Instruction sheet 3D Printing:
Bambu Lab X1C aka Erik

3D Printer Specifications

Build volume: 256 x 256 x 256 mm

Filament diameter: 1.75 mm

AMS: Four different spools (PLA, PETG)

Software: Bambu Studio (alternatief: Ultimaker Cura 4.4)

Suported file types: X3D, 3MF, BMP, OBJ, STL

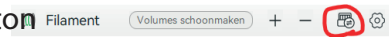
Available filament colours:

Green, white, black, teal, red, yellow, ...

STEP 1: Preparing your 3D model

1. **Open** the software Bambu Studio on a provided MaM Laptop:
- **Import** your 3D model (preferably in *.obj or .stl* format).
- Turn on the printer so that it can connect to the software. The button for this is on the back at the **bottom** on the right.

2. Adjust your **model**:
- Use the tools at the **top** of the program to move, scale, rotate or mirror your model.
- Click on your model to place it centrally and adjust its position as required.

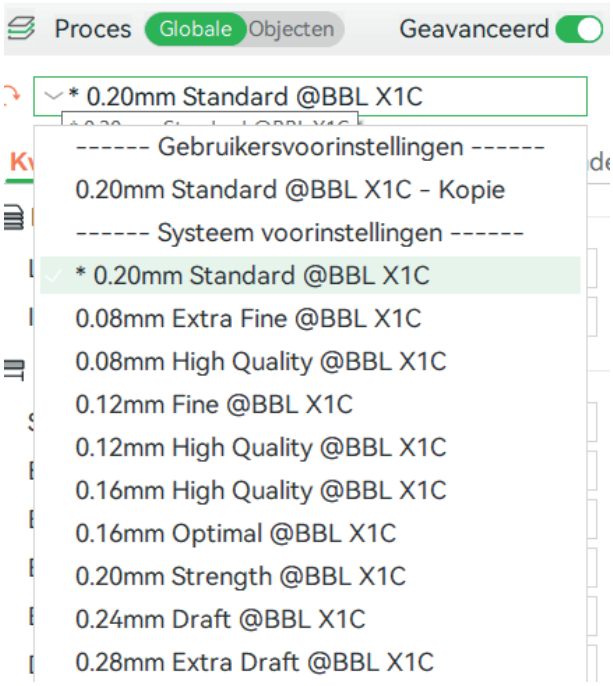
3. **Filament** selection:
- Choose the filament you want to use
- Update the printer by clicking on this button 

- Process --> Objects --> double-click it
- Choose the spool with matching filament colour
- of your choice by clicking on it. 

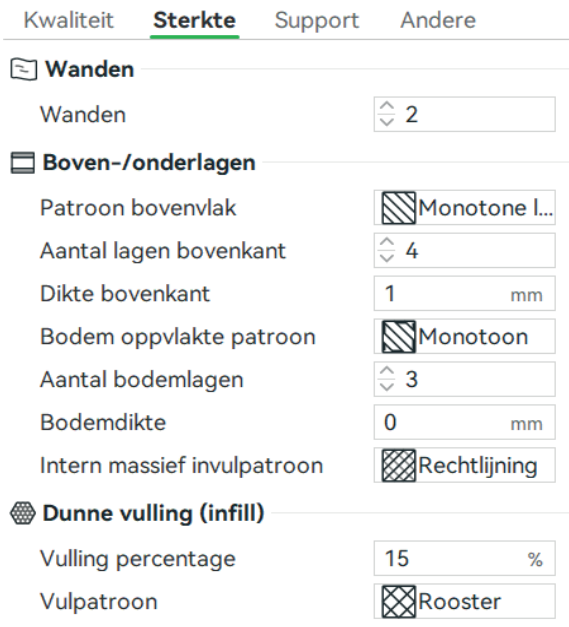
>> TIP: If you want to use a different colour filament, ask the Wally to change the filament.

4. **Setting** the print profile:

- **Quality:** Standard *0.2 mm* for medium prints; *0.12 a 0.08 mm* for finer details.

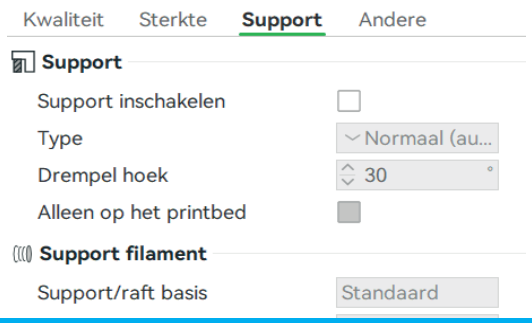


- **Density infill:** Select the percentage (e.g. *20%, 40%, 80%*) depending on the desired sturdiness. The default is **15%** is ideal for **light prints** and proto types, items that need to withstand a lot we recommend a higher % such as **40%.**



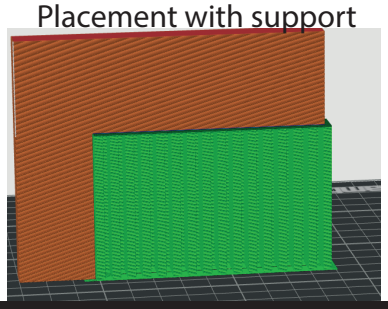
5. **Support:**

- If necessary, select support for floating parts of the model. *If necessary, the software will signal it itself.*

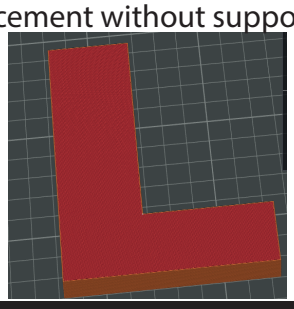


>> TIP: Try to avoid support by cleverly positioning the model.
This saves time and material.

Placement with support

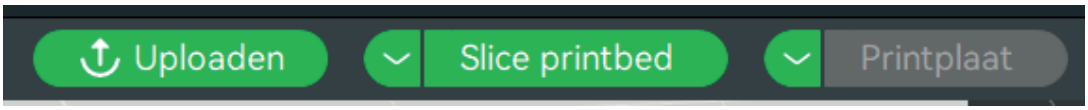


Placement without support



6. **Slicing the model:**

- Click the **Slice Printbed** button at the **top right**. The software divides your model into thin layers.



7. **Transfering the model to the printer:**

- Now click **Printplate** to send the print to the printer.
- Check that the correct filaments are indicated and press **send.**
- The printer now starts preparing the print, you don't need to do more.

STEP 2: Watch & Wait

1. Monitor the print:

- Check regularly that the print is progressing well, especially during the first few layers.

STEP 3: Postprocessing

1. Removing the print:

- Let the print cool until the 'print ready' message appears on the screen.
- If necessary, use a spatula resting on the printer to gently remove the print from the bed.

2. Postprocessing:

- Remove unnecessary parts and support structures with pliers.
- Sand or file your model for a smoother result.

3. Waste management:

- PLA (residual filament) that may be discarded can be deposited in the PLA bin under the printer.

4. Clean up:

- Confirm on the printer screen that you have removed the print.

Common Tips

- **Change filament:** ask the Wally for help if the colours or filament are not loaded correctly.
- **Improve adhesion:** Ask the Wally for help if you notice that the first print layers do not adhere well to the print bed.
- **Cool down:** Let your print cool before removing it to avoid distortion.
- **Different filaments:** We offer two types of filament PLA and PETG
    - **PLA:** This is the industry standard, provides a sturdy and affordable print ideal for prototyping.
    - **PETG:** Slightly firmer than PLA, providing an even sturdier, heat-resistant print. Ideal for prints that need to withstand a beating.