

Quick-teck PCB Standard Specification

If your PCB falls within our standard PCB specification, you can get an instant price by online order form:

<http://www.quick-teck.co.uk/onlinequote/onlinequote.php>

In the same way, you can place the order online by online order form:

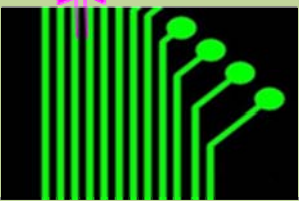
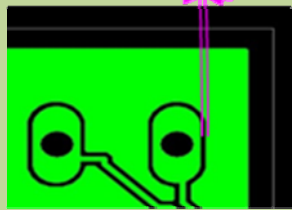

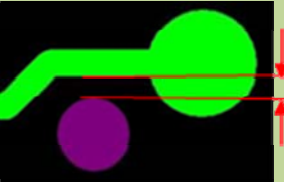
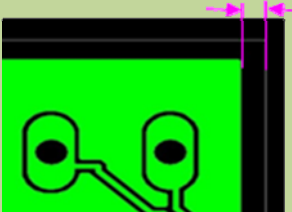
<http://www.quick-teck.co.uk/order/takeorder.php>

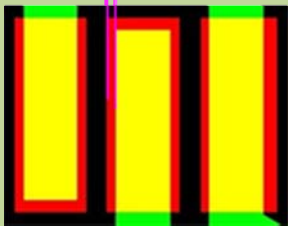
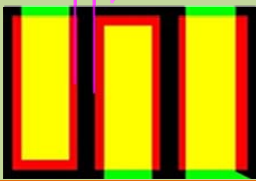
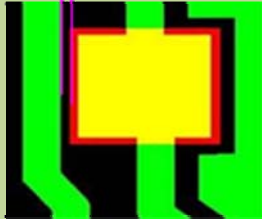
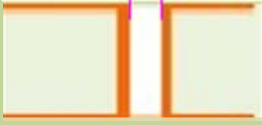
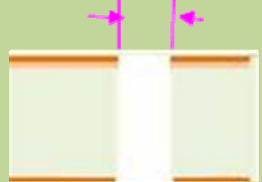
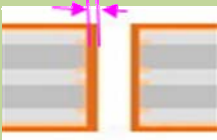
If the requirements do not conform to our standard PCB specification, we can still probably satisfy your order. However you will need to use the fast order form to get manual quote. We aim to reply to you within 2 working hours.

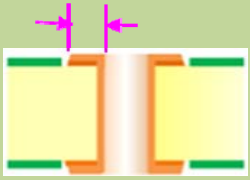
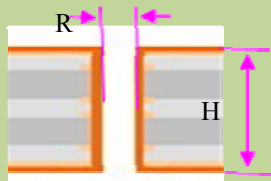
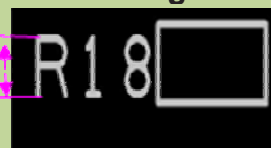

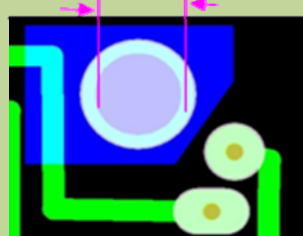
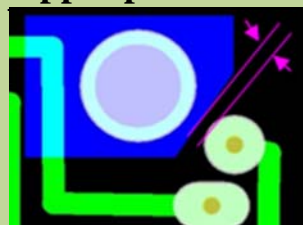
http://www.quick-teck.co.uk/fastquote/fastPCB_quote.php

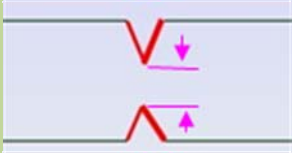

Catalogue	Specification	
PCB Dimensions	Minimum length/width	10mm
	Maximum length/width	500mm
	Additional restriction	1. For any PCBs: length +width ≤ 900mm; 2. If PCB thickness is 0.4 to 0.8mm, the maximum dimension is 200x300mm.
PCB thickness	Minimum	0.4mm
	Maximum	2.4mm
Base material	Fr4 tg-130, FR4 High tg	
Copper weight	0.5oz, 1oz, 2oz, 3oz.	
Surface finish	HASL, HASL (lead free), Electrolytic Gold, OSP, ^[1] Electroless Nickel/Immersion Gold.	
Warp & Twist	≤ 1%	
Trace width	≥ 6mil (0.15mm)	



<p>Trace clearance</p> 	$\geq 6\text{mil (0.15mm)}$	
<p>Distance between copper trace and copper flood</p> 	$\geq 8\text{mil (0.2mm)}$	
<p>Distance between copper trace and V-cut theoretical scoring axis</p> 	Minimum	20mil (0.5mm)
	Recommended	$\geq 24\text{mil (0.6mm)}$
<p>Distance between NPTH and copper trace</p> 	Minimum	7mil (0.18mm)
	Recommended	$\geq 10\text{mil (0.25mm)}$
<p>Distance between copper trace and board edge</p> 	Minimum	8mil (0.2mm)
	Recommended	$\geq 10\text{mil (0.25mm)}$

<p>Solder mask annular ring</p> 	Minimum	2mil (0.05mm)
<p>Yellow: exposed pad; Green: trace covered by solder mask; Red: exposed base material; Black: base material covered by solder mask;</p>	Recommended	≥3mil (0.08mm)
<p>Solder mask bridge</p> 	Minimum	10mil (0.25mm)
<p>Distance between solder mask clearance and copper trace</p> 	Minimum	3mil (0.08mm)
	Recommended	≥4mil (0.1mm)
<p>Drill size (PTH)</p>  <p>(NPTH)</p> 	Minimum	10 mil (0.25mm), tolerance : PTH+/-3mil NPTH:+/-2mil
	Step	1mil. If the drill size larger than 80mil, step value is 2mil (e.g., 82mil, 84mil, 86mil...).
	Maximum	236mil (5.9mm)
<p>Platted wall thickness</p> 	≥0.8mil (0.02mm)	

<p>Copper annual ring</p> 	<p>Minimum</p>	<p>0.5oz: $\geq 6\text{mil}$; 1.0oz: $\geq 8\text{mil}$; 2.0oz: $\geq 10\text{mil}$; 3.0oz: $\geq 10\text{mil}$;</p>
<p>Plating Aspect Ratio (H:R)</p> 	<p>Maximum</p>	<p>10:1</p>
<p>Silkscreen height/width</p> 	<p>$\geq 32/32\text{mil}$ (0.8mm)</p>	
<p>Silkscreen line width</p> 	<p>$\geq 5\text{mil}$ (0.13mm)</p>	
<p>Maximum hole to be plugged with peelable mask</p> 	<p>80mil (2.0mm)</p>	
<p>Distance between peelable mask and copper pad</p> 	<p>20mil (0.5mm)</p>	

<p>Core thickness after scoring process</p> 	<p>Minimum: 4mil (0.1mm), normal: 1/3 board thickness</p>
<p>V-scoring</p> 	<p>30°, 45°, 60°. Default is 45°</p>

[1]. Extra charge maybe applied if ENIG finishing area covers more than 75% proportion of the whole PCB surface.