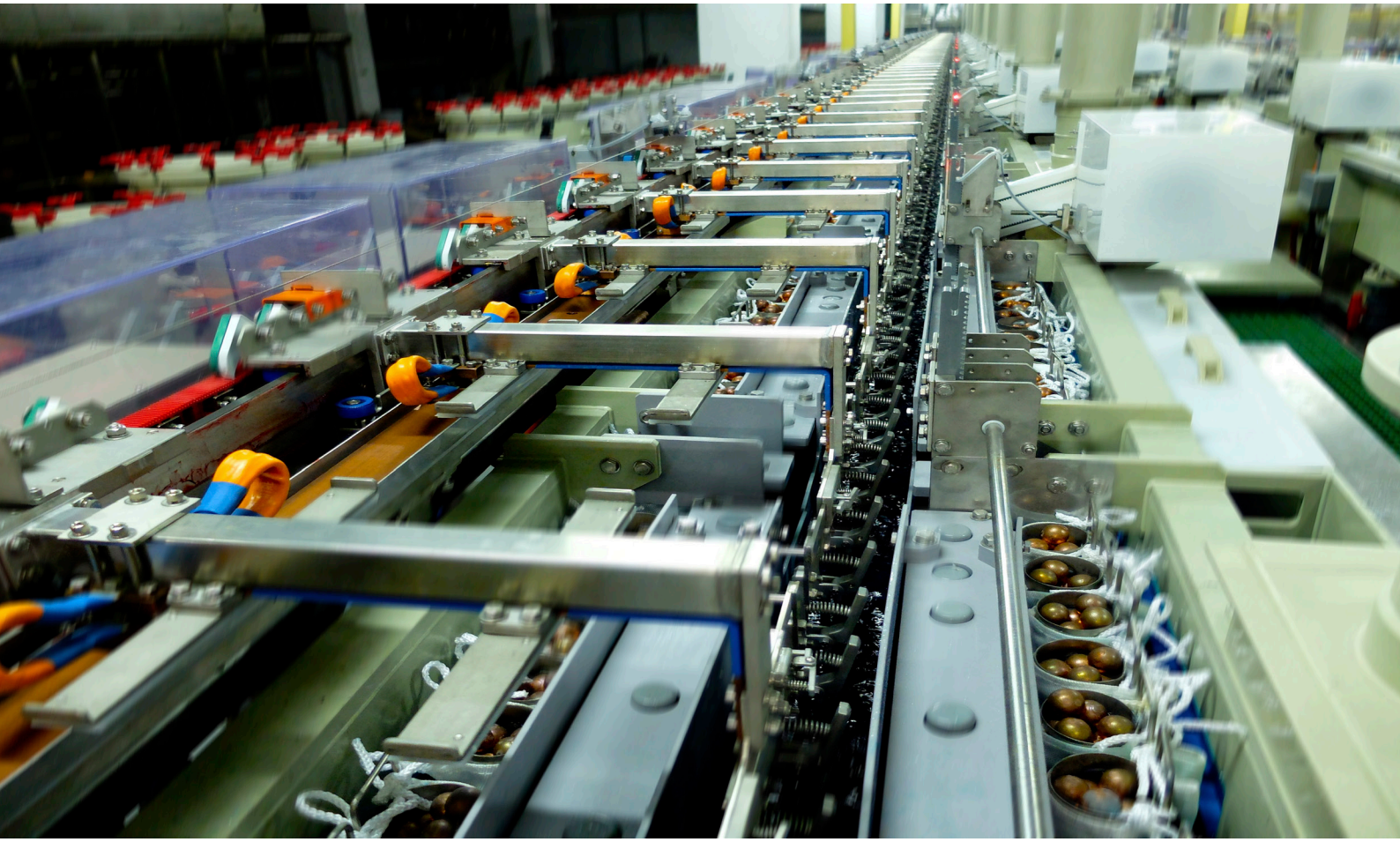
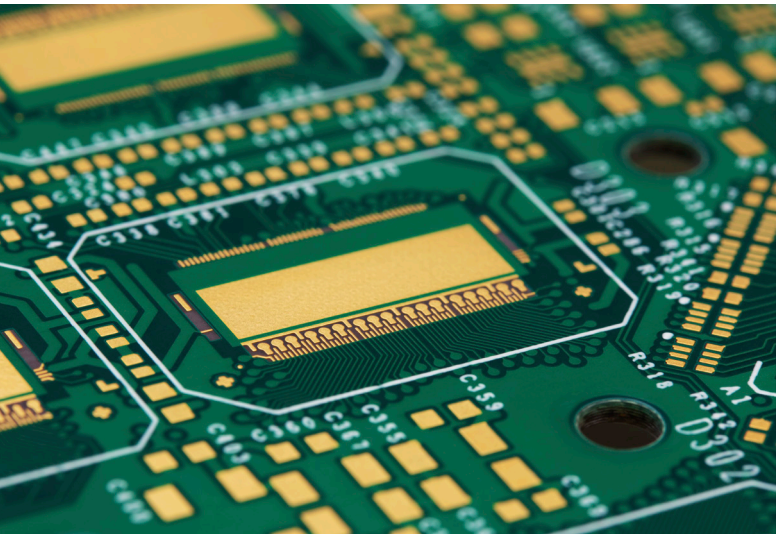
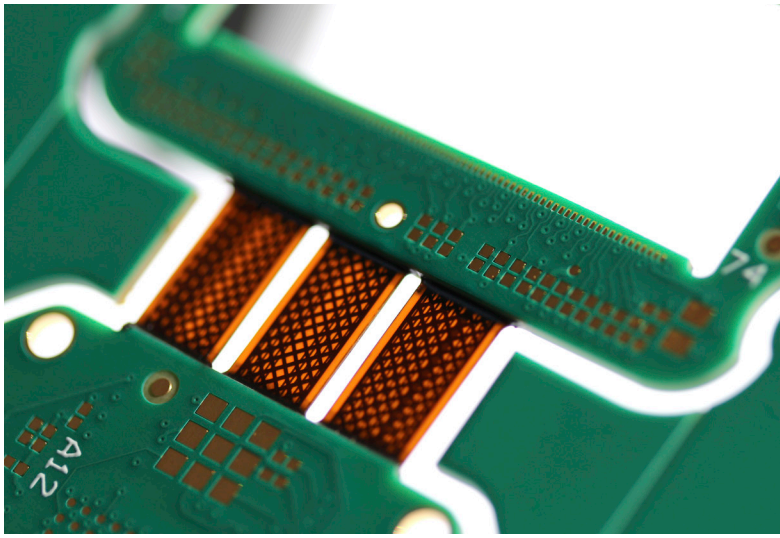
eurotronics

Design Manual

PCB rigid, flex & rigid-flex





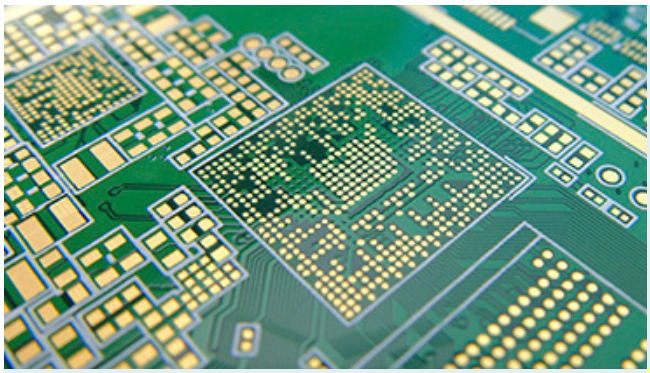


Design rules

PCB rigid, flex & rigid-flex

Eurotronics delivers highly advanced printed circuit board technologies to the unique requirements of each customer. Besides standard (HDI) printed circuit board technologies, Eurotronics adapts to the ongoing market trend towards miniaturization: ever thinner and ever more highly integrated printed circuit boards. We offer advanced capabilities ranging from ultra-fine line production with copper filled stacked micro via's to the processing of ultra-thin base materials and the manufacture of complex rigid-flex substrates incorporating bookbinder and window technology.

Our partner production facility is perfectly equipped to build highly integrated (HDI) multi-layer rigid and complex flexible circuit boards, suitable for microelectronics, COB, COF, flip chip and wire bond applications.
In order to achieve optimal quality results, we advise to the follow the applicable design rules and the designated IPC standards. This technical information only represents our basic capabilities. Together with our engineers we will evaluate our customer's layout designs to work out the best possible solution.

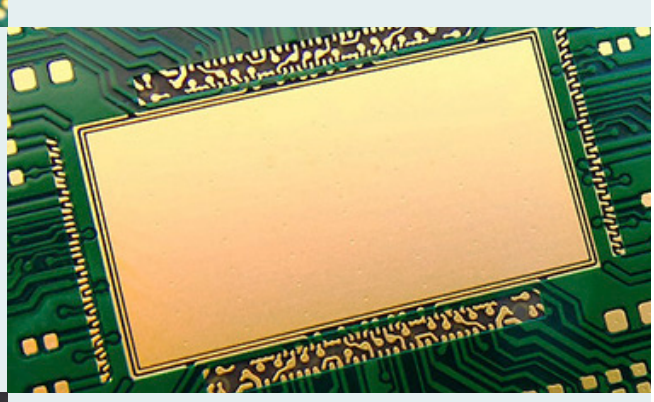


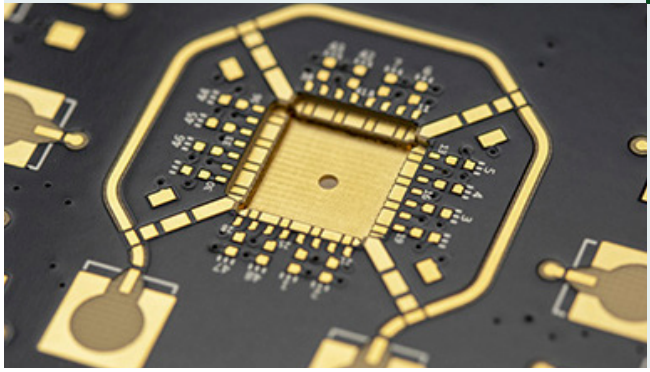
Standard

To be able to achieve the highest level of production yields and product reliability, the design values mentioned in the standard category should be followed whenever possible for layout designs.

Advanced

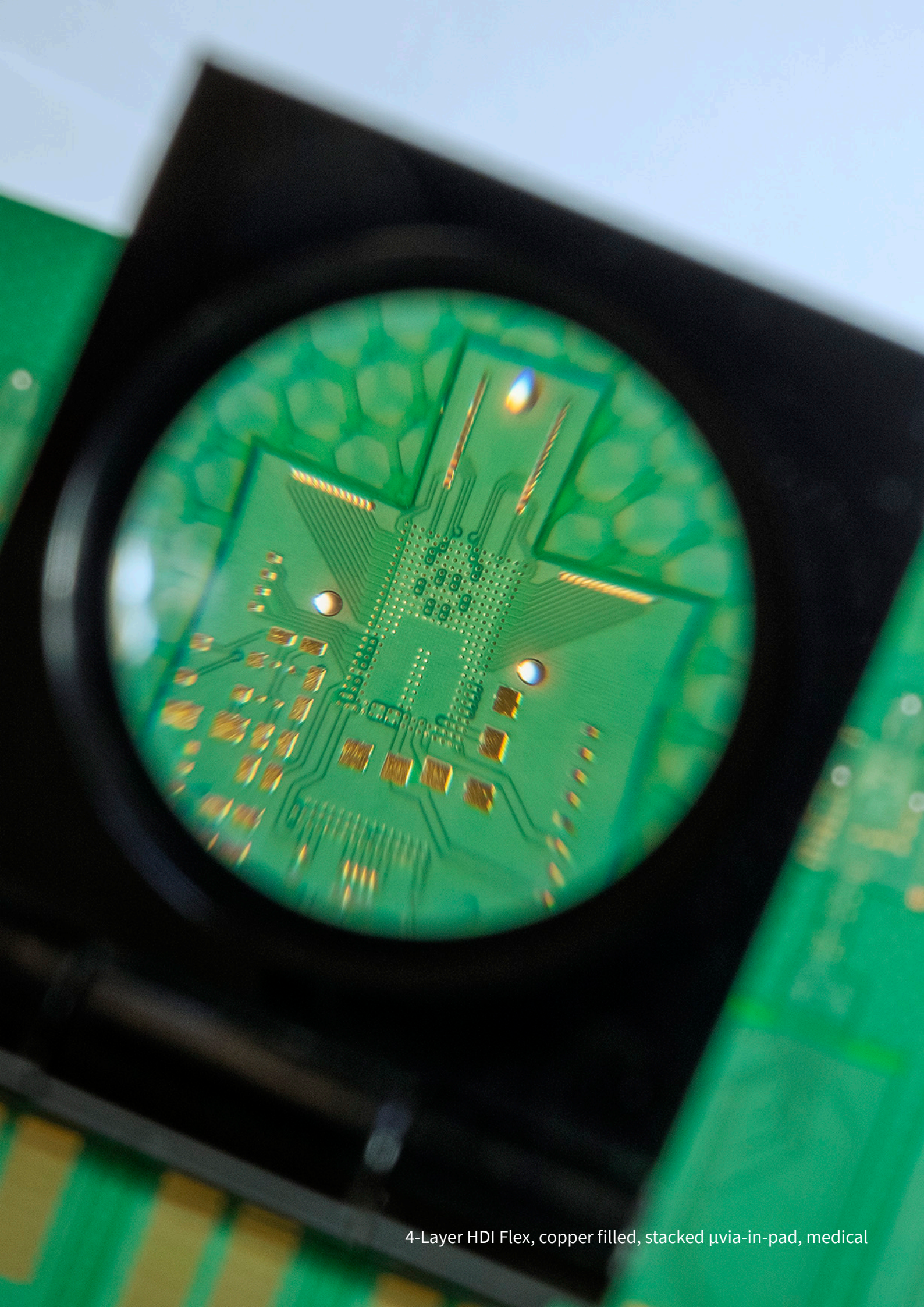
To achieve design values in this category, special materials, processes and equipments might be needed. If the design values of this category are intended to be used, Eurotronics strongly recommends consulting its engineering team during the early design phase (Design for Manufacturing).





Development

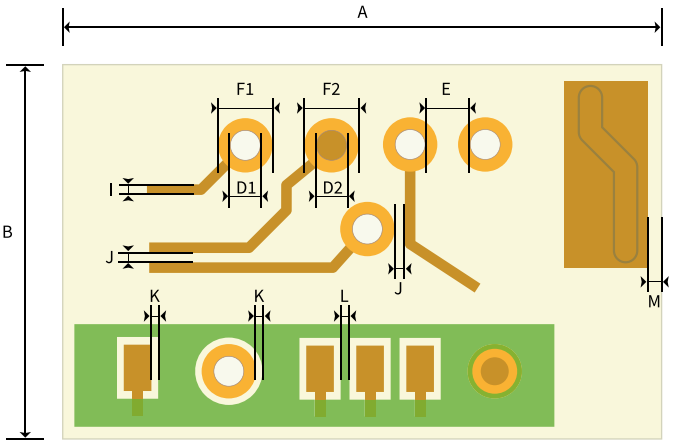
Eurotronics' engineering teams are looking forward to support your development projects. Design values that go beyond the ones mentioned in the categories "standard" and "advanced" are considered to have development status. PCB's designed using values of this category will require high level attention of our engineering teams.

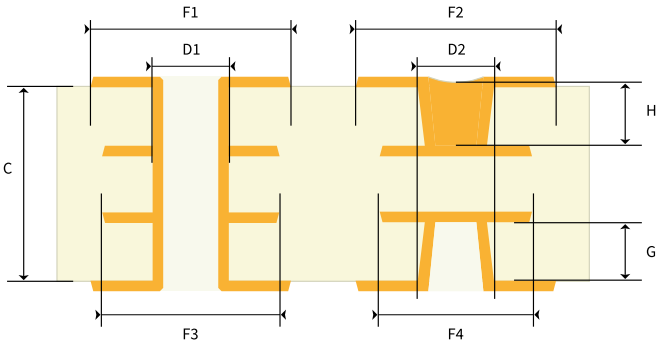


4-Layer HDI Flex, copper filled, stacked µvia-in-pad, medical

Design rules

PCB rigid, flex & rigid-flex



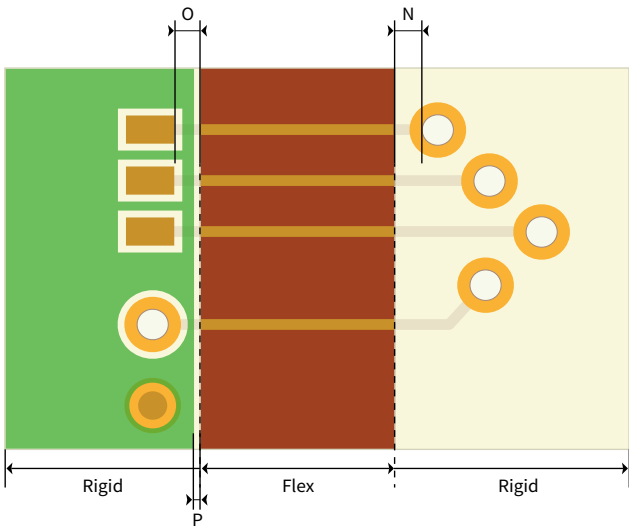


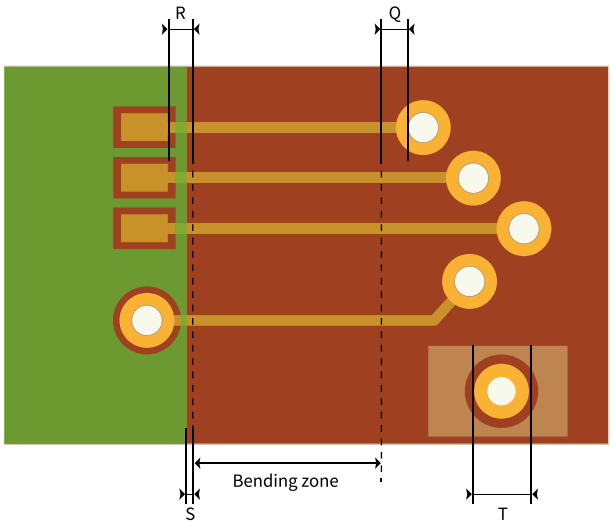
Ref.	Design parameter	Unit	Standard	Advanced	Development
A,B	Maximum board size	mm	630 x 620 (rigid), 570 x 417 (rigid-flex) 265 x 417 (flex)		
	Maximum board thickness	mm	5		
C	Minimum board thickness (substrate thickness without Cu)	µm	25	12.5	12.5
	Maximum layer count rigid PCB	-	30, HDI: 6+N+6	ELIC	
	Maximum layer count rigid-flex PCB	-	10 + 4 flex	14 + 6 flex	20 + 8 flex
	Maximum layer count flex PCB	-	6	8	> 8
D1	Minimum through hole diameter (mechanically drilled)	µm	150 ~ 250	100	75
D2	Minimum hole diameter (laser drilled, blind & through holes)	µm	75 ~100	50	40
E	Minimum distance hole - hole (mechanically drilled)	µm	350	275	200
(F1-D1)/2	Minimum annular ring outer layer (mechanically drilled)	µm	100	75	50
(F2-D2)/2	Minimum annular ring outer layer (laser drilled)	µm	70	50	25
(F3-D1)/2	Minimum annular ring inner layer (mechanically drilled)	µm	100	75	50
(F4-D2)/2	Minimum annular ring inner layer (laser drilled)	µm	70	50	25
C, D1	Maximum aspect ratio through holes	D1:C	1:10	1:12	1:16
G, D2	Maximum aspect ratio blind via's (base copper included)	D2:G	1:0.8	1:0.9	1:1
H	Copper filling ratio blind via's	%	80	90	>90
I	Minimum line width (depending on copper thickness)	µm	75	50	<50
J	Minimum spacing (depending on copper thickness)	µm	75	50	<50
K	Minimum solder mask opening	µm	40	35	20
L	Minimum solder mask dam width	µm	100	60	40
M	Minimum distance conductive material to board outline (mechanical/laser)	µm	150 / 100	100 / 75	75 / 50
	Layer to layer alignment	µm	+/- 50	+/- 40	+/- 35
	Hole to hole alignment (laser drilled)	µm	+/- 25	+/- 20	+/- 20
	Hole to hole alignment (mechanically drilled)	µm	+/- 35	+/- 30	+/- 30
	Controlled impedance tolerance	%	+/- 10	+/- 8	+/- 5
	Automatic optical inspection (AOI)	%	100	100	100

Recommended data file formats: Extended GERBER (RS-274X), ODB++, DXF, DWG.

Design rules

PCB rigid, flex & rigid-flex



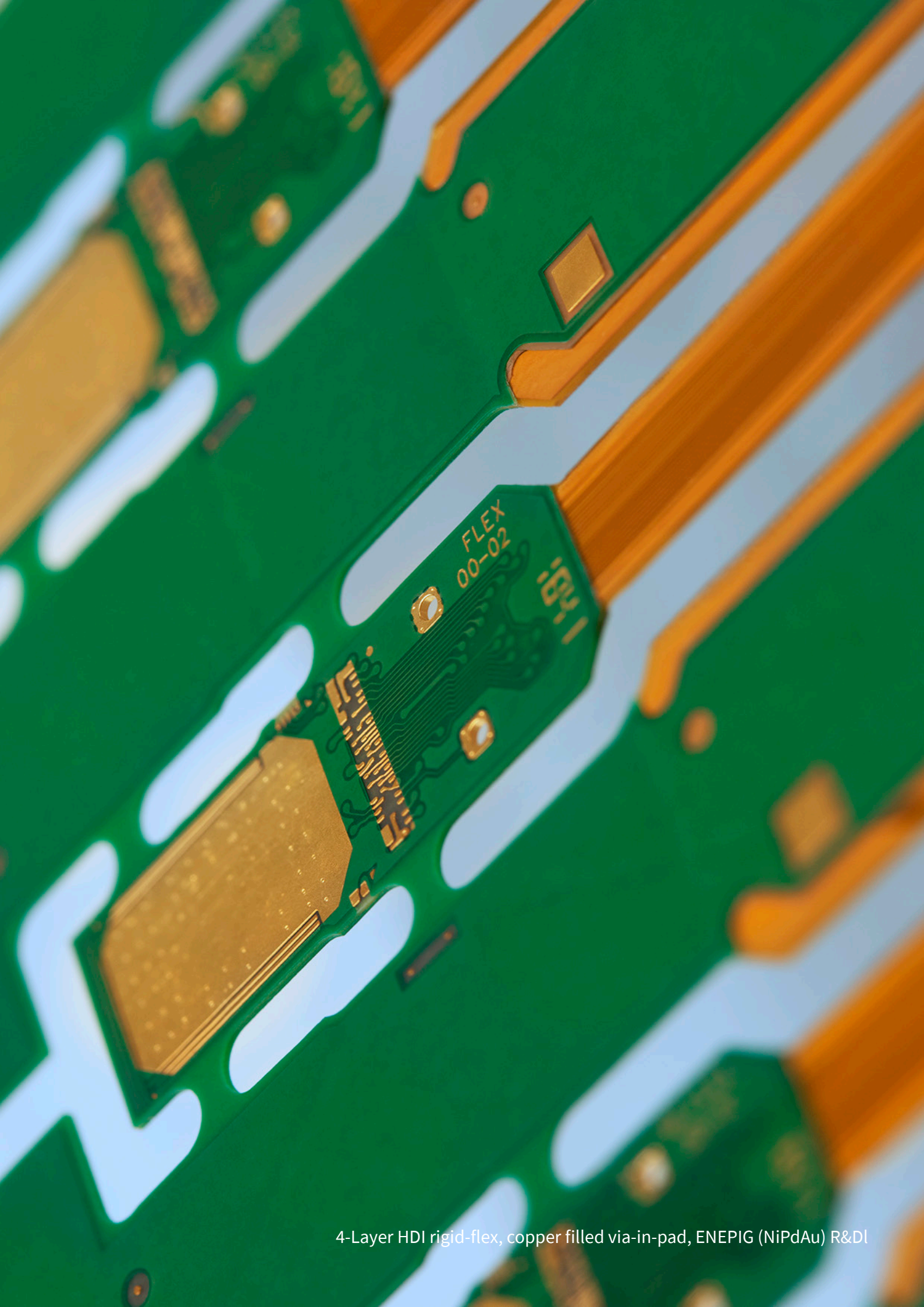


Ref.	Design parameter	Unit	Standard	Advanced	Development
N	Distance holes to transition zone rigid-flex	mm	> 1	> 0.7	> 0.5
O	Distance Cu to transition zone rigid-flex	mm	0.6	0.4	0.25
P	Distance solder mask to transition zone rigid-flex	µm	200	150	100
Q	Distance holes to bending zone flex	µm	400	300	200
R	Distance conductive pads to bending zone flex	µm	300	200	150
S	Distance solder mask to bending zone flex	µm	100	70	50
T	Coverlay opening: pad size +	µm	+ 300	+ 250	+ 200

PCB-Cavity

Multi-cavity layouts are possible, depending on the boards thickness and layer count

Design parameter	Depth tolerance	Outline tolerance	Plated cavities
Depth routing	+/- 150 µm	+/- 100 µm	++
Laser cavities (advanced)	+/- 50 µm	+/- 50 µm	++
Cavity creation by sequential lamination	+/- 10 % laminate thickness	+/- 150 µm	+



4-Layer HDI rigid-flex, copper filled via-in-pad, ENEPIG (NiPdAu) R&Dl

Design rules

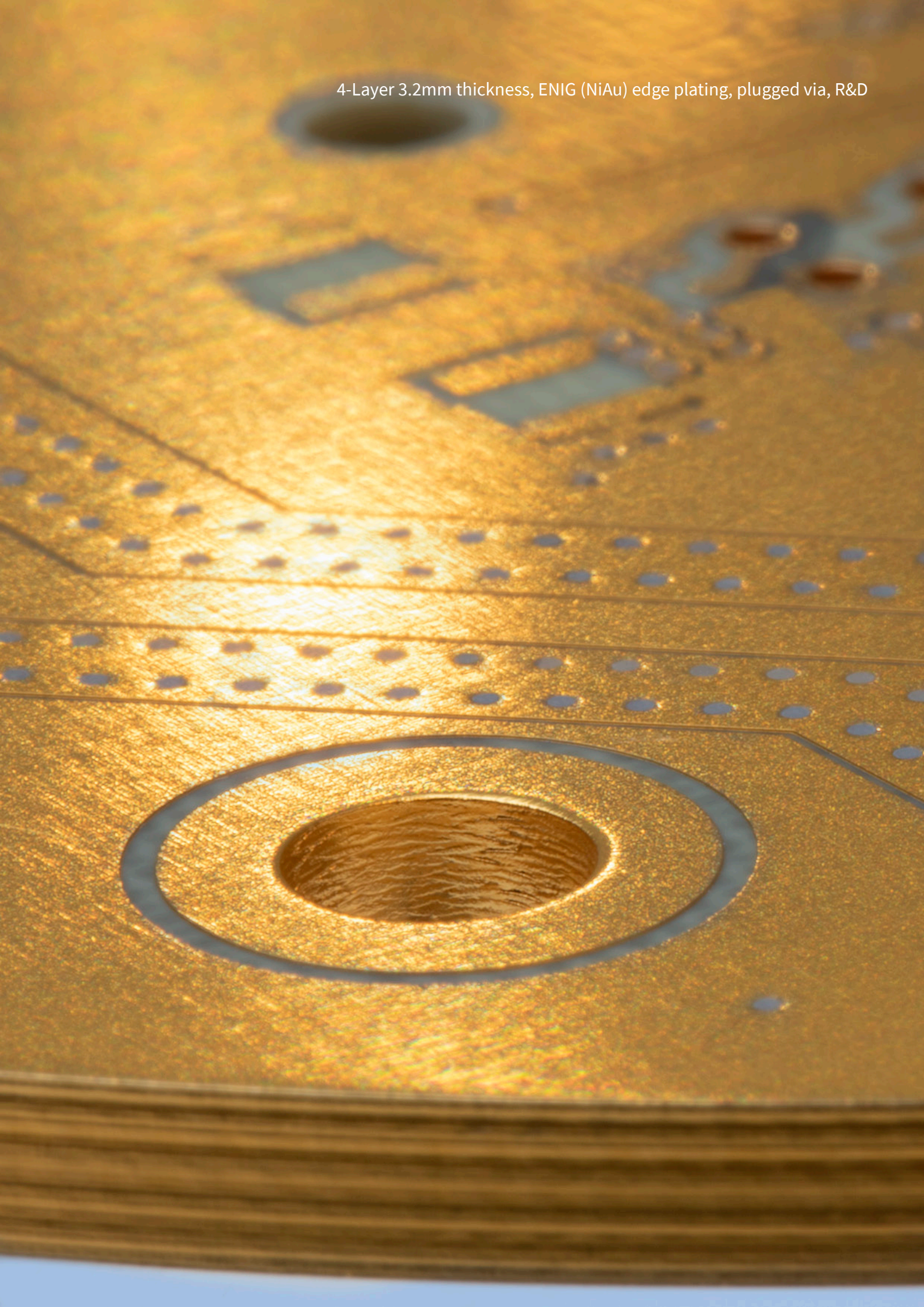
PCB base materials

Material type	Designation	Tg °C	Vendor	Remarks
Standard FR4	S1000-H	150	Shengyi	Low CTE, CAF resistant, IPC-4101 Type Designation: /21, /98, /99,/101, /121, /124
	TU-662	150	TUC	
	NP-155FTL	150	NanYa	
High Tg FR4	S1000-2M	180	Shengyi	Low CTE, CAF resistant, IPC-4101 Type Designation: /98, /99, /101, /121, /124, /126, /129
	IT-180A	175	ITEQ	
	TU-768	180	TUC	
	NP-175F	175	NanYa	
	370HR	180	Isola	
	EM-827	175	EMC	
High frequency	4000 Series	280	Rogers	Ceramic filled high speed, low loss substrate
	I-Tera MT40	200	Isola	High speed, low loss substrate
Thermal management	TC350 (Plus)		Rogers	Thermal Conductivity (1.0 W/mK) and Dielectric Constant Stability across Wide Temperatures (-9 ppm/°C)
	92ML	160	Rogers	
Polyimide film	ThinFlex-A, -G, -H, -K,...	-	ThinFlex	Adhesive-less FCCL, IPC-4104 Type Designation : /11
	Pyralux AP/AC Series	-	Dupont	
	2FP, 2LP Series	-	Taiflex	
	SF202	-	Shengyi	
	NanPao F Series	-	Nan Pao	Acrylic adhesive type FCCL
Coverlay	ThinFlex-I, -M, -Q	-	ThinFlex	
	LF, FR	-	Dupont	
	FH, FHL	-	Taiflex	
	SF305C	-	Shengyi	
	NanPao L Series	-	Nan Pao	
Bonding adhesive	ThinFlex-KC	-	ThinFlex	
	LF, FR	-	Dupont	
	BT25	-	Taiflex	
	NanPao D Series	-	Nan Pao	
Stiffener	PI, FR4, Stainless steel, Aluminium			

Above listed are commonly used base materials. Basically any type of material is available upon request.

Hybrid stack-up's and implant compatible materials are also available upon request.

We explicitly remark that printed circuit board production and assembly processes can change the physical and chemical properties of the base materials under influence of chemicals, high pressure and high temperatures. Therefore we are here to advise our customers in the best possible manner to ensure that the base materials used comply with the intended usage of the application.


4-Layer 3.2mm thickness, ENIG (NiAu) edge plating, plugged via, R&D

Design rules

PCB surface finishes

Surface finish type	Thickness (µm)	Recommended shelf life	Soldering	Al-wire bonding	Au-wire bonding	Contacts/ connectors
ENIG (NiAu)	Ni: 3 ~ 6 Au: 0.05 minimum	1 year	++	++	-	+
ENEPIG (NiPdAu)	Ni: 3 ~ 6 Pd: 0.05 ~ 0.15 Au: 0.05 minimum	1 year (for Au-wire bonding we recommend a shelf life of 6 months)	++	++	++	+
Electroplated Ni / soft Au	According to customer requirement	6 months	++	++	-	+
Electroplated Ni / hard Au	According to customer requirement	1 year	++	++	++	++
Immersion Tin (Sn)	1.2 maximum	6 months	++	-	-	-
Immersion Silver (Ag)	0.15 ~ 0.45	6 months	++	-	-	-
HASL lead free	0.2 - 0.3 - 0.5	1 year	++	-	-	-
OSP (ENTEK HT plus)	1 ~ 30	1 year	++	-	-	-

Copper Claddings and Foils

Electro Deposited (ED): 5, 9, 12, 18 and 35µm

Rolled Annealed (RA): 18 and 35µm

Other thicknesses are available upon request

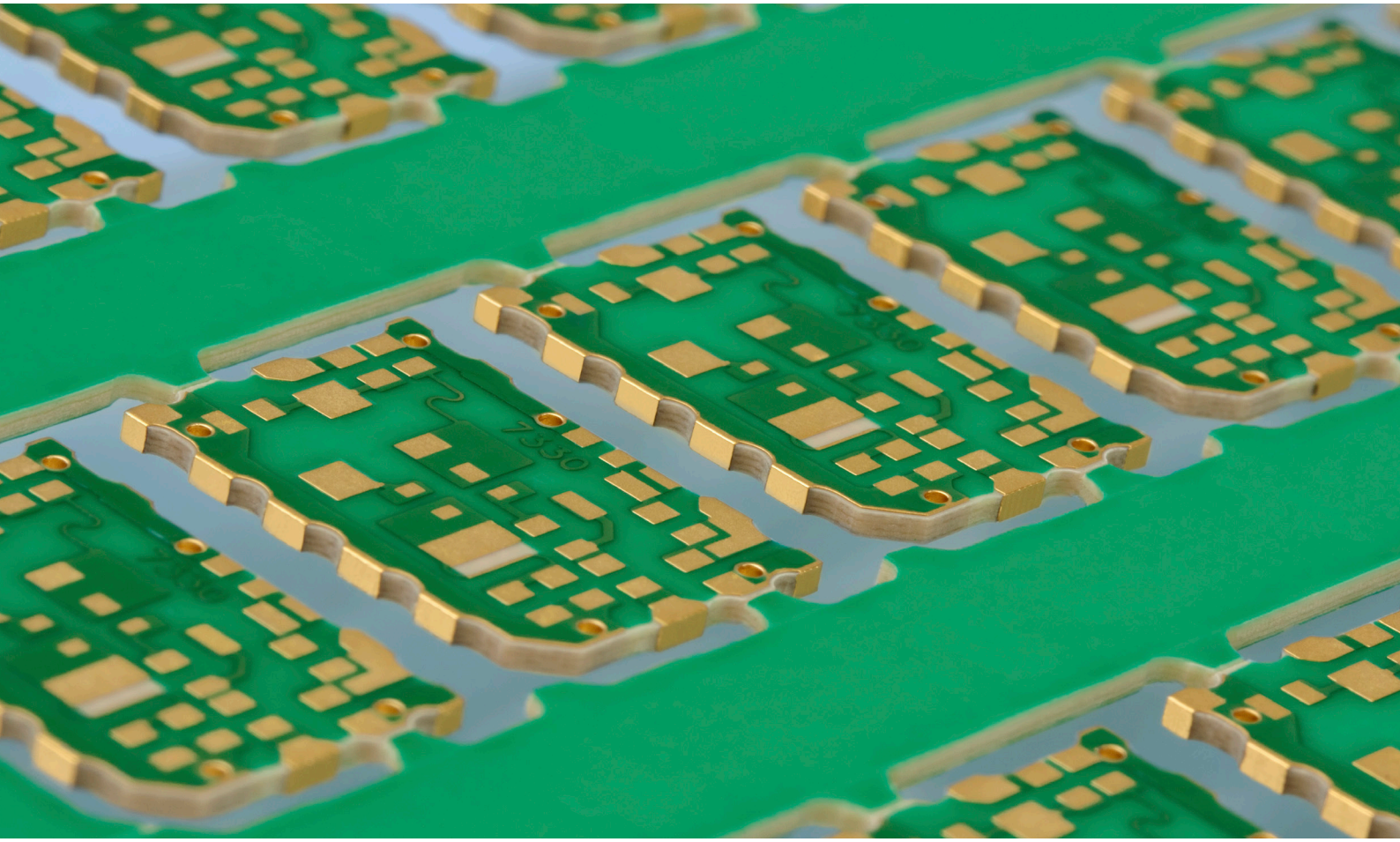
Design rules

PCB quality assurance

Eurotronics attaches great value to quality just to ensure that only PCB's that meet your quality requirements are delivered. That's why Eurotronics has build up a long term strategic partnership with one of the finest printed circuit board production facilities in Taiwan, China, Germany and Switzerland to make sure high quality printed circuit board technologies are guaranteed with respect for human rights and the strictest environmental demands.

The quality management system is guaranteed by a fully integrated ERP system and well maintained calibration processes. Every single PCB is inspected according to latest applicable IPC A-600 class 2 and pursuant to customer specifications.
At explicit request we can even guarantee IPC Class 3 or produce according to AS9100.

Quality overview
Design for Manufacturing (DfM)
Verified, validated and reproducible production processes managed by a fully integrated ERP system.
ISO 9001 certified quality management systems.
ISO 14001 certified environmental quality systems.
ISO 45001 occupational health & safety systems.
UL, RoHs, REACH & EICC-GeSi certified quality management systems.
Inspection & electrical testing according to IPC-A600 class 2, class 3 or specific customer specifications.
Interconnect stress test (IPC specified thermal cycle test).



Eurotronics BV

Grote Hondstraat 32
B-2018 Antwerpen
Belgium
+32 (0)3 281 1699
info@eurotronics.be
www.eurotronics.be