CAMD Makerspaces

3D Printing User Guide

Table of Contents

Choosing a slicer

A slicer is software that prepares 3D models for printing. It allows the user to customize settings before converting it into G-code, making the file readable by printers.

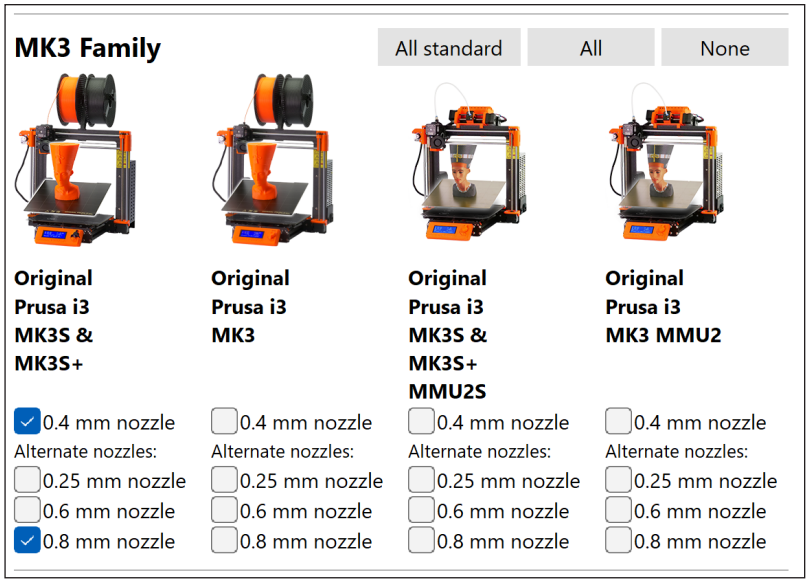
Cura is our default slicer for CAMD. With it, you can use our default profiles with both the Prusa (Ryder) and Ultimaker (Ruggles) printers for all standard prints. **Ultimaker users should use Cura, as Ultimaker is not currently supported by PrusaSlicer.**

PrusaSlicer allows for advanced functions, such as paint-on supports, optimized tree supports, and variable layer height. Both slicers prefer **STL** files.

Using PrusaSlicer

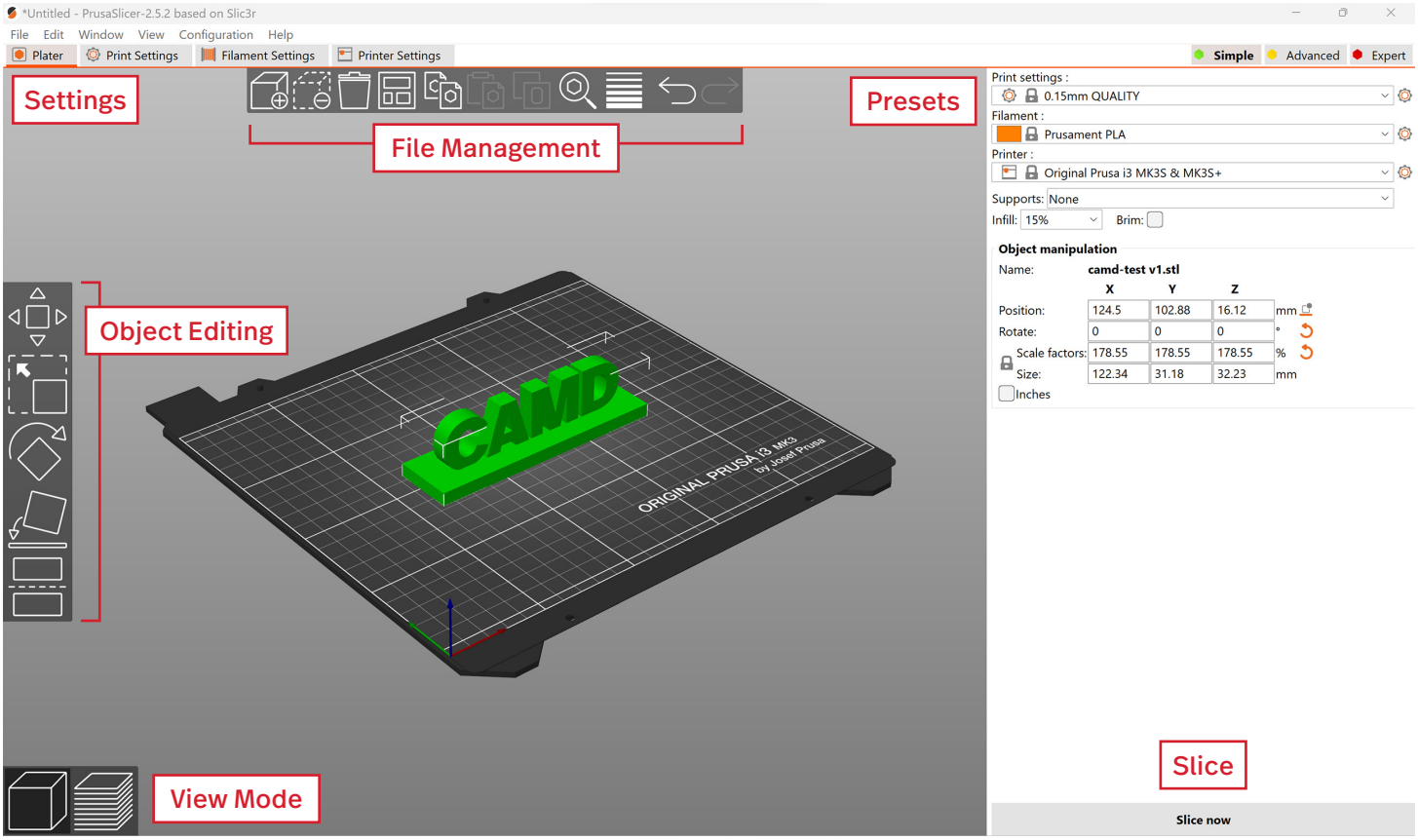
Software Installation

1) Download the slicer at https://www.prusa3d.com/page/prusaslicer_424/
2) Install for Windows/Mac/Linux.
3) Open the application. This should run the Configuration Wizard/Assistant. Scroll to the MK3 Family and check 0.4 mm and 0.8 mm nozzle under Original Prusa i3 MK3.



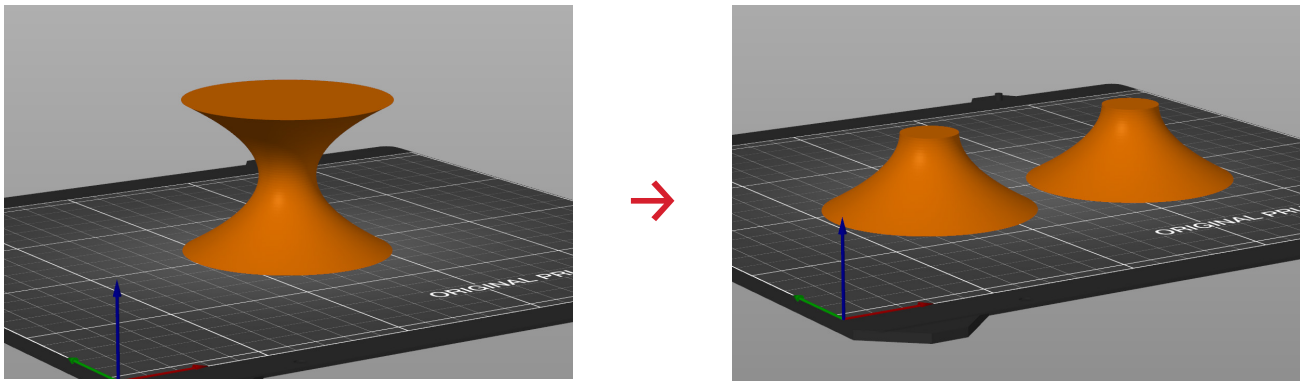
4) To include TPU in filament profiles, open “Filaments,” select the “Generic” vendor, and check “Generic FLEX.”
5) You can now press **finish**.

Prusa Basics



File Management: Insert, remove, and arrange files; variable layer height.

Object Editing: Move, scale, rotate, and cut objects. Cutting objects is ideal for optimizing prints that would otherwise need supports or be difficult to print, as shown below. These parts can be glued together later.



View Mode: Switch between editor and preview mode. Preview is a great way to ensure that everything will print normally; note that details under .4mm (the size of the nozzle) cannot be printed.

Presets: Print presets for **Settings**. Layer height, infill, and supports are essential settings to configure; read more about them in the **Optimization** section (page 9). Specifics can be set using the gear on the right or opening the tabs shown by **Settings**.

Slice: Slice and then export your file.

Settings

Most settings can be left as is. The ones you might want to change are under **Print Settings**; Filament and Printer settings should remain default.

Layers and perimeters: Layer height

Infill: Fill density

Skirt and brim: Brim type

Support material: Generate and Auto generated (both of these should be selected if using supports); Build plate only (eliminates supports that don't touch the build plate)

Using Cura

Software Installation: Mac

1) Download Cura here: https://ultimaker.com/software/ultimaker-cura/
2) Open "Ultimaker_Cura-x.x.x-Darwin.dmg" and drag "Ultimaker Cura" to the "Applications" folder.
3) Go to your applications folder and open "Ultimaker Cura"

4) You may get a notice stating that Cura cannot be opened because Apple cannot check it for malicious software. If so, complete the following steps:
    a. Click “Show in Finder”
    b. Right click on “Ultimaker Cura”
    c. Click “Open”
    d. In the dialog that appears click “Open”
    e. Cura should now launch correctly from now on
5) Continue to **Printer Setup** below

Software Installation: Windows

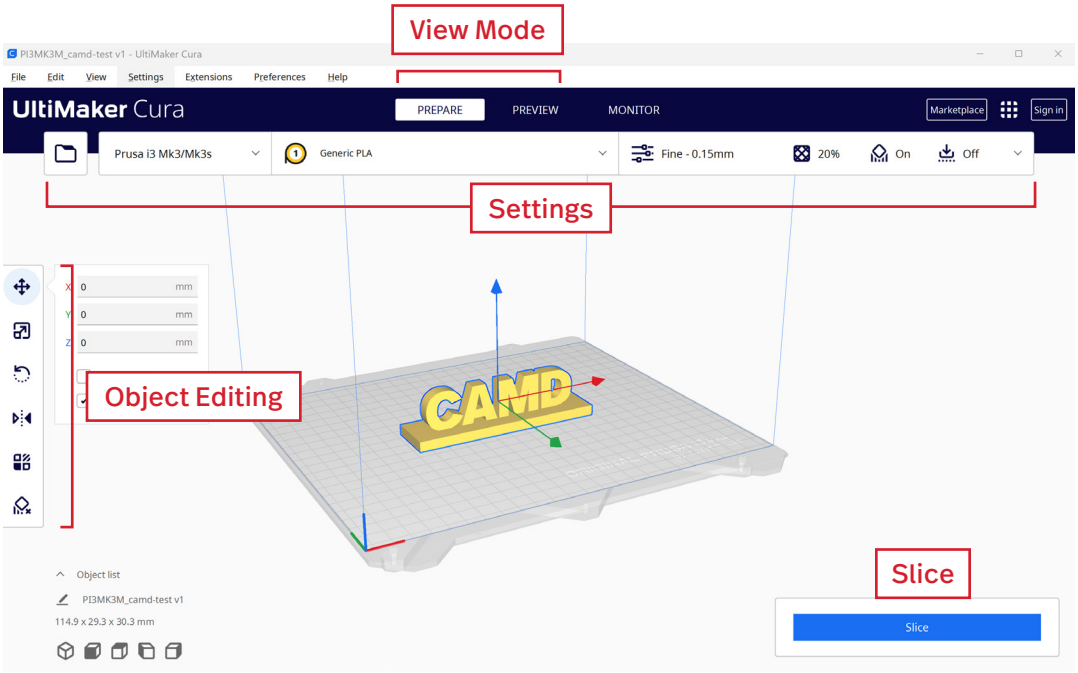
1) Download Cura here: https://ultimaker.com/software/ultimaker-cura/
2) Open “Ultimaker_Cura-x.x.x-win64”
3) Click through the installer until you reach the “Choose Components” screen
4) In addition to the items already checked, check “Open 3MF files”, “Open OBJ files”, and “Open GCODE files”
5) Click Install
6) Once Cura is done installing you should be able to open it from the start menu
7) Continue to **Printer Setup** below

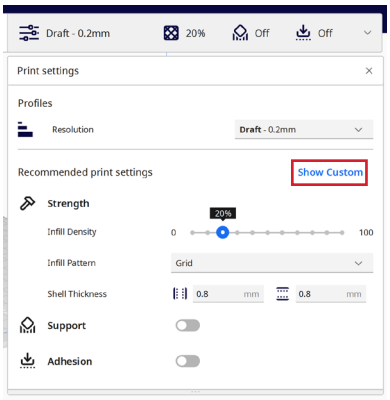
Printer Setup

1) Inside Cura, click through the prompts until you reach the “Add Printer” screen
2) For **Prusa** users:
    a. Click “Non Ultimaker printer”
    b. Click “Add a non-networked printer”
    c. Scroll down to “Prusa3D” and select the Prusa i3 Mk3/Mk3s, then click Next/Add

3) For **Ultimaker** users:
   a. Click “Ultimaker printer,” then “Add local printer”
   b. Click “Add a non-networked printer”
   c. Select “Ultimaker S5” (the larger printers) or “Ultimaker 2+ Connect” (the smaller ones) based on which you will be using
   d. Click Next/Add

Cura Basics





Expand print settings and select “show custom” to finely change layer height (quality), infill, supports, and build plate adhesion.

Settings: Add file; select printer, filament, and print settings. Layer height, infill, supports, and build plate adhesion are essential settings to configure; read more about them in the **Optimization** section.

Object Editing: Move, scale, and rotate objects.

View Mode: Switch between prepare and preview mode. Preview is a great way to ensure that everything will print normally; note that details under .4mm (the size of the nozzle) cannot be printed.

Slice: Slice and then export your file.

Printer Use

Prusa

1) To start print:
    a. Press the knob to reveal the menu.
    b. Scroll down to “Print from SD”
    c. Select file from SD card
    d. Start print
    e. Write your name + file name so we can restart your print if it fails
    f. Make sure you stay for the first few layers to ensure that it is printing smoothly!
2) To remove print:
    a. Remove build plate from printer
    b. Flex the plate to free the printed part(s)
3) If the plate is dirty, you can clean it with a sponge and hand or dish soap under a trickle of warm water. **Make sure the plate is completely dry before re-attaching it.**

Ultimaker

1) To start print:
    a. Insert USB
    b. Tap “Select from USB”
    c. Select your print
    d. Tap “Print”
    e. Make sure you stay for the first few layers to ensure that it is printing smoothly!
2) To remove print:
    a. Use a scraper to remove the print if it doesn’t come off by hand
3) If the plate is dirty, you can clean it by scraping or scrubbing off the glue under a trickle of warm water. **Make sure the plate is completely dry before re-attaching it.**

Filament Types

PLA is the standard filament used in CAMD. It is a popular and versatile filament known for its ease of use, biodegradability, and low warping tendency, making it ideal for decorative objects and prototypes that don't require high strength.

PETG is a strong and durable filament with good heat resistance, making it suitable for functional parts, mechanical components, and objects exposed to moderate stress. It is a bit more flexible than PLA.

TPU is the most flexible filament out of the three. It has rubber-like properties, offering elasticity and impact absorption. It is commonly used for creating soft and bendable objects such as phone cases, gaskets, and prosthetics.

Optimization

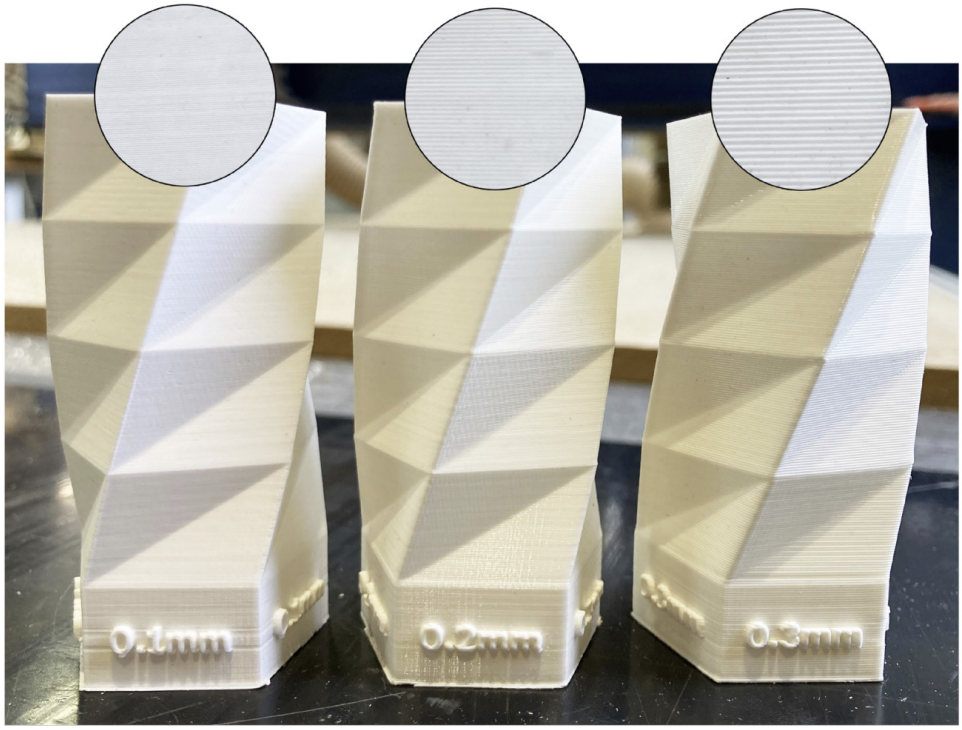
Overview

There are many options to consider when 3D printing. This basic guide will outline ways to reduce printing time and how to choose the best methods for printing your part. The two major ways to reduce printing time are **increasing the layer height and lowering the infill percentage.**

Layer Height

Models are printed in layers, and the layer height is the thickness of each layer. This is typically the greatest factor when determining print time, and increasing layer height can reduce print time substantially.

A smaller layer height (between 0.1mm and 0.15mm) increases the quality and resolution of a print. This should be used if your model has fine details. For basic models with not much detail, a larger layer height (between 0.2mm and 0.3mm) is the best option. Below are examples of print times and quality for different layer heights for a 70mm tall object.



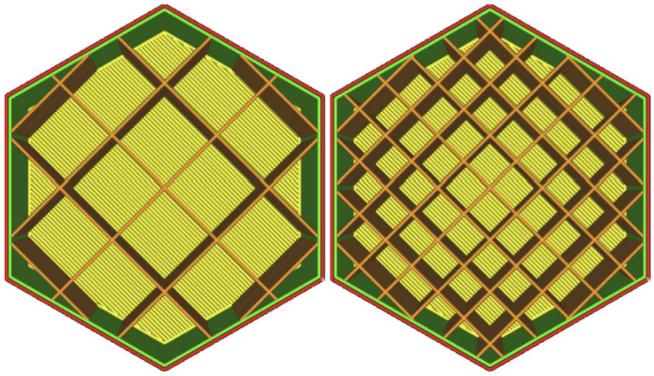
0.1mm: **3 hrs**

0.2mm: **1.5 hrs**

0.3mm: **1 hr**

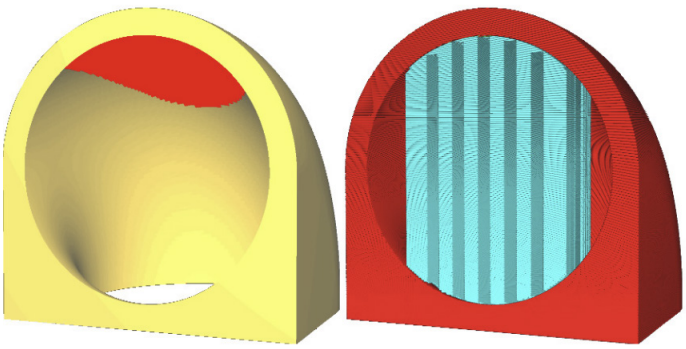
Infill

Infill is the material inside of a 3D printed part. The higher your infill, the denser your part is. If your part is purely for display or does not have structural needs, an infill of 10-20% is sufficient. Lowering infill can greatly reduce printing time.



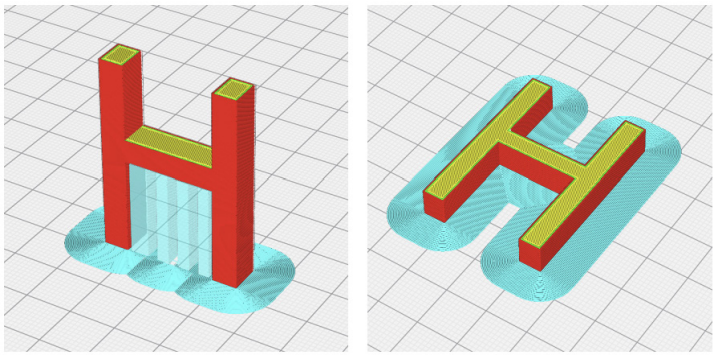
Supports

Using supports is necessary when your model has significant overhang. Overhang refers to when a layer extends beyond the previous layer and is indicated by red areas when preparing a file in Cura. Having overhang at an angle greater than 45 degrees increases the risk of sagging and possible print failure, so it is recommended to enable supports in these instances. Supports can be removed with pliers.

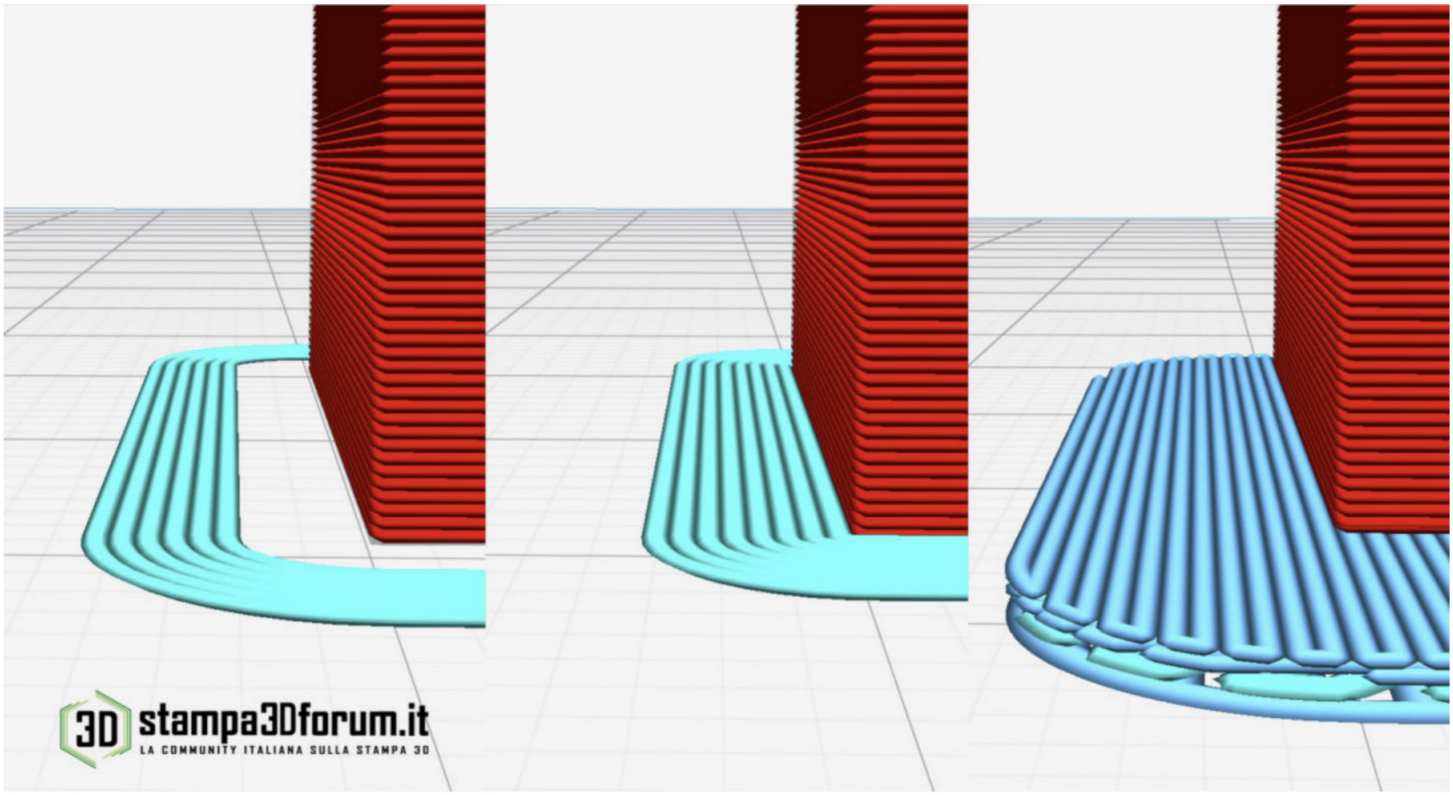


Orientation

It is important to orient your model in an efficient way to avoid using supports if possible. This will save time and give a lower chance of print failure. Sometimes, overhang can be avoided by rotating the model to a better position. Ensure that the models are touching the build plate. If possible, it is also a good idea to separate models into different parts, print them separately, and then glue them together.



Build Plate Adhesion



Skirt

Provides **no additional adhesion** to bed

Primes nozzle before print

Good for hygroscopic materials (PVA, Nylon)

Useful when your print has enough surface area to hold itself down to print bed

Leaves minimal postprocessing for finished model

Small risk of damaging model during removal

Highly recommended for Prusas

Brim

Provides **additional adhesion** because of added width/surface area to your model

Provides a very strong hold during the print – very good for models prone to warping

Primes nozzle before print

Brim is easily peeled away from base of model but may leave burrs

May be difficult to remove model from buildplate

Highly recommended for Ultimakers

Raft

Provides **substantial adhesion** to print bed or parts with minimal contact surface

Ideal for small/fragile prints

Primes nozzle before print

Adds significant time to print

Finished model should peel easily from raft; raft gets discarded

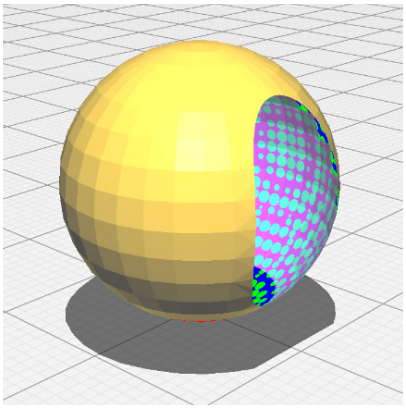
Leaves minimal postprocessing for finished model

Not recommended for standard prints

Common Errors

Ensure the following to avoid print failure:

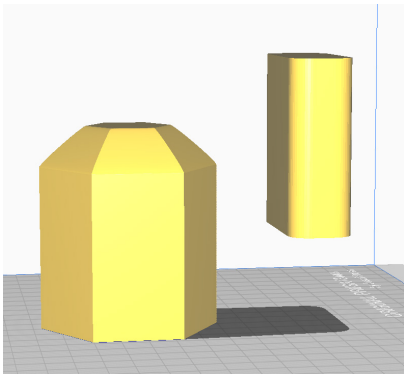
1) Ensure that models are **watertight.** Parts that aren't watertight occur when they have no volume or thickness. This error is signified in Cura with the cyan/magenta texture shown on the right. Components and details need a minimum thickness of either 0.4mm or 0.8mm, depending on the printer nozzle being used.



A model that is not watertight

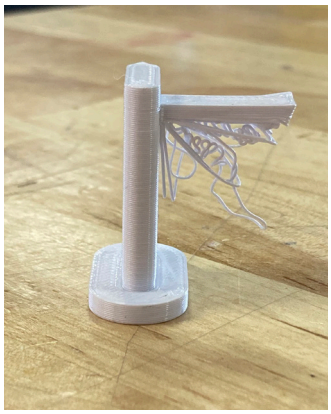
2) In your slicer, make sure you have the **correct printer and filament** selected.

3) **All objects must be touching the build plate.** A common reason for floating objects is if they are grouped into one file are not all on the same plane.

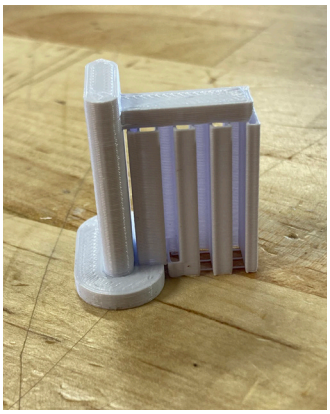


A model with a floating part

4) Make sure supports are enabled if you have significant overhang. Otherwise, the nozzle will extrude filament with nowhere to adhere to and the print will fail.



Model A with no supports



Model A with supports before removal

Model A with supports removed