

Cura Instructions Manual




MAKERSPACE - CITY COLLEGE OF NEW YORK



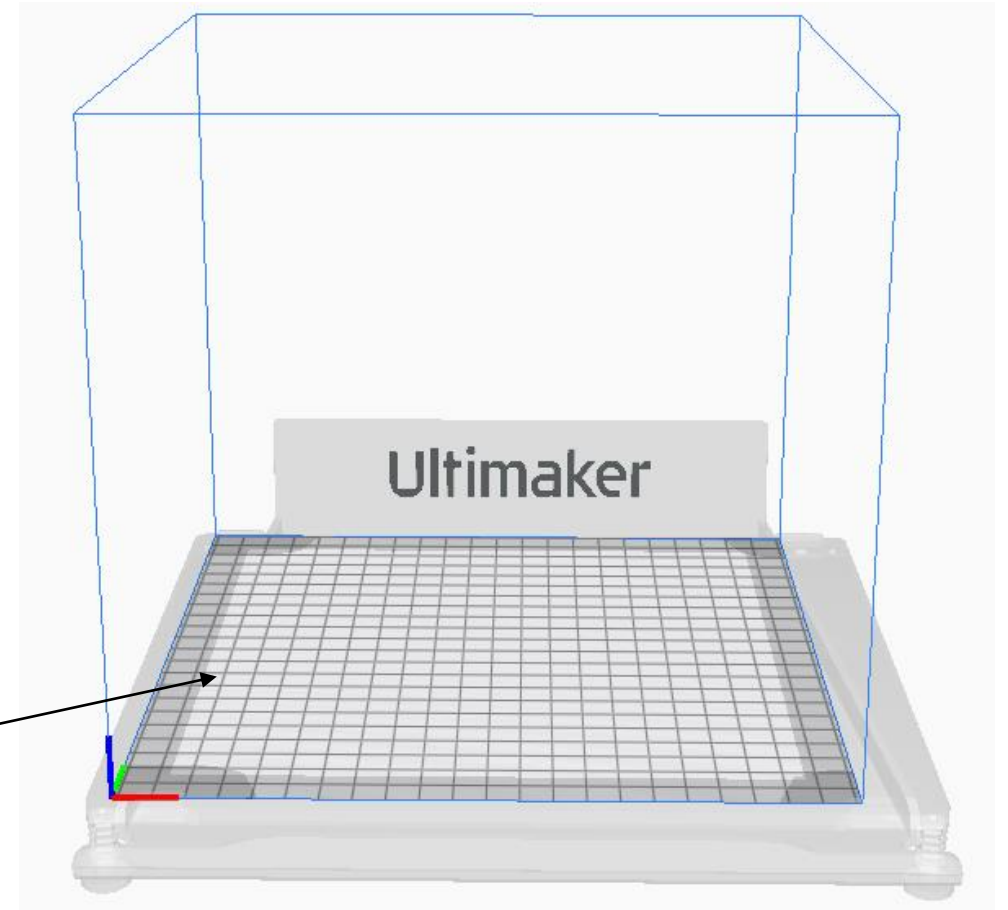
File Formats for Printing with Cura

- Cura can open the following file types:
 - 3MF File (.3mf)
 - AMF File (.amf)
 - COLLADA Digital Asset Exchange (.dae)
 - Compressed COLLADA Digital Asset Exchange (.zae)
 - Open Compressed Triangle Mesh (.ctm)
 - STL file (.stl)
 - Stanford Triangle Format (.ply)
 - Wavefront OBJ File (.obj)
 - X3D file (.x3d)
 - glTF Binary (.glb)
 - glTF Embedded JSON (.gltf)
- We use STL (.stl)

Placing an Object into Cura

- 3 ways to load a part
 1. Click on  -> locate file to open.
 2. Click on file -> open file -> locate file top open
 3. Locate file -> click and drag into build plate area

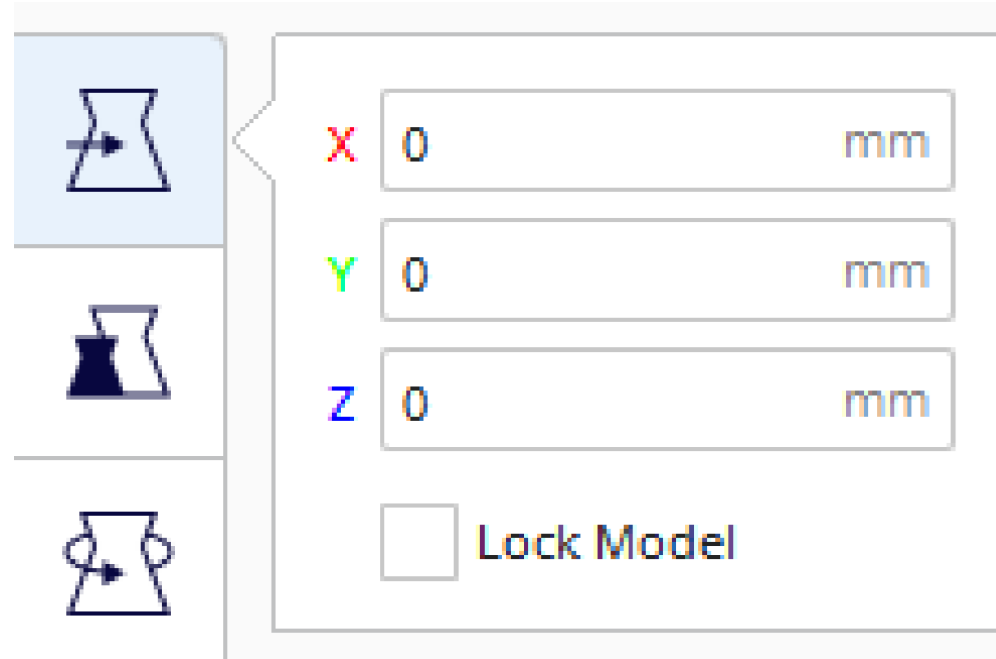
Build plate area



Object Settings

- Moving the Object

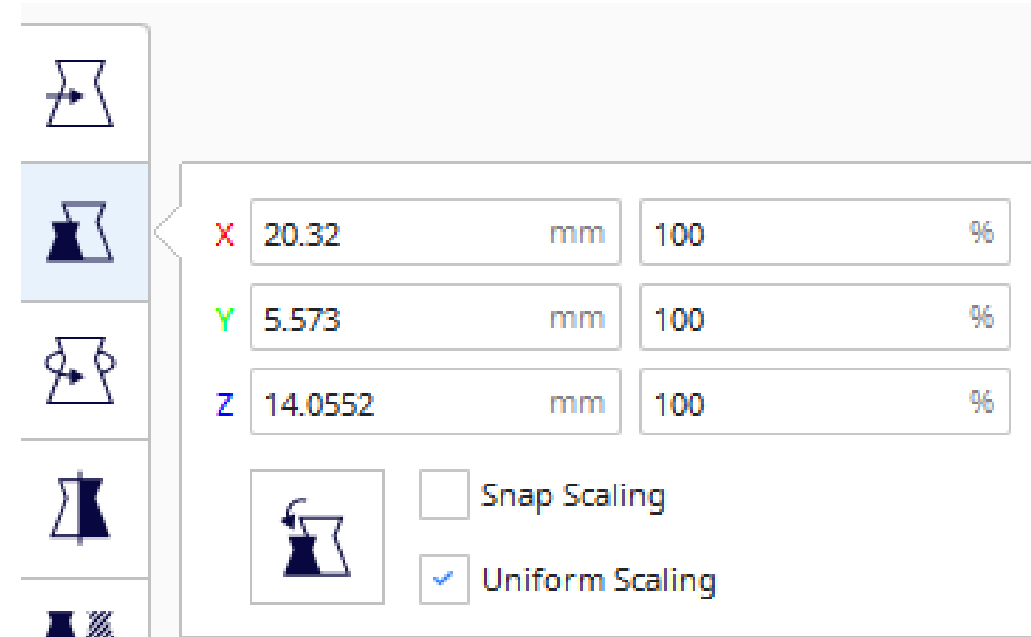
- When you click on the part a menu on the left-side will appear
- Click on the top icon to move either in x,y,z directions
- Can type in numbers
- Can also click and drag the part anywhere on build area
- Check the box to lock in that position



X	0	mm
Y	0	mm
Z	0	mm
<input type="checkbox"/> Lock Model		

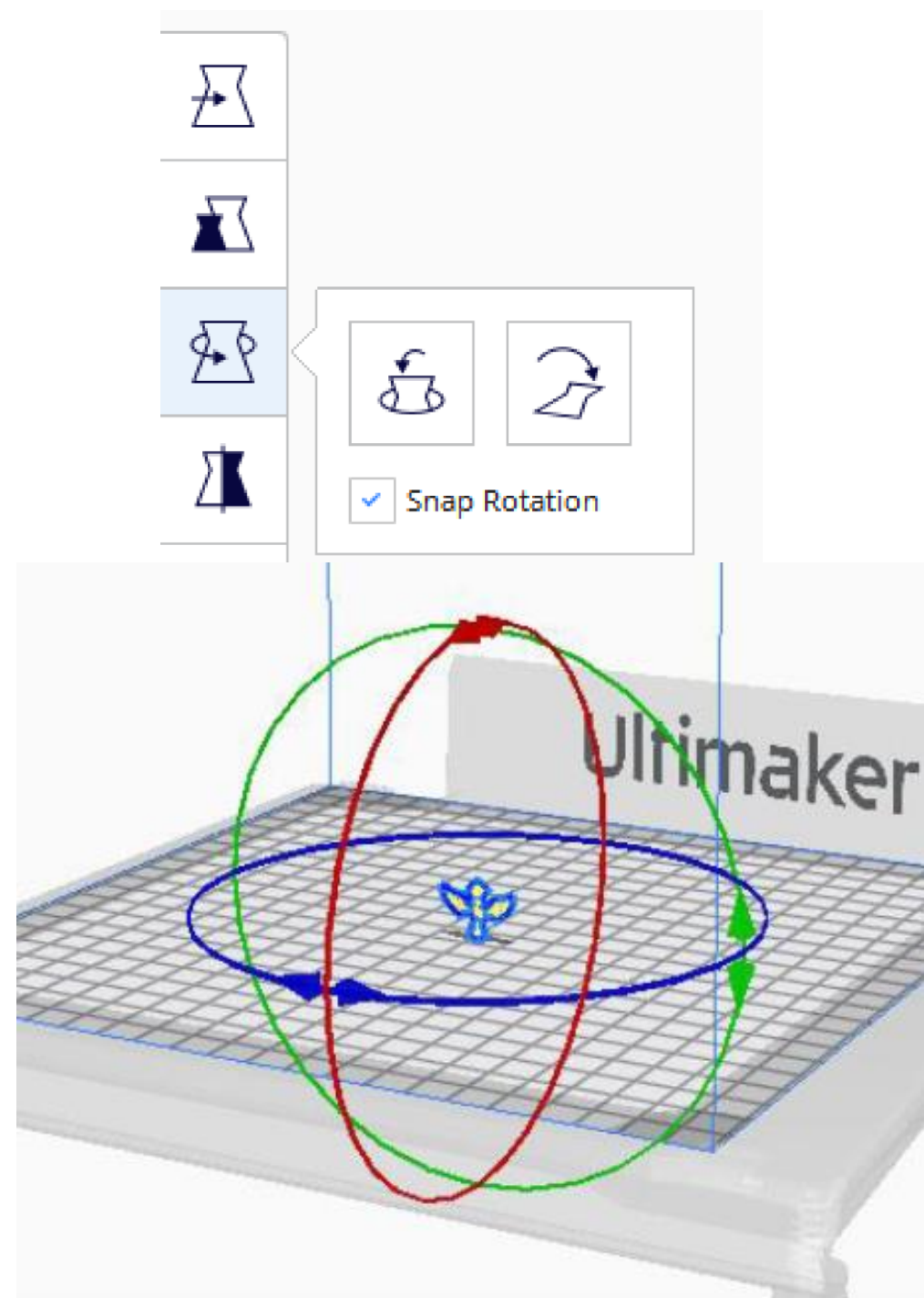
Object Settings

- Scaling the Object
 - When you click on the part a menu on the left-side will appear
 - Click on the 2nd icon to scale either in x,y,z directions
 - Can type in numbers in mm or %
 - Can also click and drag the arrows on part
 - Uniform scaling scales all directions to keep the same ratio as you change one parameter
 - Snap scaling scales by increments of 10
 - Bottom left button resets to original dimensions



Object Settings

- Rotating the Object
 - When you click on the part a menu on the left-side will appear
 - Click on 3rd top icon to rotate either in x,y,z directions
 - Click and drag the arrow anywhere on part
 - Check the box to snap rotation by increments of 15°
 - Left button resets to original orientation
 - Right button lays part flat onto build plate



Material Settings

- Only 2 parameters:
 - Nozzle: 0.4mm
 - Plastic: PLA (generic/Ultimaker)

1 Generic PLA
0.4 mm

Custom

Material

PLA

Nozzle

0.4 mm

1 Ultimaker Black PLA
0.4 mm

Custom

Material

Black PLA

Nozzle

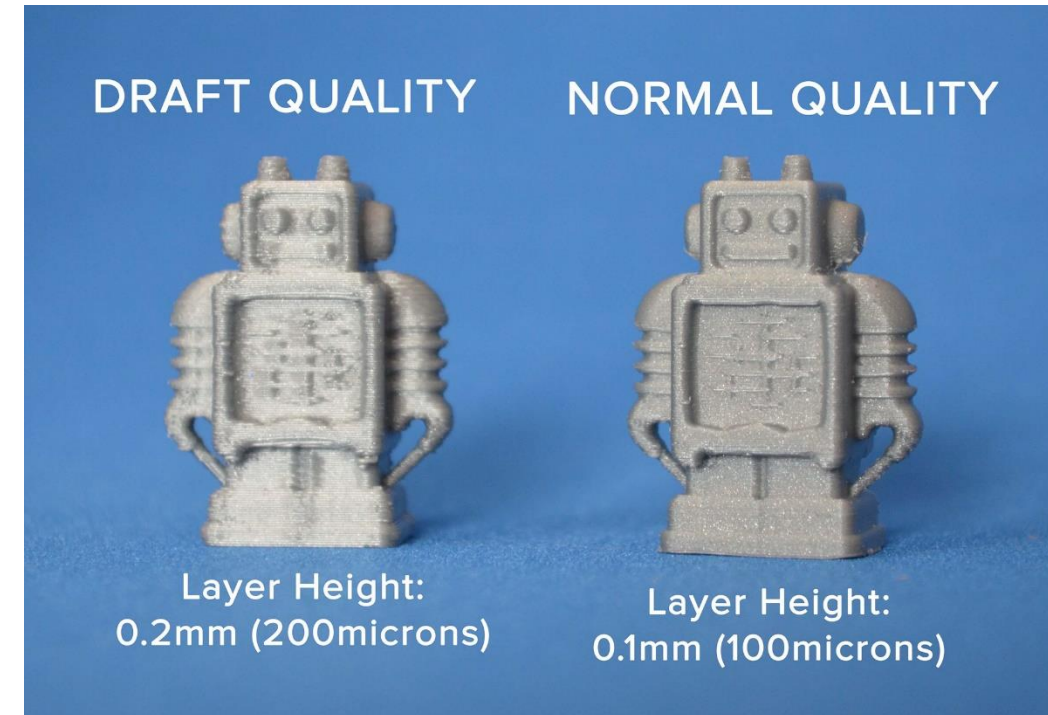
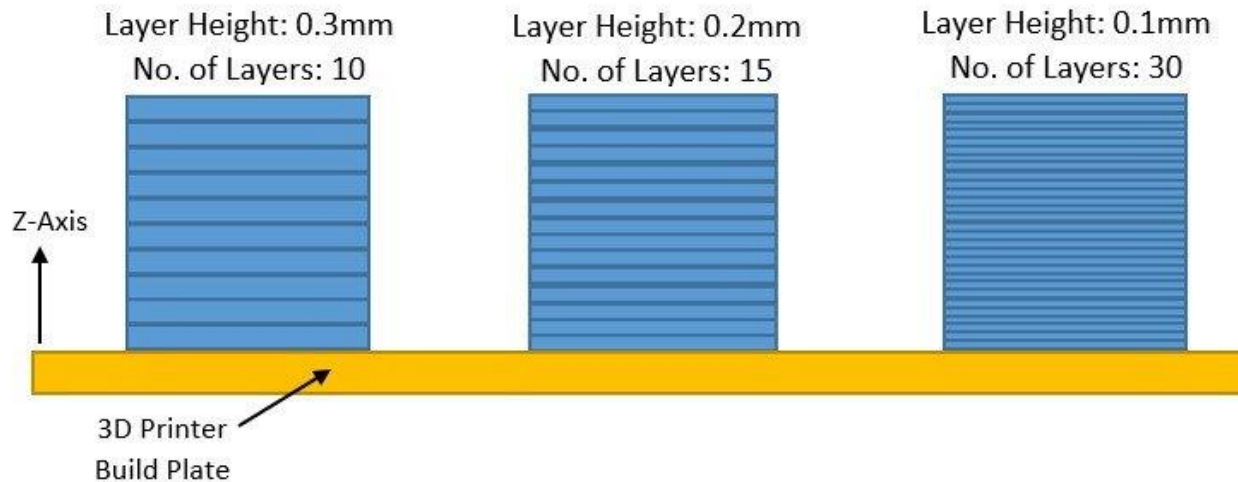
0.4 mm

Printer Settings

- Several parameters that should be changed per job:
 - Layer height/profile/quality
 - Infill
 - Support
 - Build plate adhesion

Printer Settings

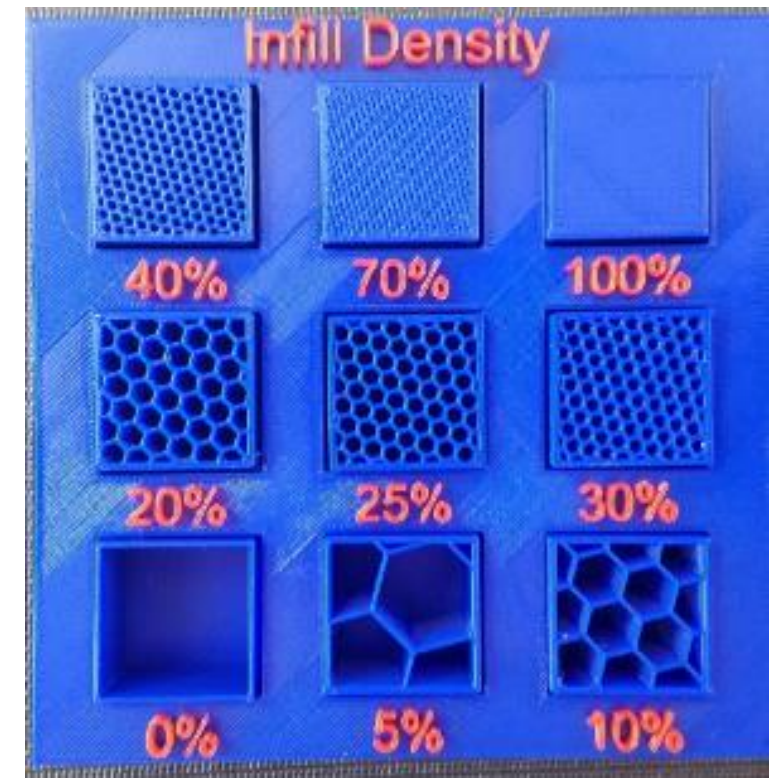
- Layer height/profile/quality: how thin of a layer it prints
 - Affects quality of print – fine, normal, draft
 - Smaller layer heights take longer to print – results in smooth, detailed part
 - Larger heights are quicker to print – fine details will be missed
 - Fine: 0.1mm
 - Normal: 0.15mm
 - Draft: 0.2mm



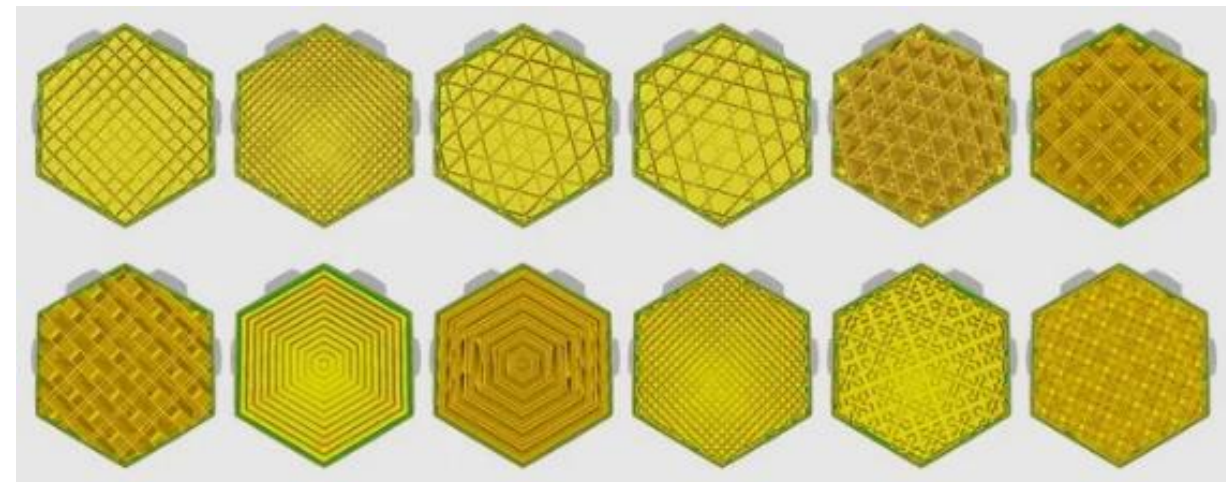
https://pro2-bar-s3-cdn-cf5.myportfolio.com/0ef57db35ca69e98c25cf8f88b6f790a/66d2fa21-bb6e-49d0-8205-ade017e81f9b_rw_1920.jpg?h=3e4cbe5274b5d26344476db09add7696

Printer Settings

- Infill: plastic filling in between the outer walls of the object
 - Gives it structural integrity and strength
 - Ranges from 0-100%: 0% is none, 100% is completely filled dense part
 - Can choose different geometries: some are stronger than others
 - More infill, stronger object, longer to print
- Recommendations:
 - Display: 10-15%, grid
 - Normal: 40%, grid
 - Heavy Use: 80-100%, cubic subdivision



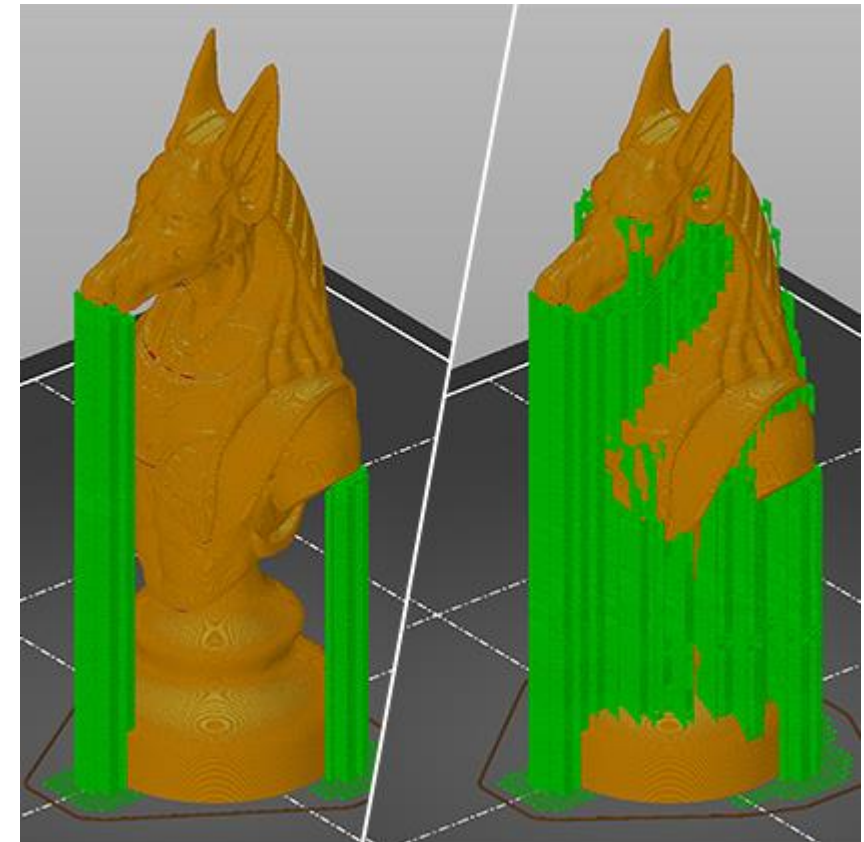
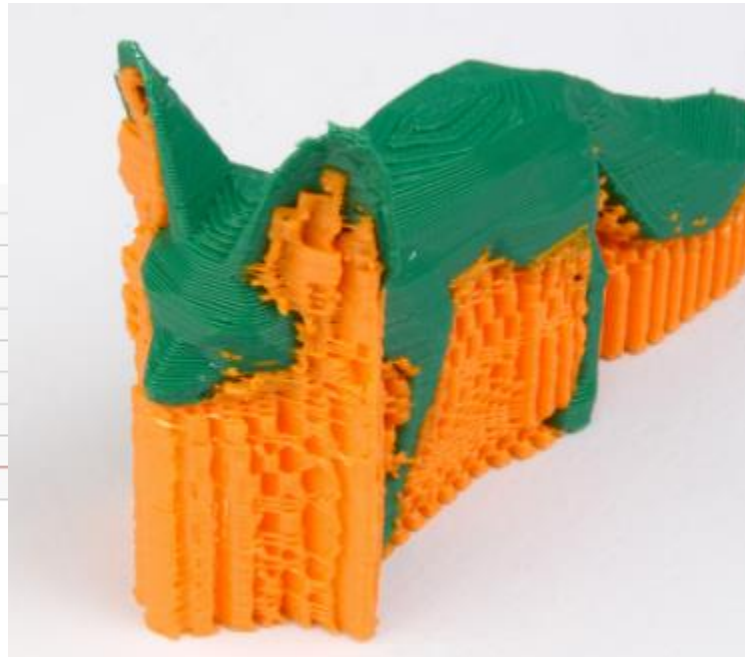
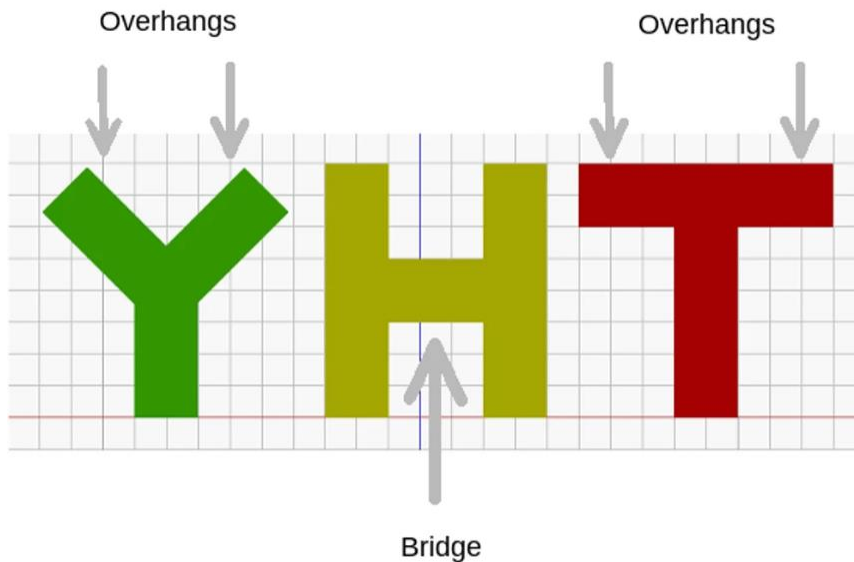
<https://assets.pinshape.com/uploads/image/file/98144/display-tray-for-infill-pattern-and-infill-density-3d-printing-98144.jpg>



https://cdn.thingiverse.com/renderers/03/1b/b0/54/2e/5ddd44d2650279cae9da5f9089f45495_pre-view_featured.jpg

Printer Settings

- Supports: Extra removable plastic to help print overhangs and bridges
 - Necessary for angles above 45-50°
 - Supports can be printed to build plate or part



https://i.all3dp.com/cdn-cgi/image/fit=cover,w=1000,gravity=0.5x0.5,format=auto/wp-content/uploads/2018/04/26153119/yht_overhang_bridge.png

<https://encrypted-tbn0.gstatic.com/images?q=tbn:ANd9GcQp5taBRZmBBGt78y32w3iGlcWLzc7deJSjKg&usqp=CAU>

<https://cdn.help.prusa3d.com/wp-content/uploads/supports.png>

Printer Settings

- Build plate adhesion: to reduce chances of print failures and prime extruder
 - Skirt: A single layer around the part but doesn't touch it
 - Brim: Single layer and the part right on top of it
 - Raft: Thick foundation and then part on top

