CIRCUITS IMPRIMÉS DE BELLÊME

TECHNICAL SKILLS – ROAD MAP

September 2011

OUTLINE

TECHNICAL SKILLS – ROAD MAP 1
1 Purpose 2
2 Overview and Roadmap 2
3 Types of PCB's 3
4 Size limits 3
4.1 Dimensions 3
4.2 Machining tolerances (routing) 3
4.3 Machining versus layout positionning (routing) 3
5 Currently used substrates : 4
6 PCB thickness limits 4
7 Copper base layer thickness (int or ext) 4
8 Minimum space between traces, and their tolerances (iT*) relative to copper base layer thickness 5
8.1 Inverse Copper etching (PTH PCB's external layer) 5
8.2 « Direct » etching (internal layers, PCB without PTH) 6
9 Drilling 7
9.1 Mechanical "Through" holes (platable / max drilled thickness) 7
9.2 Z axis controlled depth drilling 8
9.2.1 Ratio e/D and over drilling α 8
9.3 Minimum space " drilled hole / copper entity" (Δ) 9
9.3.1 General values 9
9.3.2 Special requirements 9
9.4 Minimum space "PTH/PTH" (drilled hole/ drilled hole gap) 9
10 Stack –up examples 10
11 Finishing layer 11
12 Miscellaneous 12

CIBEL

1 Purpose

This non-contractual document is intended to present and explain our current knowledge base. The various competencies and performances are classified into three categories . Actually, we use the term of "*Classical*" for traditional production and "*Technical*"for advanced production techniques. This document also includes the "*Special*"classification (very low volumes, particularly difficult to realize), for which the manufacturing process is not yet fully qualified because of a lack of experience . Therefore, whenever one or more circuit specification is issued from this "*Special*" classification , then the applications become restricted to prototyping, experimental and draft version PCB . It also mean that absolutely no NS3 circuit ("Class 3" from IPC A-600 , including EN9100 products) must present those "*Special*" charateristics unless the customer being informed of all the risks and formally accept the configuration.
***Nota**: When adding various and different "Technical" specifications to a circuit , the circuit classification may consequently change to "Special" categorie. Same change and attention may happen for a circuit having different " Special " specifications. In that particular case, together sales and technical competencies will align a conclusion and bring recommandations back to the customer.*

2 Overview and Roadmap

		2011- 2012			2013…	
	see §	« Classical »	« Technical »	« Special »	« Classical »	« Technical »
Max rigid PCB dimensions	4.1	650mm x 550mm	650mm x 550mm	650mm x 550mm	650mm x 550mm	650mm x 550mm
Max flexible PCB dimensions	4.1	370mm x 270mm	550mm x 270mm	550mm x 400mm	550mm x 270mm	550mm x 400mm
Max PCB thickness	6, 9.1	e ≤ 3,2mm	e ≤ 6mm	e > 6mm	e ≤ 4mm	e ≤ 6mm
Max layers count	-	e ≤ 24	e ≤ 40	n > 40	e ≤ 30	e ≤ 40
Min pre-preg thickness	6	90µm	60µm	45µm	90µm	45µm
Min drilled hole size	6, 9.1, 9.2	∅ ≥ 0,150mm	∅ ≥ 0,100mm	∅ < 0,100mm	∅ ≥ 0,150mm	∅ ≥ 0,100mm
Tolerance on Z axis drilled holes	-	iT ≥ 50µm	iT =35µm	iT <35µm	iT ≥ 40µm	iT = 30µm
Through holes Aspect Ratio	9.1, 9.2	R ≤ 10/1	R ≤ 15/1	R > 15/1	R ≤ 12/1	R ≤ 18/1
Z axis drilled holes (blind) Aspect Ratio	9.2.1	R ≤ 0,8 / 1	R ≤ 1 / 1	R > 1,1 / 1	R ≤ 1/1	R ≤ 1,1 / 1
Vias filling	-	pre-preg or plugging paste	pre-preg or plugging paste	pre-preg or plugging paste	pre-preg or plugging paste	pre-preg or plugging paste
Min. Line and spaces	8.2	≥ 110µm	≥ 75µm	< 75µm	≥ 100µm	≥ 70µm
Min "drilled hole / copper" distance	9.3	≥ 150µm	≥ 120µm	< 120 µm	≥ 150µm	≥ 120µm
Impedance control	-	±10%	±5%	±5%	±10%	±5%
Max sequential drilling	10	3	5	7	4	6
Max sequential lamination		2	4	6	3	4

CIBEL

3 Types of PCB's

	Classical	Technical	Special
Single layer	x	x	x
Double layers with or without PTH's	x	x	x
Multilayers	x	x	x
Flexible	x	x	x
Rigid-Flexible		x	x
Hi-Frequency		x	x
Multilayers with blind/burried vias		x	x
Impedance control		x	x
Heat sinks		x	x
Micro-vias (Mechnical drilling. Ø ≤ 0,100mm)		x	x
« Press-fit » PCB	x	x	x
Mixed substrates	x	x	x

4 Size limits

4.1 Dimensions

- Rigid PCB's max : 650 x 550mm , except for HAL finishing (TinLead or SN100C = 550 mm x 400 mm)
- Flexible / Rigid-flex max : 370 x 270mm
-

4.2 Machining tolerances (routing)

	« Classical »	« Technical »	« Special »
	iT (µm)	iT (µm)	iT (µm)
Rigid PCB	iT ≥ 200	200> iT ≥ 100	< 100
Flexible PCB	iT ≥ 300	300> iT ≥ 150	< 150

4.3 Machining versus layout positionning (routing)

	« Classical »	« Technical »	« Special »
	iT (µm)	iT (µm)	iT (µm)
Rigid PCB	iT ≥ 250	250> iT ≥ 125	< 125
Flexible PCB	iT ≥ 400	400> iT ≥ 200	< 200

5 Currently used substrates :

- FR4
- FR4 HTg
- Polyimides (including Kapton)
- BT Epoxy
- Substrates « Hyper » (of which RO 3…, RO 4…, Cuclad…., Diclad…, TMM… DUROÏD, etc…)

6 PCB thickness limits

	« Classical »	**« Technical »**	**« Special »**
Rigid “thick” PCB thickness	$e \leq 3{,}2$mm	$3{,}2 < e \leq 6$mm	$e > 6$mm
Rigid “thin” PCB thickness	$e > 100\mu m$	$e = 100\mu m$	$e < 100\mu m$
Flexible PCB thickness	$e \geq 50\mu m$	-	$e < 50\mu m$
Min inter-layer thickness (pre-preg)	$e \geq 90\mu m$	$90 > e \geq 60\mu m$	$e < 60\mu m$

7 Copper base layer thickness (int or ext)

	« Classical »	**« Technical »**	**« Special »**
Min copper base thickness (internal and external)	$e \geq 17{,}5\mu m$	$e = 9\mu m$ or $12\mu m$	$e = 5\mu m$
Max copper base thickness (internal and external)	$e \leq 70\mu m$	e 105µm or 210µm	$e > 210\mu m$

CIBEL

8 Minimum space between traces, and their tolerances (iT*) relative to copper base layer thickness

8.1 Inverse Copper etching (PTH PCB's external layer)

	Classical Products		Technical Products		Special Products	
Copper base thickness (µm)	Min. spacing between traces (µm)	iT	Min. spacing between traces (µm)	*iT*	Min. spacing between traces (µm)	*iT*
5	-	-	-	-	*60*	*10*
9/12	-	-	*80*	*20*	*65*	*15*
17,5	120	*30*	*90*	*25*	*70*	*20*
35	150	*40*	*120*	*35*	*90*	*25*
70	210	*60*	*150*	*45*	*110*	*30*
105	310	*100*	*200*	*70*	*150*	*50*
210	500	*150*	*320*	*130*	*250*	*100*

* iT = tolerance variation (ΔMin/Max) : 0/-x iT for **traces**
0/+x iT for the **spaces**
or + /- 1/2x iT for both **traces and spaces**

CIBEL

8.2 « Direct » etching (internal layers, PCB without PTH)

	Classical Products		Technical Products		Special Products	
Copper base thickness (µm)	Min. spacing between traces (µm)	iT	Min. spacing between traces (µm)	*iT*	Min. spacing between traces (µm)	*iT*
5	-	-	-	-	*50*	*10*
9/12	-	-	*75*	*10*	*55*	*10*
17,5	110	*20*	*90*	*15*	*60*	*15*
35	135	*35*	*105*	*30*	*80*	*25*
70	180	*50*	*140*	*40*	*110*	*30*
105	250	*100*	*180*	*60*	*150*	*50*
210	500	*150*	*320*	*130*	*250*	*100*

* iT = tolerance variation (ΔMin/Max) : 0/-x iT for **traces**
0/+x iT for the **spaces**
or + /- 1/2x iT for both **traces and spaces**

CIBEL

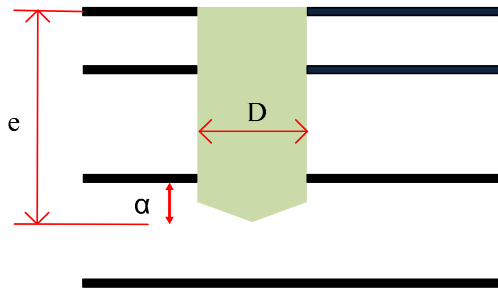
9 Drilling

9.1 Mechanical "Through" holes (platable / max drilled thickness)

Ø drilling	Max PCB thickness e (mm)	Up to number of drilled 17,5µm layers	Ratio e / Ø
0,060	0,8 0,6	2 4	13 10
0,080	1,6 1,4	2 6	20 18
0,100	2,0 1,6	2 6	20 16
0,125	2,4 2	2 6	19 16
0,150	2,6 2,2	2 8	17 15
0,175	2,8 2,4	2 10	16 14
0,200	3	→ 18	15
0,200 à 0,450	4	→ 24	20 à 9
0,500 à 0,750	6	→ 38	12 à 8
0,800 à 1	6,5	→ 46	8 à 7
Ø > 1	6,5	→ 46	/

CIBEL

9.2 Z axis controlled depth drilling



9.2.1 Ratio e/D and over drilling α

	Ratio
« Classical » products	0,8
« Technical » products	1
« Special » products	1,2

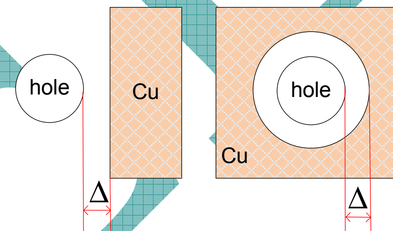
overdrilling α
$50\mu m < \alpha < 5\mu m$
$35\mu m < \alpha < 5\mu m$
$20\mu m < \alpha < 5\mu m$

CIBEL

9.3 Minimum space " drilled hole / copper entity" (Δ)

9.3.1 General values

- Δ mini recommended : 150µm - Conductive Anodic Filament (CAF) risk

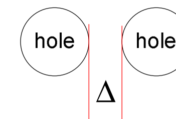


9.3.2 Special requirements

- Please contact us for needs between 150µm and 80µm distance (restrictions concerning duration and using conditions)
- Absolute min 80m

9.4 Minimum space "PTH/PTH" (drilled hole/ drilled hole gap)

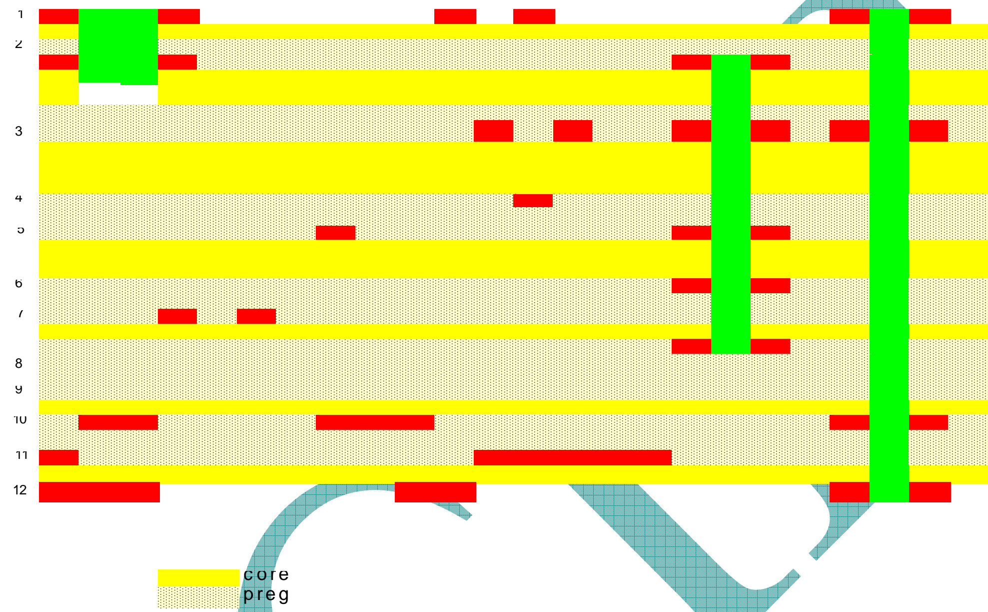
- Min Space recommended: 200µm (CAF!)
- Absolute min Δ : 100µm (with restrictions concerning duration and using conditions: CAF !!)



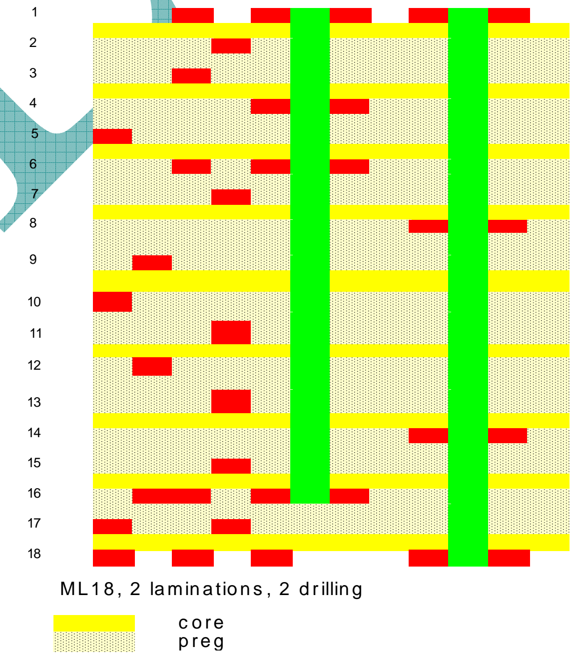
❖ Note: these values may change depending on the substrate material. Please contact us for any substrates else than FR4 and FR4 highTg

CIBEL

10 Stack –up examples



ML12, 2 laminations, 3 drilling (with one Z axis controlled drilling

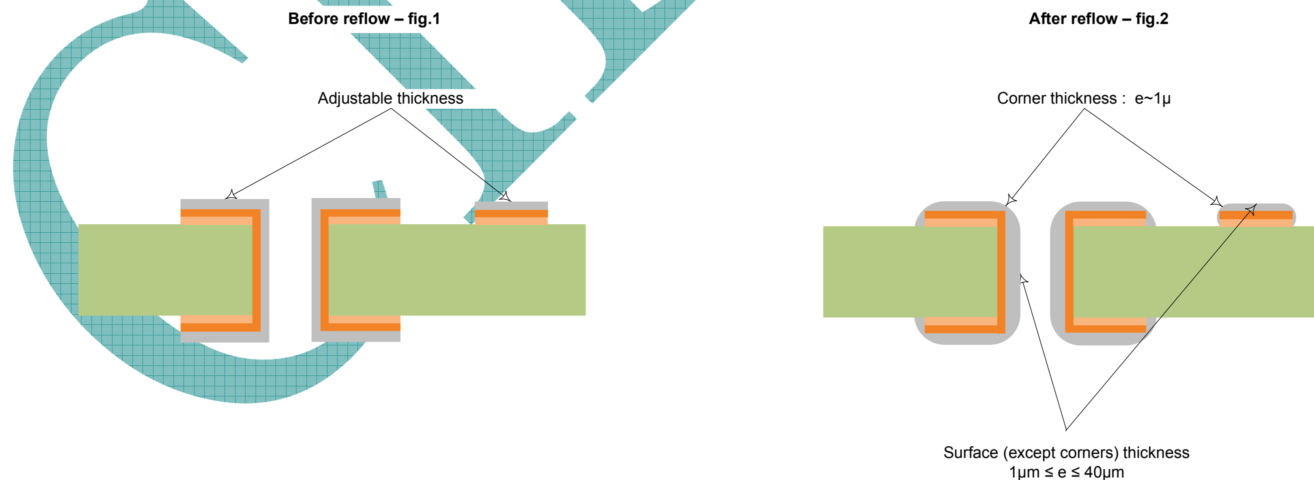


ML18, 2 laminations, 2 drilling

11 Finishing layer

Material	Range of possible thicknesses	TOLERANCE
- COPPER deposit	From 5µm to 50µm	± 20%
- Flash Electrolytic NICKEL / GOLD (+ cobalt) for soldering	Ni from 3 to 5µm + Au : 0,2µm to 0,4µm	± 20%
- Electrolytic NICKEL / GOLD (+ cobalt) for contacts	Ni from 7 to 25µm + Au : 1µm to 1,5µm	± 20%
- NICKEL / GOLD for « Bonding » (pure Gold)	Ni from 5 to 10µm Au : 3µm ± 1µm	N.A.
- ENIG	Ni from 3 to 10 µm + Au < 0,1µm **maximum**	± 20%
- NICKEL – chemical process	Ni from 3 to 10 µm	± 20%
- TIN *	1,3µm	-0,2 +0,1µm
- HAL leadfree (SN100C)**	From 1 to 50µm (Based upon configuration, figure 2)	N.A.
- TIN-LEAD **(not ROHS compliant!)** (63% Sn/37%Pb)	Non reflowed electro SnPb « : From 3 to 10µm (Figure 1)	± 20%
	reflowed electro SnPb: From 1 to 40µm (Based upon config. Figure 2)	N.A.
	HAL : from 1 to 50µm (Based upon config. Figure 2)	N.A.

* Tin : strongly not recommended for Rigid-Flexible and polyimide circuits.
** SN100C not for any NS3 (Class 3 of IPC A-600) including EN9100, Rigid-Flexible and high freqency circuits.



CIBEL

12 Miscellaneous

- SOLDER MASK: Photo imageable green color (other colors available)
- SOLDER MASK: Fine Special Photo imageable (for fine thickness)
- SOLDER MASK for flexible PCB's

- COVERLAY
- PHOTOIMAGEABLE COVERLAY (About 60µm thickness)

- LEGEND PRINTING: Ink-jet process (white) or screen-printing (other colors)

- IMPEDANCES
  - ➢ Calculation and control of the files by POLAR
  - ➢ Measurement made with Reflectometer POLAR INSTRUMENTS CITS 200

- ELECTICAL TEST: 3 FLYING PROBE TEST SYSTEMS ATG type A2 and A3
  - ➢ Standard Value : 10V (up to 500V adjustable)
  - ➢ Insulation : > 10MOhms
  - ➢ Continuity threshold < 10 Ohms
  - ➢ Adjustable stress and measurement voltage

- OPTICAL TEST and INSPECTION: with CAMTECH ORION 868

- THICKNESS MEASUREMENT:
  - ➢ CAVIDERM (micro-ohmmeter) for non destructive hole copper thickness measurement
  - ➢ Cross-sections for qualitative measurements and controls
  - ➢ FISCHER (Fluorescence X) for deposit thickness measurement and for chemical bath metal concentration measurement

CIBEL