
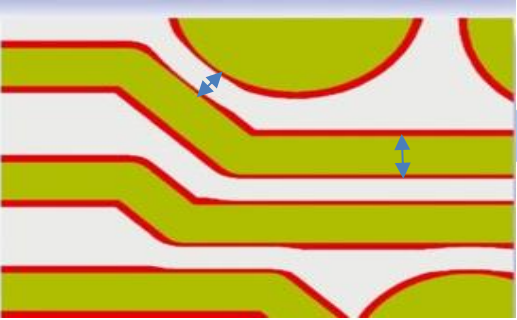

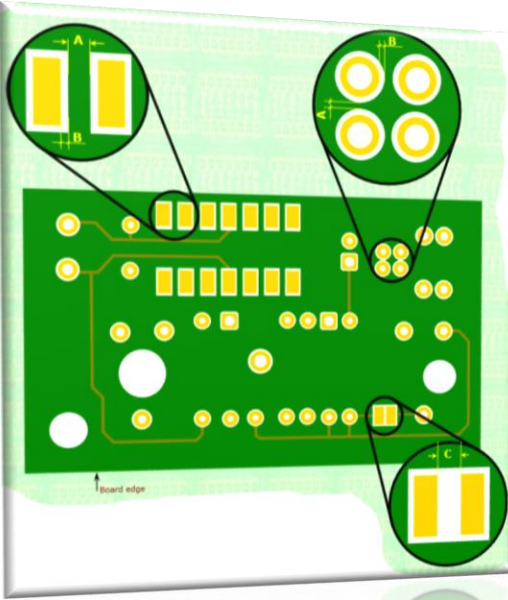
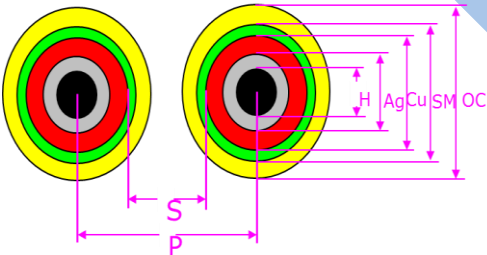
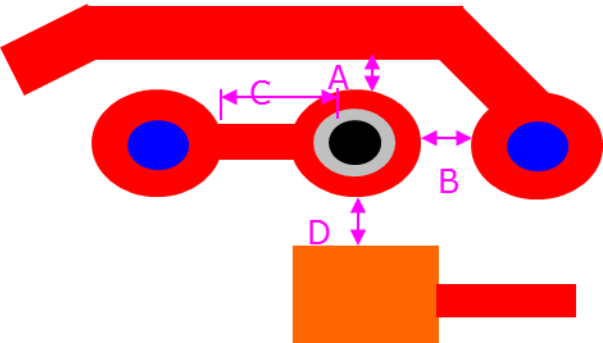
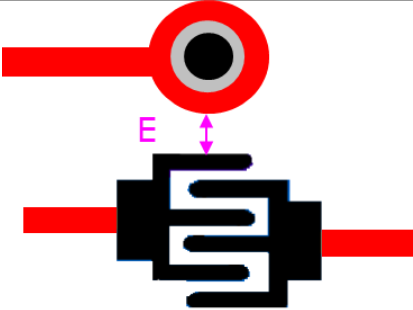
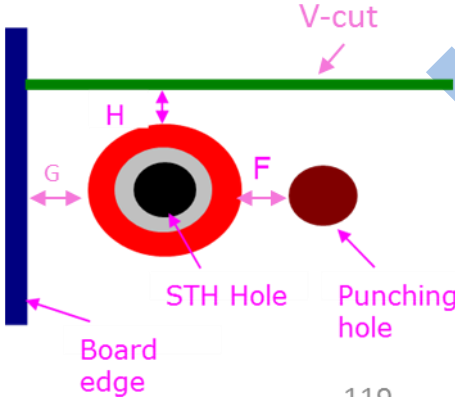


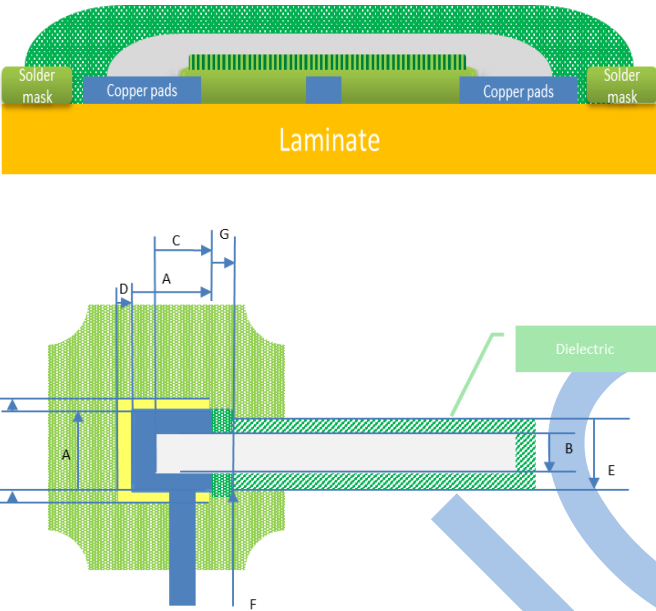


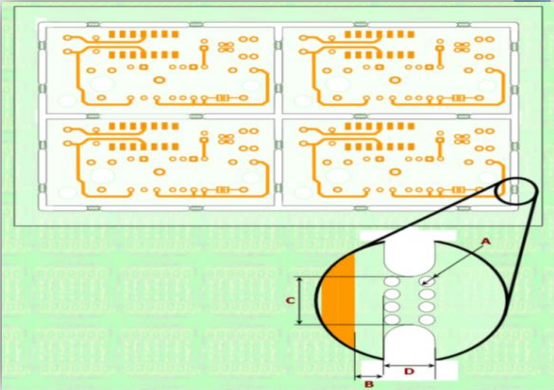
ICAPE General capability for Single side & polymer pcb

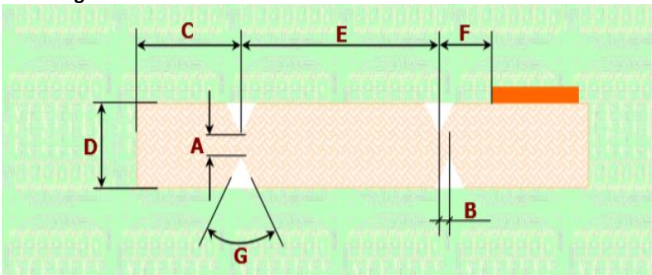
Item	Description	Sym.	Standard	Advanced
Raw material 	FR-1, FR-2 FR-4 CEM-1, CEM-3	Thickness mm	1.6mm 0.8 ~1.6mm 1.6mm	N/A 0.4 up to 3.2 N/A
	Max.board dimension	mm	610 x510	On request
Copper Thickness	Thickness of the copper on each layer of the pcb	µm or Oz	17.5µm => 0.5 oz (H/0) 35 µm => 1 oz (1/0) 70 µm => 2 oz (2/0)	105µm =>3 oz
Copper Line/Space 	 Distance line to line/pad - Line Width	µm	>180µm by screen printing	<180µm by Photo Exposure

Item	Description	Sym.	Standard	Advanced
Solder Mask 	A- Web		$\geq 0,25\text{mm}$	0,2
	B- Clearance		0,2	0,05
	C- web between Surface mount pads			$\geq 0,10\text{mm}$
Silver Through Hole 	Best Through Hole size (H)	ϕ	0.6~0.7mm	0.4
	Hole Spacing (P)	mm	$\geq 2\text{mm}$	$\geq 1.5\text{mm}$
	Land Spacing (S)	mm	$\geq 0.5\text{mm}$	$\geq 0.3\text{mm}$
	Copper Land diameter(Cu)	ϕ	$\geq 1.5\text{mm}$	$\geq 1.2\text{mm}$
	Solder mask opening diameter(SM)	ϕ	$\geq 1.6\text{mm}$	$\geq 1.3\text{mm}$
	Silver Land diameter(Ag)	ϕ	$\geq 1.1\text{mm}$	$\geq 0.9\text{mm}$
	Over coat diameter(OC)	ϕ	$\geq 2.3\text{mm}$	$\geq 2\text{mm}$

Item	description	Sym.	Standard	Advanced
Silver Through Hole 	A Land to trace	mm	$\geq 0.3\text{mm}$	
	B Land to Pad for different net	mm	$\geq 0.5\text{mm}$	
	C Land to Pad for same net	mm	$\geq 0.5\text{mm}$	
	D Land to SMD pad	mm	$\geq 0.5\text{mm}$	
	E Land to Carbon Ink	mm	$\geq 1\text{mm}$	
	F Land to punching Hole	mm	$\geq 1.5\text{mm}$	
	G Land to board edge	mm	$\geq 2\text{mm}$	
	H Land to V-cut	mm	$\geq 1\text{mm}$	

Item	description	Sym.	Standard	Advanced
Copper Through Hole	Hole Resistance	Ω	$\leq 100\text{m}\Omega/\text{hole}$	
	Best Through Hole size	Φ	$0.7 \pm 0.1\text{mm}$	
	Hole Resistance	Ω	$< 50\Omega/\text{hole}$	
Single Side with conductive jumpers 	A min copper Area B Min Width of Silver/Carbon crossover C Overlaying strap on Copper D Window SM Vs Copper area (3 sides only) E Width of Dielectric under Crossover F Distance between Dielectric & copper Area G Solder mask covers Copper area		1,6mm x 1,6mm 1mm \pm 20% 0,8mm 0,1mm (B) +1mm 0 0,2	
Carbon Printing	Min.carbon track width		0.4mm	
	Min.carbon track space		0.4mm	

Item	description	Sym.	Standard	Advanced
CNC	Drilling <ul style="list-style-type: none"> Min Hole diameter 		0,45mm±0,05	
	Routing <ul style="list-style-type: none"> Mini radius for inside corner Mini distance between edge and hole edge Min distance between trace and routing 		1 mm ±0.2 1.27mm 0.3mm	
Punching	<ul style="list-style-type: none"> Min Hole diameter Mini distance between edge and hole edge Min distance between trace and routing 		0,65 ±0,1 2.54mm 0.5mm	
Breakaway tabs 	A Drilled Hole B Conductive Pattern C Web D Routing width Space between 2 breakaway tabs		0,7mm 0,25mm 3mm 2mm 75mm	60mm

Item	description	Sym.	Standard	Advanced
V-scoring 	A Web		0.4 ± 0.1	
	B Displacement of score		0 ± 0.3	
	C scoring to board edge		$5\text{mm} \pm 0.1$	
	D Pcb overall Thickness		0.8up to 3.2mm	
	E Scoring to scoring		5mm	3mm
	F Scoring to conductive pattern		1.2mm	
	G Angle of scoring		$30-45^\circ$	