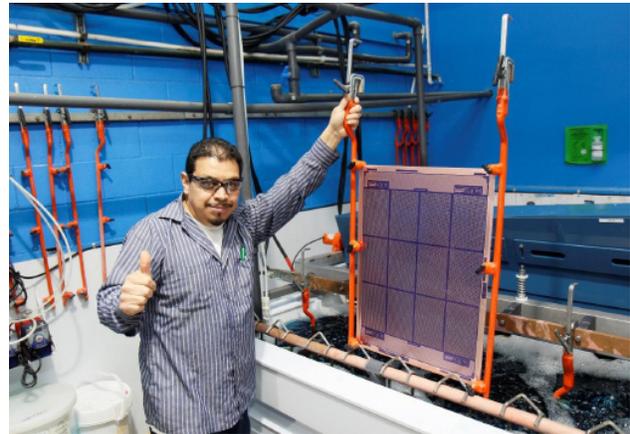




7 ESSENTIAL QUESTIONS TO ASK YOUR PCB FABRICATOR

Does your PCB fabricator add value to your business? Or does your PCB fabricator cause you to lose sleep at night?

While many manufacturers claim quick turns and high quality, it can be hard to discern advertising from reality. When deciding who to trust with your PCB designs, it's important to "look under the hood." Use these 7 questions to help guide your evaluation process and uncover the right PCB fab partner for your next project.



1) How Fast Can You Spin a Board?

Can I have a board by the end of the week? How about tomorrow? What about by the end-of-day? The answer depends both on your design as well as the capabilities of your manufacturer.

PCB manufacture is a multi-step process that is controlled by a mixture of physics, chemistry, planning, and the mechanical limitations of manufacturing equipment. While your manufacturing partner can reduce turn time by using the latest technology, you can also reduce turn time with some key design considerations.

How Many Types of Laminate Do You Stock?

Each board begins life as a schematic. Engineers usually know early on if they will have unusual requirements for a project, such as an exotic dielectric material required for a PAM4 bus (e.g. Duroid 5870), or an unusually thick piece of mid-layer copper (e.g. 6 oz) for a power-design. Ask your board shop if they have the material in stock, or if they can pre-order your stack-up materials while you complete the routing on your design. Also, ask what materials your [fabricator regularly stocks](#) – if you can choose those materials for your design, you will save yourself time and money.



**ROYAL CIRCUIT
SOLUTIONS**



One of the on-hand storage areas at Royal Circuits. We stock a [wide variety of laminate materials](#) and copper sheets so can start production as soon as the DFM checks are complete.

What Ovens Do You Use?

PCBs are made up of layers of conductors separated by layers of insulators. These layers are aligned and glued together in a high-temperature, high-pressure oven, a process that takes hours with the previous generation of hot-air ovens. If a misalignment occurs, the board is scrapped, and the manufacturer must start over and try again. The ovens are

used again at the end of the process to cure the silkscreen – requiring another hour to ninety minutes.

It is physically impossible to do “miracle-turns” if you have a single machine that consumes 25% of the production time. It is advisable to select a PCB manufacturer that uses nano-steam instead. Nano-steam ovens reduce the process time for most lamination to 10 minutes, and silkscreen curing to 5 minutes. While the nano-steam ovens cannot be used for all stack-ups, they can be used for most orders, greatly reducing turn-time.

Material Removal

PCBs need holes that are made through two processes – drilling and routing. The last-generation of manufacturing machinery required the processes to be done on two different machines. A few manufacturers, like Royal Circuit Solutions, invested in the latest generation of LENZ manufacturing machinery that drills and routes on the same machine. This saves time and improves overall accuracy.

But even these machines have their limitations. As drill size decreases, spindle speed must increase. The smallest mechanical drill is 6 mils, which requires moving boards to a different machine, which will increase both the time it takes to make your board and the overall cost of your project.



**ROYAL CIRCUIT
SOLUTIONS**



These 6 mil drills create the smallest machined vias possible. Smaller holes require laser drilling. →

So how soon can I get my board?

Talk to your PCB manufacturer early in the process and find out the limitation and capabilities of the machinery as well as the stock on hand. Simple design decisions (such as using a 0.006" via instead of a 0.005" via) can end up saving significant amounts of time and money. Royal Circuits, for example, can manufacture up to 250 ten-layer boards in under a day, and some orders can be manufactured in a single shift.



2) Does Your Fabrication Shop Offer Free DFM Service?

Did you know Electrical Engineers can graduate from many universities without ever taking a course in PCB design? This unfortunate fact leads many engineers to have a fundamental misunderstanding of manufacturing constraints and industry best-practices. Approximately 75% of designs are put on hold or returned to the engineer because they cannot be manufactured as submitted. And it is often the little things that get you. Does your manufacturer help you fix problems and learn along the way?



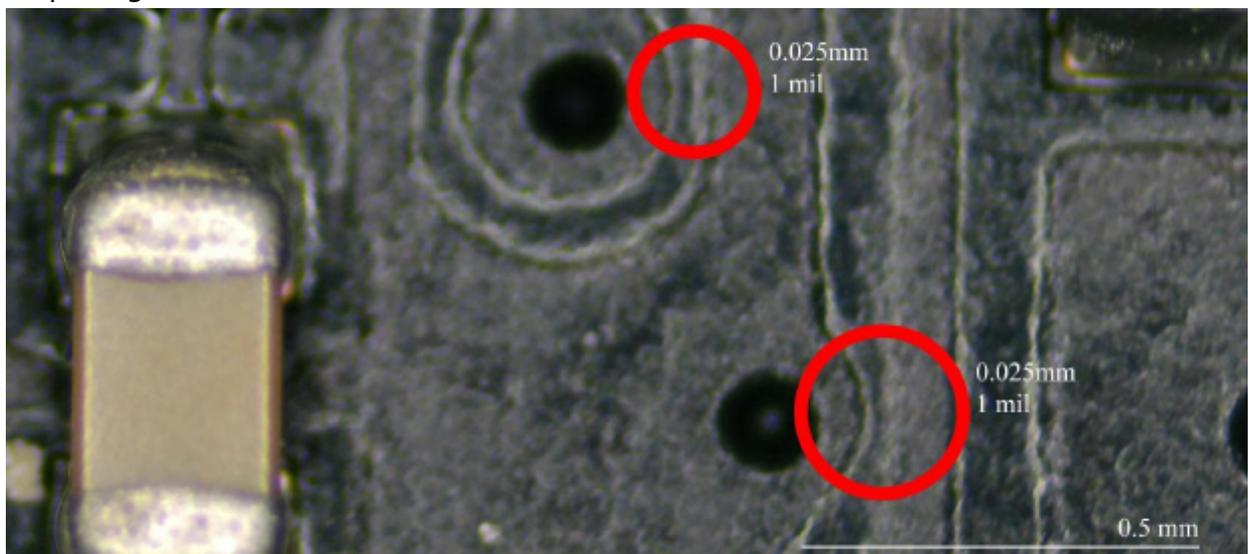
PCBs are Complex and Errors are Common

Electrical Engineering is a complicated subject, and pitfalls are everywhere. For example, 0.25 oz copper can handle 3/3 mil trace and space. On the other end of the spectrum, 6 oz copper is limited to 9/15 mil trace and space. Did you properly program your design rules in your cad program to account for those differences? Did you know that 3 mil traces can lift away from the substrate in certain geometries? Did you properly specify the minimum annular ring diameters for 5 mil vias? Did you accidentally move signal traces so they now cross plane layers? Do your plated through-hole pads have corresponding drill instructions? Is your silk-screen text so small that it will look like a blob even under 100x magnification? Will your design partner help you learn and grow as a professional, or will they make whatever design you submit them and force you to learn about mistakes the hard way?

The DRCs in your cad program only look at some of the possible issues that will affect the manufacturability of your design – that is because they work on theoretically ideal PCBs. But real-world manufacturing has variations and anomalies. You need to account for manufacturing variation in your designs.

Example: Trace & Space Violation

As an example, the following microscope image shows a production board from a well-known manufacturer that has design flaws. The board was released in March of 2019. This company is on both the NASDAQ-100 index and the S&P-100 index. Microscopic examination shows that annular rings around vias violate spacing guidelines in more than one location on the board. This might be due to a design rule that requires a minimum diameter annular ring that has a higher priority than a spacing rule.





ROYAL CIRCUIT SOLUTIONS

As you can tell in other parts of the image, copper doesn't etch into perfectly straight lines. This board design has annular rings that have 1 mil or less spacing from adjacent nets. Minor variations in etching could easily allow the vias to short circuit with adjacent traces. The result is that many of the boards will fail the flying-probe net-check and must be rejected.

Would your board shop have caught this error before you ordered 10,000 boards? Ask your potential PCB fabrication partner if it performs free, automated Design-For-Manufacturability (DFM) checks at the quote stage. Also, make sure the automated software is backed by experienced engineers who check your design immediately after submission and notify you of problems that your DRC might have missed.

3) Does Your Fabrication Shop Charge NRE Fees?

Non-Recurring Engineering (NRE) fees are one-time charges for tooling, testing, templates, and the time needed to set up a production run.

Why do NRE fees exist?

PCB manufacturing is a multi-step process that requires tool creation and machine programming. Companies may charge fees for design-checks, photolithography, stencil creation, machine setup, etc... Some shops are very transparent about these charges while others hide them in the total cost of the board. Either way, the customer ends up paying more than they need to.

But, there are some shops that never charges NRE fees.

Why doesn't Royal Circuits charge NRE fees?

Royal Circuits' ongoing commitment to annual machinery upgrades means it doesn't have as many NRE costs during manufacturing as other fab houses, because the machines don't have the same requirements. The few NRE steps it does have are not passed on to customers.

Since the company has fewer manufacturing steps, it can produce more boards per shift. Our success come from high customer volume, not high customer fees.





4) Does Your Fab Shop Charge Extra for Custom Solder Mask Colors?

Solder mask is a thin layer of colored lacquer that is applied to a PCB in the final steps of fabrication, just before silk-screening. This coating gives PCBs their characteristic color and the coating serves several purposes:

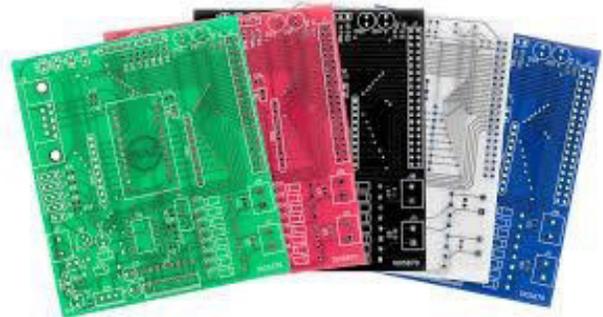
- Protects the copper from oxidation
- Helps protect against short circuits
- Helps thin traces adhere to the substrate
- Slows humidity induced board changes

Historically, the only available color of solder-mask was green. Then black, white, blue, and red boards gained in popularity. Most fabrication shops upcharge hundreds of dollars for these custom solder mask colors. They do this because they can, not because they need to.

Can I have a non-standard solder-mask color?

Find a PCB fabrication partner that allows you to use any color of solder-mask you want without extra changers.

Red, Orange, Purple, Pink, Cyan, Yellow, Chartreuse, Burgundy, etc.... it should not cost to use custom colors.



5) Does Your Fab Shop Treat You Like a True Partner?

Have you fallen in love with one manufacturing shop, or do you move from one shop to another based off search engine results?

Once you find the right partner, you should want to stay with that partner. Can you tour the facility? Do parents and their children work on the same production line?

- Do they provide two solder samples with every order to allow you to fine-tune your reflow profile setup?
- Do they provide 3-5 extra boards with every order because emergencies, and accidents, happen?
- Do they provide automatic order status and shipping status updates?



**ROYAL CIRCUIT
SOLUTIONS**

- Do they help you design impedance-controlled boards?
- Do they tell you the tips and tricks they've learned over the past 30 years that allow you to reduce the manufacturing cost of your boards?
- Do they help you move offshore and handle all questions needed to move to volume production?
- Do they give back to the community by helping to educate the next generation of electrical engineers?



Royal Circuits' Mihir Shah gives a factor tour to Department Chair Jack Bedell, PhD and his Electrical Engineering Students from California State University, Fullerton.

6) Can Your Fabrication Shop Take My Project from Prototype to Production?

Some customers need 2 boards by the end of the week, some customers need 250 boards by tomorrow, and some customers need 250,000 boards by the end of the month. Can your fab partner fulfill all your needs, no matter the quantity? Will they deliver the boards the next day? Will they help you reduce the cost of your board?

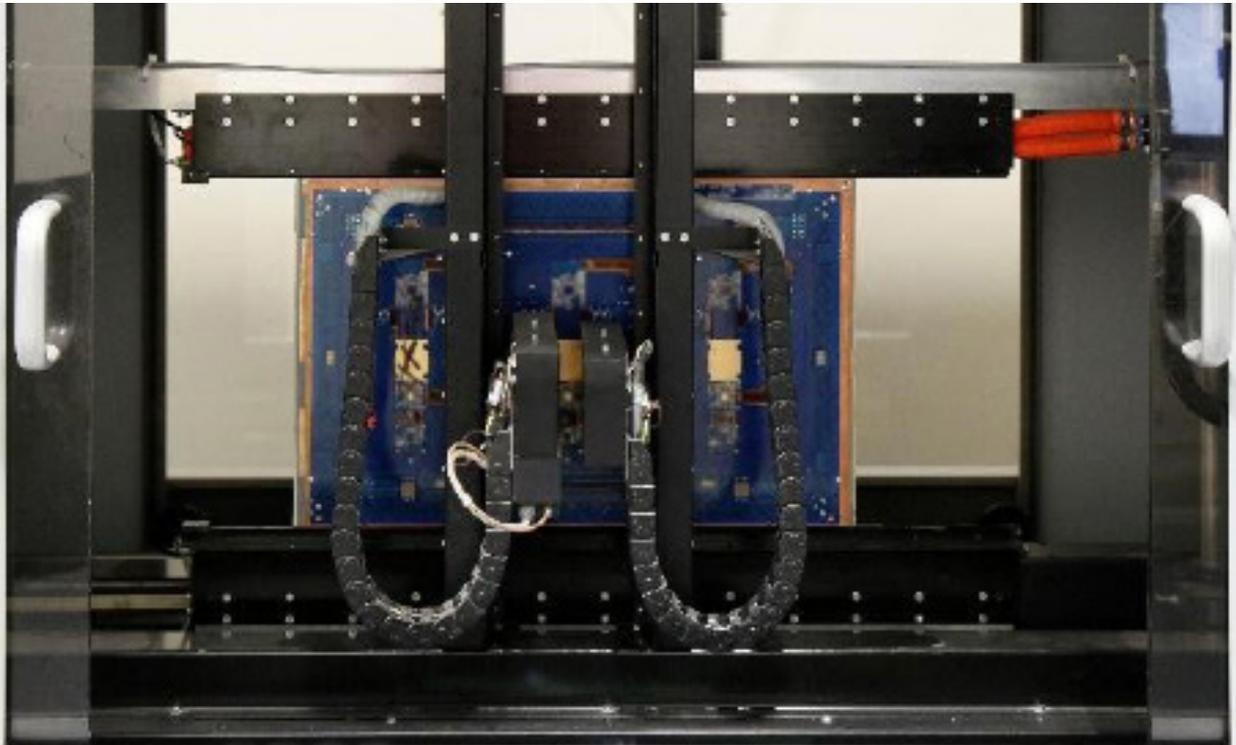
Moving from prototype quantities to large-volume production quantities requires a separate skillset. Does your fab house have offshore partners who can fulfill volume orders at low cost? Can your fab-house handle all the overseas interactions and answer all the CAM questions so that you don't have to?



**ROYAL CIRCUIT
SOLUTIONS**

Will your fab partner ensure that every board is 100% netlist checked with a flying probe machine, so you don't waste parts and time populating boards with open-circuit or short-circuit flaws?

Find a PCB partner that can handle any quantity of boards and can help you work with an overseas partner when necessary so that you can focus on other aspects of your project.



This flying probe tester checks 100% of the boards that we deliver.

7) What Are Some of Your Capabilities?

Electrical Engineers are faced with numerous challenges and competing priorities when they design a board. As complexity increases, the chances of a mistake rise as well. Does your fab house have the experience and know-how to make sure your board works on the first spin?

How complicated can it be?

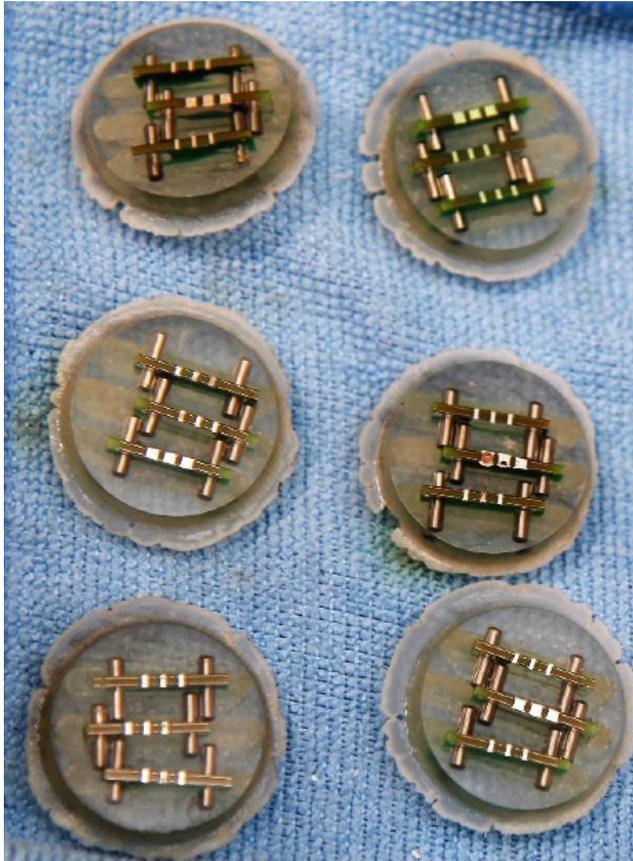
ICs have lower operating voltages than they did 10 years ago, which means that components can shrink in size. Smaller components allow engineers to pack today's boards with more parts per square inch than ever before. And while the parts now



ROYAL CIRCUIT SOLUTIONS

consume a smaller surface area, the vias and the traces have reached their lower design limits for manufacturability using standard processes.

Will your fab partner help you analyze your board for crosstalk noise? Or look for signal nets that cross copper pour boundaries? Will your eye-diagram close because you chose the wrong dielectric or the wrong weave pattern? Will your fab partner



help you with other High-Density Interconnect issues? Do they help you determine the actual impedance of your board stack-up with FEA, so you can use appropriate trace/space on single-ended and differential pair signals? What is the smallest micro-via they can make? Can their engineers help you design high-speed and RF-circuits? Will they teach you how to make planar capacitors on interior layers?

The current generation of PCBs already have components so small that they can only be seen with microscopes. And components, and boards, are only going to continue to shrink in size. Will your fab partner help you transition from yesterday's technology to tomorrow's technology?

About Royal Circuit Solutions

We are a manufacturing facility dedicated to quick-turn prototype printed circuit board (PCB) fabrication for thousands of customers nation-wide, including engineers at some of the world's biggest tech companies. Founded in 1998, we've grown exponentially, honed our skills, invested millions in advanced high-tech equipment, and built an extraordinary team that is second-to-none. We'll manufacture your boards right, giving you high-speed service with precision quality. We're one of Silicon Valley's best kept secrets. Give us a try and contact us today at (831) 636-7789 or email us at sales@royalcircuits.com. To learn more about us, visit www.royalcircuits.com.