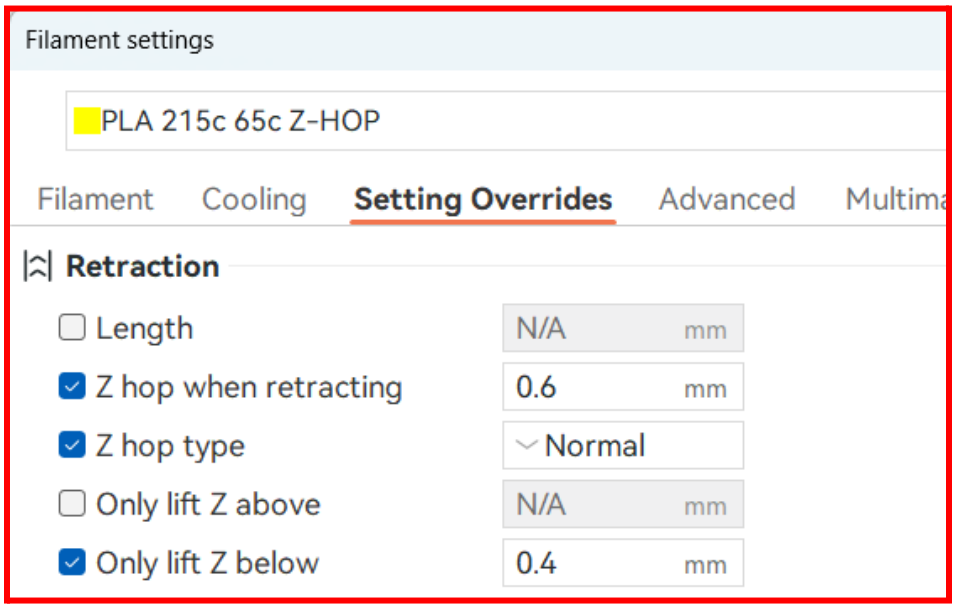
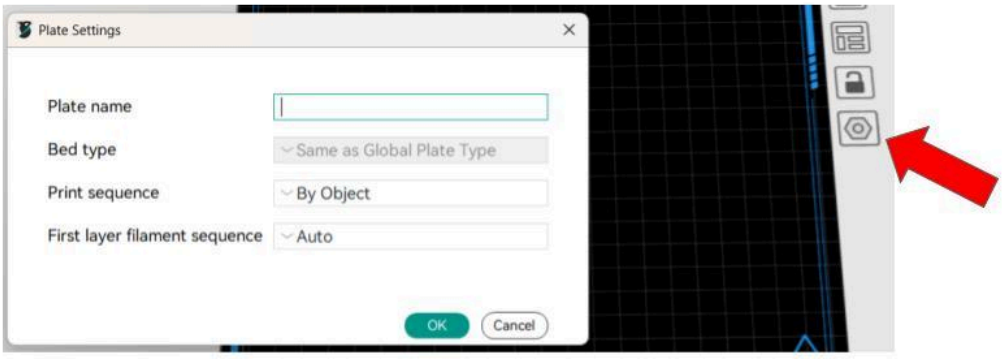
Inlay Technique with ORCA slicer

STEP 1 - Create a setup in Orca

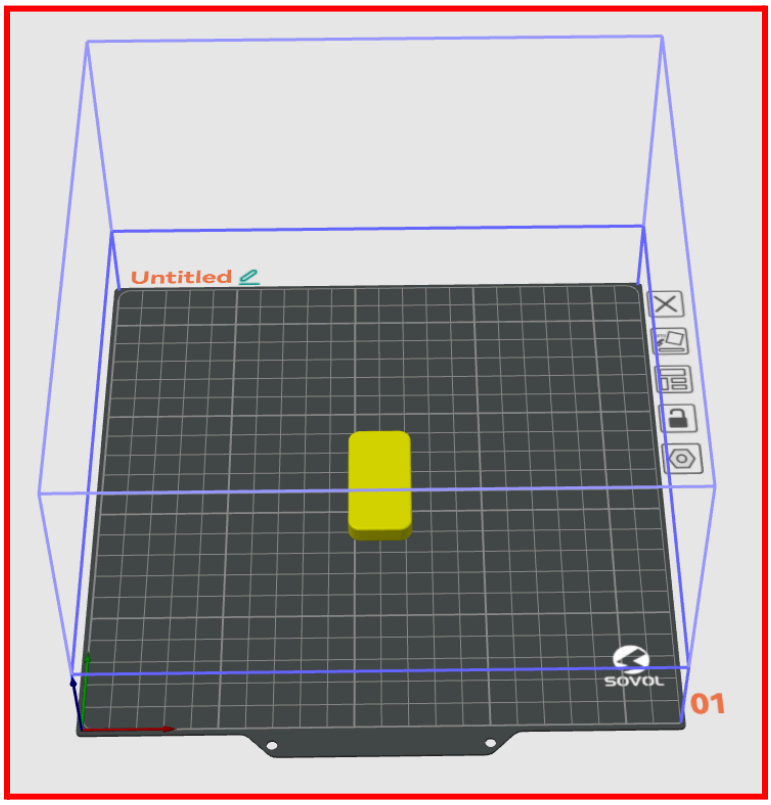
1. Create a filament with Z-HOP
   (These settings are for a layer height of 0.2 and a two layer tall inlay of 0.4)



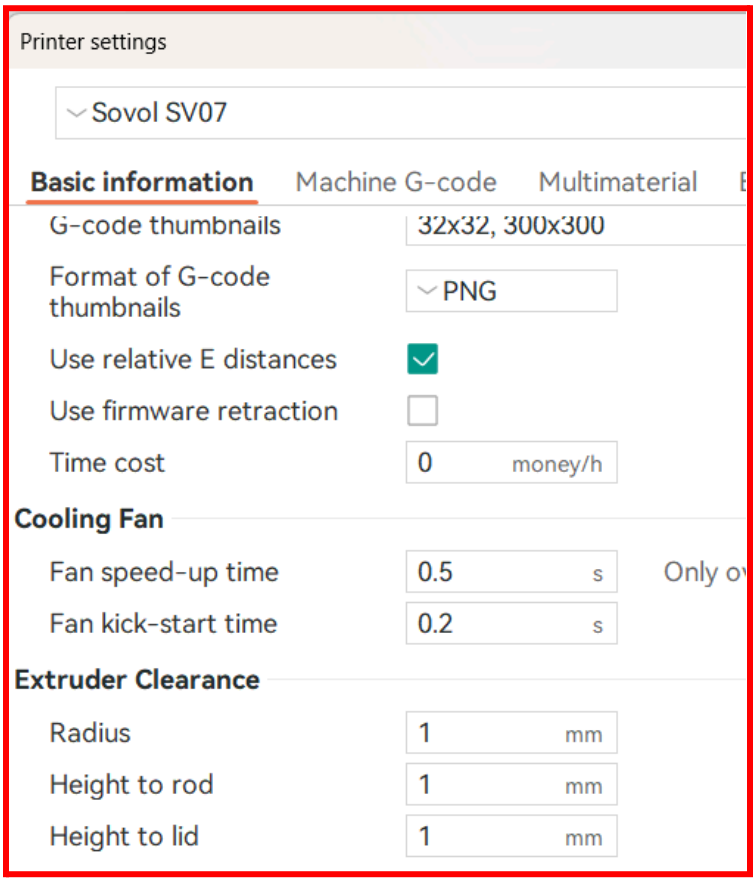
2. Adjust the plate Print Sequence to “By Object”



3. Your build plate will have this blue wire frame to show the print-by-object boundaries.

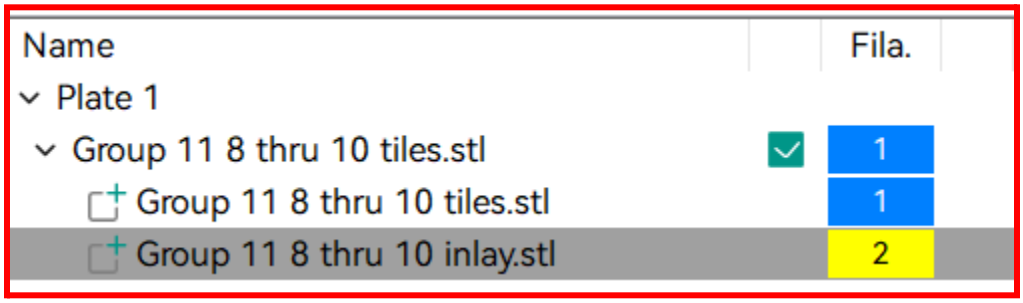


If you want to get rid of them, change your extruder clearances to 1:



STEP 2 - Import your parts

Import the main part into Orca (not the inlay)
Right click and choose ADD PART - LOAD and add the inlay part
Switch to Objects mode and change the inlay to filament 2



Slice the plate and preview. Use the path and layer sliders to check your setup. On the first layer, it will print the main part first, then the inlay. On the second layer, it should continue with the inlay filament, then switch to the main part filament.