Capability and Design Guidelines

Approvals	ISO 9001:2008 Manufactured and Inspected to IPC600 up to and inc Class 3 UL RoHS Trusted Kitemark
CAD Data Format	RS274-X Gerber RS274-D (Gerber with aperture list) ODB++ Excellon MDA DXF Please contact us for other data formats.
Printed Circuit Types	Single Sided Double Sided Multilayer up to 40 Layers Flex, Flex Rigid & Multilayer Flex rigid up to 16 layers. High Temperature circuits for down Hole Applications. Microwave & RF Hybrid Dielectric combinations IMS Insulated Metallic Substrates Blind & Buried via's Controlled Impedance Buried Resistors Edge Plating Edge Plated Half Hole
Base Material Types	FR4, Isola, EMC, Panasonic, please contact for full list. Halogen Free, High TG FR4, BT Epoxy, Polyimide Rigid (Arlon), Polyimide (Kapton) Flex & Multi Flex Rigid(Du-Pont FR, LF&AP), LCP Flex (Rogers) IMS (Thermal Clad Metal Substrates), PTFE Microwave (Rogers & Arlon) Copper Invar Copper
Specialised Processes	Edge Plating. Half routed through plated holes. Resist filled via's. Buried Resistors. Resin Filled Via's

Capability and Design Guidelines

- **PCB DIMENSIONS.**

Maximum Circuit Size	**Double Sided& Multilayer:**	**508mmx571.5mm**
	Single Sided:	**508x720mm**
Maximum number of layers For Rigid Multilayer:		**40**
Maximum number of layers For Flex & Flex Rigid Multilayer:		**16**
Thickness Rigid Multilayer:		**0.4mm to 5.0mm**
Thickness Flex & Flex Rigid Multilayer:		**0.2mm to 3.8mm**
Flatness:		**1% of the longest edge**

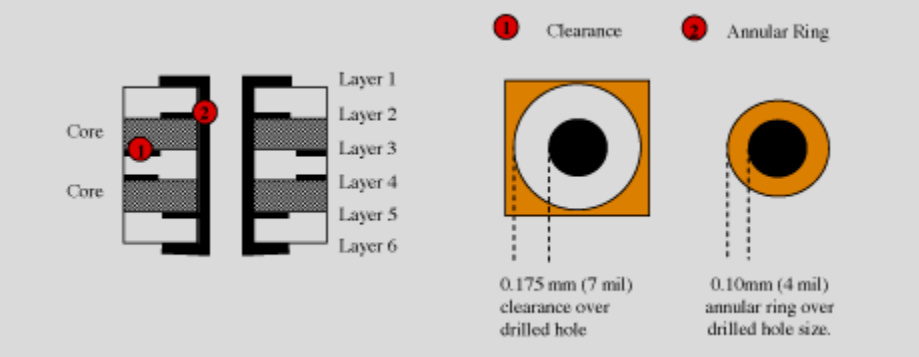
- **HOLES, PADS & CLEARANCES.**

Hole sizes:	**0.15mm to 6.35mm Larger holes available through Routing (please contact Sales)**
Smallest drilled hole:	**0.1mm (4Thou)**
Largest drilled blind hole:	**6.35mm (25.4Thou)**
Hole size tolerance:	**Plated +/- 0.05mm**
Hole Positional tolerance:	**+/- 0.025mm**

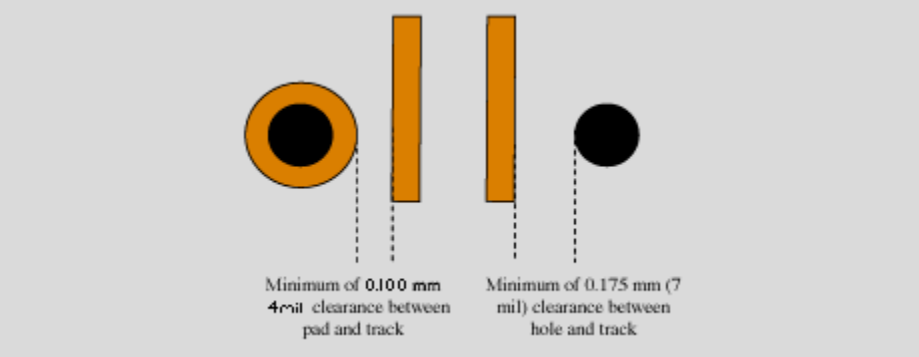
Capability and Design Guidelines

Annular Ring Clearances

Cross-section after plating



Track to Feature Clearances



- **TRACKS & GAPS.**

Start Copper	Minimum Track Outer Layers	Minimum Track Inner Layers	Minimum Gaps Outers & Inners
0.5oz (17.5µm)	0.100mm	0.100mm	0.100mm
1.0oz (35µm)	0.125mm	0.100mm	0.125mm
2.0oz (70µm)	0.250mm	0.225mm	0.200mm
3.0oz (105µm)	0.300mm	0.250mm	0.250mm

Capability and Design Guidelines

SOLDER RESIST AND COMPONENT IDENT

SOLDER RESIST

Type	Liquid photo-imageable
Colours	Green, Red, Blue, Black, Yellow, Purple, Pink, White and Clear
Nominal Thickness	35µm surface features & 12-15µm knee of track
Clearance around copper feature	0.075mm
Minimum web (dam) between pads	0.1mm

Component Identification	
Minimum Line Width	0.1mm
Ink Types	Ink Jet Printer (white only) Liquid photo-imageable (others Colours)
Colours	White (standard), Yellow, Black and Others

Capability and Design Guidelines

- **SURFACE FINISHES**

Immersion Gold over Nickel - Thickness 0.06 to 0.12µm Gold Over 3 to 5µm Nickel

Lead Free Hot Air Solder Level (LFHAL) - Thickness 4 to 40µm

Immersion Tin - Thickness 0.7 to 1.1µm

Silver - Thickness 0.25 to 0.4µm

OSP (organic) - monomolecular structure (1 atom thick)

Lead Hot Air Solder Level (HAL) - Thickness 4 to 40µm

- **EDGE CONNECTOR & KEY PADS.**

Gold edge connectors

Hard gold thickness 1, 2.5 or 5um

Keypads

Hard gold thickness 1, 2.5 or 5um

- **ROUTING**

Rout bit diameters 0.6 to 2.4mm Standard = 2.4mm

Dimensional tolerance +/- 0.1mm

- **SCORING**

Location tolerance +/- 0.2mm

Scoring web thickness (1.6mm PCB) 0.4mm +/- 10%

Jump scoring is available, this requires a routed clearance of 20mm each side of the jump

Capability and Design Guidelines

CONTROLLED IMPEDANCE.

At high frequencies PCB traces do not behave like simple connections, we need to ensure that signals are not degraded as they route around the PCB.

When should you consider impedance control?

- Fast digital applications
- ICT
- Data and signal processing
- RF communication

Tolerance:

+/- 10%