HDI DESIGN GUIDELINES 1.1

ISO 26000 CONSCIOUS COMPONENT ENVIRONMENTAL, ETHICAL & SOCIAL

NCAB GROUP
Integrated PCB Production

Construction types

TYPE I

Cycles

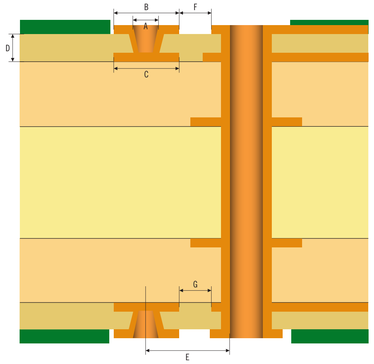
1	2	1
Bond	Drill	Plating

Availability

LOW — MEDIUM — HIGH

Density

LOW — MEDIUM — HIGH



TYPE II

Cycles

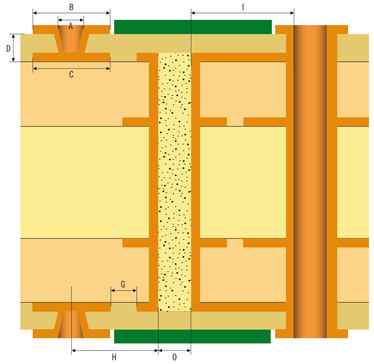
2	3	2
Bond	Drill	Plating

Availability

LOW — MEDIUM — HIGH

Density

LOW — MEDIUM — HIGH



TYPE III

Cycles

3	4	3+
Bond	Drill	Plating

Availability - Stacked / Standard via

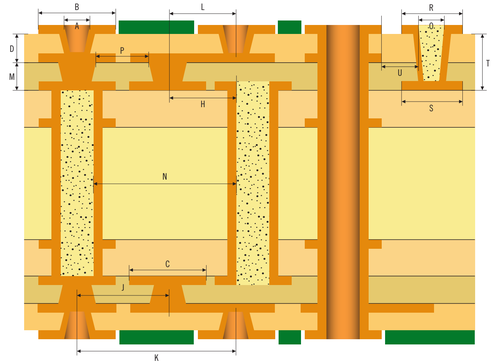
LOW — MEDIUM — HIGH

Density

LOW — MEDIUM — HIGH

Availability - Skip via

LOW — MEDIUM — HIGH



	Features (dimensions um)	RECOMMENDED	ADVANCED
A	Microvia size / diameter	100	80
B	Capture land	325 (class 2) 350 (class 3)	250* (class 2) 250* (class 3)
C	Target land	300 (class 2) 325 (class 3)	250* (class 2) 250* (class 3)
D	Dielectric L1 - L2 microvia	60-80	60-100
E	Microvia center to PTH edge	380	300
F	Outer layer space	100	76
G	Inner layer space	100	76
H	Microvia to buried hole	375	300
I	Buried hole to PTH	450	430
J	Pitch - internal microvia (different net)	425	325*
K	Pitch - outer microvia (different net)	525 (soldermask web) 425 (no soldermask)	325*
L	Pitch - staggered microvia	400	225
M	Dielectric for internal microvia	60-80	60-100
N	Buried hole to buried hole	450	350*
O	Buried hole size / diameter	250	150
P	Microvia to microvia	300	220
	Buried via target land	Buried via + 250	Buried via + 250*
Q	Skip via microvia size / diameter	300	200
R	Skip via capture land	500 (via + 200)	400 (via + 200)
S	Skip via target land	600 (via + 300)	500 (via + 300)
T	Dielectric L1-L3 skip via	200	160
U	Skip via to copper on L2	250	150

* For design tighter than the given value, please consult your local NCAB technical contact to discuss specific projects case by case.

Design tips

1 Aspect ratio recommended as 0,8:1 for laser drilled microvia (L1-L2), advanced is 1:1.
2 For better reliability, we recommend that microvia should be 100um when copper filling.
3 If possible entry and capture pads should be 200um greater than the microvia size.
4 Epoxy plugged via holes should be the same size if possible. Certainly no more than 0,15mm variation in size.
5 Always copper fill microvia in SMD pad. It is more costly, but it will provide a better and more reliable option.
6 Best practice is to stagger, rather than stack microvias on buried via holes as this relieves stress on the microvias.
7 Where possible we recommend 400um between microvia holes, 300um at lowest to ensure no processing issues.
8 Always plug through hole via in SMD pad according to IPC-4761 type VII.
9 Skip via structures are not preferred, staggered structures are recommended.
10 Always resin fill skip via holes.
11 Aspect ratio recommended as 0,67:1 for skip microvia (L1-L3), advanced is 0,8:1.

Material

IPC-4101	99	124	125	126	128	129	130	131
ANSI	FR4	FR4	HF-FR4	FR4	HF-FR4	FR4	HF-FR4	HF-FR4
Fillers > 5%	Yes	N/A	N/A	Yes	Yes	N/A	Yes	N/A
Tg	>150°C	>150°C	>150°C	>170°C	>150°C	>170°C	>170°C	>170°C
Td	>325°C	>325°C	>325°C	>340°C	>325°C	>340°C	>340°C	>340°C
CTE 50-260°C	<3,5%	<3,5%	<3,5%	<3,0%	<3,5%	<3,5%	<3,0%	<3,5%
CTE PPM < Tg	<60	<60	<60	<60	<60	<60	<60	<60
CTE PPM > Tg	<300	<300	<300	<300	<300	<300	<300	<300
T260 (min)	>30	>30	>30	>30	>30	>30	>30	>30
T288 (min)	>5	>5	>5	>15	>5	>15	>15	>15
T300 (min)	---	---	---	>2	---	>2	>2	>2
Availability	High	High	Low	High	Low	Low	Low	Low

IPC-4101/99 or 124
Thickness ≤ 1,6mm, 4-12 layers, copper ≤ 70um, moderate no. of reflows (≤260°C), demanding environment / temp

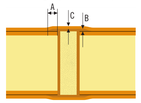
IPC-4101/126 or 129
Thickness > 1,6mm, ≥ 6 layers, copper > 70um, multiple no. of reflows (≤260°C), very demanding environment / temp

The above is only a recommendation - it is critical to determine the characteristics of the assembly process - exactly what thermal stresses will be experienced, as this will guide material selection. If this is unknown then you don't know what it takes for the material to survive.

Minimum track and gap for cap plating IPC-4761 TYPE VII PLUGGING / VIPPO / POFV

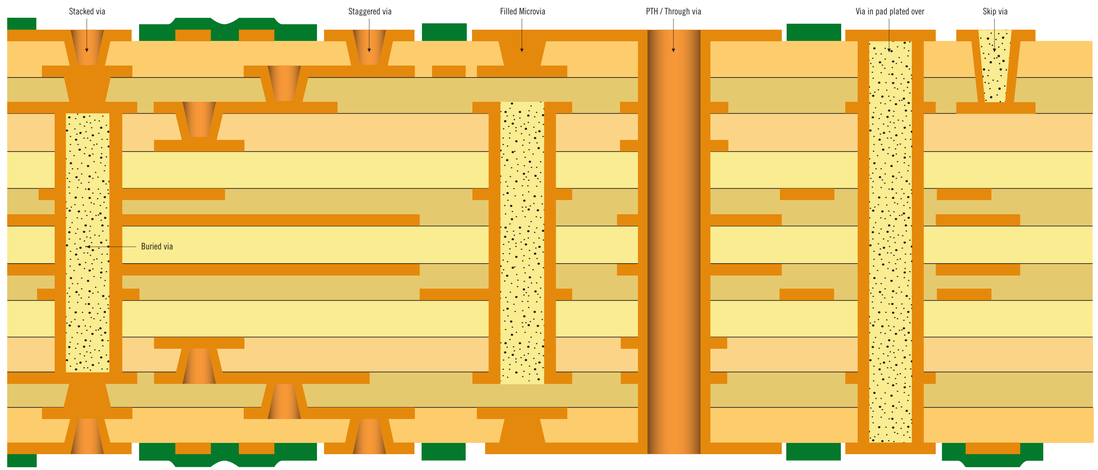
LOCATION	IPC DEMAND (Copper wrap & cap plating)	BASE COPPER (oz)	GENERAL TRACE	GENERAL GAP	MODERATE TRACE	MODERATE GAP**	ADVANCED TRACE	ADVANCED GAP**
Outer layer (PTH holes)	Class 2	1/3	N/A		0.10	0.10	0.075	0.09
		1/2	N/A		0.10	0.12	0.075	0.10
	Class 3	1/3	N/A		0.12	0.12	0.09	0.10
		1/2	N/A		0.12	0.12	0.09	0.10
Inner layer (Buried holes)*	Class 2	1/3	N/A		0.09	0.09	0.075	0.075
		1/2	N/A		0.09	0.09	0.075	0.075
	Class 3	1/3	N/A		0.10	0.10	0.09	0.09
		1/2	N/A		0.10	0.10	0.09	0.09

* Valid only for the outer phase of a buried core which needs cap plating - typically when a microvia is stacked on a buried hole.
** For design tighter than this, please consult your local NCAB technical contact to review specific projects case by case.



A Minimum copper wrap distance 25 um for class 1, class 2 and class 3
B Minimum copper wrap thickness 5 um for class 2 and class 3
C Minimum cap plating 12 um for class 3
C Minimum cap plating 5 um for class 2

Build capability



Getting it right from the start

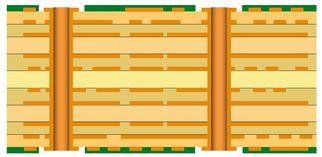
Nothing affects the PCB's cost and quality as much as the initial design. As modern electronic products are expected to offer more and more advanced functions, while the products themselves are becoming smaller and smaller, this puts greater demands on the PCB design and the importance of making the right choices at the design stage.

More than 30% of the Gerber data packs we receive do have some issues, generally, ambiguous information, errors, design rule conflicts, missing information and contradictions between the data and specifications. To prevent that it gets wrong from the start, we have put together these design guidelines, to use as a checklist…

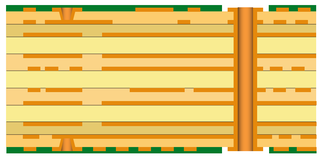
When using combinations of parameters, you should always consult your local NCAB technical contact person.

Cost drivers

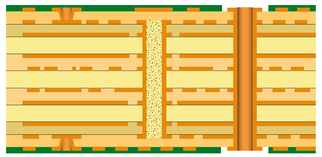
10L PTH = BASE PRICE



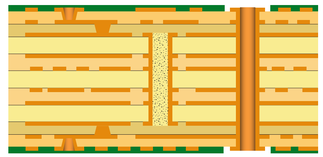
10L TYPE I (1+8+1) = + 40 - 70%



10L TYPE II (1+8B+1) = + 80 - 120%



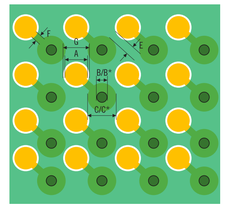
10L TYPE III (2+6B+2) = + 180-280%

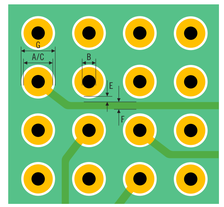


BGA Layout

0.80MM PITCH		RECOMMENDED DOG BONE	RECOMMENDED VIA IN PAD	ADVANCED DOG BONE	ADVANCED VIA IN PAD
A	BGA pad	Max 400	N/A	Max 400	Max 400
B*	Via hole (PTH)	200 (PTH)	N/A	N/A	N/A
C*	Via pad (OL / IL)	450	N/A	N/A	N/A
B	Microvia	N/A	N/A	100	100
C	Capture land	N/A	N/A	300	300
D	Target land	N/A	N/A	300	300
E/F	Track+Gap (outerlayer)	100 /125	N/A	75/100	75/100
E/F	Track+Gap (innerlayer)	100/125	N/A	75/75	75/75
G	Soldermask opening	BGA PAD+100	N/A	BGA PAD+100	BGA PAD+100
0.65MM PITCH					
A	BGA pad	Max 350	Max 350	Max 350	Max 350
B	Microvia	125	125	100	100
C	Capture land	Min 300	Min 300	Min 250	Min 250
D	Target land	Min 300	Min 300	Min 250	Min 250
E/F	Track+Gap (outerlayer)	100 /100	100 /100	75/100	75/100
E/F	Track+Gap (innerlayer)	100 /100	100 /100	75/75	75/75
G	Soldermask opening	BGA PAD+100	BGA PAD+100	BGA PAD+100	BGA PAD+100
0.50MM PITCH					
A	BGA pad	N/A	Max 300	Max 250	250
B	Microvia	N/A	125	100	100
C	Capture land	N/A	Min 250	Min 250	Min 250
D	Target land	N/A	Min 250	Min 250	Min 250
E/F	Track+Gap (outerlayer)	N/A	N/A	75/87	75/87
E/F	Track+Gap (innerlayer)	N/A	75/87	75/87	75/87
G	Soldermask opening	N/A	BGA PAD+100	BGA PAD+80	BGA PAD+80
0.40MM PITCH					
A	BGA pad	N/A	250	NA	250
B	Microvia	N/A	100	NA	80
C	Capture land	N/A	250	NA	250
D	Target land	N/A	250	NA	250*
E/F	Track+Gap (outerlayer)	N/A	NA	NA	NA
E/F	Track+Gap (innerlayer)	N/A	NA	NA	NA
G	Soldermask opening	N/A	BGA PAD+80	NA	BGA PAD+80

* For design tighter than 250 microns, please consult your local NCAB technical contact to discuss specific projects case by case.





For class 3 demands, please contact your local NCAB technical representative for details.

JUNE 2020 TULA NE