

Design Guide and CAD Parameters Standard

		Standard 1	Standard 2	Notes
	Via (Internal / External Layers)	<p>Land Diameter $\phi 0.60$</p> <p>Drill Diameter $\phi 0.30$</p> <p>Open Space $\phi 0.90$</p> <p>Dummy Pad</p>	<p>Land Diameter $\phi 0.50$</p> <p>Drill Diameter $\phi 0.30$</p> <p>Open Space $\phi 0.80$</p> <p>Dummy Pad</p>	
	Via Pitch Pitch (Internal / External Layers)	<p>Land Diameter: $\phi 0.60$</p> <p>0.75mm Min</p>	<p>Land Diameter: $\phi 0.50$</p> <p>0.65mm Min</p>	
	Surface Via Hole (Internal / External Layers)	<p>Land Diameter $\phi 0.40$</p> <p>Drill Diameter $\phi 0.15$</p> <p>Open Space $\phi 0.70$</p> <p>Dummy Pad</p>	<p>Land Diameter $\phi 0.35$</p> <p>Drill Diameter $\phi 0.15$</p> <p>Open Space $\phi 0.65$</p> <p>Dummy Pad</p>	Surface via hole (SVH) can be used to connect most external layers and adjacent layers electrically.
	Surface Via Hole Pitch (Internal / External Layers)	<p>Land Diameter: $\phi 0.40$</p> <p>0.55mm Min</p>	<p>Land Diameter: $\phi 0.35$</p> <p>0.50mm Min</p>	
	IVH Interstitial Via Hole (Internal Layers)	<p>Land Diameter $\phi 0.60$</p> <p>Drill Diameter $\phi 0.30$</p> <p>Open Space $\phi 0.90$</p> <p>Dummy Pad</p>	<p>Land Diameter $\phi 0.50$</p> <p>Drill Diameter $\phi 0.30$</p> <p>Open Space $\phi 0.80$</p> <p>Dummy Pad</p>	
	IVH Pitch (Internal Layers)	<p>Land Diameter: $\phi 0.60$</p> <p>0.75mm Min</p>	<p>Land Diameter: $\phi 0.50$</p> <p>0.65mm Min</p>	
	Minimum Trace (Internal Layer)	<p>Trace 0.12mm</p> <p>Trace Space 0.12mm</p>	<p>Trace 0.10mm</p> <p>Trace Space 0.10mm</p>	Regarding adjacent Via, refer to "Via Pitch" section.
	Minimum Trace (External Layer)	<p>Trace 0.12mm</p> <p>Trace Space 0.12mm</p>	<p>Trace Width 0.10mm</p> <p>Trace Space 0.10mm</p>	Regarding adjacent Via, refer to "Via" section. Gap = 0.15mm

Design Guide and CAD Parameters Standard

		Standard 1	Standard 2	Notes
	Board Edge Trace Clearance (External Layer) with Solder Mask			
	Board Edge Trace Clearance (External Layer) with Coverfilm			
	Board Edge Trace Clearance (Internal Layer)			
	Board-Cable Edge Area Trace Clearance (Internal / External Layers)			

Design Guide and CAD Parameters Standard

		Standard 1	Standard 2	Notes
Board Edge Solder Mask Clearance		0.3mm Min	0.15mm Min	
Board Edge Coverfilm Clearance				Coverfilm outline locates at the same area as board edge.
Pad Solder Mask Strip Between Pads		Space 0.10mm Min Solder Mask Width 0.15mm Min	Space 0.075mm Min Solder Mask Width 0.10mm Min	
Pad, Solder Mask, Trace Distance		0.1mm Min 0.1mm Min (*1)	0.075mm Min 0.075mm Min (*1)	(*1 Solder mask opening must be larger than pad.
Pad, Coverfilm, Trace Distance		0.3mm Min 0.3mm Min (*1)	0.25mm Min 0.25mm Min (*1)	(*1 Coverfilm opening is larger than pad.
Pad, Coverfilm, Trace Clearance		0.15mm Min 0.12mm Min 0.3mm Min (*2)	0.15mm Min 0.1mm Min 0.25mm Min (*2)	(*2 Coverfilm opening is smaller than pad.

