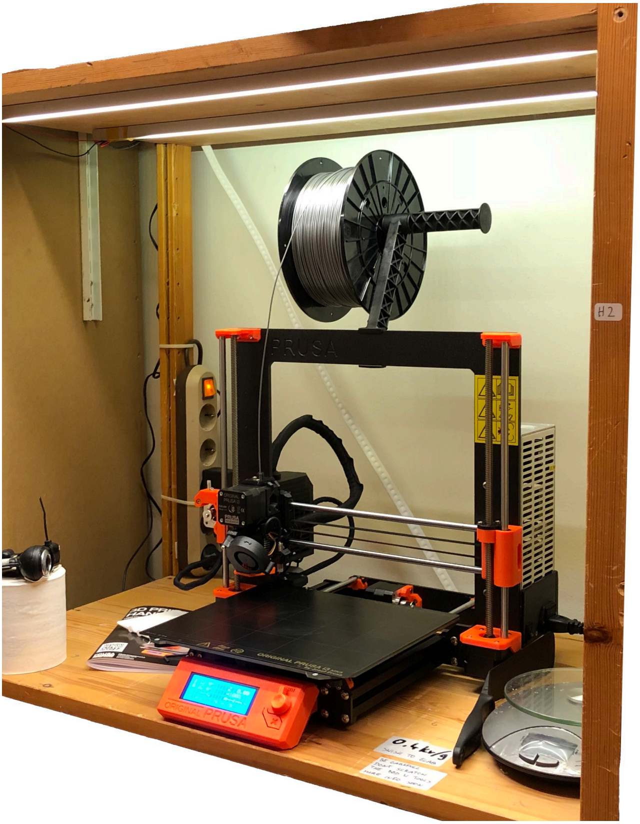
How to print using the ELAB printer



Most important rules:

- **Don't use any tools on the print surface!** The surface is fragile!
  Read the guide on how to remove your print from the bed
- **Don't adjust any screws**, belts, motors or sensors
  All calibration should be done in software only
- Use the correct setting on the printer for the material you are using
  PLA, ABS, PETG all have different settings!
- Run only .gcode generated specifically for this printer
  Preferably from Slic3r Prusa Edition
- Remember to **pay for the filament**
  Use the scale, pay 0.5 SEK per gram to ELAB, swish or cash
- Make sure that the nozzle isn't scraping the bed on the first layers
- If something looks fishy, turn off the power immediately
- If you have any problems, ask #3dprinting on the ELAB slack
  instead of breaking the expensive printer
- If something breaks, don't worry, just contact #3dprinting on slack

Tools and downloads:

This is a **Prusa i3 MK3S** and belongs to ELAB

It has a standard **0.4mm nozzle** and takes **1.75mm filament**
We have various filaments, including PLA, ABS and PETG.

Download the "Drivers and apps" for the Prusa i3 MK3S from this link:

https://www.prusa3d.com/drivers/

Preparing the print file

You need a 3d design in an `.stl` file. Some 3D CAD tools can export such files (example: Fusion360), and this is a common format for 3d printable designs (like on thingverse.com).

1. Open “**Slic3r Prusa Edition**” - part of the download package mentioned before. If asked, select the printer and nozzle that we have.
2. Import your `.stl` file by file->open or just drag-and-drop.
3. Select a preset from the “Print settings” dropdown list on top right. **”0.15mm QUALITY MK3”** is a very good default setting.
4. Pick the filament material from the dropdown list. PLA is the most reliable.
5. Make sure that the right printer is selected - Original Prusa i3 MK3S.
6. If you need to re-orient your print, click the lowest of the three buttons on the left, and click on the surface, that you want to be the new BOTTOM.
7. If the print needs supports, set them to “Everywhere”. This will turn off bridging perimeter detection, and that’s ok.
8. If you need to, adjust the infill percentage (more info on the next page).
9. If your print is relatively thin and tall, you should enable the “brim”.
10. Press “Slice now”!
11. At this point you might want to switch to “preview” and inspect the simulated print. For better insight, you can view each layer separately. With some practical reasoning and a bit of experience one can identify potentially problematic areas, and address the issues before wasting time and filament.

    Note that the printer bed in the 3d preview is shown to scale, and each square is 1cm.
12. Press “Export G-code” and save it on the SD card of the printer.
13. Now, go and print.

A quick word on infill and print strength

Feel free to skip if you know these things.

The printed objects are not solid plastic. Even if your 3d design suggests a solid cube, the slicer software will construct it with a “shell” and a low density infill structure inside.

The default settings in the “0.15mm QUALITY MK3” profile are reasonably strong, but not recommended for load-bearing parts. If your print is supposed to work mechanically, you might want to increase its strength. The best way to do so is to increase the **thickness of the outer** shell as well as the **fill density**.

- In the main settings you can find the fill density option
- In the “Print settings” tab, in “Layers and perimeters” you can find:
  - “Perimeters” count in the “Vertical shells” section
  - “Solid layers top” and “bottom” in the “Horizontal shells” option
- The table below shows reasonable numbers to use.

Setting	Light	Default	Strong	Very strong
Perimiters (vertical)	2	2	3	4
Top solid layers	6	7	10	12
Bottom solid layers	4	5	8	10
Infill percent	10%	15%	25%	40%

Note that increasing the strength will make the print take significantly longer, and cost more in filament. Increasing infill above 50% does not help much more, and increasing above 80% may cause the print to fail, as there is less room for the material to expand, or for feed inaccuracy.

The material also plays a part in strength. PLA is decently hard, but brittle - it does not work well for pieces that need to bend or snap in place. PETG is better suited for such prints, but on the other hand it is more prone to scratching.

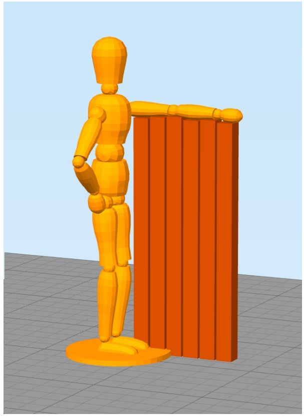
A quick word on supports and print orientation

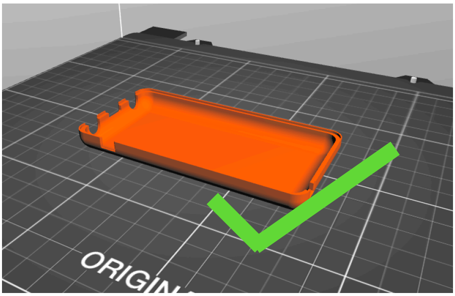
Feel free to skip if you know these things.

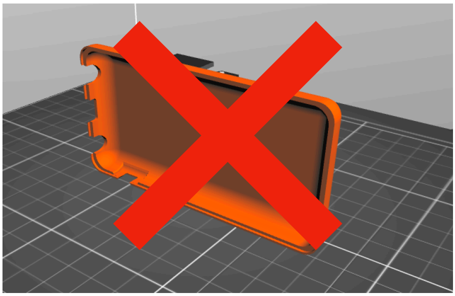
Since the printer prints from the bottom up, and it can't print "mid air", some shapes will require a support structure - a thin tower that supports the piece while the printer is printing it.

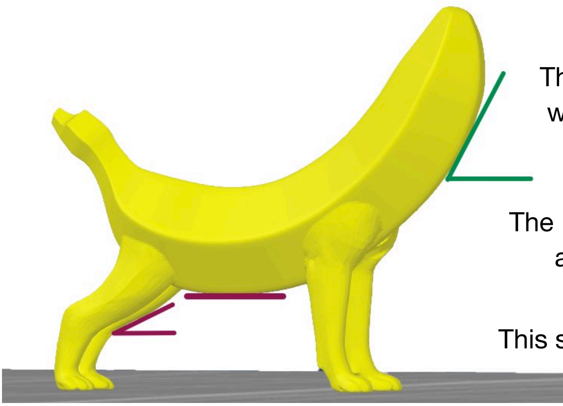
The picture shows a support structure, (dark orange) underneath the arm of the puppet, that would otherwise be unprintable. After the print is done, you can break off the supports by hand.

Usually you can orient the object in a way that will greatly reduce the need of supports. See the picture below for an example.









The steep angle of this overhang will print easily without supports

The lowest point is hanging mid air, and therefore requires supports

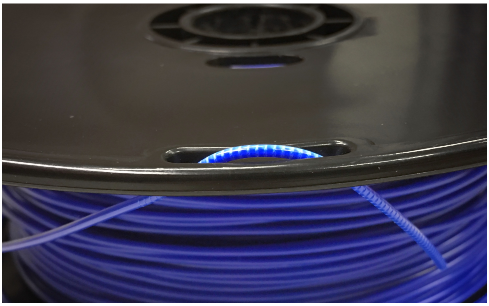
This shallow-angled overhang might also require supports.

Operating the printer

1. Turn on the power strip mounted on the left side of the shelf. The power switch on the right side of the printer should always be on.
2. Make sure that the **proper material is installed** on the printer - the type (PLA, ABS etc) has to match the one you selected in Slic3r! See "changing the filament spool" for a detailed instruction.
3. Make sure that the print **bed is clean**. You can remove it by pulling upwards on one of the front corners for easy cleaning. Bend the bed gently to remove any plastic parts easily with your bare hands. If necessary, use the spray bottle with red alcohol and a paper towel to remove oily stains.
DO NOT SCRAPE THE BED WITH METAL TOOLS
4. Plug in the SD card in (metal contacts facing you), go to "Print from SD" and pick your file.
5. Keep an eye on the printer for at least the first couple of minutes. If the nozzle scratches on the bed, stop the printer (by pushing the "X" key or just switch off the power).
6. Now, you wait.
7. Once the print is done, **detach the magnetic bed** by pulling upwards on one of the front corners, and then **gently bend the bed sheet** while pulling on your printed object. You should be able to easily release it from the bed without using any tools.
8. Put your print, any previous failed attempts and all support material on the scale and weight it. **Pay ELAB 0.5 SEK per gram of used material.**

Changing the filament spool on the printer

1. Check the type of filament currently loaded in the printer (PLA, ABS etc)
2. Go to “Unload filament” and pick the material that **is currently loaded**
3. Follow the onscreen instructions
   - If you can’t get the filament out, try to “Load filament” from the menu and again “Unload filament”
   - Sometimes the printer “forgets” to heat up the nozzle when doing this operation. You might need to go to “Preheat” to manually start the heaters.
4. **Please** secure the unloaded filament like this: (this prevents tangling)

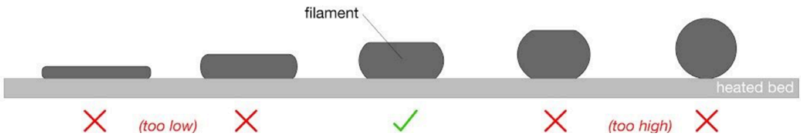


5. Put the new spool on the printer, make sure it unrolls neatly
6. If the end is melted or messy, take a pair of cutters and snip a couple of centimetres off, so it’s nice and sharp.
7. Go to “Load filament”, follow the onscreen instructions.
8. The printer will push some filament out during loading, wait a couple of seconds and then remove that piece with your fingers. Try not to touch the nozzle because it’s hot.

Troubleshooting

The print material does not stick to the bed on the first layer of print

- Clean the bed (“Operating the printer”, step 2)
- Check if the material loaded is of the same kind as what you picked in the slicer application (PLA or PETG or ABS etc)
- Run the “preheat” option from the printer menu
- If none of that helps, run “Calibration” -> “First layer cal.” from the menu
  The printer will start a test print, while you get to precisely adjust the height of the nozzle by spinning the knob. A “more negative” value will push the nozzle down and make the filament more squished against the bed, a “more positive” number will pull the extruder up. Adjust it so that it sticks to the bed properly and looks like on the illustration:



After a while of printing, my object detaches from the bed and is dragged by the printhead

- Use the “brim” option when generating your print file.
- If your design has a very small area contacting the bed, enable supports with the “on build plate only” option.
- If you are using ABS filament, don’t.

The Load / Unload filament options don’t work

- Force the printer to preheat manually, from its the main menu.