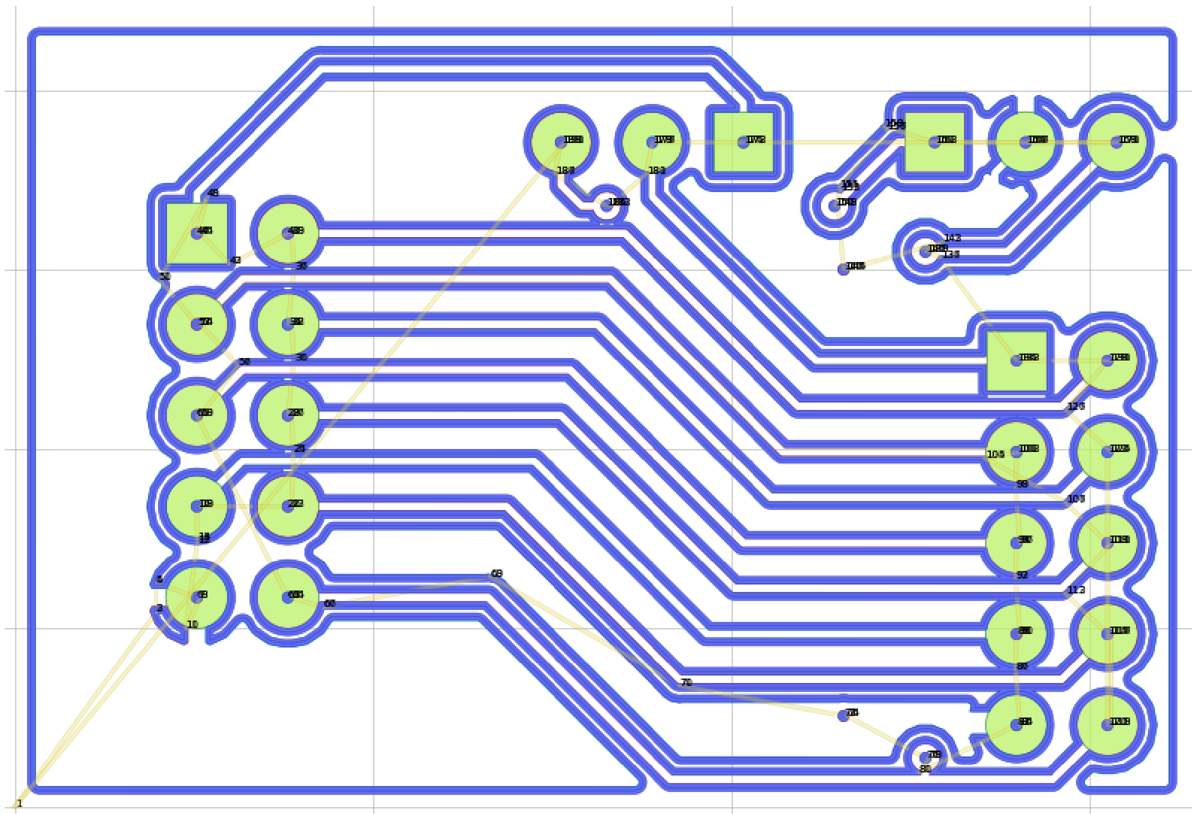
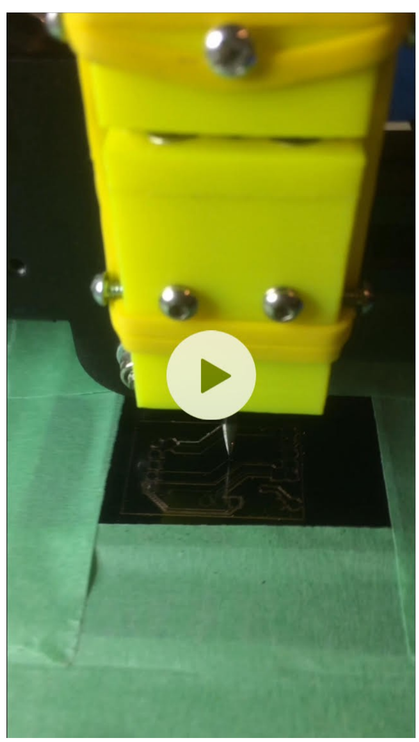
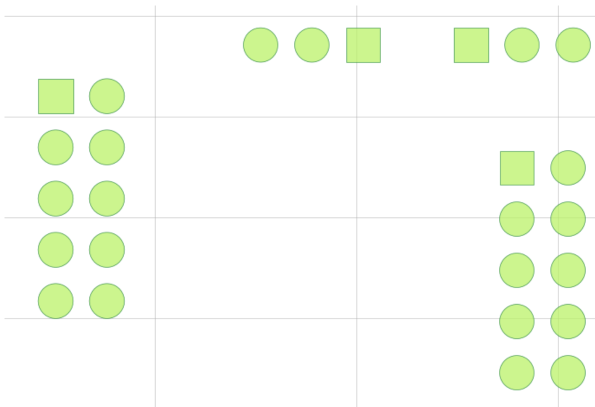
Request regarding KiCAD and FlatCAM to produce a modified « solder mask » gerber file to « scrape » the pcb pads after applying a UV solder mask

1) « add_pilot » application incorporates the .drill file in the gerber file in such a way that flatCAM will produce a file with small holes in each pad requiring to be drill or etched. THAT IS A GREAT ADD-ON that I use prior to import my design in « flatCAM » to engrave with my 3D printer (see below).





2) Now, after eching the pcb (and prior to drilling holes for the trough holes) I would like to add UV sensitive solder mask all over the pcb and « scape » the pads (smd as well as trough holes) with my 3D printer and a file produced by flatCAM, using a « modified solder mask gerber file » as an input to flatCAM. In the picture below the pads would be « scraped ».



Do you think that using an approach similar to the one used in « add_pilot » it would be possible for you to produce a gerber file that once read by flatCAM would produce the desired result for the entire surface of each pad?

Thank you so much for your time and commitment.

Regards,

R-J. Mercier
2018-12-03