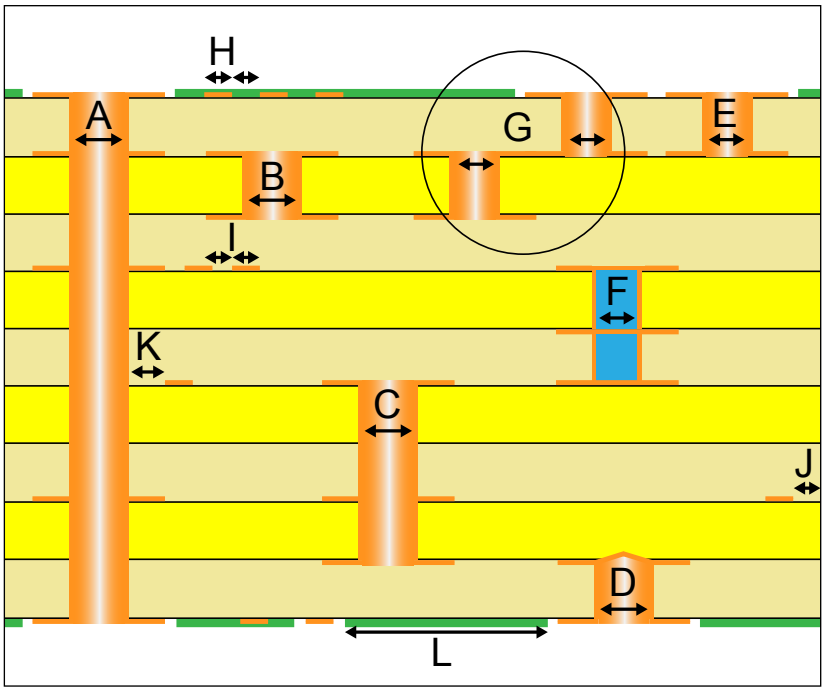
multi-cb
PRINTED CIRCUIT BOARDS

# BASIC DESIGN RULES

## 1. Design Parameters

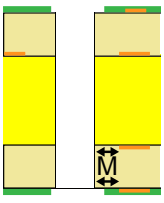



| Inner layers: copper thickness / min conductor | | |
|---|---|---|
| **copper thickness** | **conductor trace width / space** | **annular ring min.** |
| 12µm | 75µm / 100µm | 90µm |
| 18µm | 90µm | 90µm |
| **35µm** | **100µm** | **100µm** |
| 70µm | 150µm | 150µm |
| 105µm | 250µm | 250µm |
| 140µm | 300µm | 300µm |

| Outer layers: copper thickness / min. conductor | | |
|---|---|---|
| **copper final-thickness** | **conductor trace width / space** | **annular ring min.** |
| 30µm | 75µm / 100µm<br>90µm | 90µm<br>90µm |
| **35µm** | **100µm** | **100µm** |
| 70µm | 150µm | 150µm |
| 105µm | 250µm | 250µm |
| 140µm | 300µm | 300µm |
| 210µm | 500µm | 500µm |
| 400µm | 900µm | 900µm |

| Name (Parameters exemplary for 35µm copper) | | Standard (min.) | | | | Special production (min.) | | | |
|---|---|---|---|---|---|---|---|---|---|
| | | aspekt ratio | Ø | via-pad | annular ring circular | aspekt ratio | Ø | via-Pad | annular ring circular |
| A, B, C | via, buried Via (component hole: annular ring circular 25µm larger) | 1:10 | 200µm | 400µm | 100µm | 1:12 | 150µm | 330µm | 90µm |
| D | blind via, mechanical | 1:1 | 200µm | 400µm | 100µm | 1:1.2 | 150µm | 350µm | 100µm |
| E | blind via, laser | - | - | - | - | 1:1 | 100µm | 280µm | 90µm* |
| F | stacked vias Disproportional high effort. | - | - | - | - | - | - | - | - |
| G | staggered vias (microvias) | 1:1 - 1:10* | 200µm | 400µm | 100µm | 1:1 - 1:12* | 100µm | 300µm | 90µm |
| H, I | conductor traces outer<br>conductor traces inner | width \| space<br>width \| space | | 100µm \| 100µm<br>100µm \| 100µm | | width \| space<br>width \| space | | 75µm \| 100µm<br>90µm \| 90µm | |
| J | conductor, pad <> milling edge<br>conductor, pad <> scoring edge | space<br>space | | 200µm<br>500µm | | space<br>space | | 200µm<br>500µm | |
| K | conductor, pad <> via | space | | 200µm | | space | | 200µm | |
| L | solder-stop, green | clearance<br>bridge | | 50µm circular<br>100µm | | clearance<br>bridge | | 25µm circ. (BGA)<br>80µm | |
| | solder-stop, other colours | -<br>- | | -<br>- | | clearance<br>bridge | | 50µm<br>125µm | |

* see buried via, blind via | * For special production (min.) possible data check necessary.




### NPT - Holes
min. Ø: 200µm
aspekt ratio: 1:10 (o.r. 1:12)

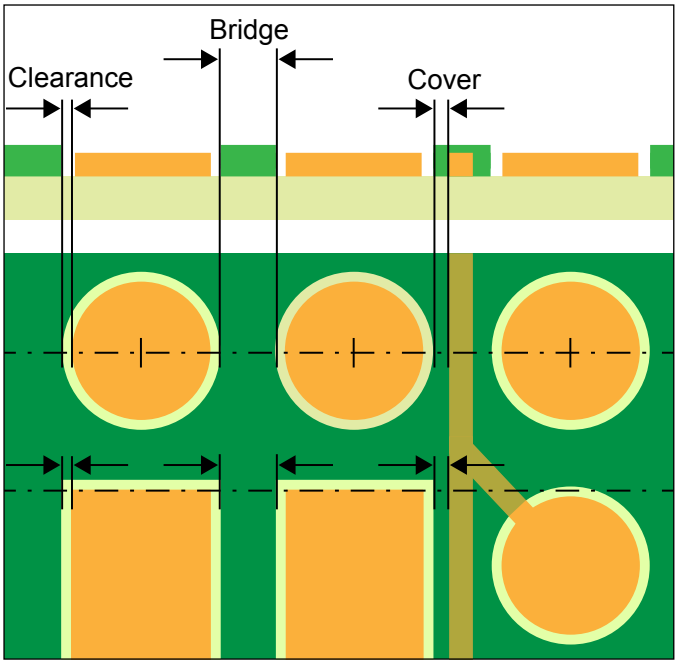
M conductor, pad <> NPTH: min. 200µm

Pilot or mounting holes (usually with Ø = 3,05mm) should be created in the same drill program as NPT-holes. Please label mounting holes in the dimension layer, as such.

### Coil
Coils on the inner layers:
min. conductor width / space of 125µm.

Coils on the outer layers:
min. conductor width / space of 100µm.

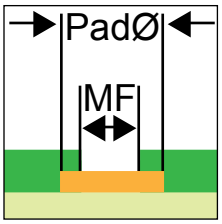
## 2. Solder-stop




| Solder-stop = green | | |
|---|---|---|
| | **standard** | **on request (data)** |
| **clearance** | 50µm | 40µm \| 25µm (BGA) |
| **bridge width** | 100µm | 80µm |
| **cover** | 100µm | 80µm |

| Solder-stop = black, blue, white, red, yellow | | |
|---|---|---|
| | **standard** | **on request (data)** |
| **clearance** | 50µm | 40µm |
| **bridge width** | 125µm | 100µm |
| **cover** | 150µm | 125µm |

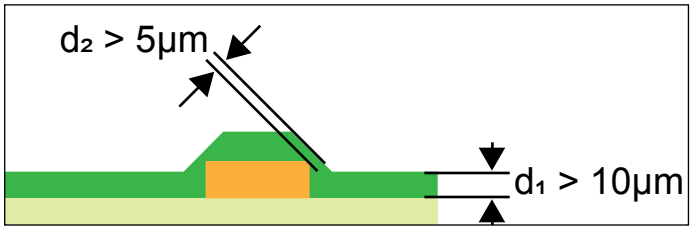
### SMD-Pads (Solder-Mask-Defined Pads)




For solder pads, which are defined by the solder-mask, please use the following parameters:

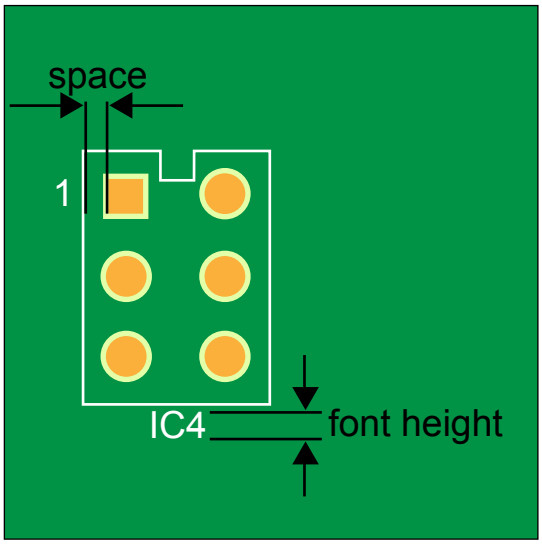
MC Ø (Mask Clearance) = Pad Ø - 80µm

Process capable for drill Ø ≥ 0,3 mm




| Solder-stop Parameters | |
|---|---|
| | **thickness** |
| d1: on the PCB | > 10µm < 25µm |
| d2: on the conductor edge | > 5µm < 25µm |
| | |
| **electric strength** | 500VDC min. |

## 3. Marking print




| Marking Print Parameters | | |
|---|---|---|
| **font height** | **ideal font width** | **min. font width*** |
| 1,2mm | 150µm | 100µm |
| 1,5mm | 180µm | 125mm |
| 1,8mm | 200µm | 150µm |
| | | |
| **spacing to pad min.** | 150µm | |
| spacing to solder-stop clearance | 100µm | |
| Never place marking print on pads > will be clipped by Multi-CB before production. | | |

* Can lead to surcharge > special production

### For EAGLE-Users

Before exporting your data, you should always activate the option

- “Always vector font”

which is found under: Options/User interface. Otherwise your marking print will very probably be incorrectly applied (EAGLE V. 5+).

## 4. Tolerances and Design Limits

The production of printed circuit boards is carried out according to the valid IPC guidelines and standards and on the basis of following technical specifications. HDI or MFT boards can be produced with smaller tolerances. Differing requirements of the customer must be explicitly agreed!

| Pattern tolerances | |
|---|---|
| | **Tolerance** |
| Drilling (PTH) to conductive pattern outer layers | ±0,10mm |
| Drilling (PTH) to conductive pattern inner layers | ±0,15mm |
| Drilling (PTH) to milling pattern / contour | ±0,10mm |
| Drilling (NPTH) to milling pattern / contour | ±0,10mm |
| Drilling (PTH) to marking print | ±0,15mm |
| | |
| Conductive pattern to solder resist | ±0,10mm |
| Conductive pattern to marking print | ±0,20mm |
| | |
| Hole to hole, one pass*<br>PTH-PTH or NPTH-NPTH | ±0,05mm |
| Hole to hole, two passes PTH-NPTH | ±0,10mm |
| * Also applies for PTH-NPTH if they are drilled in one run (e.g. location holes for SMD stencils) | |

| Conductor (acc. to IPC-6012) | | |
|---|---|---|
| Conductor width | min. 80% | in comparsion to the data |
| Conductor space | max. 30% | reduction in comparsion to data |

| Impedance control | |
|---|---|
| Tolerance (normal) | 10% |
| Tolerance (extended) | 5% |

| Milling | |
|---|---|
| | **Tolerance** |
| Milling offset | ±0,10mm |
| Z-Axis milling depth | ±0,20mm |

| Base material | |
|---|---|
| | **Tolerance** |
| FR4 thickness | ±10% |
| The information about the base material thickness exclusively defines the thickness of the dielectric including base copper. The other layer structures such as electroplated Cu layers or solder resist layers result in increased final thickness. | |

| Rigid PCB thickness | |
|---|---|
| **Type** | **Tolerance** |
| Producibility level B (standard) | ±10% or ±178µm |
| Thickness tolerances for pressed multilayers according to IPC-2222A. The higher value is valid. When measured over metallizations or coatings, those thicknesses and tolerances must be considered. | |

| Flexible PCB thickness | |
|---|---|
| **Type** | **Tolerance** |
| Flexible part thickness | ±50µm |
| Stiffener thickness | ±50µm |

| Vias & Drills | |
|---|---|
| | **Final-Ø** |
| Plated-through-holes (PTH) and component holes | ±0,10mm |
| Non-plated-through-holes (NPTH) | ±0,08mm |
| Press-fit technology (drilled) | ±0,05mm |
| > on request | +0,10mm/-0 |
| Press-fit technology (milled*) | ±0,075mm |
| * From a final diameter of approx. 6.0mm (depending on the surface) the holes are milled, not drilled. | |

| Cu min. thickness of throughplating | | |
|---|---|---|
| | **Class 2*** | **Class 3** |
| Via (> 150µm) | 20µm - 25µm | 20µm - 25µm |
| Microvia (≤ 150µm) | 18µm - 20µm | 20µm - 25µm |
| Blind Via | 10µm - 12µm | 10µm - 12µm |
| Buried Via | 10µm - 12µm | 10µm - 12µm |
| * Standard | | |

| Scoring | |
|---|---|
| | **Tolerance** |
| Offset (to PCB center) | ±0,10mm |
| Drilling (PTH) to scoring pattern | ±0,15mm |
| Drilling (NPTH) to scoring pattern | ±0,20mm |
| PCB dimension x/y | ±0,15mm |
| Scoring depth | ±0,20mm |

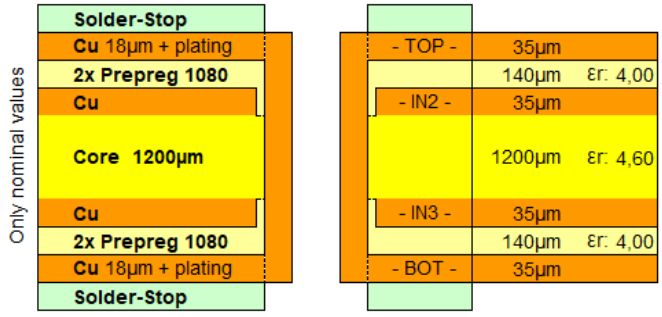
| Bow & Twist | |
|---|---|
| | **Tolerance** |
| For PCBs ≥ 0,8mm thickness | 0,75% with SMD |
| | 1,50% without SMD |
| Please note that the twist & bow value is increased above average, if the copper balance of the PCB is locally very unequal or if the circuit board is very thin. | |

| Delivery quantity | |
|---|---|
| **Pieces** | **Tolerance** |
| 1 - 20 pcs. | Excess or short delivery 0% |
| from 21 pcs. | Excess or short delivery of up to10% |
| On request: quantities of more than 20 pieces in exact quantity.<br>The (single) piece rule also applies when ordering as panel. | |

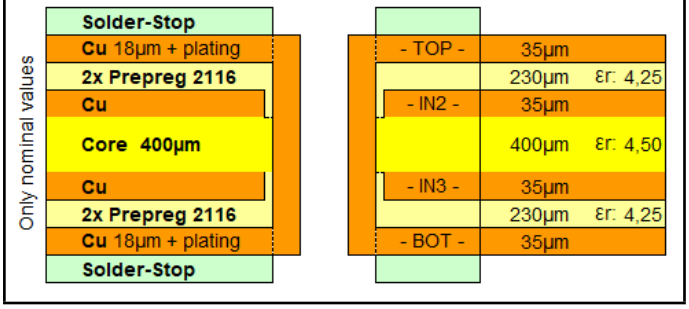
## 5. Defined layer buildup

For certain applications it is necessary that the layer isolation thicknesses are defined, e.g. for impedances. For this purpose, Multi-CB offers the following defined layer buildups at no extra charge. You can achieve your desired values with our layout examples for impedances (approx.).

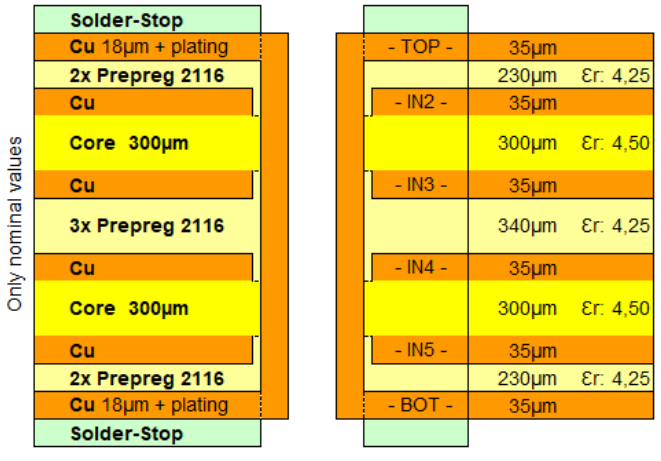
**4 layers: 4L-01 (1.6mm)**



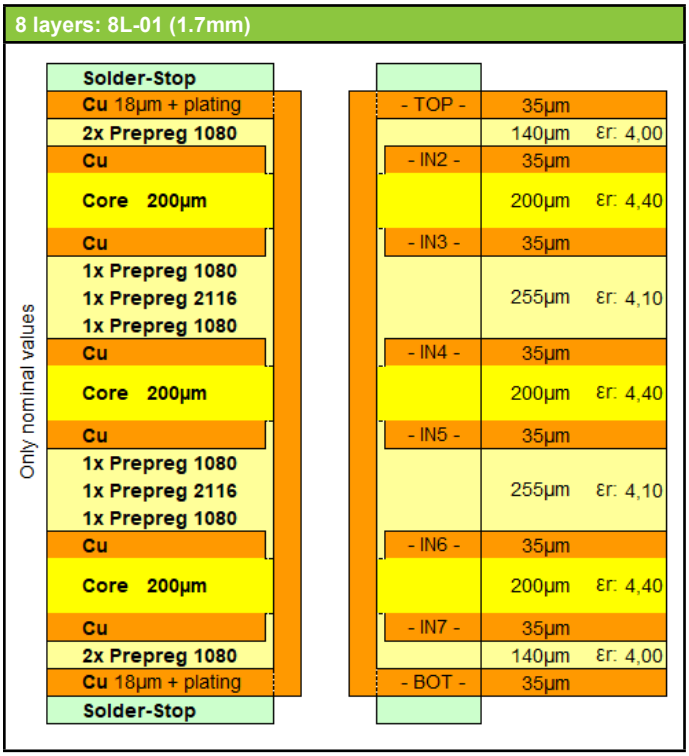

**4 layers: 4L-02 (1.0mm)**




**6 layers: 6L-01 (1.6mm)**



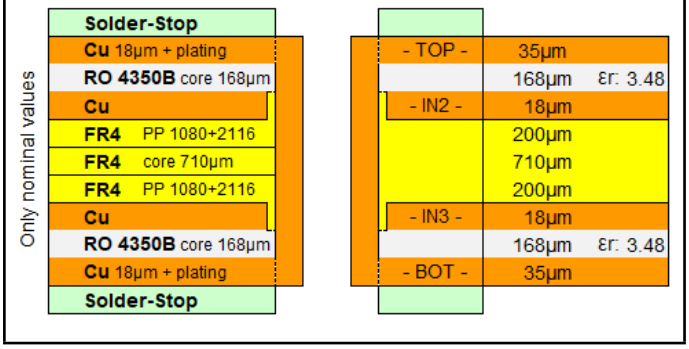

**8 layers: 8L-01 (1.7mm)**



## 6. Hybrid layer buildup Rogers 4350B/FR4

A hybrid layer buildup of Rogers 4350B and FR4 material combines several advantages. The critical layers are built up with a high-tech Rogers core and then pressed with common FR4 prepreg. Your advantages: Optimised low loss material properties in the high-frequency layers, lower price due to material mix, greatly improved mechanical stability, defined layer buildup with hybrid buildup - layout examples for impedances (approx.).

**4 layers: 4L-H01 (1.55mm) Hybrid Rogers 4350B 168µm core**




**4 layers: 4L-H02 (1.55mm) Hybrid Rogers 4350B 254µm core**