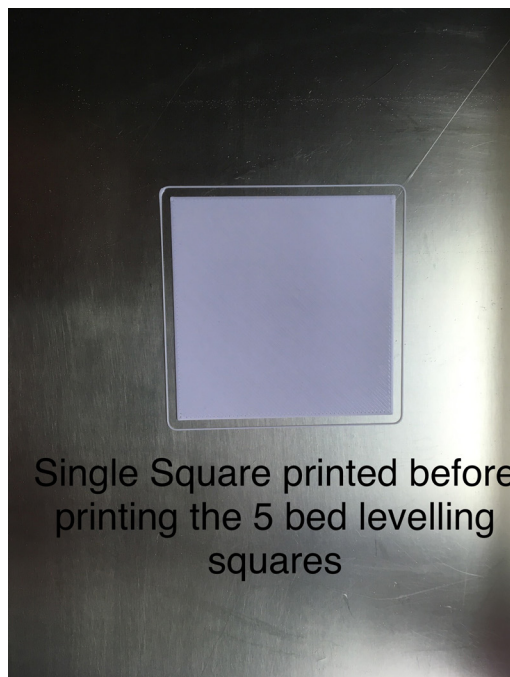
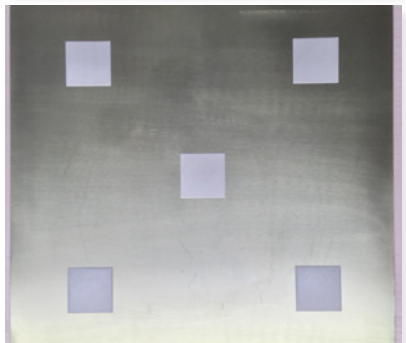
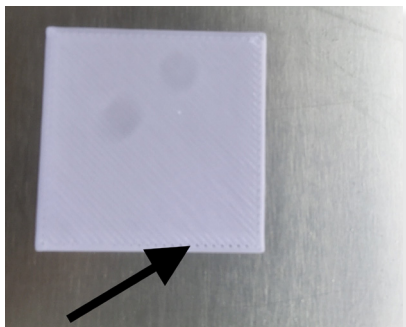


## Bed Levelling Squares

Hi guys,

I printed out the bed levelling squares given in the course material and tweaked the Z distance to get a good result. Majority part of the the squares seems good but all the squares have little gaps near the edges and corners (Please see the attached images). I tried tweaking the Z distance some more but if i reduce the distance the gaps aren't visible but the surface becomes rough and uneven and if I increase the distance the individual layers don't stick. How do I fill up the small gaps with the same Z height ? or am I doing something wrong all together?



Here are the settings I used ;

1 : Material : PLA

2 : Nozzle Temp : 210

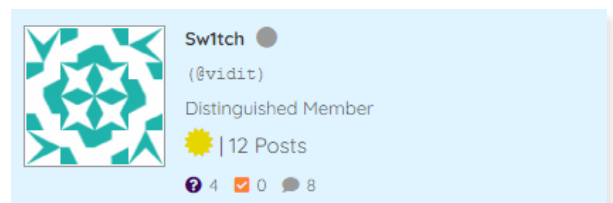
3 : Bed Temp : 60

4 : Layer Height : 0.2

5: Printer : Prusa I3 Mk3s

(All other settings were default values predefined in the slicer for PLA)

PS: I printed the single centre square before printing the 5 squares and encountered the same problem.

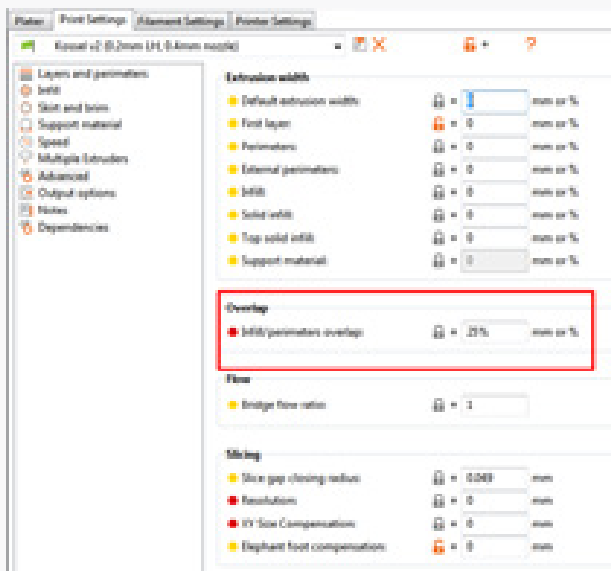


Hi SwItch,

Welcome to the community! :)

Your squares look almost perfect, but to fill in those gaps you want to increase the perimeter overlap. That makes the nozzle print a little further into the perimeters at the end of a line so will add more plastic near the points where you are currently getting holes. It might have a slightly negative effect on the quality of the surface (since the nozzle will rub on any extra plastic at the skin-to-perimeter joint and give it a matte finish).

Assuming you're using PrusaSlicer, you can find the setting under Print Settings -> Advanced -> Overlap:



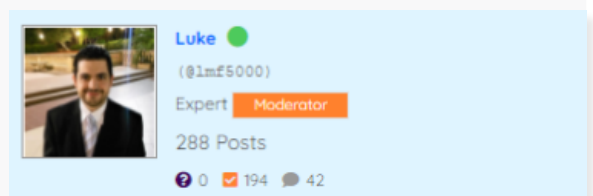
Start by increasing it by 5% and see how it goes. If need be increase by a further 5% and iterate as necessary. You want to find the sweet spot between having those gaps (overlap too little) and having overextrusion and bad surface quality where the skin meets the perimeters (overlap too big).

After that exercise, if you could print something higher, like a 20mm calibration cube, and post photos of the top layer, I will check if your extrusion multiplier needs a slight tweak, since that also influences this slightly (I ask for a cube as the nozzle-to-bed spacing also plays a significant role in the shape of the plastic for the first few layers, so

although your multiplier looks spot on, small prints like bed levelling squares are not ideal for fine-tuning the extrusion multiplier).

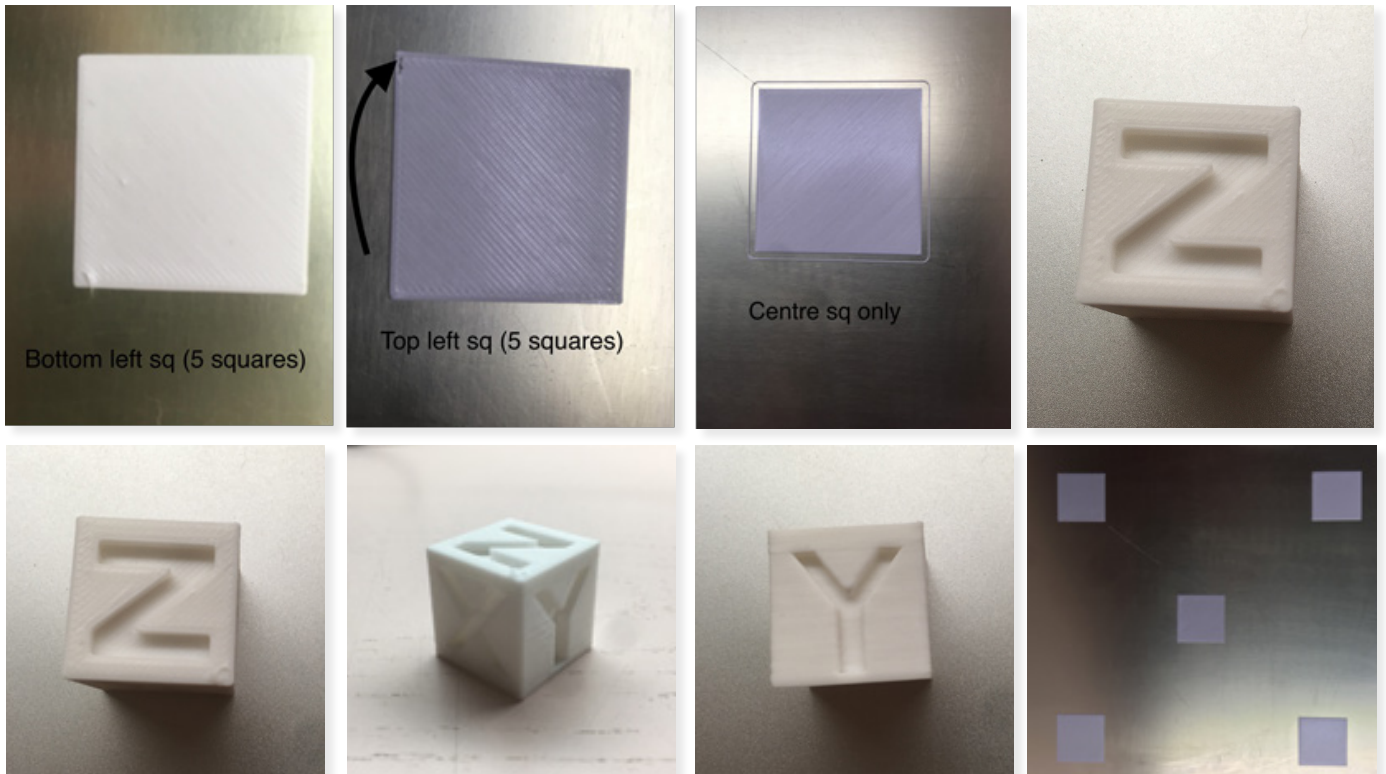
Let me know how it goes with the perimeter overlap :)

Regards,  
Luke



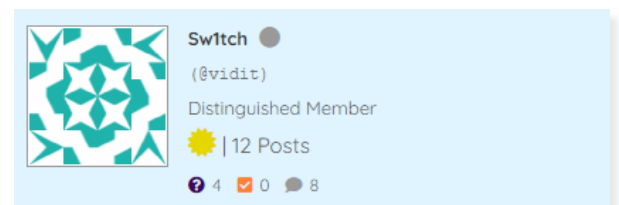
Hi Luke,

I did what you told me to and here are the results.



I had to increase the infill overlap by 15% to get these results. Also, I'm having trouble printing the top left square because the gaps in it still do not fill up completely. At times the perimeter walls doesn't print correctly. The bottom left square feels rough to touch as well with this setting but the gaps are gone, rest of them seem good. I think I'm still not there quite yet with the first layer but still went ahead with the xyz cube. Please let me know what you think.

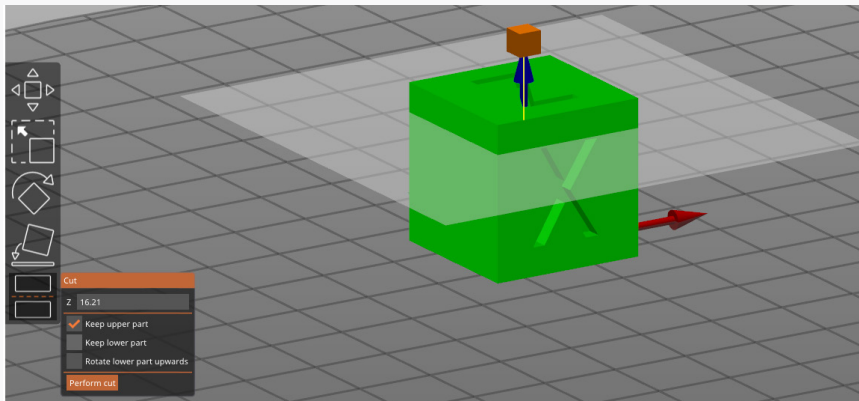
Thank you so much for your advice btw. :)  
Cheers.



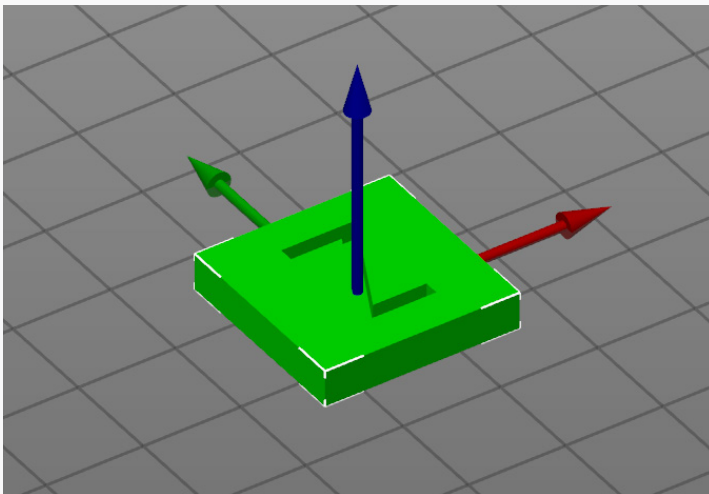
Hi Sw1tch,

That's some good progress! You appear to be very slightly over-extruding (which is why the top surfaces look a little rough), so lower the extrusion multiplier (under Filament -> Filament Settings -> Extrusion multiplier). Start by reducing it by 0.01 (that's 1%) and keep going in 1% increments until the print quality stops improving. You might need to decrease by up to 0.05 before that happens, so about 5 iterations.

For the sake of this testing, you don't need to print the whole cube (since we're just interested in the top layer and a bit of the sides), so you could use the plane cut feature like so:



And that way you print just the top few millimeters of the print and save 80% of the time and material



To get back to the original issue with the perimeter gaps, I would say your settings are 95% perfect at this point and what you're noticing now is the imperfection of the 3D printing process - by which I mean if you repeat the print again with identical settings, you might very well get a completely perfect print, or you might get a defect in a completely different place :)

Nonetheless, we'll double-check the physical setup first. Can you measure the height of

your bed levelling squares with a caliper (accurate to 0.01mm) or micrometer? The height should be roughly equal to, or slightly lower than your first layer height setting (which should be 0.2mm if you kept the Prusa defaults, meaning actual square height in the range of 0.15-0.21mm is acceptable). If the squares are much thicker (0.25-0.3mm) then you would benefit from lowering your nozzle-to-bed distance slightly (say by 0.05mm) by tweaking your probe offset, or babys-tepping, or setting a Z-offset in the slicer. The extra "squish" makes the filament weld better to every other part of the print (perimeters, skin and eventually other layers in a real print).

Other things that will help at this point are slightly lowering the first layer speed (so the filament will weld more neatly to the perimeters instead of being pulled away prematurely by the nozzle), and increasing the perimeter overlap by yet another 5% (this will degrade surface quality, but the reduction in extrusion multiplier will counteract it somewhat).

Let me know how it goes :)

Regards,  
Luke

