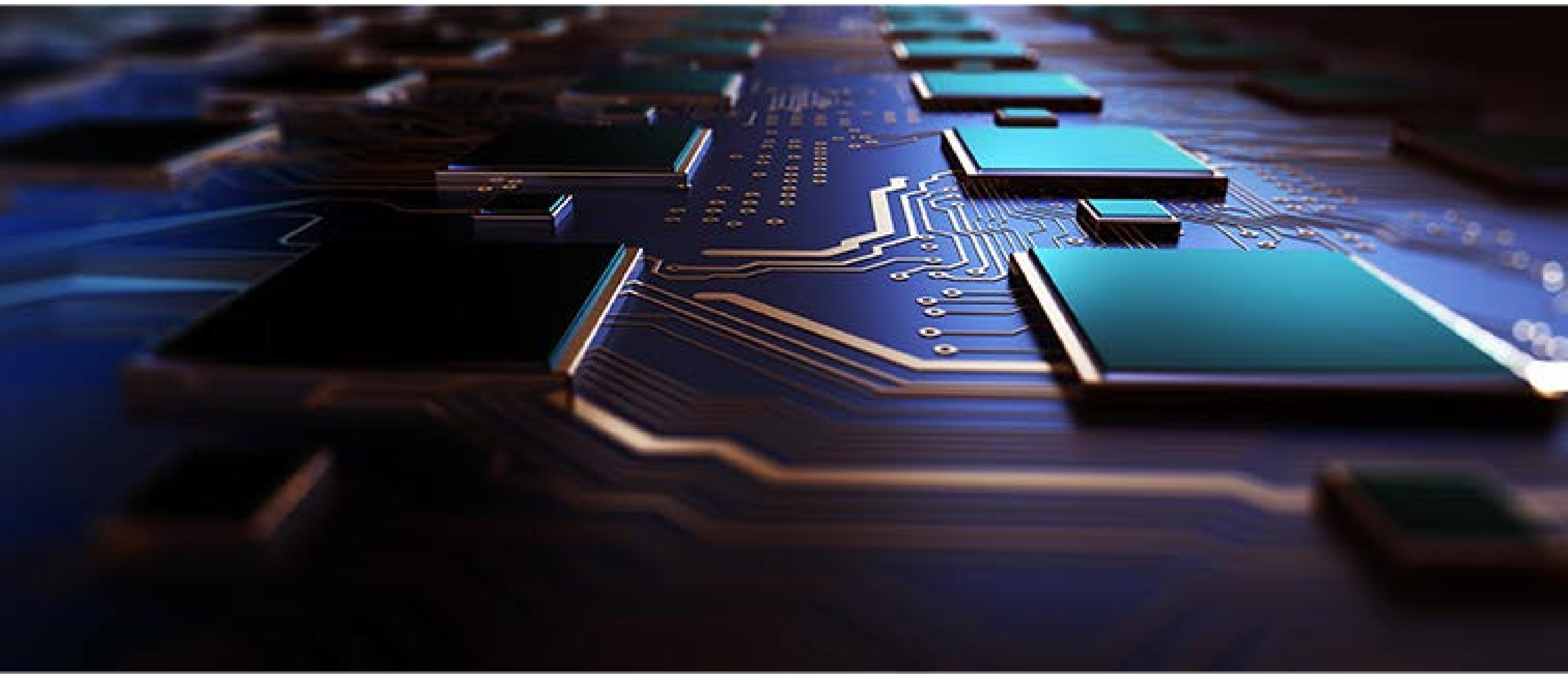
PCB Fabrication Cost Drivers

Here is a short summary of some major cost drivers involved in fabrication. Be sure to consider each of these areas when creating a PCB manufacturing cost estimation.

Cost driver	Effect on manufacturing costs
Number of boards	Increase boards → decreased cost per board
Number of components	Increase component count → decreased assembly cost per board
Plating type	Tin-lead or HASL is usually standard. Using ENIG/ENEPIG, immersion silver, OSP, or other plating tends to increase per-board costs by up to ~10%.
Via hole sizes	Smaller via holes → increased tooling costs and per board costs
Conductor size and spacing	Smaller size/spacing → increased costs
Copper weight	Heavier copper → increased cost
Silkscreen and solder mask	Negligible effects on cost (less than 0.1% effect)
Mechanical holes	Depends on size, shape, and plating
Substrate material	FR4 materials are most popular and carry lower costs. Materials like Rogers or polyimide tend to carry higher costs.



Regarding the number of boards being fabricated, the main driver is not really the number of boards, it's the cost per panel. For a small run, per panel cost and NRE costs will determine your per board cost. At high volume, the per overall per board cost comes down as variable costs dominate the overall production run cost. The other points above are just the starting point for manufacturing as they only focus on bare board fabrication. There are also assembly costs to consider..



Assembly Cost Drivers

The cost drivers involved in assembly boil down to the following:

Number of unique components: More unique components equates to higher cost, although ordering components in bulk reduces the price per component.

Type of components: All-SMD assembly can cost up to ~50% more through-hole assembly.

Double-sided vs. single-sided assembly: Double sided assembly adds to costs as your board essentially has to run down the assembly line twice.

Lead pitch on SMD components: I've seen that any premium on fine-pitch SMD components varies by manufacturer. If you're already using all-SMD, some manufacturers will not charge extra for finer pitch components.

BGAs: Single-sided or double-sided BGA component assembly can cost ~20% more than other SMD components, which is in addition to any premium that's already charged for double-sided assembly. Finer pitch BGAs below some limit could carry additional costs, depending on fab and assembly capabilities.

IPC 6012/IPC-A-600 Class 3: Production and assembly up to IPC-A-600 Class 3 can carry significantly higher costs, although the quality of the final product will be much higher. The costs are also higher due to additional inspections required to verify compliance with the higher performance and inspection specifications imposed on Class 3 products.

ITAR compliance: If you're working in the U.S., your project may need to be ITAR compliant. Finished PCBs will need to adhere to ITAR requirements on data integrity, PCB integrity, and shipping. These costs get passed on to you as the designer.

Unless assembly is being done by hand, assembly costs will not always increase linearly with the number of boards. In other words, with automated assembly, doubling the number of boards does not always double the cost of assembly. Some of the assembly cost goes into programming machines, while the remainder effectively pays for machine time.

Component costs follow a similar structure. When you order more components, you might reach a new price tier with your component distributor where the cost per component drops. This is one reason it's better to buy from a single distributor rather than mix-and-match multiple distributors.

Overseas vs. Local Production

We live in a globalized environment, and it's never been easier to outsource fabrication and/or assembly to a region with lower labor costs. The choice to do so depends on the production volume, level of service required, the risk involved in exposing a design oversees, lead time equired, and potential cost savings.

For basic designs that have been thoroughly qualified, I’ve found that Chinese manufacturers can do the job just fine, although they nickel-and-dime you on small additions to the order (e.g., certain screen printing features, testing, solder mask color). I've had clients tell me horror stories where the design they sent in did not at all resemble the product they received; apparently the manufacturer made many modifications to the design in order to cut costs as low as possible, then they pocketed the difference. There are also intellectual property concerns, and the risk of exposing design data to an overseas party could be unacceptably high. For mil-areo designs in the US, export control restrictions (EAR) and ITAR requirements will limit your ability outsource fabrication and ship overseas; similar restrictions apply in Canada and Europe.

There are some real benefits to staying local, even if the cost is higher. I prefer local manufacturers as it gives me more control over the entire process, especially for prototyping or low-volume production. I might end up paying a more per unit at low volume, but myself and my clients are willing to pay for the extra level of service, responsiveness, and security in knowing our designs won't be stolen. If you're willing to accept the risk involved in going overseas for fabrication, you could work with a contract manufacturer (CM) that uses a qualified overseas fabrication house and assembles locally. For more advanced designs, or if the design carries some specific testing/inspection requirements, I’d rather use a local manufacturer as I can always talk to someone on the phone directly.

In terms of cost, the per-unit price for a board produced locally (in the U.S.) can be a factor 5 as large as the price for the same board produced overseas with the same lead time. Assembly is another matter altogether, as not all manufacturers will provide on-site assembly, and you'll need to get a quote from an assembler company. to ship your boards from your manufacturer to your assembler also increases your lead time, which adds intangible costs to your new product.

Be careful before you send your design overseas for production and carefully consider the risks.

Testing, Inspection, and Qualification

Testing and inspection are two important activities that also need to be performed when a new board is produced. Production and inspection up to IPC-A-600 Class 3 costs more than Class 2 or Class 1; the same applies to other relevant standards and classes. You might also need to produce to a specific set of industry standards, such as IPC-13485, NADCAP, or AS9100, but it's hard to make generalized statements regarding costs with these manufacturers. Qualification to UL standards or other specific industry standards normally carries an added cost and will not be available from every manufacturer. This is where different manufacturers can have big differences in capabilities, which will determine costs involved in testing, inspection, and qualification. Make sure to account for these points when requesting quotes from manufacturers.

Last But Not Least: Components!

With all this talk about fabrication and assembly costs, it's easy to forget about the costs for the actual components. In today's environment of semiconductor shortages, all the big EMS companies are moving from a just-in-time to just-in-case supply chain model. This means designers need to place a role in risk management by performing sourcing activities at the start of the design, not at the end of a design.

This isn't necessarily a cost driver for fabrication or assembly, but it can create the need for a redesign when you suddenly find that your critical components go out of stock.

1. Identify critical components with few or no replacements early, and then procure minimum required stocks.

2. Ideally, instead of #1, use components that have some available replacements (ideally pin compatible) and plan to procure the required quantities before production.

3. For specialty parts where you'll deal directly with a manufacturer, nail down the requirements at the beginning of the design and place an order with the component manufacturer early. The lead time might be long, but you'll have parts in transit before you finish the design.

If you're part of a service bureau or larger engineering firm, make sure to account for these points early in the design process. It may take some commitment from customers, but locking in these requirements early helps you eliminate a lot of risk and cost increases before going to production

When you’re building a new PCB, Altium Designer® gives you the tools you need to create products that include advanced features while complying with your manufacturer’s DFM requirements. You’ll also be able to quickly generate deliverables and documentation that help ensure you'll get an accurate PCB manufacturing cost estimation. When you're ready to release your project for a PCB design review, you can use the Altium 365 platform to share design data with your manufacturer and get to market quickly.