

	Technical	Id.- Nr.: CM-SO-010 Rev.: B Datum: 23.01.2008
--	------------------	--

 **Important Information!**

Base material

Standard Industry Quality FR4 1,55mm (+- 0,13mm)

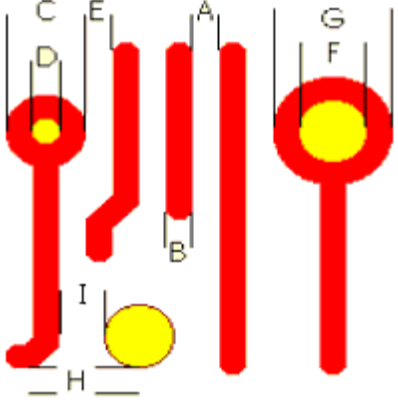
Additional Layerbuild info:

Copper (30 – 50 µm /layer), Soldermask (10 – 30µm / layer) + Plating on pads (11 – 15 µm)

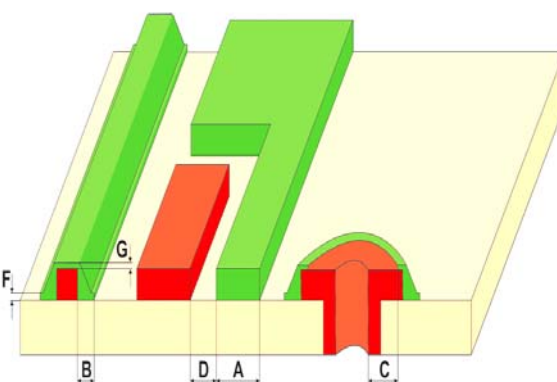
Surfaces


Surface	Layerbuild	Layer Thicknesses	Shelf-life	Soldering temperature	Remarks
Sn	HAL Lead free	SnCu0,7Ni	>= 10µm	> 12 Months	250 – 260°C

Design Overview

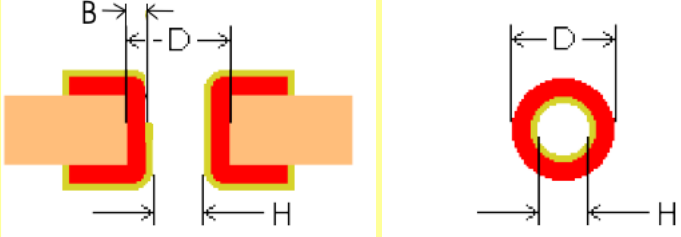
	<table> <tr> <td>Minimum Air-gap (A)</td> <td>0,15mm</td> </tr> <tr> <td>Minimum track width (B)</td> <td>0,15mm</td> </tr> <tr> <td>Smallest annular ring on vias (C-D)</td> <td>0,30mm</td> </tr> <tr> <td>Smallest annular ring on drills (G-F)</td> <td>0,40mm</td> </tr> <tr> <td>Minimum clearance for rout contour (H)</td> <td>0,50mm</td> </tr> <tr> <td>Minimum clearance for npt holes (I)</td> <td>0,50mm</td> </tr> </table> <p>Attention, in case of single-sided PCB's</p> <table> <tr> <td>Track widths and air-gaps (A-B) must be</td> <td>0,20mm</td> </tr> </table>	Minimum Air-gap (A)	0,15mm	Minimum track width (B)	0,15mm	Smallest annular ring on vias (C-D)	0,30mm	Smallest annular ring on drills (G-F)	0,40mm	Minimum clearance for rout contour (H)	0,50mm	Minimum clearance for npt holes (I)	0,50mm	Track widths and air-gaps (A-B) must be	0,20mm
Minimum Air-gap (A)	0,15mm														
Minimum track width (B)	0,15mm														
Smallest annular ring on vias (C-D)	0,30mm														
Smallest annular ring on drills (G-F)	0,40mm														
Minimum clearance for rout contour (H)	0,50mm														
Minimum clearance for npt holes (I)	0,50mm														
Track widths and air-gaps (A-B) must be	0,20mm														

Soldermask

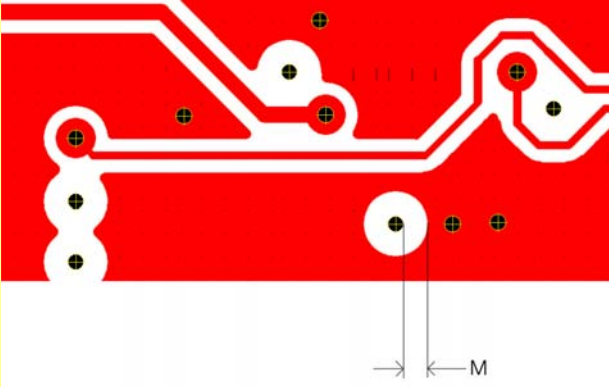
	<table> <tr> <td>Minimum Soldermask pad (A)</td> <td>0,15mm</td> </tr> <tr> <td>Minimum Soldermask width (B)</td> <td>0,075mm</td> </tr> <tr> <td>Minimum gap for drills (C)</td> <td>0,15mm</td> </tr> <tr> <td>Minimum gap on solder surfaces (D)</td> <td>0,075mm</td> </tr> <tr> <td>Soldermask layer thickness (E)</td> <td>0,02mm – 0,05mm</td> </tr> <tr> <td>Minimum soldermask layer-thickness over tracks (G)</td> <td>0,008mm</td> </tr> </table>	Minimum Soldermask pad (A)	0,15mm	Minimum Soldermask width (B)	0,075mm	Minimum gap for drills (C)	0,15mm	Minimum gap on solder surfaces (D)	0,075mm	Soldermask layer thickness (E)	0,02mm – 0,05mm	Minimum soldermask layer-thickness over tracks (G)	0,008mm
Minimum Soldermask pad (A)	0,15mm												
Minimum Soldermask width (B)	0,075mm												
Minimum gap for drills (C)	0,15mm												
Minimum gap on solder surfaces (D)	0,075mm												
Soldermask layer thickness (E)	0,02mm – 0,05mm												
Minimum soldermask layer-thickness over tracks (G)	0,008mm												

	Technical	Id.- Nr.: CM-SO-010 Rev.: B Datum: 23.01.2008
--	------------------	---

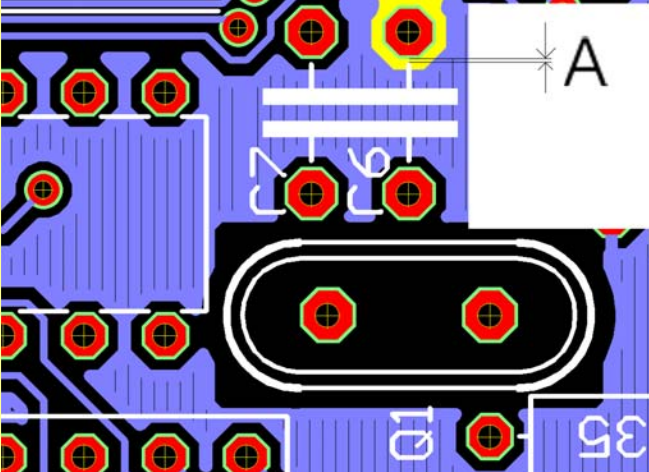
Through-hole plating

	<p>Minimum layer-thickness in the hole (B) 0,04mm (0,03mmCu + 0,010mmHAL)</p> <p>Tool diameter (D) Finished diameter after manufacture (H)</p>
---	---

Multilayer

	<p>Multilayer clearance recess on innerlayers(M) 0,5mm (Gap between Drill and Copper)</p> <p>Unfortunately blind and buried vias are not possible in PCB-POOL® but we will be happy to offer you a “NonPool” quote for this kind of technology, just contact our sales team.</p>
--	--

Silkscreen / Position print

	<p>Silkscreen will be clipped back (to a diameter of 0,1mm) from any corresponding soldermask pad in order to prevent pads being printed on (A)</p> <p>Recommended</p> <p>Mminimum script thickness 0,15mm Script height 2,00mm</p>
---	---