Four channel PCB status

PCB requirements

- **Clearance between sapphire pads 65 um**
- **Sufficient HV track insulation for 1400V**
- **Removable connectors for all connections**
- **Gold plating for good adhesion of bonding wires.**

• Clearance between sapphire pads

65 nm is a compromise between feasibility and costs.

Although for pcb manufacturer is possible to go down to pad-pad clearances of 40 um, this leads to a decrease in the thickness of copper and thickness of through holes metallization (less than 20um of IPC rules)

With pad-pad separation of 65nm and pads width of 135nm it is possible to make a bonding that is not perfectly aligned but achievable

• HV track insulation

Although it is not expected to reach these voltages,
since the HV filter capacitor supports up to 2KV and
the HV connector 1.4KV,
all tracks affected by HV have been sized for a clearance of 2KV.

- **Removable connectors for all connections**

For convenience of construction and use

it is necessary that all connections are removable.

Output connectors can be LIMO or SMP

• Gold plating

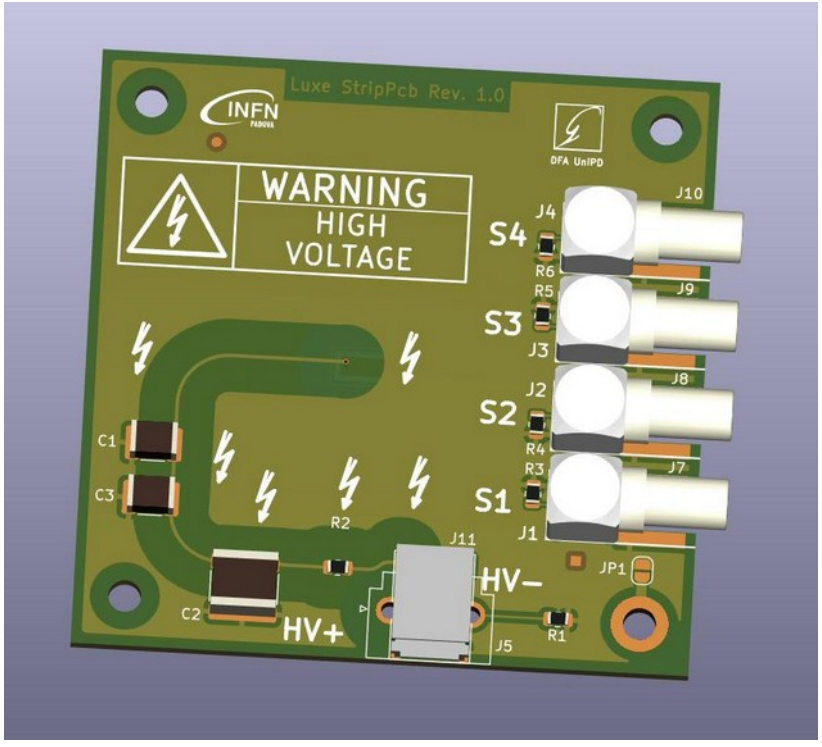
To get a good connection of the bonding wires all pads are gold plated.

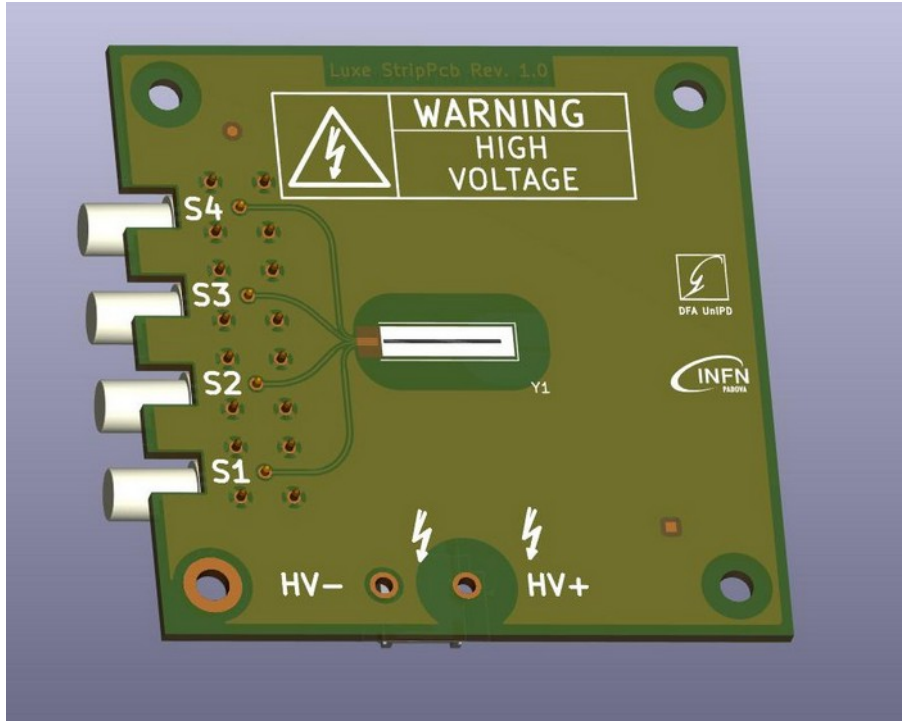
This treatment should also improve the connections

between the two sides of pcb (vias) which, due to the reduced thickness of copper needed to achieve clearance of 65um,

are of reduced thickness compared to the IPC standard

Final appearance





Pads of sapphire

To avoid obstacles in case of rebonding the sapphire is mounted on the back of the pcb