

WOOF3D

How to 3D Print



Download and install Ultimaker Cura (free)

Ultimaker Cura

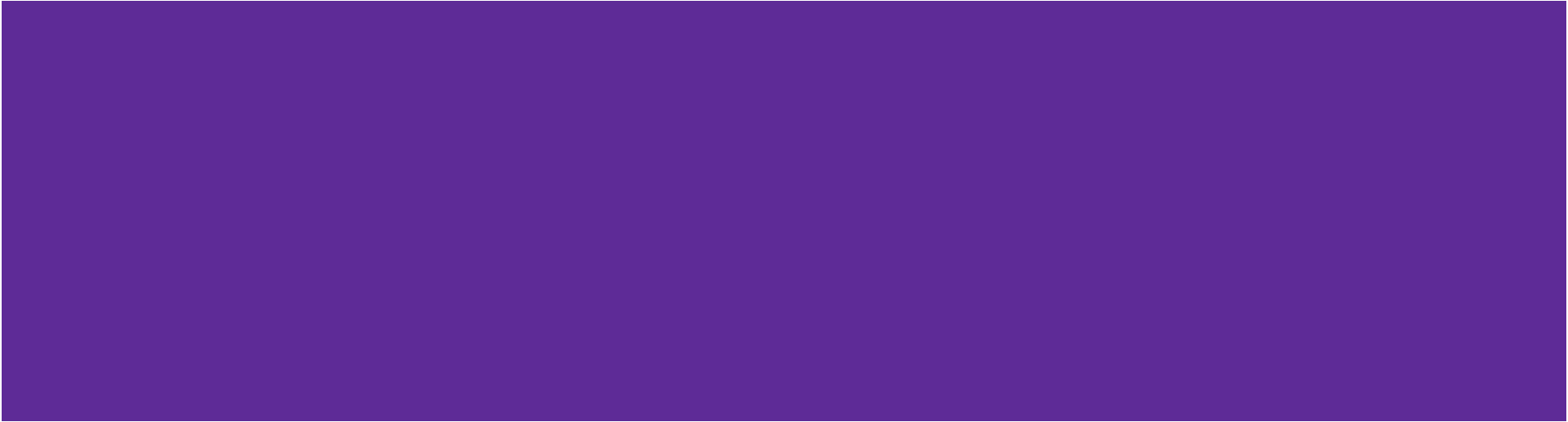
Trusted by millions of users, Ultimaker Cura is the world's most popular 3D printing software. Prepare prints with a few clicks, integrate with CAD software for an easier workflow, or dive into custom settings for in-depth control.



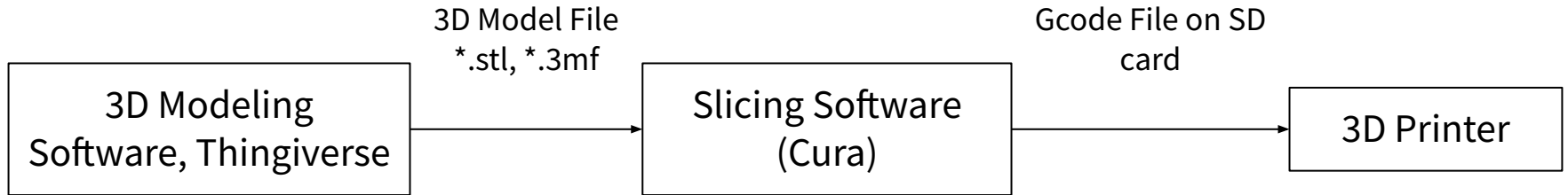
Ultimaker Cura 4.11

[Download for free](#)

How to 3D Print



Workflow



Thingiverse



Considerations when Using Thingiverse

- Check Thing Details for suggested settings
 - Will often show supports, bed adhesion, etc
 - If not, use your best judgement
- Comments for additional insight into prints
 - More makes is generally better but not always
- Download your .stl file from the “Thing Files” tab

Thing Details

1
Thing Files

63
Comments


591
Makes

67
Remixes

0
Apps

Low-Poly Bulbasaur


Download All



bulbasaur_starter_1gen_flowalisti...
17 kb | Updated 05-11-2014 | 194638 Downloads

Download

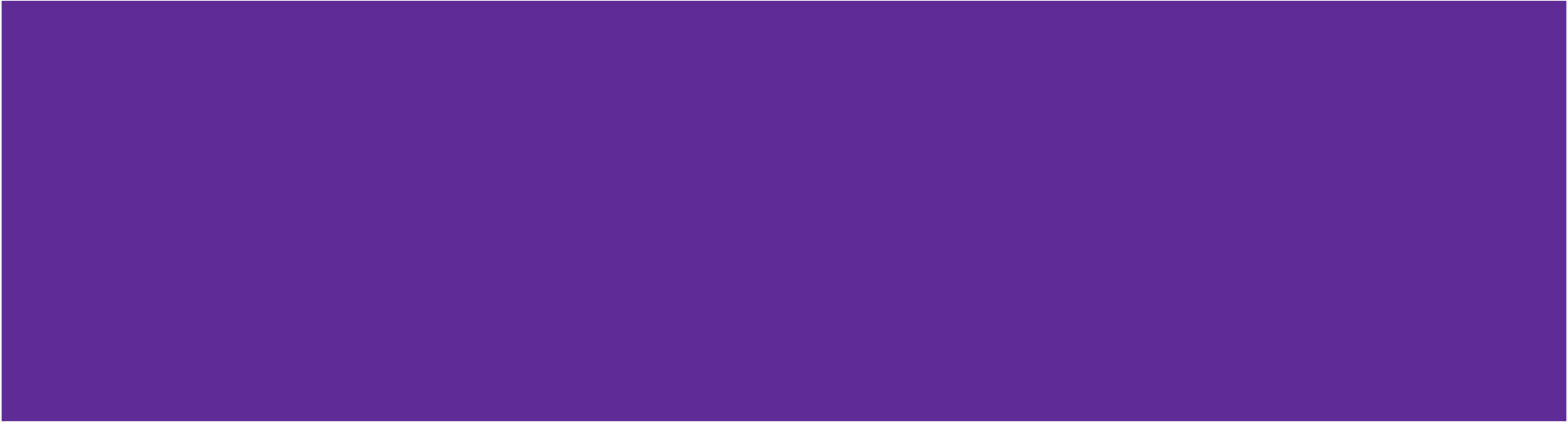
License



Low-Poly Bulbasaur by FLOWALISTIK is licensed under the [Creative Commons - Attribution - Non-Commercial - Share Alike](#) license.

By downloading this thing, you agree to abide by the license: Creative Commons - Attribution - Non-Commercial - No Derivatives

Slicing Software (Cura)

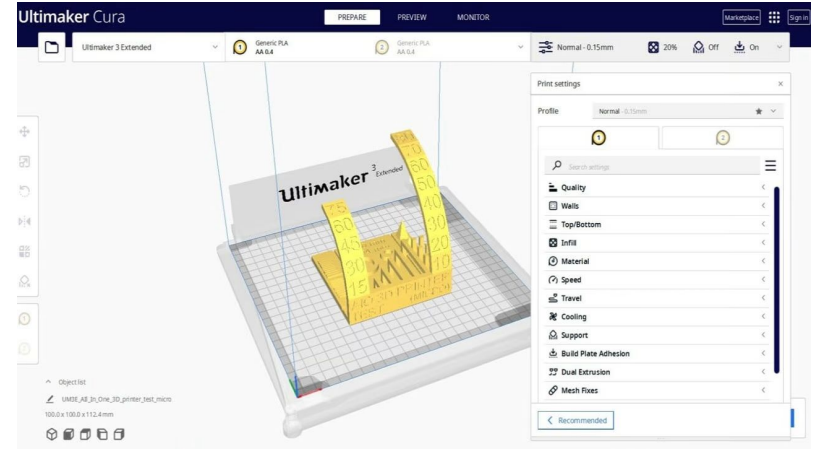


Ultimaker Cura

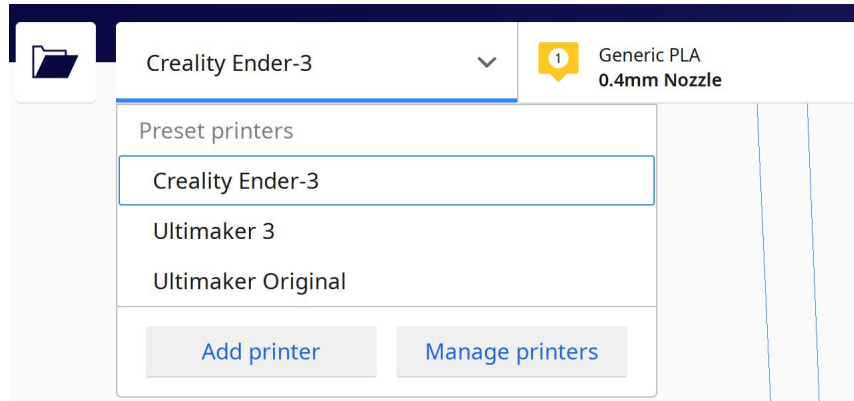
- Most popular slicer that supports many different brands of 3D printers
- Ender 3 uses Cura

Notes:

- For Prusa's, PrusaSlicer is preferred
- For Dremel, DigiLab (Cura-based)



Getting Started with Cura



- Download off of the Ultimaker Website
- <https://ultimaker.com/software/ultimaker-cura>
- Set up your printer profile (Creality Ender 3)
 - Should be correct, but always a good idea to check
 - 235x235 bed, 0.4mm nozzle

Important Settings

- Layer Height
- Infill
- Supports & Part orientation
- Bed Adhesion
- Temperatures

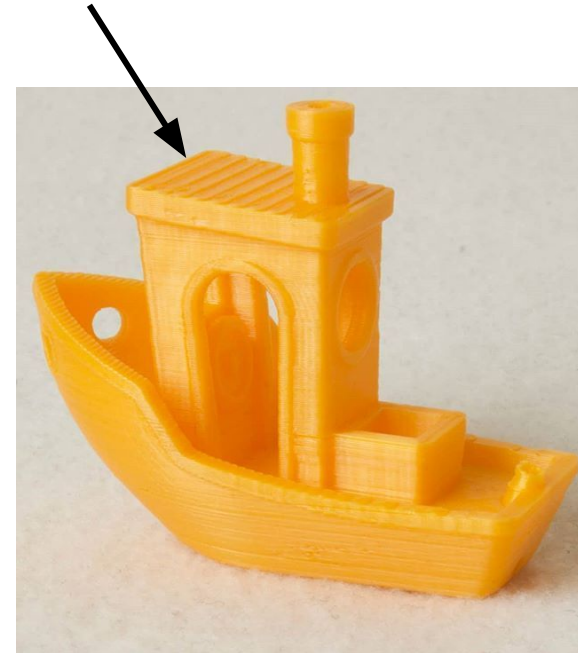
Other settings are for troubleshooting prints, we may cover them another meeting.

Some of these settings will take some experimentation to find what works best

Layer Height

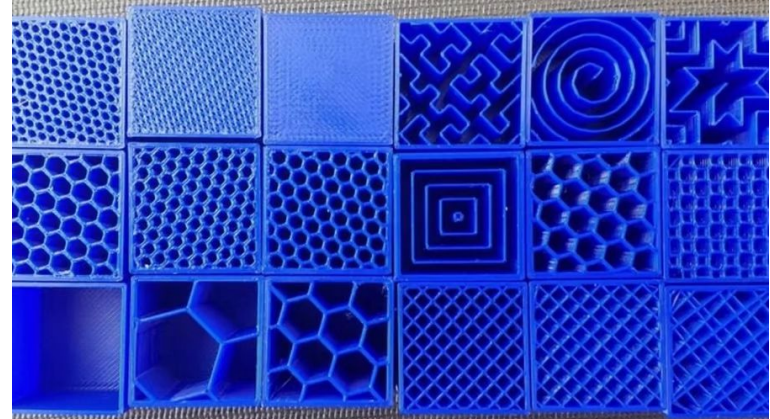
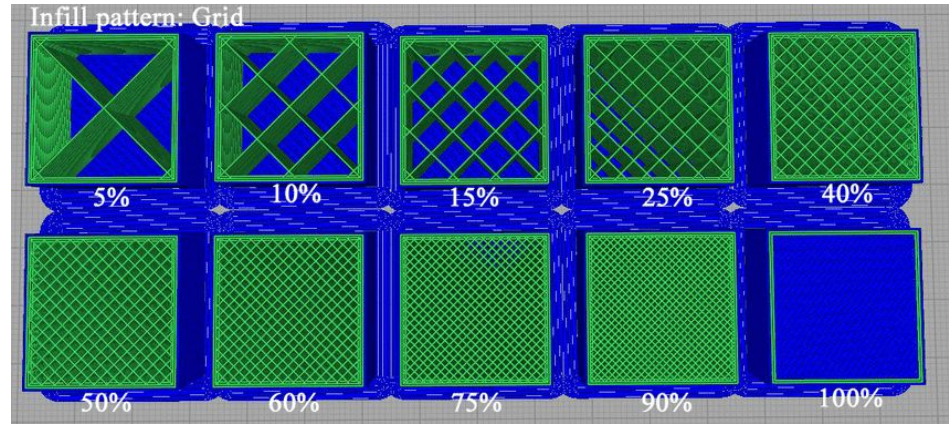
- The thickness of each layer printed
- Thicker layers = less layer needed to finish print, faster print
- Thinner layers = less visible layer lines, nicer looking print
- ~0.05mm to 0.35mm
- Some shapes make layer lines more visible than others
 - Top of sphere
 - Gradual incline
- Larger prints have less need for smaller layers

0.20 mm	0.10 mm
1 h : 50min	3 h : 45min



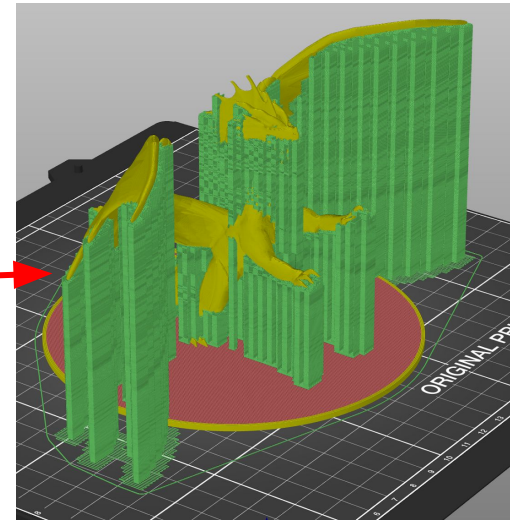
Infill

- 3D prints are not solid plastic, they are partially hollow
- Filled with a pattern called infill to maintain rigidity while decreasing material and time needed to print
- 10%-30% density is all that's necessary for most prints
- Different infill patterns to choose from, default is probably the best, most of the others are silly



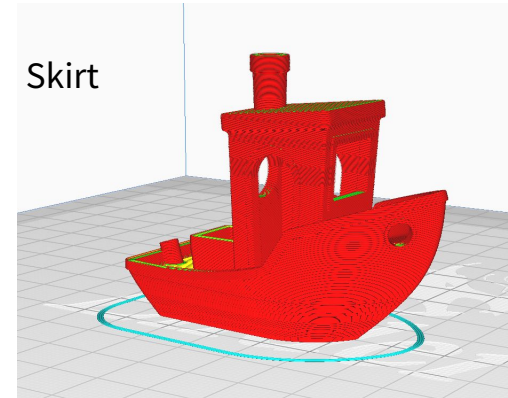
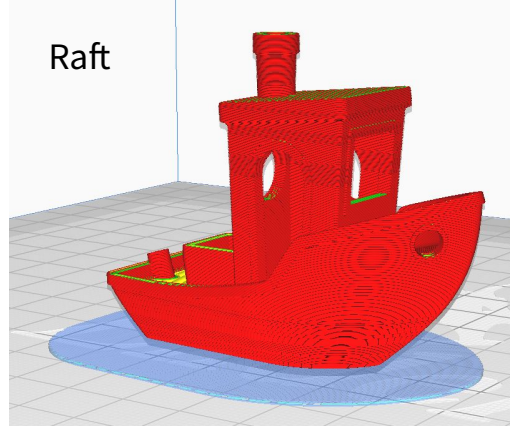
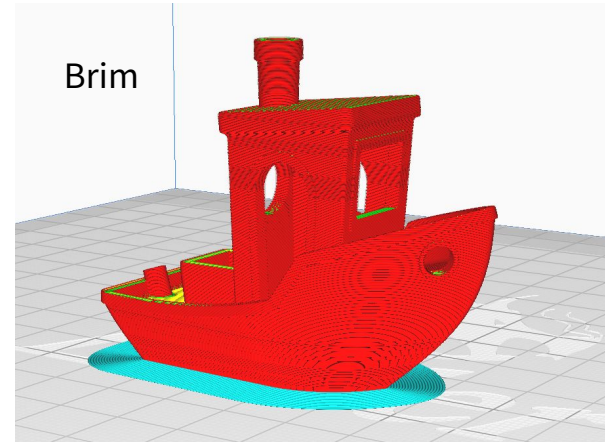
Supports & Part Orientation

- Since 3D printers stack layers from the bottom up, some details might start floating in thin air
- This can cause prints to fail
- Support material is used to give these details something to build on, and are removed once the print is finished
- Supports extend print time, use extra material and are difficult to remove
- Removing supports from thin/fragile sections can destroy a print
- Reorienting a model can reduce the amount of supports needed
- Many models designed for 3D printing use as few supports as possible



Bed Adhesion

- Prints need to stay adhered to the print bed for the duration of the print
- Models with low surface area in contact with the bed are at risk of coming off mid-print
- Some material can be added to help the print stick down better, is removed after printing
- **Brim**: Extends the first layer, adding extra perimeters around the print. Difficult to remove
- **Raft**: Creates a platform underneath the print, uses more material, can be difficult to remove
- **Skirt**: Doesn't help with print adhesion, use if not using a brim or raft

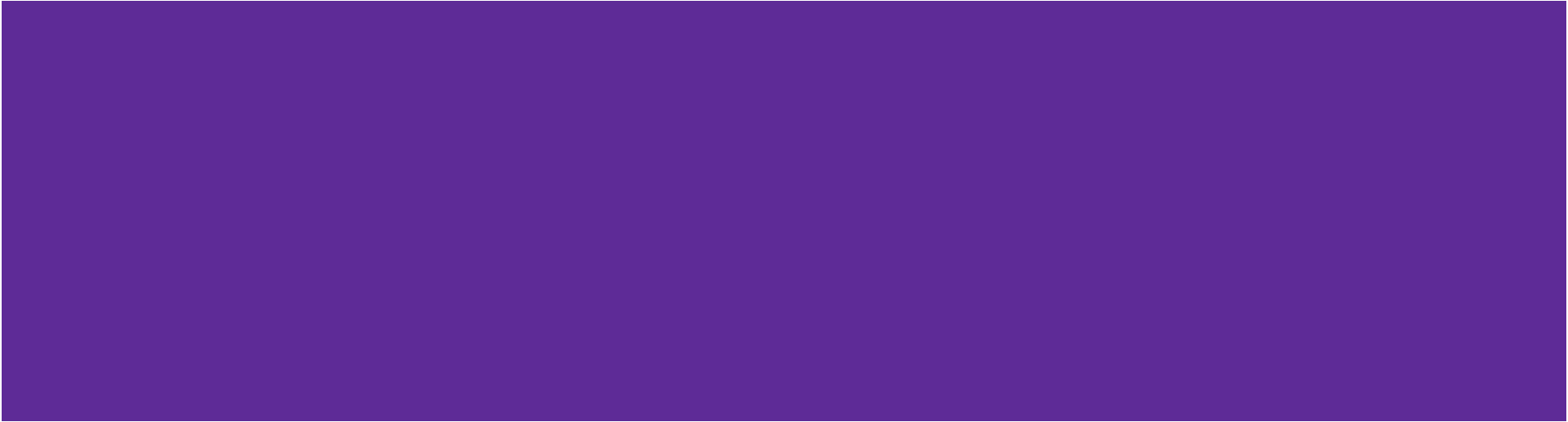


Print Temperature

- Determined by the material you use
 - We will only be using PLA for safety
- Spools are generally labelled with extruder temperature ranges
- PLA is roughly 180°-220° C
- Bed temperatures are sometimes labelled on spools
- PLA bed temperatures are generally 40°-70° C
- May take some experimentation to find the best extruder/bed temperature



Starting a print on an Ender 3



Setup Process

- Check if bed is clean/apply adhesive if needed
- Insert microSD
- Autohome, (possibly) check levelling
- Start preheat
- Load filament
- Start up; what to watch for in a first layer
- Quickly go through screens you will need to use on the Ender

