



Royal Circuit Solutions  
21 Hamilton Ct, Hollister, CA 95023

(831) 636-7789  
[www.RoyalCircuits.com](http://www.RoyalCircuits.com)

ROYAL CIRCUIT  
SOLUTIONS

# DESIGN TIPS AND COST-SAVING TRICKS

Our customers often ask how to save money and time on their designs. We created a webinar to answer questions and talk about some of the common issues we see in the sales office and on the production floor.

DESIGN TRICKS & COST SAVING TIPS

With  
THE GRACIA BROTHERS

TUESDAY  
JANUARY 28TH  
11 A.M. PST

ELIJAH RIKI

ROYAL CIRCUIT SOLUTIONS

The graphic features a dark background with a circuit board pattern. It includes two circular portraits of men, Elijah and Riki, with their names below them. The text is in a mix of serif and sans-serif fonts, with 'DESIGN TRICKS & COST SAVING TIPS' in a large, bold, serif font at the top. The date and time are in a large, bold, sans-serif font. The Royal Circuit Solutions logo is in the bottom right corner.

The Gracia brothers, Elijah and Riki, are two of Royal's most valuable employees. You can a recording of the webinar on our website [www.royalcircuits.com](http://www.royalcircuits.com) to learn even more tips and tricks!

## FABRICATION STEPS

When a customer orders a printed circuit board, one of our engineers creates a "traveler" that lists each step required to fabricate that board. If you want to bake a cake you need a recipe, if you want to make a PCB you need a traveler.



# ROYAL CIRCUIT SOLUTIONS



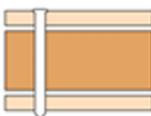
**HOT!**



PAGE 1 OF 4  
TRAVELER 1 OF 1

IPC CLASS 2

CUSTOMER: JOE DOE	DUE DATE: 3/3/2020
LAYER COUNT: 2	RELEASE DATE: 3/1/2020
SM COLOR: CYAN 	QTY BOARDS DUE: 2
BOARD SIZE: 19" x 22"	QTY BOARDS REL: 2
PANEL SIZE: 21" x 24"	OVERALL THICKNESS: 62

LAYER	BASE CU/PLT	THICK	TYPE	STACKUP	MATERIAL
SILKSCREEN		0.0			TAIYO-SS - WHITE
SOLDERMASK		0.6			TAIYO-SM-CYAN
LYR 1	0.5 Oz / STD	1.8	S		KB-6167-59.0 MILS
CORE		57.8			
LYR 2	0.5 Oz / STD	1.8	S		
SOLDERMASK		0.6			TAIYO-SM - CYAN
SILKSCREEN		0.0			TAIYO-SS - WHITE

<b>1</b> DRILL - 63	VERIFY MATERIALS	<input type="checkbox"/> ACC	<input type="checkbox"/> REJ
<b>2</b> DEBURR - 65	DEBURR DESMEAR	<input type="checkbox"/> ACC	<input type="checkbox"/> REJ
<b>3</b> DM RESISTANCE	DR	<input type="checkbox"/> ACC	<input type="checkbox"/> REJ

This mock-up of a traveler shows the first three steps of a PCB manufacturing process.

Each additional step in the process takes time, manpower, materials, and adds risk to the process. So to reduce the cost of your board, try to prevent unnecessary steps in the traveler. For example - microvias might add 9 additional steps and buried vias add 9-11 additional steps.

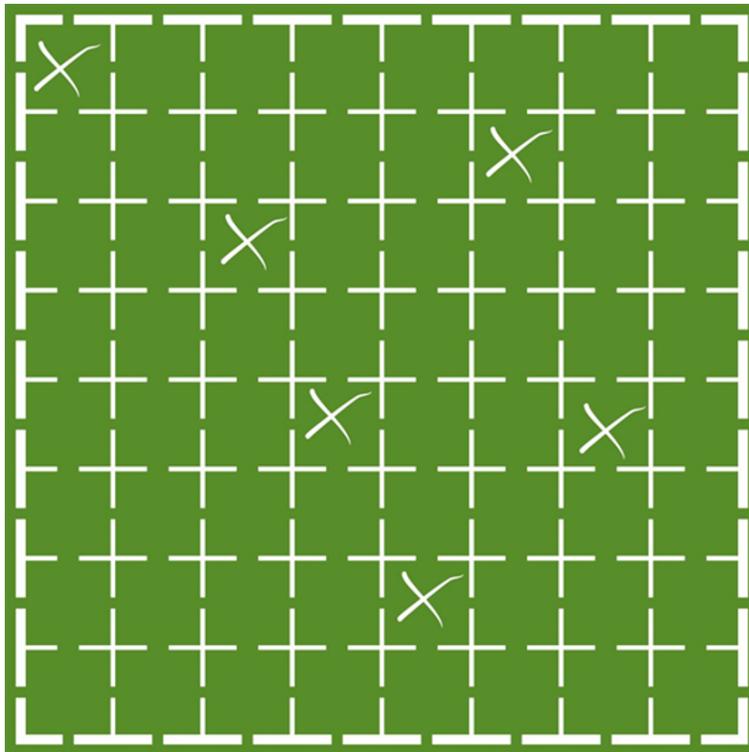


## ROYAL CIRCUIT SOLUTIONS

### MANAGING RISK

The processes and materials used in manufacturing are not perfectly uniform. So there's a bit of uncertainty in everything that we do. If we are making a board with a 25-page traveler, and something fails on page 23, 8 steps from the end, we have to throw the board in the trash and start over. We're very good at what we do, so that doesn't happen too often, but we have to include a risk-factor with each board. And very risky processes will increase the cost of your board.

I remember one customer a few months ago ordered a 16-layer board with multi-depth back-drilled vias on both sides of the board. The hard part on that job was the multiple drilling depths over a very large board. I think we spent three days making and remaking that board to get it just right.



We test each and every board that we produce. So if there is a problem, we are going to cross-out that board and not deliver it to the customer.

Boards are combined in a panel. If any board fails inspection, that board will be rejected.

The overall number of boards per panel is called the yield - and some decisions, such as very narrow trace and space or running copper to the edges of a board are going to decrease the yield.

### IPC CLASS 2 AND CLASS 3

We don't have a class 2 production line. All of our processes are designed for class 3 production. If you order Class 3, and some of the boards on your panel fall short of even one IPC specification, those specific boards are crossed out from your panel, even though they are still fully functional boards.



## ROYAL CIRCUIT SOLUTIONS

You aren't losing connectivity or impedance control on the class 2 boards. You are losing longevity - these boards will fail a few years before the other boards. For prototypes and respins, it's simply not an issue.

For prototypes, class 2 boards are usually sufficient for your design. Aerospace, Automotive, Downhole, and other High-Reliability Production situations require Class-3 design. An Arduino shield will work just fine as a class 2. Remember - we're aiming for class 3 regardless, so even if you order a class 2 board, there's a good chance you will get a class 3 product.

Here's another cost-saving tip- you can tell your board supplier you want certain design aspects to be class 3 while others can be class 2. Maybe vias are really important to you - then put teardrops on your pads and tell the supplier they must be class 3.



Our hot-oil lamination press permanently bonds the various layers of a PCB.

### Material Choice

The cost of your board is affected by the choice of materials in your design as well as the number of times we have to laminate it. We will talk about vias in a bit - but adding microvias, blind, or buried vias to your design requires additional lamination cycles. Multiple lamination cycles are very, very risky because the layers can shift. And when risk goes up, the price goes up.



ROYAL CIRCUIT  
SOLUTIONS

## STACKUP SYMMETRY

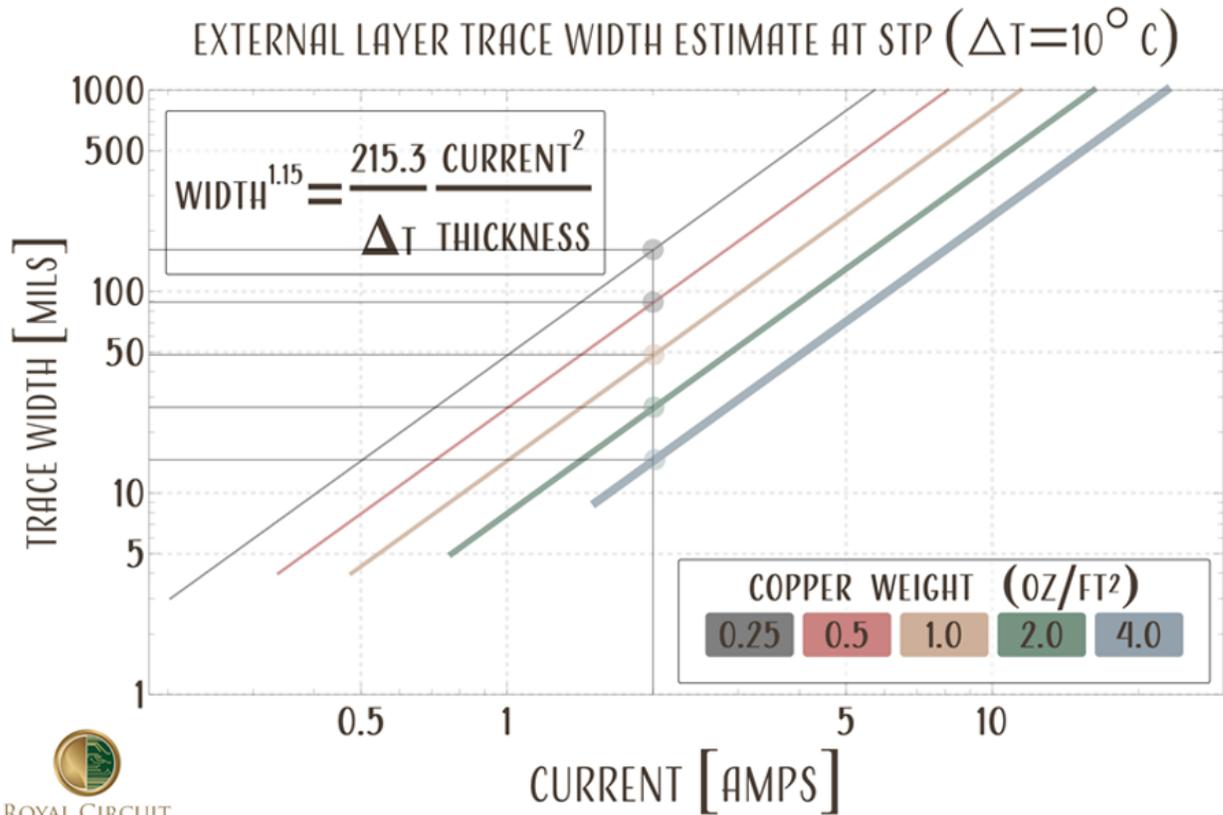


Your stack up should have symmetric usage of copper and dielectric materials. If you use an asymmetric stack up, the board will warp. Even if we can correct the movement during fabrication, it will absolutely warp during assembly - this is caused by differences in the coefficient of thermal expansion of the various materials. Pick and place machines aren't designed to place parts on bent boards, and even if you manage to get them on the board, the variable expansion will cause the solder connections to break during thermal cycling.

We have up to 4-oz copper on the shelves ready to go. If you need heavy copper on the interior layers of your PCB, try to use the foil that is already manufactured to that thickness - do not call out additional plating if you are trying to save money. We will always have to electroplate outer layers of boards to form vias - so plate there if you need to increase thickness of some nets. But if you ask us to plate interior layers, or create a board with excessive plating thickness, your board is going to take a long time to fabricate.

## HEAVY COPPER

Heavy Copper is used to carry high-current traces. As a rule-of-thumb, any trace that carries more than 500 mA deserves some attention since you can't run high currents through minimum-sized traces.



This graph shows the expected current-carrying capacity of a trace on an exterior layer. See our article on Copper Trace to learn more.

Here's an example from one of our design charts based on IPC-2152. 2 Amps of current will cause a temperature increase of  $10^{\circ}C$  in  $\frac{1}{4}$ -ounce 200-mil trace. But that same current can be transported by a 4-ounce 20-mil trace.

Use the allowable **temperature increase** along with the **expected current** to **determine the copper weight and trace width**. Contact us for more resources on that - we have a whole webinar on using heavy copper.

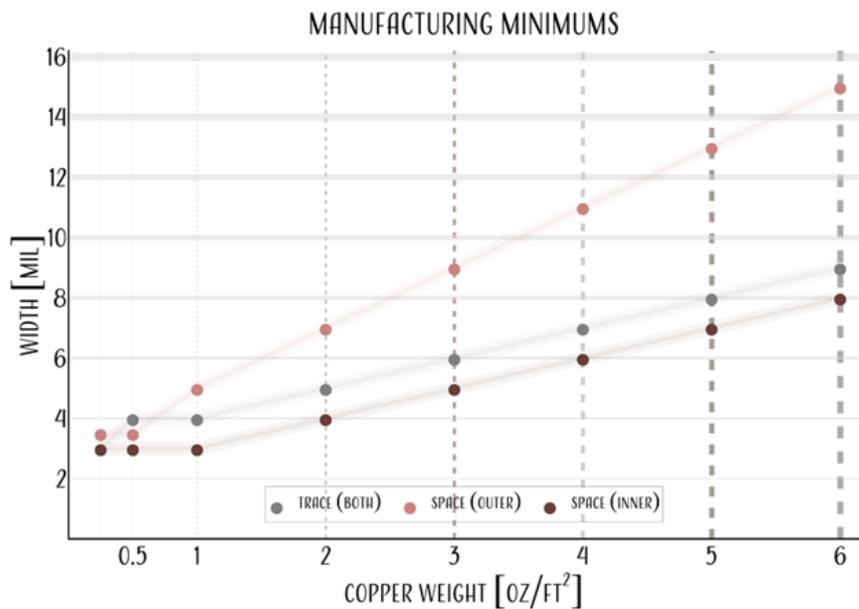


ROYAL CIRCUIT  
SOLUTIONS



ROYAL CIRCUIT  
SOLUTIONS

COPPER WEIGHT [OZ/FT <sup>2</sup> ]	TRACE (BOTH) [MIL]	SPACE (OUTER) [MIL]	SPACE (INNER) [MIL]
0.25	3	3.5	3
0.5	4	3.5	3
1	4	5	3
2	5	7	4
3	6	9	5
4	7	11	6
5	8	13	7
6	9	15	8



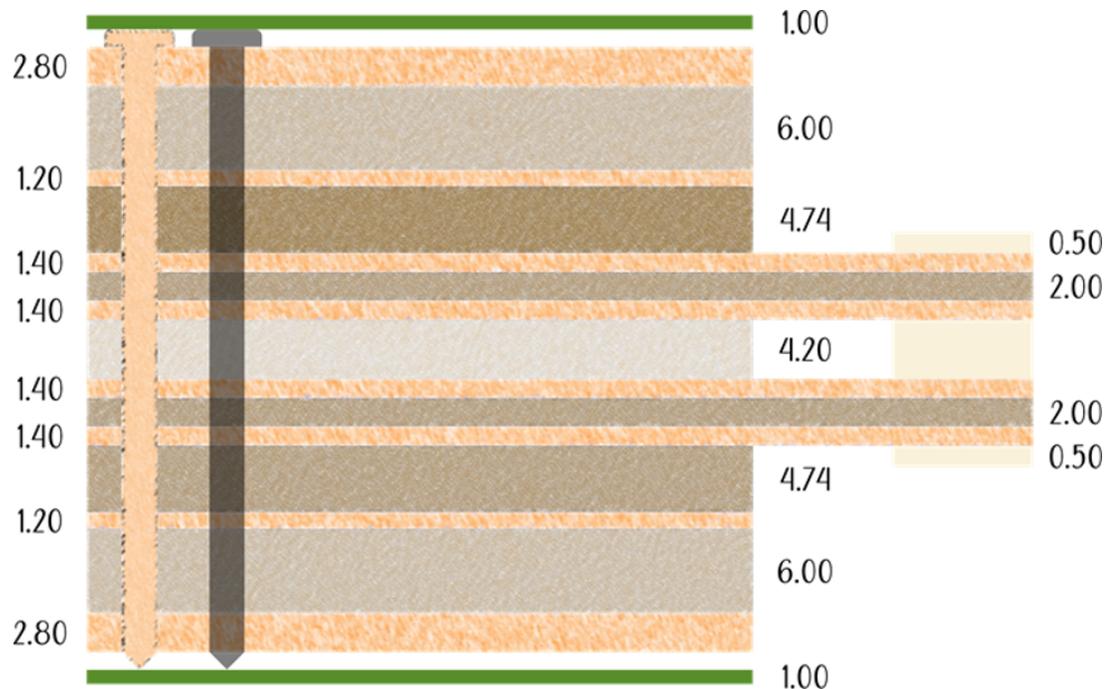
Our minimum trace and space guidelines are not the absolute manufacturing minimums. They are the minimums that provide a 95%-100% yield. We can push the limits further at the expense of yield.

After you select your copper weight, find out what your manufacturers trace/space minimums are. For example, it's simply not possible to have a 3-mil trace in 4-oz copper (5.6 mil). The aspect ratio would be too high, and the trace would roll over onto the board. We need trace and space minimums to ensure that the traces are etched all the way down to the dielectric and ensure that the copper is thick enough at the bottom.

Also remember that if you are asking the manufacturers to exceed their preferred minimums, they can likely do it - but at the expense of the yield. Sure, they might be able to etch 1 mil traces in ¼-ounce copper, but maybe the yield will be 40% -- that drives the cost up.



## STACKUP SUGGESTIONS



This fictional stackup for a rigid-flex PCB shows the expected thickness of the various layers.

Your layer stackup isn't just the copper and dielectric. We need to use adhesives and stiffeners as well. So if you are designing a rigid-flex PCB for a precise thickness or bend-radius, be sure to talk to your board house about what the final thickness will be.

A common misconception is that thicker boards cost more than thinner boards. That's not necessarily true. Very thin boards can be quite expensive due to special handling and machining requirements. For example, anything less than 14 mils thick must be laser drilled, which is a significant price increase.

So how can you save money on your design? Go with industry standards 31, 62, and 93 mil boards (0.8 mm, 1.6 mm, 2.4 mm for our friends across the pond).

If you are able to use one of our pre-approved stackups then we can combine your order with some of our other customer's orders and run them all through at the same time.

For Royal, that's 62 mil board thickness, HASL or ENIG finish, 1oz copper, green LPI and white silkscreen.



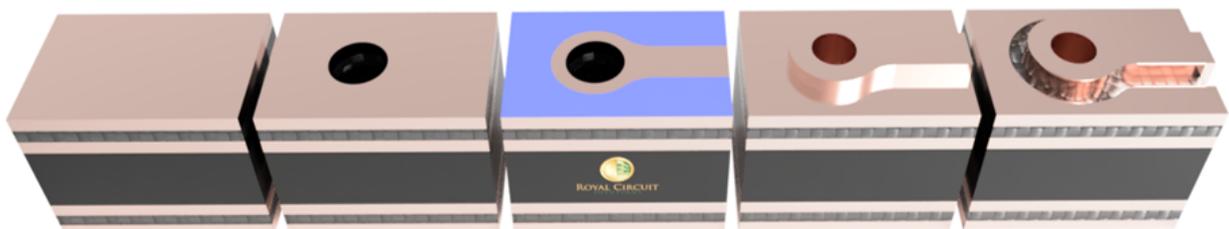
# ROYAL CIRCUIT SOLUTIONS

Layer	Base CU / Plt	Thick	Type	Stackup	Subs	Imp	Material	Dk	Df
Silkscreen		0.00					Taiyo-SS - White		
Soldermask		0.60					Taiyo-SM - Green	2.70	0.033
Lyr1	0.5oz / Std	1.80	S						
Prepreg		7.94					370HR - 2x1080/1x106	3.82	0.025
Lyr2	1oz	1.20	P						
Core		39.00					370HR - 39.0mils	4.71	0.021
Lyr3	1oz	1.20	P						
Prepreg		7.94					370HR - 1x106/2x1080	3.82	0.025
Lyr4	0.5oz / Std	1.80	S						
Soldermask		0.60					Taiyo-SM - Green	2.70	0.033
Silkscreen		0.00					Taiyo-SS - White		

Our standard 4-layer board stackup uses 1 oz copper on all layers (after plating), green solder-resist mask, and white silkscreen.

## VIAS

Vias are made after a stack or sub-stack has been assembled. First, we drill a hole through all the layers in the board. Then we plate the electroplate additional copper onto the board which thickens the outer layers and forms the via walls. Finally, we etch away copper to create air-space between the nets.



Vias are formed by first drilling a hole, then plating additional copper atop and inside the hole. Finally, copper is etched away from the perimeter to isolate the net.

We use this same process for through-hole vias, blind-vias, and buried vias. In fact, through-hole vias become blind vias or buried vias only after we laminate additional layers onto the board. To keep costs low, keep the lamination cycles to a minimum. It's better to use a blind-vias in a board that requires two lamination cycles than to use stacked microvias in a board that requires four lamination cycles.

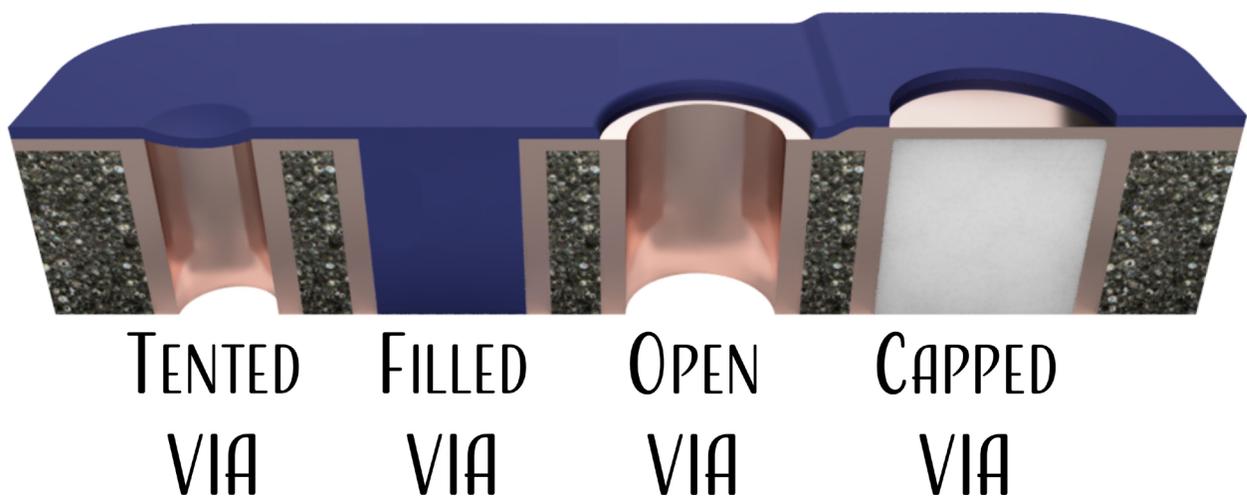


## ROYAL CIRCUIT SOLUTIONS

If you add blind vias, buried vias, or add them at different depths, we will have to make your board in several stages - that increases the risk, the turn-time, and the cost of your project.

If you are using a via in the middle of your board, you can leave it open, or tent it by removing the hole in your LPI soldermask layer.

If your board is going into a high-reliability environment such as aerospace or downhole, you should consider using a LPI soldermask fill. There we basically force additional LPI into the hole to provide some measure of protection against moisture and corrosives.



### SUMMARY

This is just a brief list of ways you can create a better design for less money. If you'd like to know more, please contact our engineering department at 1-831-636-7789.

### About Royal Circuit Solutions

Royal Circuit Solutions is a company focused on quick-turn, prototype printed circuit board manufacturing. The company specializes in next day turns and complex PCB boards including rigid, rigid-flex and flex. Royal Circuit Solutions provides fast quotes and high quality to meet the PCB needs of all design engineers. For more information visit [www.royalcircuits.com](http://www.royalcircuits.com) or call 1-831-636-7789.