

## ACI – BASIC DESIGN RULES

All these rules are basic rules. For any specialty please do not hesitate to contact us so we can discuss and find a solution to fulfill your needs.

ACI available products			
Material	Layers	Material thickness (in mm)	Copper thickness (in $\mu\text{m}$ )
FR4	2	0.1	12
FR4	2	0.1	17
FR4	2	0.1	35
FR4	2	0.2	105
FR4	2	0.3	12
FR4	2	0.5	12
FR4	2	0.5	17
FR4	2	0.8	5
FR4	2	0.8	12
FR4	2	0.8	17
FR4	2	0.8	35
FR4	2	1.2	17
FR4	2	1.2	35
FR4	2	1.6	5
FR4	2	1.6	12
FR4	2	1.6	35
FR4	2	1.6	105
FR4	2	2.4	17
FR4	2	2.4	35
FR4	2	3.2	17
FR4	2	3.2	35
FR4	1	0.8	35
FR4	1	1.2	35
FR4	1	1.6	35
FR4	1	2.4	35
FR4	1	3.2	35
Kapton	2	0.1	18

Upon request we may process other types of material (Rogers, etc.) or different thicknesses. Material will be provided by the customer.

## Standard Dimensions (in mm)

Description	PCB size max.	Panel size max.
Small	148 x 180	170 x 250
Medium	228 x 230	250 x 300
Large	228 x 310	250 x 380

## Hole Diameter (in mm)

Material Thickness	Min. Hole Diameter	Max. Hole Diameter
	(c)	(c)
0.1	0.25	6.2
0.2	0.25	6.2
0.3	0.3	6.2
0.5	0.3	6.2
0.8	0.3	6.2
1.2	0.4	6.2
1.6	0.4	6.2
2.4	0.65	6.2
3.2	0.75	6.2

### Available drills:

- From 0.25 up to 3.25 mm, by steps of 0.05
- From 3.30 up to 6.2 mm, by steps of 0.1
- Drill diameters above 6.2 mm must be placed in the Outline file, GERBER RS-274X format

### Files needed:

- 1 file showing the layout of one side of the PCB, scale 1:1 (with dimensions if bigger than A4), PDF format (PDF 1:1 du circuit)
- 1 file TOP GERBER RS-274X format (Face dessus)
- 1 file BOT GERBER RS-274X format (Face dessous)
- 1 file Solder Mask TOP GERBER RS-274X format (Masque anti-soudure face dessus)
- 1 file Solder Mask BOT GERBER RS-274X format (Masque anti-soudure face dessous)
- 1 file Outline GERBER RS-274X format (Fichier de découpe mécanique)
- 1 Excellon 2 drill file with header (Fichier de perçage)

### We do not provide silk-screen (for component legend)

**Important: do not forget to add the EPFL logo in the design. If not present, ACI will add it, unless it is specified in the order that it should not.**

EPFL logos can be found under: [http://documents.epfl.ch/groups/a/ac/aci-unit/www/epfl\\_logos/](http://documents.epfl.ch/groups/a/ac/aci-unit/www/epfl_logos/)

For Altium a True Type editor can be found here: <http://wiki.epfl.ch/pcb/altium/logo>

Gerber Files Format Extensions				
Description	Protel	Orcad	Eagle	Other
Top Copper	gtl	top	cmp	gbr, gbx
Bottom Copper	gbl	bot	sol	gbr, gbx
Top Soldermask	gts	smt	stc	gbr, gbx
Bottom Soldermask	gbs	smb	sts	gbr, gbx
Top Silkscreen	gto	sst	plc	
Bottom Silkscreen	gbo	ssb	pls	
NC Drill	drl	thruhole.tap	drd	nc, txt, g01
Outline	gm1			gbr, gbx, gm1

## Finishing:

ACI offers the following finishing:

- Copper (Cuivre)
- Chemical tin (Etain)
- Gold (Nickel-Or)
  - **ENEPIG** - Electroless Nickel Electroless Palladium Immersion Gold. This finish has many advantages like excellent flatness for fine pitch components, very good oxidation properties, etc. It should also be the perfect choice for wire bonding.  
It is made of: 3.0 – 8.0 microns of Nickel, 0.05 – 0.25 microns of palladium and 0.05 – 0.3 microns of gold.
  - **ENIG (Chemical gold)** – Electroless Nickel Immersion Gold. This finish is also absolutely flat, as very good oxidation properties and should be used in case of edge card connection and press fit connectors because the gold is harder and more resistant.  
It is made of: 3.0 – 8.0 microns of Nickel and 0.05 – 0.3 microns of gold.
- Gold (**without Nickel**)
  - **EPIG** - Electroless Palladium Immersion Gold. Due to the absence of Nickel this finish is really useful in case of medical or high- frequency applications.

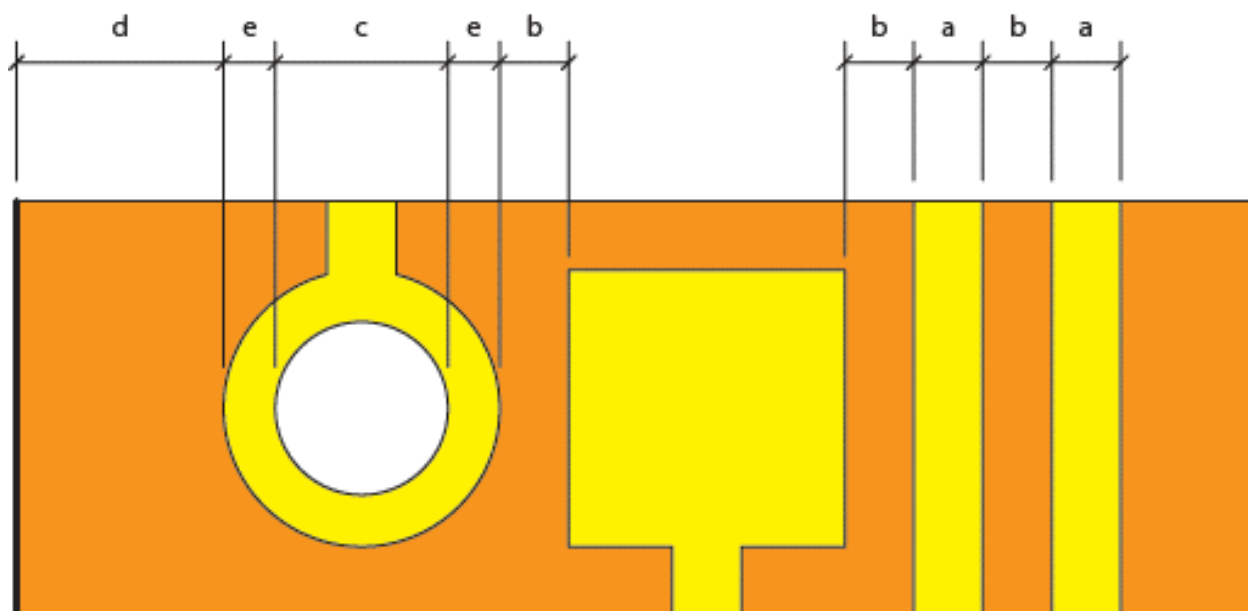
The type of gold finish requested, ENEPIG, Chem-Gold or EPIG, must be mentioned in the "Remarques" window of the order form.

Gold finish is done externally and therefore an extra 5 working days delivery should be added.

To order ACI services

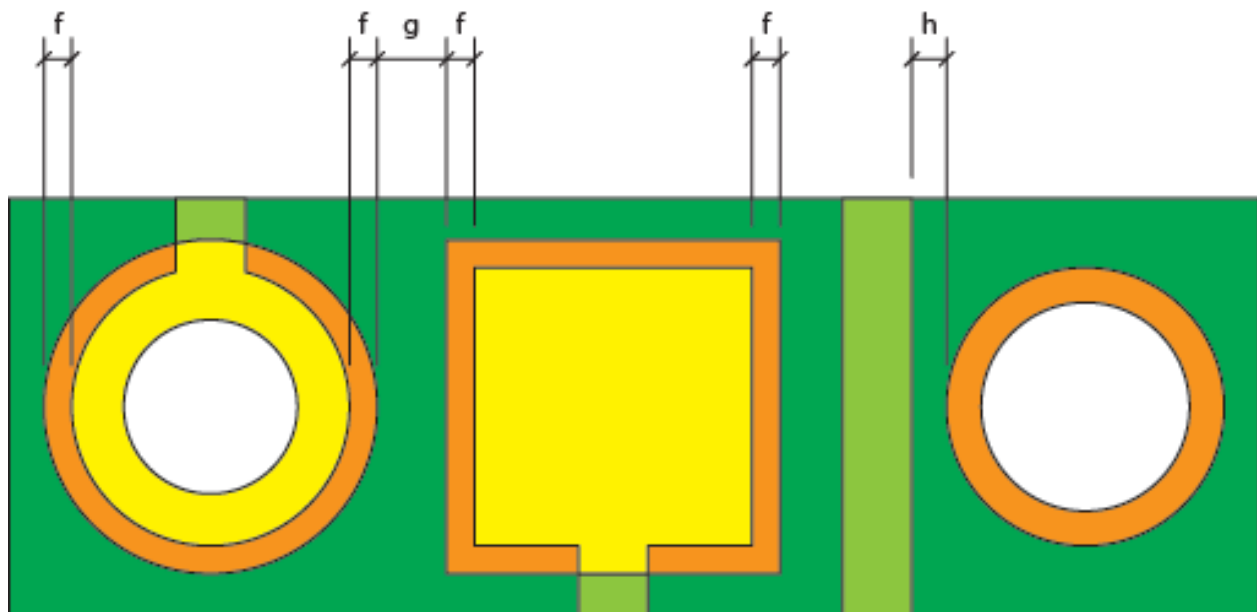
**[aci-commandes.epfl.ch](http://aci-commandes.epfl.ch)**

Copper design rules (in $\mu\text{m}$ )				
Copper thickness	Min. Trace width	Min. Space width	Min. Annulus width	Min. space between copper and PCB outline
	(a)	(b)	(e)	(d)
5	100	100	100	300
12	100	100	100	300
17	100	100	100	300
35	150	150	150	300
105	400	400	400	300



### Solder Mask (in $\mu\text{m}$ )

Clearance (f)	Bridge (g)	Cover (h)
100	100	100



EIDGENÖSSISCHE TECHNISCHE HOCHSCHULE LAUSANNE  
POLITECNICO FEDERALE DI LOSANNA  
SWISS FEDERAL INSTITUTE OF TECHNOLOGY LAUSANNE



---

*Atelier de fabrication des circuits imprimés ACI*  
*Bâtiment ELE CH - 1015 LAUSANNE (Switzerland)*  
*TÉLÉPHONE: +41-21 / 693 46 99*