FABLAB BRUSSELS
Prototypes! Not Powerpoints

Prusa 3D-printer

INTRODUCTION

AND TROUBLESHOOTING GUIDE

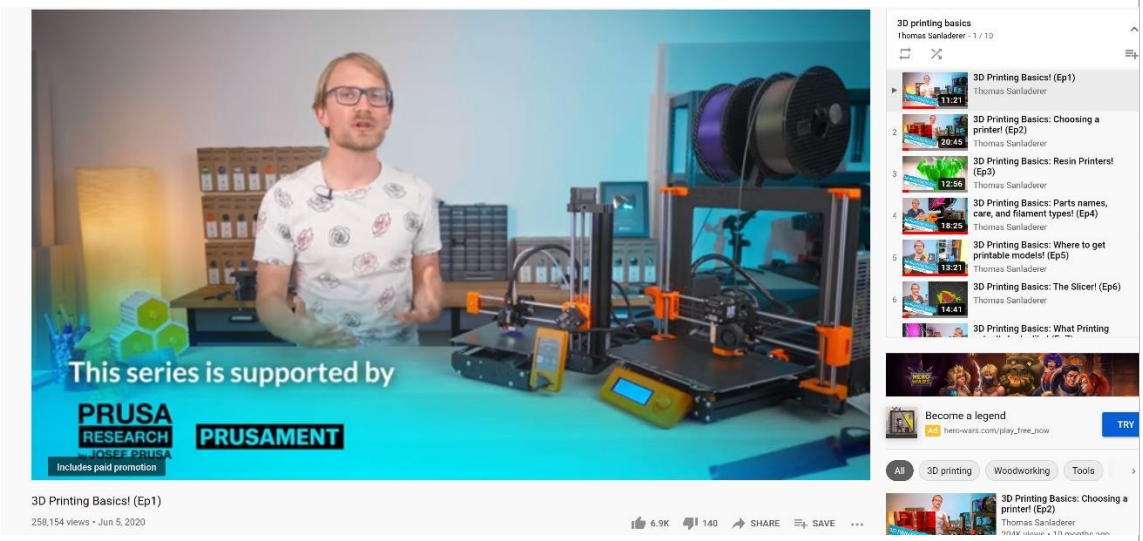
A digital version of this tutorial can be found at: http://www.fablab-brussels.be/fablab-english/tutorials/

Please do not remove the physical copy from the lab.

Version 2021-03-10 – fablab@vub.be

	# About this tutorial You can find three icons in the margins.
	This is an important point about safety.
	Here you should stop and ask a lab manager or assistant to look at you work or check it.
	Here you must perform an action
	It takes about 30 minutes to complete this introduction. The introduction assumes that you have all the necessary software installed on your own laptop.

1. Video alternative



Thomas Sandladerer on Youtube is a great resource for 3D-printing information.

The link & QR-code below take you to a 10-part introduction on 3D-printing, using the same Prusa printers we use in the fablab.

If you prefer watching video over reading this tutorial, then this series will tell you everything you need to know. (you can skip part 3 on resin printers)

https://www.youtube.com/watch?v=nb-Bzf4nQdE&list=PLDJMid0lOOYnkcFhz6rfQ6Uj8x7meNJJx

2. What is 3D printing?

3D printing is an "additive technique" whereby material is combined to make an object. There are various sorts of techniques and printers that use different sorts of media (e.g. vapour, liquid, powder).

The Prusa printers are FDM or Fused Deposition Modelling printers. They build up objects layer by layer to create the design using molten plastic.

These printers are not state-of-the-art printers, and you will find that prints fail. Even though they work pretty well, they are not plug-and-play, and require a certain amount of calibration. It will take you a certain amount of practice before you het the knack of using the printer optimally.

Please note that our printers require a bit of TLC (Tender Loving Care) from time to time.

3. Types of plastic

There are many sorts of plastic used in 3D printing. The three most common sorts used in the Fablab are **PLA, PETG** and **ABS** (not very often anymore).

PLA

PLA (polylactic acid) Is a polymer based on maize and is biodegradable. It is harder than ABS, but more brittle and temperature sensitive.

PLA shrinks less than ABS, and this means that it can be printed onto an unwarmed bed, although we do recommend heating the bed up to 60°-65° C.

PLA is the easiest material to use in order to get a good resulting print.

ABS

ABS (acrylonitrile butadiene styrene) is the plastic from which LEGO blocks are made. It is tougher than PLA, but also more toxic.

It dissolves in acetone, and by this means you can stick/glue pieces of ABS together. It melts, and hence prints at a higher temperature.

ABS shrinks more than PLA as it cools, and has a higher tendency to curl or buckle. For this reason it is unwise to print large flat pieces in ABS.

PETG (XT, UPET ...)

PET (polyethylene terephthalate, the plastic used in drink bottles) is a newer 3D printing material which prints more easily than ABS and is stronger than PLA.

This material is food safe.

In most cases at the Fablab we use PETG in its transparent form, as this is cheaper, but it is available in many colours.

	Print temperature	**Print quality**	**Strength**	**Temperature resistance**
PLA	210° - 220 °C	***	**	*
ABS	250 °C	**	***	**
PET, XT	245 °C	**	***	**

4. Recognizing plastics

As 3D printer filament comes is all colours and thicknesses, it is not so easy to identify what sort of plastic they are made from. It is important to know the variety of plastics so that you use the right one for your print, and it's also useful to know what the printer's previous user was printing with.

The easiest way to recognize which plastic is which is with a simple burning test.

PLA smells of burning sugar (but more unpleasant) and burns easily with a blue flame.

ABS smells of polystyrene and burns with a yellow smoky flame.

Take a piece of leftover MDF from the bin next to the lasercutters and a lighter. Ask for a piece of PLA and a piece of ABS. light each piece of filament in turn over the piece of wood. Make sure you don't get molten plastic on your fingers!

Observe the colour of the flame and smell the smoke.

Tidy up when finished.

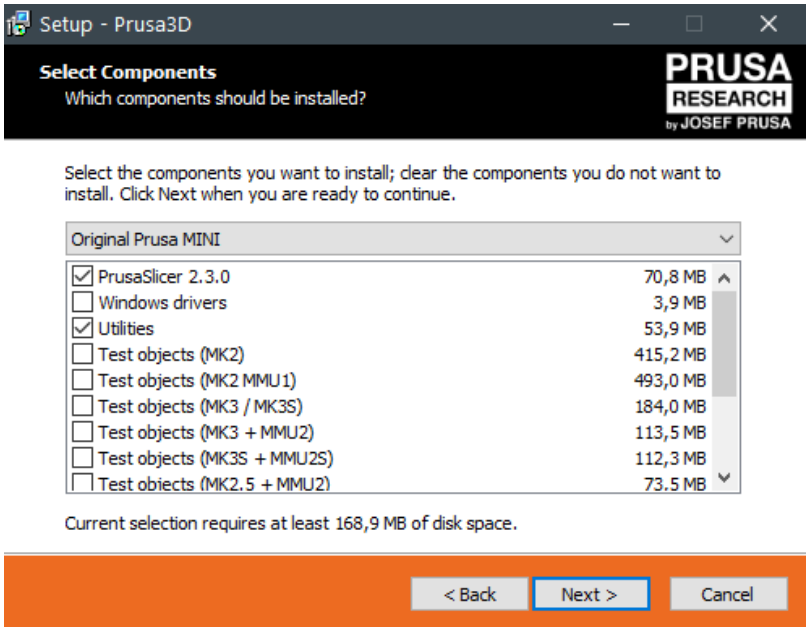
5. Software: Prusaslicer

5.1. Install

Go to https://www.prusa3d.com/prusaslicer/ and download the installer for your computer.

Launch the installer and go through all of its steps.

The installer asks you what to install. Make sure "PrusaSlicer x" is selected together with "Utilities". If you want, you can also install the sample objects for the different types of printers. We have the MK3S, MK2.5S and Mini.

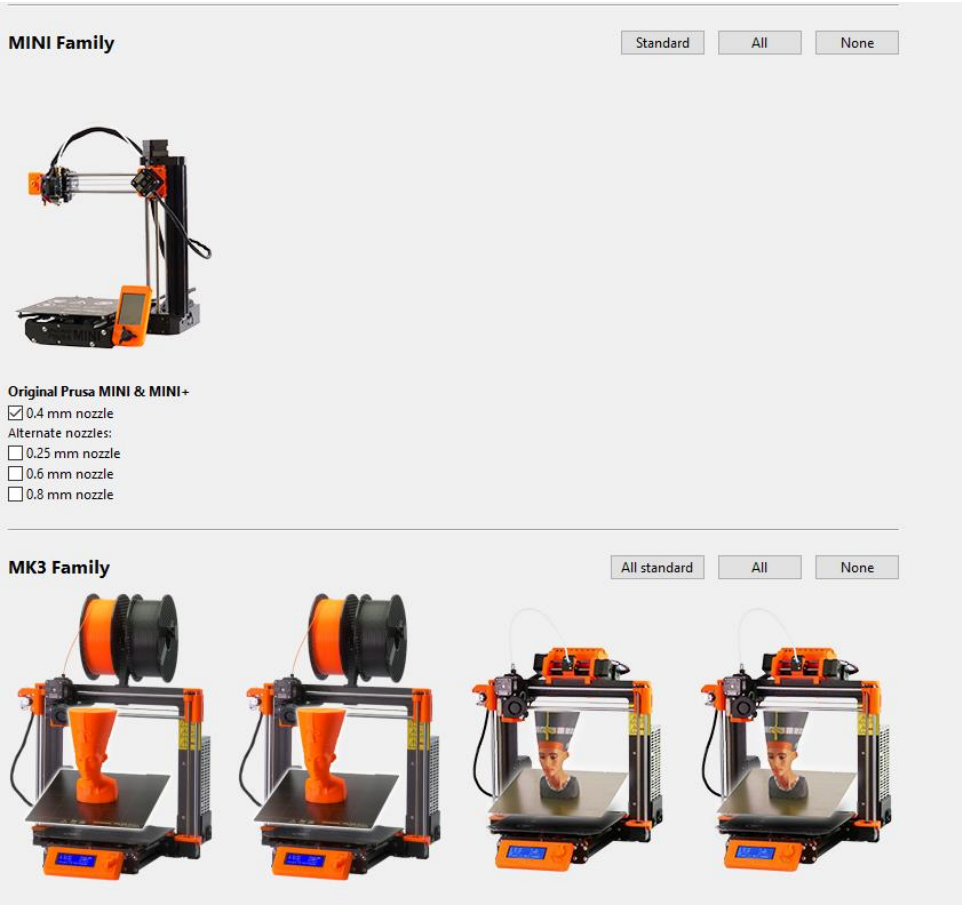


5.2. Opening Prusaslicer for the 1st time

When you have installed PrusaSlicer, run it for the first time. A pop-up screen will appear to set everything up for the printer settings.

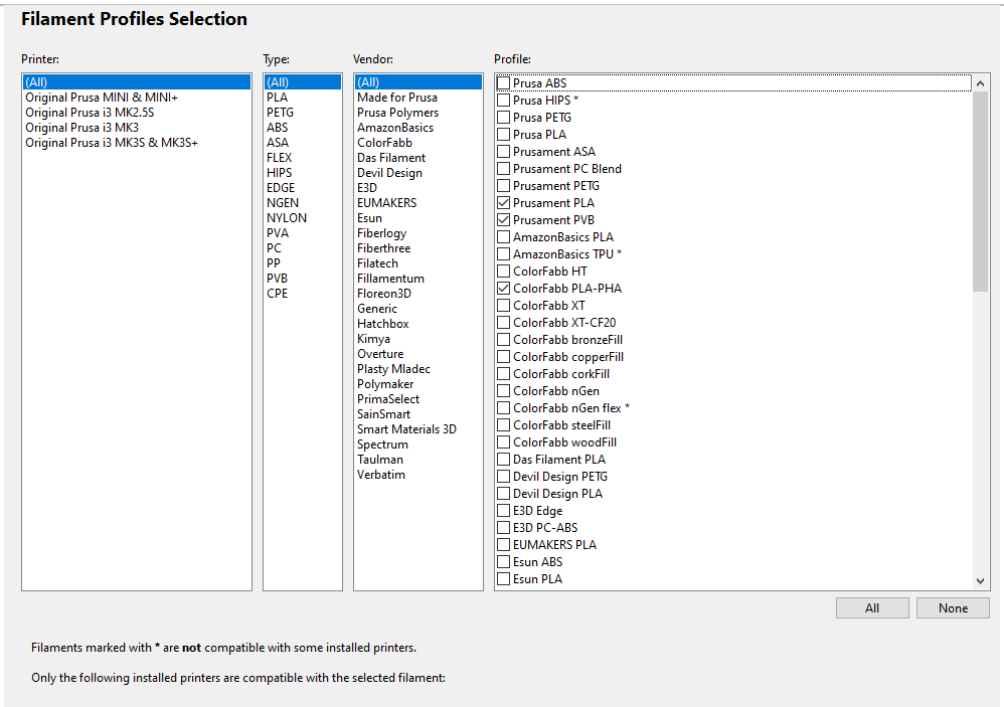
The software will ask you which printers you want to install. Select the following printers:

- Prusa Mini with 0.4mm nozzle
- Prusa MK3S with 0.4mm nozzle
- Prusa MK2.5S with 0.4mm nozzle



Click "Next" to continue the setup. You will have the option to install other printers; this is not needed for this tutorial.

You will then arrive on the page where you can select the filaments used. "Prusament PLA" and "Prusament PVB" are already selected. Also select the "ColorFabb PLA-PHA" as we mostly use this filament next to the Prusament filament.



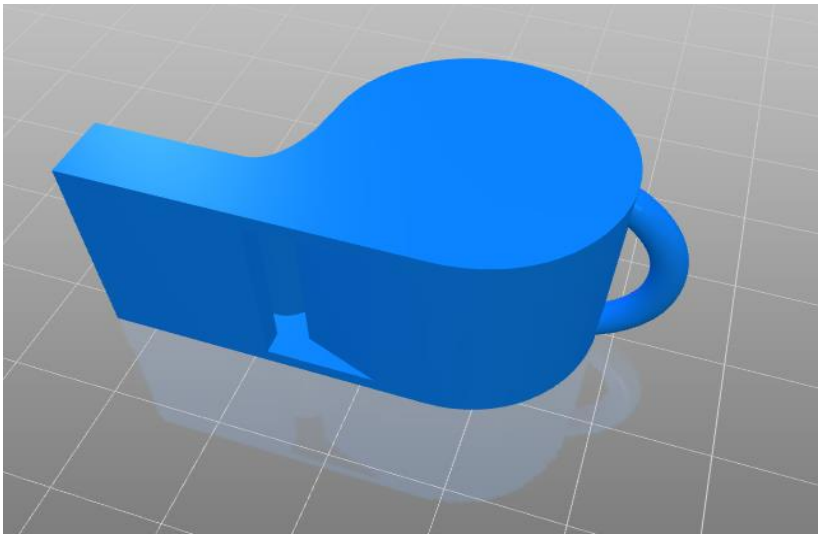
Click “Finish” to complete the setup.

Congrats, the software is now successfully installed and configured!

5.3. Preparing 3D-model

In this part you will 3D-print a whistle.

If you wish to print your own object, you will need a file in STL format.



This whistle was designed by Zaggo and published on thingiverse in september 2009.

It was one of the first things printed on our Makerbot Cupcake.

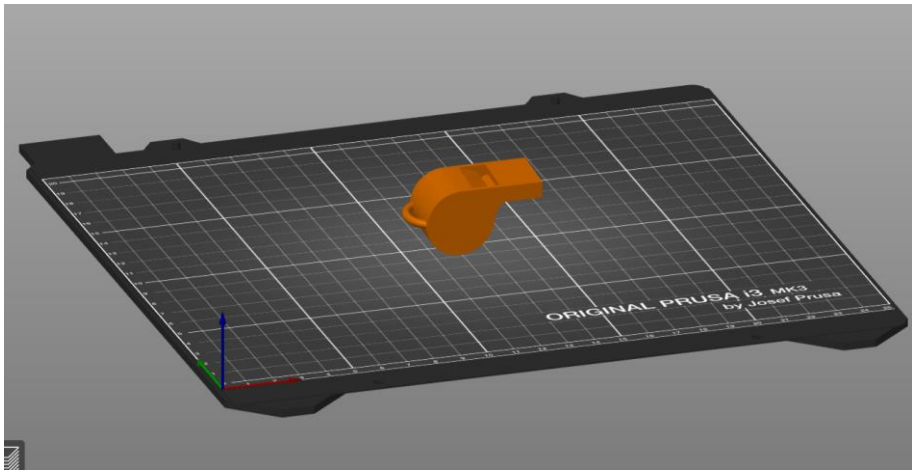
You can find files for many 3D objects on websites such as **thingiverse.com** and **youmagine.com** but before printing your own thing, you must do this example. (Remember prints can often go wrong and a standard object makes calibration of the machine easier.)

Go to http://www.fablab-brussels.be/fablab-english/tutorials/

Download the file **miniwhistle.stl**

Save the file on your computer.

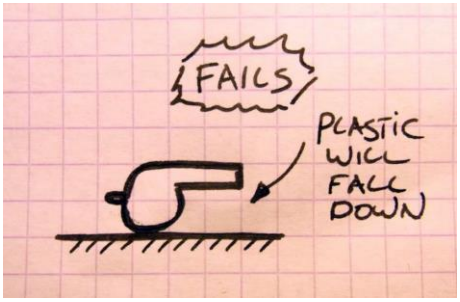
Open Prusaslicer and go to *File → Import → Import STOL/OBJ/…* and select the file **miniwhistle.stl** that you downloaded. The screen should look as following once loaded:



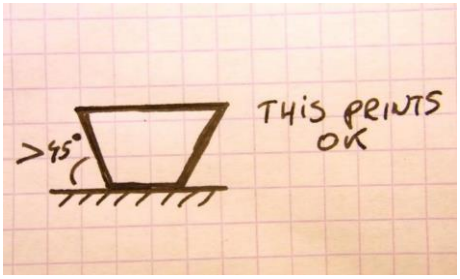
Now guess why the whistle is an interesting example? It's because you can't print it like this.

Printers build up a 3D-print layer by layer, but you can't print so much plastic above thin air.

I.e. you cannot 3D-print every shape.

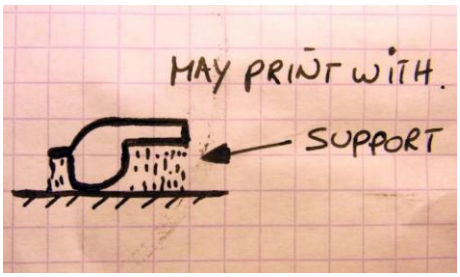


You can print overhangs up to about 45 degrees:



There are options available to print the whistle in this position by printing a "**Support**" underneath the mouthpiece.

Sometimes a support will be necessary, but the surface of your print becomes slightly uglier and the print takes longer (and more filament). The chance of the print failing is also greater.



Our preference is to design objects carefully and if necessary rotate them in order that they print well.

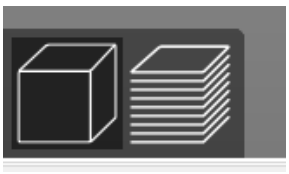
So for the whistle:

Find the command to rotate an object. This is located at the left in the Prusaslicer software along with some other handy tools.

Turn the whistle onto its side, this way it will print nicely.

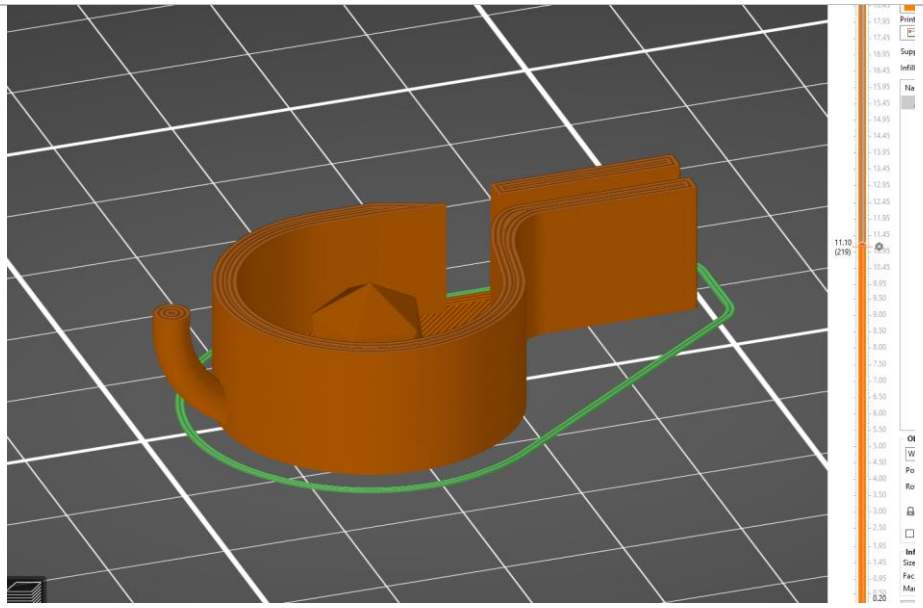
There is a hidden problem with this object; inside the chamber there is a little ball that also must be printed; This will only print properly if the whistle is lying on the correct side.

You can check this by clicking on the **layer preview** button in the left bottom corner.



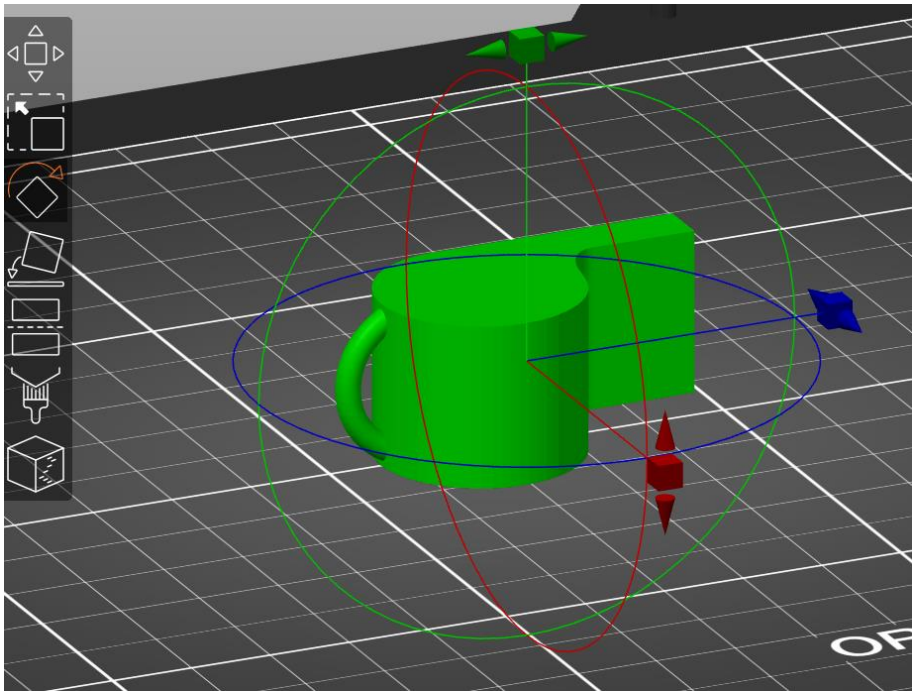
Look carefully at the file. You'll see a little ball inside. This must lie on the bottom, otherwise the print will not come out properly.

You can go through the layers by clicking and dragging the orange bar on the right when in **preview** mode. Drag it down until you see the ball inside the whistle.

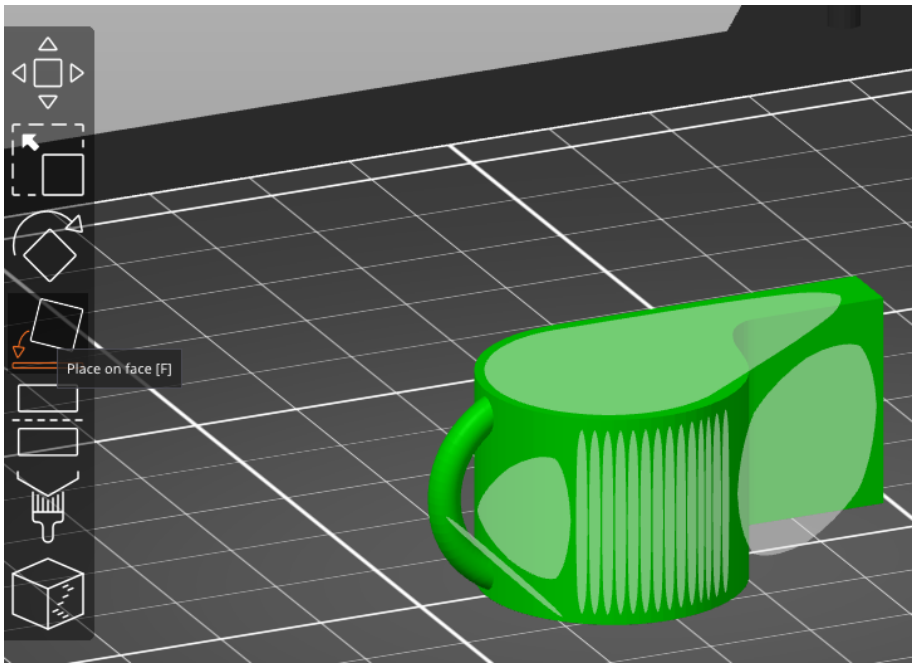


In order to rotate the object so that it can be printed properly, you have 2 options:

1. Use the rotate tool in the toolbar on the left en rotate the object until you get the desired orientation.



2. Use the "**place on face**" tool from the same toolbar. This lets you select the face you want to lie on the printing bed. You'll get some white faces shown that you can select.



Functional design for 3D-printing

"Functional design for 3D-printing" can be found in the Fablab bookcase. It is a superb (and short book) about how to design objects in a smart way so that they print properly.

Highly recommended.

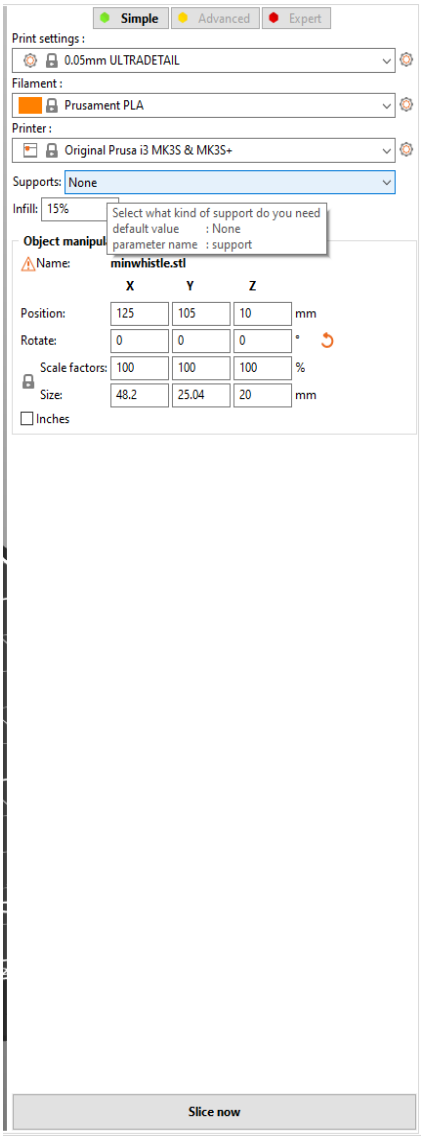
In particular, 2 tips from the book that are worth mentioning here:

- The Z-direction is less strong than the X ad Y directions in your print;
- Sometimes you can make an object more simply, faster and stronger by splitting the design into 2 or more parts.

5.4. Prusaslicer settings

In the top right corner of the screen, you can see 3 options: "**Simple**", "**Advanced**" and "**Expert**". This opens a list of settings, similar to the picture below.

When you move your cursor over a setting, you'll see a short explanation.



Print settings:
Here you can choose from different profiles where the 0.05mm is the layer height. You can mostly print at 0.15mm. 0.05 is ultradetailed but will take a lot more time. Faster prints can be achieved by using a 0.2mm layer height.

Filament:
Make sure you select the correct filament type and brand since this impacts the nozzle and bed temperature along with the printing speed. If the filament you're using isn't in the list, you can try adding it. We mostly only use Colorfabb or Prusament.

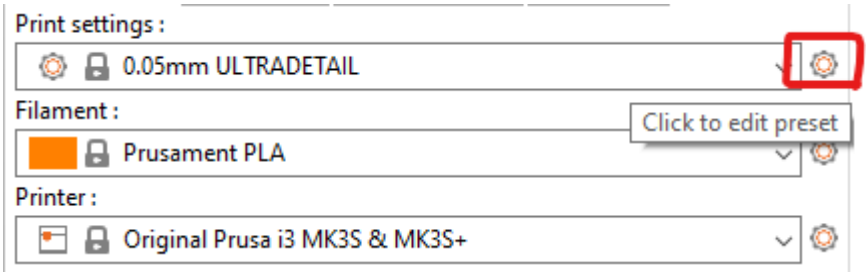
Supports:
Select "**support on build plate only**". This means that support will only be printed if it can start from the buildplate up.
"**Everywhere**" will allow the slicer to create supports where the program thinks it is necessary, not being limited by touching the buildplate.

Infill is how densely your object is printed. We seldom use more than 50% infill, if an item must be particularly strong and most printing can be done at 15-30% infill.

Brim:
This prints a kind of edge around your piece that acts as a suction cup and makes your piece stick better to the plate. We use this almost always.

If you want to further adapt settings, you can use the "**Advanced**" or "**Expert**" tab. An other way to change settings is by clicking the gear icon next to the print settings. The same applies for settings regarding the filament.

Go and select the settings for the filament. Make sure that the temperature for the nozzle is set to at least 215 °C when using PLA.



5.5. File on SD card.

Make sure the following is set correctly:

- Print settings → 0.15mm QUALITY
- Filament → generic PLA
- Nozzle temperature → 215 °C
- Bed temperature → 60°C
- The correct printer is selected.
- Infill is at 20%

To export the file to an SD-card or USB drive, click on the button "**Slice now**" in the bottom right corner. The software will now convert you model to something that the 3D-printer understands.

Once the object has been "sliced" you can click "**Export G-code**". Save the file to an SD card (Prusa I3 MK3s) or USB drive (Prusa Mini).

Call over a lab manager or assistant so he/she can check your work

Notes for the supervisor:

- Check settings of the Prusaslicer
- Assign a 3D printer for the next chapter and mark this on the printer so that other users don't use it in the meantime.
- Provide USB drive or SD card.

6. The printer

https://www.prusa3d.com/new-user-guides/

If a printer has finished its previous print, you can use it (even if the print is still there).

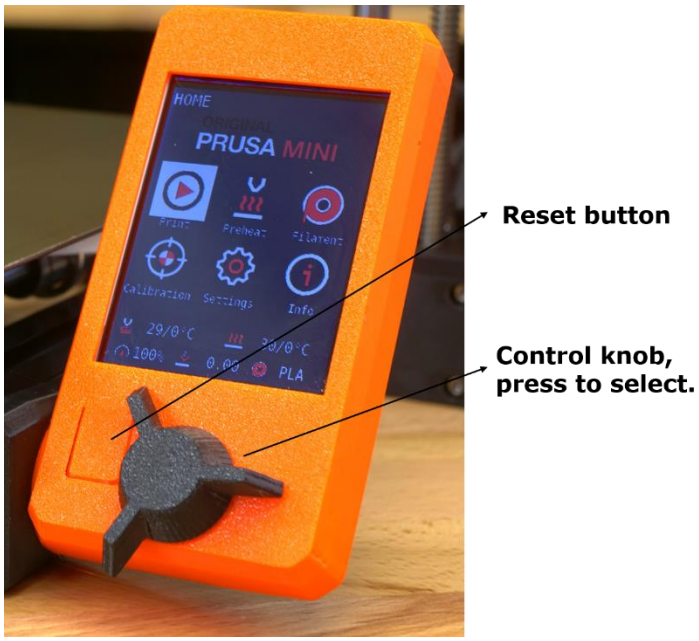
We have different models of the Prusa printers:

- MK2.5S (base consists of black threaded rods)
- MK3S (base consists of black aluminium profiles)
- Prusa Mini

These are the controls for the 3D printers:

- MK2.5 and MK3:

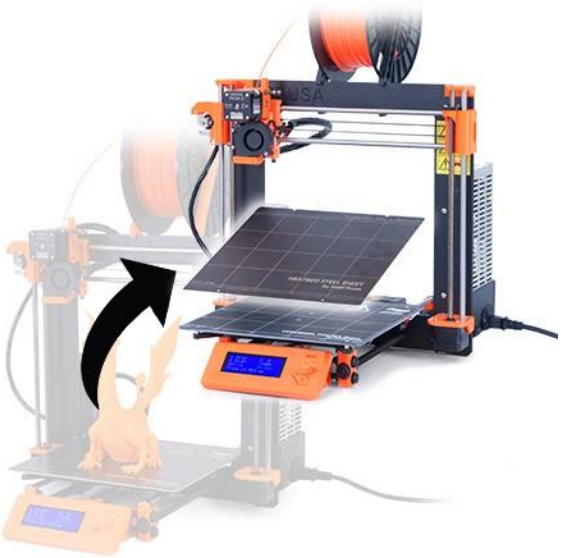
- Prusa Mini:



6.1. If the print of the previous user is still on the bed

It is your responsibility to ensure that the print of the previous user is placed in the box of completed prints without damage or dirt.

Wait for the bed to cool down to room temperature and take of the magnetic steel bed once cooled down.



To remove the 3D print, simple bend the sheet inwards and outwards. If the prints don't come loose, turn the sheet 90° and bend again. Do not overbend the steel sheet as you may damage it permanently by doing so.

	Put the 3D-print in the box marked "Afgewerkte prints" (finished prints). If the printing underway has failed, you can switch it off. Place the printed object next to the machine with a note explaining when and why you cancelled the print.
	6.2. Choosing filament for the Prusa's It may be necessary to change the filament being used by the machine: the current filament has been used up, you'd like a different colour, you need another type of material … To change the filament do the following: 1. Switch the printer on. The MK3 printers have a power switch at the back right side, the Mini's at the right side. If the machine is hooked up to the badge system you will not find the power switch but you will need to badge in. The LCD screen will light up. 2. Go to "Change filament". ▪ MK3S: press the control knob and rotate it until you are at "Change filament" ▪ Mini: go to "Filament" and select "Change filament" 3. Select the correct type of filament you want to load. Make sure you select the right type as a wrong type will result in endless failed prints. 4. If you just want to unload filament or there is no filament present and you want to load one, just select "Load filament" of "Unload filament".
	Call over a lab manager or assistant. You will do this step together.
	Notes for the supervisor: • Check if the user has understood how to change/load/unload the filament and that he/she can perform all the steps him/herself. • Point out that some filament may still remain stuck in the nozzle due to a blockage. Ends of filament must me tidied up. • Check which material is being reloaded and that this is the correct plastic for the chosen job.

6.3. Starting your first 3D-print.

1. Take of the steel sheet from the bed. If the previous print has not yet been removed, do this according to the instructions in 5.1.

   Look for a bottle of Isopropyl alcohol (disinfecting alcohol) and take some paper. Spray some alcohol on the bed and rub over the bed with the paper. This way you are removing the grease deposited on the bed by touching it with your hands/fingers. The grease will make it more difficult for the print to adhere, therefore we remove it.

   Put the steel sheet back on the bed but make sure not to touch it with your fingers.

2. Insert the USB drive or SD card depending on the printer you're using.
3. Go to "**Print**" or "**Print from SD**" and select the file you want to print.
4. The printer will first home itself and calibrate the distance to the bed.
5. Always stay with the printer until the first few layers have been printed. If these are printing correctly, than usually the entire print will be ok.

6. Troubleshooting

- **No material coming out of nozzle**:
  Check the following steps:
  - Is the nozzle hot enough? (215 °C for PLA)
  - Is the filament tangled?
  - Is the side cooling fan on when the printer is printing?
  - Is the temperature setting correct for the chosen filament?

- **First layer is not clean/does not adhere properly**
  Firstly, you have to clean the steel sheet that is on the heated bed. Follow the steps from 5.3. to do this.

  If this does not help, you can check the first layer calibration.
  https://help.prusa3d.com/en/article/first-layer-calibration_112364