

3D Printing AG Thiele (Room U108)

Version: 16.08.2021

Before using the 3D printers for the first time, please check the *calendar for availability* and *book a slot*. If you encounter any problems during the printing process, please reach out to someone from the group for help. For any further queries please do not hesitate to get in touch *via e-mail: 3d-makerspace@thieleg.dialup.fu-berlin.de*.

Please prepare the 3D model *beforehand* and bring the stl-file along to your booked slot. The printing job needs to be completed by 8:00 am the next morning. To obtain a time estimate of the printing job, install *Cura 4.10.0* add the printer (see below), and slice the model according to the guidelines below.

Preparing the 3D model

- A full 3D model of the object to be printed is required, some models can be found in the folder “3D Drucker” on the desktop of the computer in U108 or under <https://www.thingiverse.com/>
- Customised 3D models can be created for example by using the freely available software *Microsoft 3D Builder*
- Very delicate objects such as molecules need to be cut in half (in *3D Builder* “aufteilen” → “beide behalten”) to ensure as many flat surfaces between the model and the printer bed as possible with minimised overhanging material → hydrogen atoms of methyl groups pose difficulties as the structure for their support is not stable enough
- Please save the 3D model as a stl-file and bring this along on, e.g. a USB-stick

Slicing of the 3D model

- Computer U108, **Account:** *3D Maker Space*, **Password:** *PLA*
- To translate the 3D model into printable data, the model needs to be sliced, this can be done with the freely available software *Cura 4.10.0*
- Open the stl-file of the 3D model by dragging it into an open window of *Cura*
- Ensure the correct printer “Anycubic Mega S” is selected in the dropdown menu at the top left in the window

- Please check the chosen parameters on the right hand side are correct for the given printer: “Normal-0.1mm” → “Custom”

Standard print settings for PLA:

Print settings
✕

Profile Normal - 0.2mm ★ ▼

☰

Quality

▼

Layer Height 🔗 ↺ 0.1 mm

🏠

Walls

▼

Wall Thickness ↺ ⓘ 0.8 mm

Wall Line Count 2

Horizontal Expansion 0.0 mm

🏠

Top/Bottom

▼

Top/Bottom Thickness ↺ 0.6 mm

Top Thickness 0.6 mm

Top Layers ↺ ⓘ 4

Bottom Thickness 0.6 mm

Bottom Layers ↺ ⓘ 4

🧩

Infill

ⓘ ▼

Infill Density ↺ 25.0 %

Infill Pattern ⓘ Zig Zag ▼

||||

Material

▼

Printing Temperature ↺ ⓘ 195.0 °C

Build Plate Temperature 🔗 60 °C

🕒

Speed

ⓘ ▼

Print Speed 50.0 mm/s

🛤️

Travel

ⓘ ▼

Enable Retraction

Z Hop When Retracted

🌀

Cooling

ⓘ ▼

Enable Print Cooling

Fan Speed 100.0 %

🏠

Support

ⓘ ▼

Generate Support 🔗

Support Placement 🔗 ↺ Everywhere ▼

Support Overhang Angle 🔗 60.0 °

⚡

Build Plate Adhesion

▼

Build Plate Adhesion Type 🔗 Skirt ▼

🏠

Dual Extrusion

▼

Print settings for molecules with PLA:

Print settings ✕

Profile Normal - 0.2mm ★ ▼

☰

Quality ▼

Layer Height 🔗 ↺ mm

Walls ▼

Wall Thickness ↺ ⓘ mm

Wall Line Count

Horizontal Expansion mm

Top/Bottom ▼

Top/Bottom Thickness mm

Top Thickness mm

Top Layers ↺ ⓘ

Bottom Thickness mm

Bottom Layers ↺ ⓘ

Infill ⓘ ▼

Infill Density ↺ %

Infill Pattern ⓘ ▼

Material ▼

Printing Temperature ↺ ⓘ °C

Build Plate Temperature 🔗 °C

Speed ⓘ ▼

Print Speed ↺ mm/s

Travel ⓘ ▼

Enable Retraction

Z Hop When Retracted

Cooling ⓘ ▼

Enable Print Cooling

Fan Speed %

Support ⓘ ▼

Generate Support 🔗

Support Placement 🔗 ↺ ▼

Support Overhang Angle 🔗 °

Build Plate Adhesion ▼

Build Plate Adhesion Type 🔗 ↺ ▼

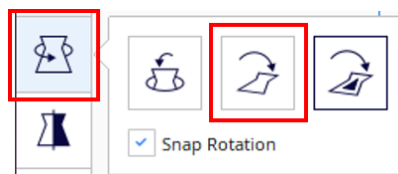
Dual Extrusion ▼

→ Please select under “Infill” whether the object should be printed as a solid object “100”% or hollow with reinforcements, “20”% is a good starting point, though to ensure objects made from PLA do not float in liquid nitrogen, an infill of “75”% is required

- After selecting the object it can be edited using the icons on the left, e.g. rotation, mirroring or scaling



- Please ensure through rotation that the object is lying as flat as possible on the build surface “Rotate” → “Lay flat”



- Slicing the 3D model “Slice” results in a preview in which the different layers and any support structures can be viewed, as well as an estimate of the time and material required for the print → check that there is sufficient filament on the filament holder and that the printing job will be finished by 8:00 am the next morning
- If the printing job lasts longer than 8 h, please select “Brim” under “Build Plate Adhesion” to ensure an improved adhesion to the build plate throughout the printing period
- The SD-cards for saving files can be found in the printers, use the adapter to connect the SD-card to the computer and save the sliced model to it “Save to Disk” → please choose short and concise titles and avoid “_” in the file names
- If the filament needs changing, please ask a group member for help with this

Starting a 3D print

- Insert the SD-card on the right hand side of the printer “SD Card inserted” (appears) → “OK”
- “Tools” → “Preheat” to start the heating process
- “Tools” → “Home” → “Home all” to move the print head to the starting position
- Select the wanted file under “Print” → check the temperatures “Bed Temp” (Build Plate Temperature) and “E0” (Printing Temperature) are correct, these can be edited *via* “Return” → “Setup” → “Temperature” → correct values → “OK”
- Please wait until the printer has printed the first layer successfully to ensure sufficient adhesion to the build plate, this could take up to a few minutes as the temperatures need to be reached
- Remove any filament which runs out of the extruder during the heating process carefully using tweezers – *Caution! Danger of burning on the extruder and printing bed!*
- After successful printing of the first layer and full adhesion to the printing bed, the print can be left unsupervised

Removing the printed object from the printing bed

- If the printing bed is not at the front, it can be shifted under “Tools” → “Axis” → “Y-Axis +10”, repeat this multiple times until the plate is at the front of the printer and the printed object can be accessed easily – *Caution! Danger of burning on the extruder and printing bed!*
- Wait until the printing bed has cooled to room temperature
- Use the spatula next to the printers to carefully loosen the object from the plate – do not apply too much pressure as this could damage the surface of the printing bed
- Surplus material can be removed using the provided cutter, please dispose of this material into the according container
- The spatula and cutter should always remain with the printers and not be used for anything else

Please leave the working space in a clean and tidy manner.

Addition of a printer in Cura

- “Add printer” → “Add a non-networked printer” → “Anycubic i3 Mega” → “Printer name “Anycubic i3 Mega S”
- “Manage Printers” → “Machine Settings”:

Machine Settings ✕

Anycubic i3 Mega S

Printer	Extruder 1
Printer Settings	Printhead Settings
X (Width) <input type="text" value="210.0"/> mm	X min <input type="text" value="-20"/> mm
Y (Depth) <input type="text" value="210.0"/> mm	Y min <input type="text" value="-10"/> mm
Z (Height) <input type="text" value="205.0"/> mm	X max <input type="text" value="10"/> mm
Build plate shape <input type="text" value="Rectangular"/> ▾	Y max <input type="text" value="10"/> mm
Origin at center <input type="checkbox"/>	Gantry Height <input type="text" value="0.0"/> mm
Heated bed <input checked="" type="checkbox"/>	Number of Extruders <input type="text" value="1"/> ▾
Heated build volume <input type="checkbox"/>	Apply Extruder offsets to GCode <input checked="" type="checkbox"/>
G-code flavor <input type="text" value="Marlin"/> ▾	
Start G-code	End G-code
<pre>G21 ;metric values G90 ;absolute positioning M82 ;set extruder to absolute mode M107 ;start with the fan off G28 X0 Y0 ;move X/Y to min endstops G28 Z0 ;move Z to min endstops G1 Z15.0 F{speed_travel} ;move the platform down 15mm G92 E0 ;zero the extruded length G1 F200 E3 ;extrude 3mm of feed stock G92 E0 ;zero the extruded length again G1 F{speed_travel} G0 Y20 F{speed_travel} M117 Printing... G5</pre>	<pre>M104 S0 ; turn off extruder M140 S0 ; turn off bed M84 ; disable motors M107 G91 ;relative positioning G1 E-1 F300 ;retract the filament a bit before lifting the nozzle, to release some of the pressure G1 Z+0.5 E-5 ;X-20 Y-20 F{speed_travel} ;move Z up a bit and retract filament even more G28 X0 ;Y0 ;move X/Y to min endstops, so the head is out of the way G1 Y180 F2000 M84 ;steppers off G90 M300 P300 S4000</pre>

Ancubic i3 Mega S

Printer

Extruder 1

Nozzle Settings

Nozzle size	<input type="text" value="0.4"/>	mm
Compatible material diameter	<input type="text" value="1.75"/>	mm
Nozzle offset X	<input type="text" value="0.0"/>	mm
Nozzle offset Y	<input type="text" value="0.0"/>	mm
Cooling Fan Number	<input type="text" value="0"/>	

Extruder Start G-code

Extruder End G-code

Close