Inventor Parts instructions for 3D printing

**Download + Configure PrusaSlicer (cable 3D printing software)**

PrusaSlicer download:

- default download

upon first opening:

Original Prusa i3 MK3S & MK3S+ 0.4 und 0.8 mm nozzle

Original Prusa i3 MK3S & MK3S+ + MMUS (multimaterial) 0.4 mm nozzle

- chose Filaments Profiles Selection:

. Vendor: Extrudr

. Profile: Extrudr GreenTEC Pro Carbon \*, Extrudr PETG \*, Extrudr PLA NX2 \*, Extrudr XPETG CF \*

- Chose View mode : Expert mode

For actual print (default options should be this)

- go for 0.2 mm Quality Print setting (larger z-axis layer height -> more stability piece, may not look as nice)

- PLA default for printing

- low right printer: i3 MK3S / S+

- Print Settings -> Vertical Shells -> Perimeter -> 4 shells minimum (for any piece that needs mechanical stability.)

**Print 3D part**

- load .stl file (available as export from Inventor)

- Orient the part with the menu on the left for optimal printing (consider easiest printing orientation and which holes need to be modified)

- Add support where needed. Be careful about adding internal support (example to holds, usually avoid this)

- Slice part(s)

- Check by looking from top and sliding down height visible on right that all support that is needed is present AND that no parts hangs and starts from nothing (no angles >45° from vertical= z axis without support)

- get SD card from printer in machine shop, insert it into PC

- export G-code to SD card

- insert SD card into 3D printer,

- press button 3D printer for Enter, wheel to Select between prints

**Make holes 3D printed**

- Cut the holes of at 45° to vertical and have tangential lines coming up, forming a teardrop shaped hole -> no overhang greater than 45°. Optionally the topmost 0.5 mm of the teardrop tip can be cut away horizontally in the CAD

- chamfer all holes on both sides with 0.2 mm AFTER making the teardrop

**Make a part with thread for 3D printing:**

Note: Only threads with pitch min. 1 mm and at least M6 come out well. For larger than M6

From discussion with Ronny: Instruction to make threads for 3D printable piece

- Download + install CoolOrange thread modeler

- Close + reopen inventor + unblock thread Modeler

- open part with existing hole

- thread hole with thread command (under 3D Model, modify)

- Under coolOrange->threadModeler select hole and click ok

- Under 3DModel->Modify->Combine: combine thread solid and main solid (with 2 pieces, maybe more possible)

- Add phase=chamfer to thread for screw to enter AND exit: Make sketch projecting hole. Use Chamfer (0.2 mm) or use Countersink seat option with outer diameter given by phase norm for thread AND use 1 mm inner diameter to avoid affecting actual thread

**3D printing, Post-processing threads**

For threads that did not come out nicely (pitch <1 mm or size < M6) need to be drilled afterwards.

- get taps (Gewindebohrer) of proper size + pitch with hand drill.

Bosch Drill in machine shop: Arrows on left and right side of drill change drilling forward <-> backward.

- Load tap by holding front of drill and drilling in one direction or rotating front of drill, both reduce / increase tap holding the chuck (Gewindefutter)

- Hold the tap perpendicular to the hole and drill the tap through.

- If the inner diameter of the threaded part is larger than the shaft (end part) of the tap, then drill all the way through, remove the tap from the drill and pull the tap out through the hole. Otherwise drill through, remove all the plastic that was drilled off, reverse drilling direction and drill the tap back out again. (If the plastic trash if not removed, it can destroy the thread on the way back out)