Instruction manual for CraftBot2 3D-printing

Version 2.0

By Niklas Valentin Roed Jensen 01/08 -2018

Revised 15/8 - 2019

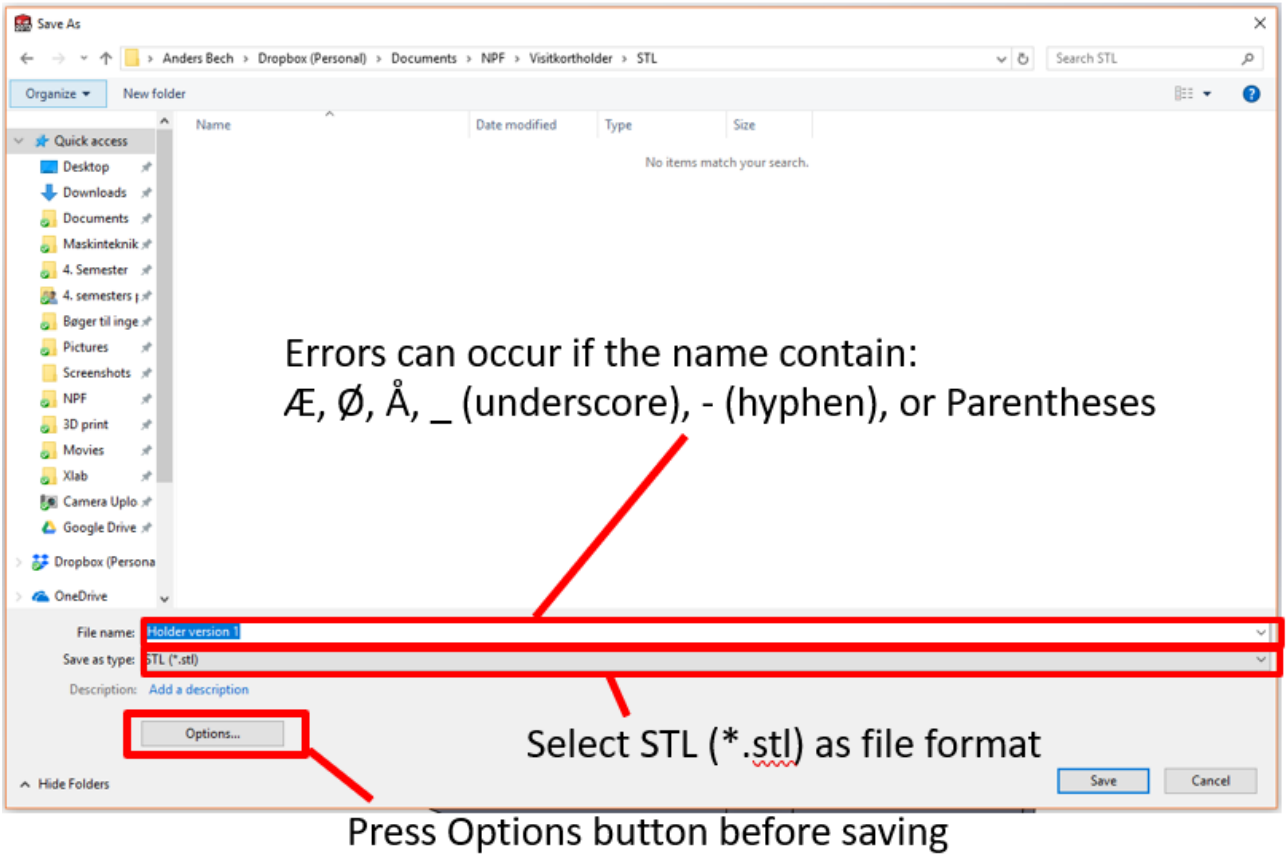
201805742@post.au.dk

Contents

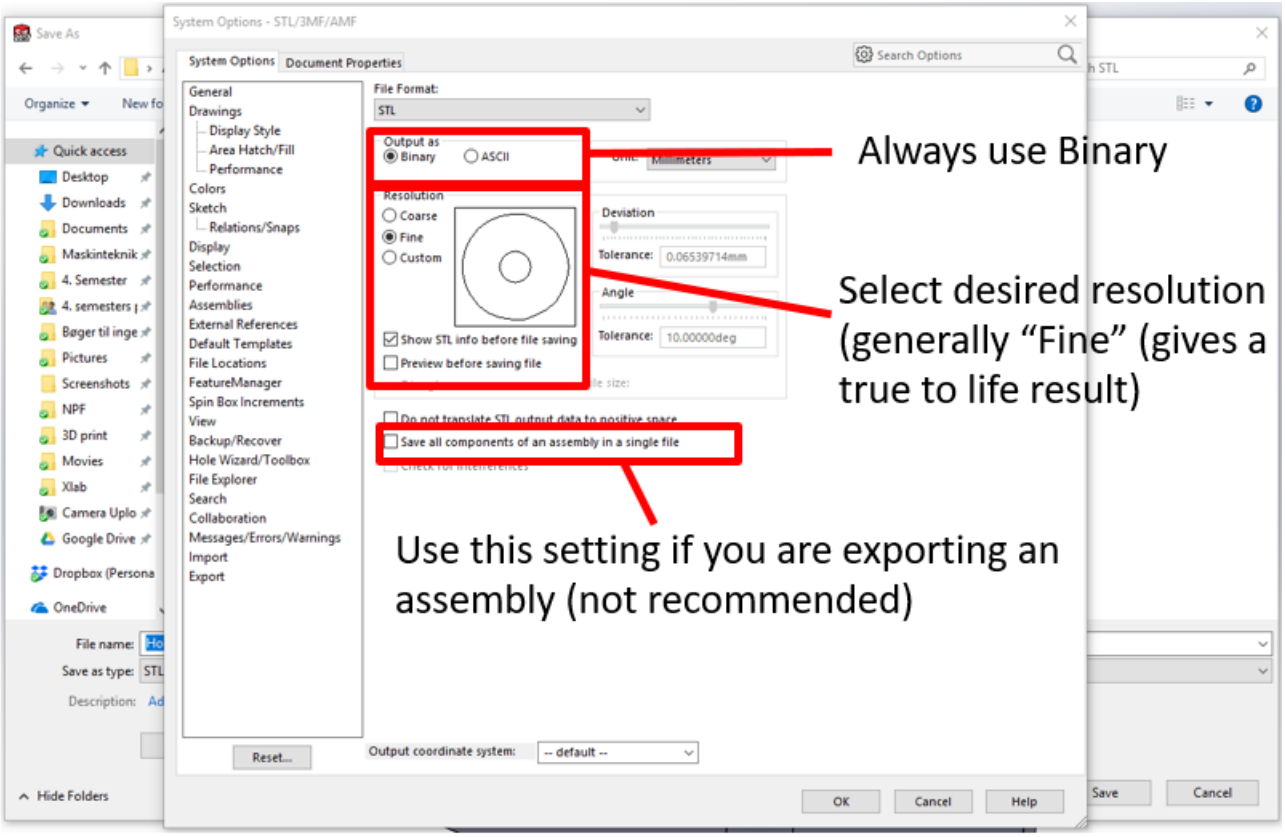
Save your model

When you have drawn your part(s) in SolidWorks, you have to export the file(s) as STL-files. (*.stl)

s



- Press ”Options”
- Make sure that ”Output as” is set to “Binary”
- The “Resolution” option is set to “Fine”
- If you’re printing an assembly, make sure to check “Save all components of an assembly in a single” If this is checked the whole assembly will be printed as one part. This isn’t recommended, rather print the parts of the assembly separately.

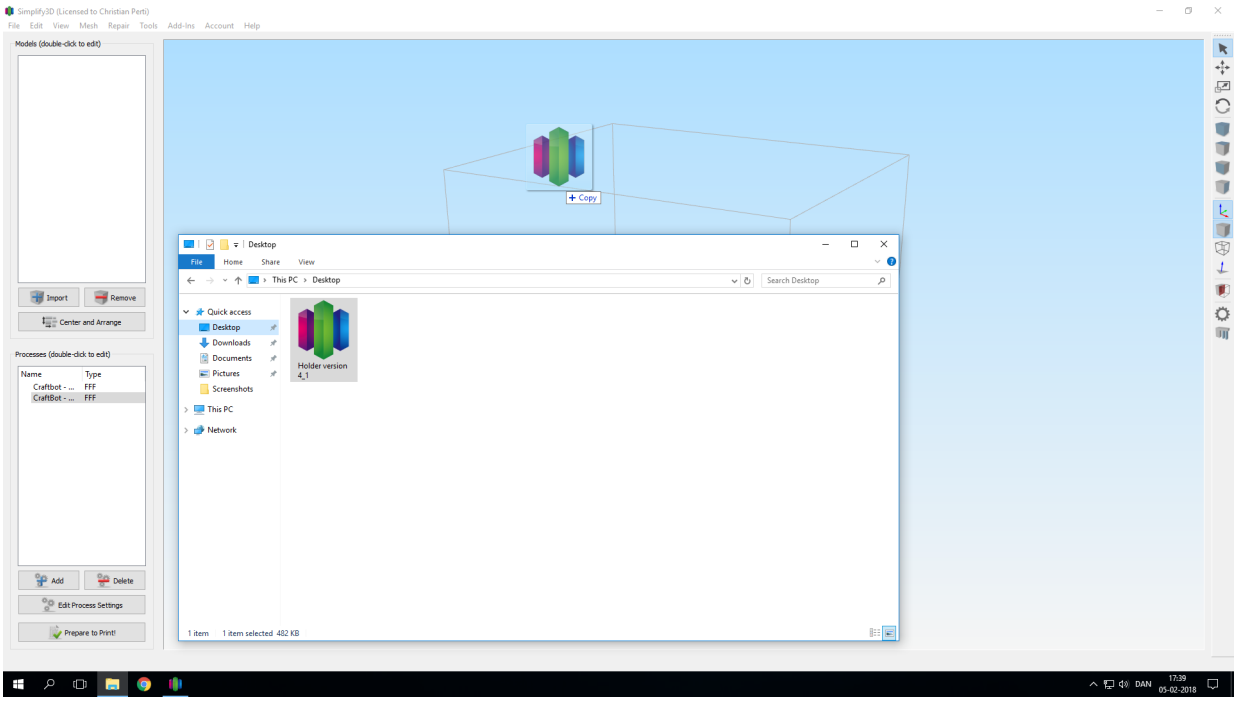


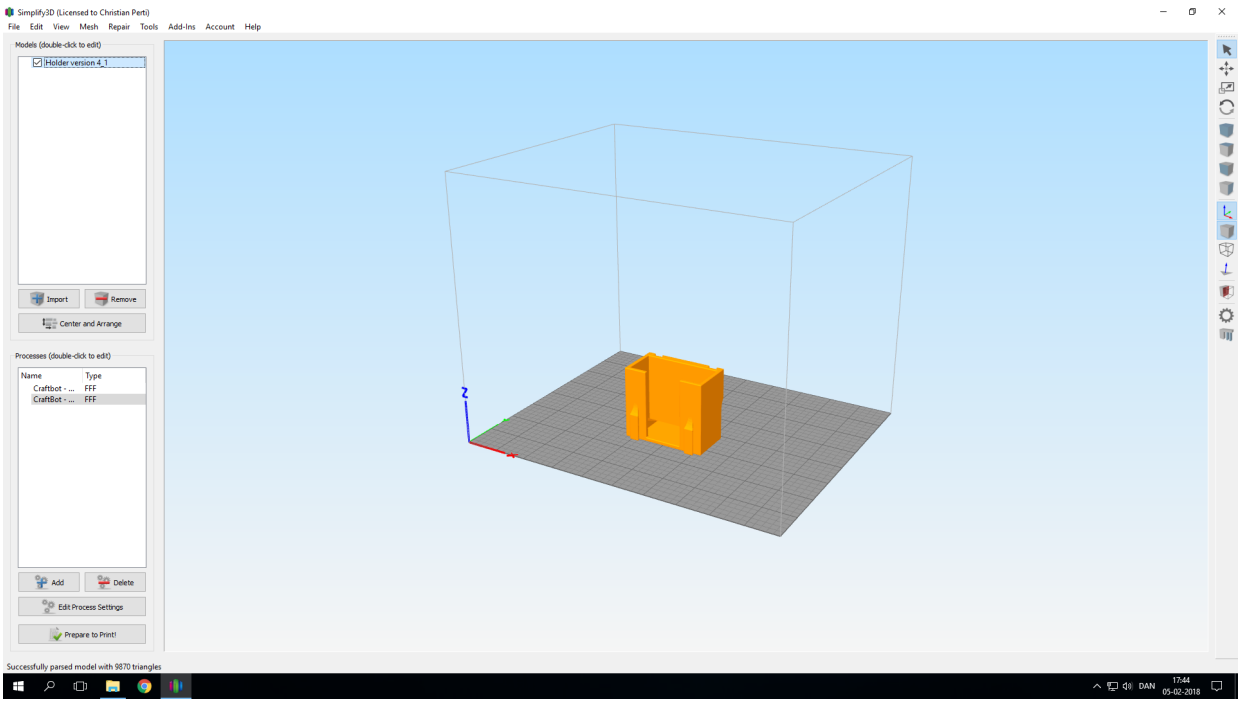
Transferring STL-file

When printing on the CraftBot2 3D-printers, the PC placed along with printers are to be used, as this PC contains the slicer-software used later. The login credentials for the PC are found directly below the display on the screen. To transfer your file(s) you have to use the internet, using either e-mail, Dropbox, Google Drive, OneDrive, Facebook or whatever you prefer, as long as you **aren’t** physically transferring the file using a USB-stick, memory card etc.

Slicer-software (Simplify3D)

When the STL-file(s) have been transferred to the PC in X-Lab it need to be “sliced” (prepared to be 3D-printed). To do this use Simplify3D, a shortcut is found on the desktop. To import your file(s) simply drag-and-drop them into Simplify3D, see below:

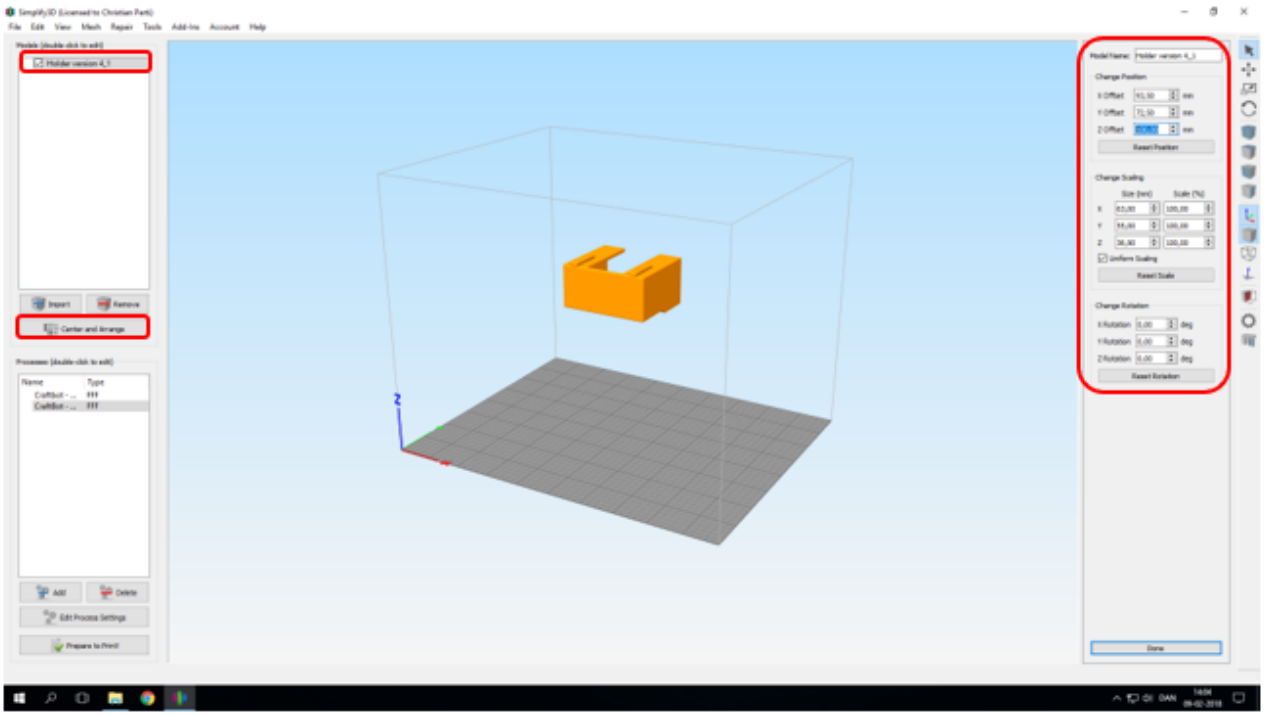




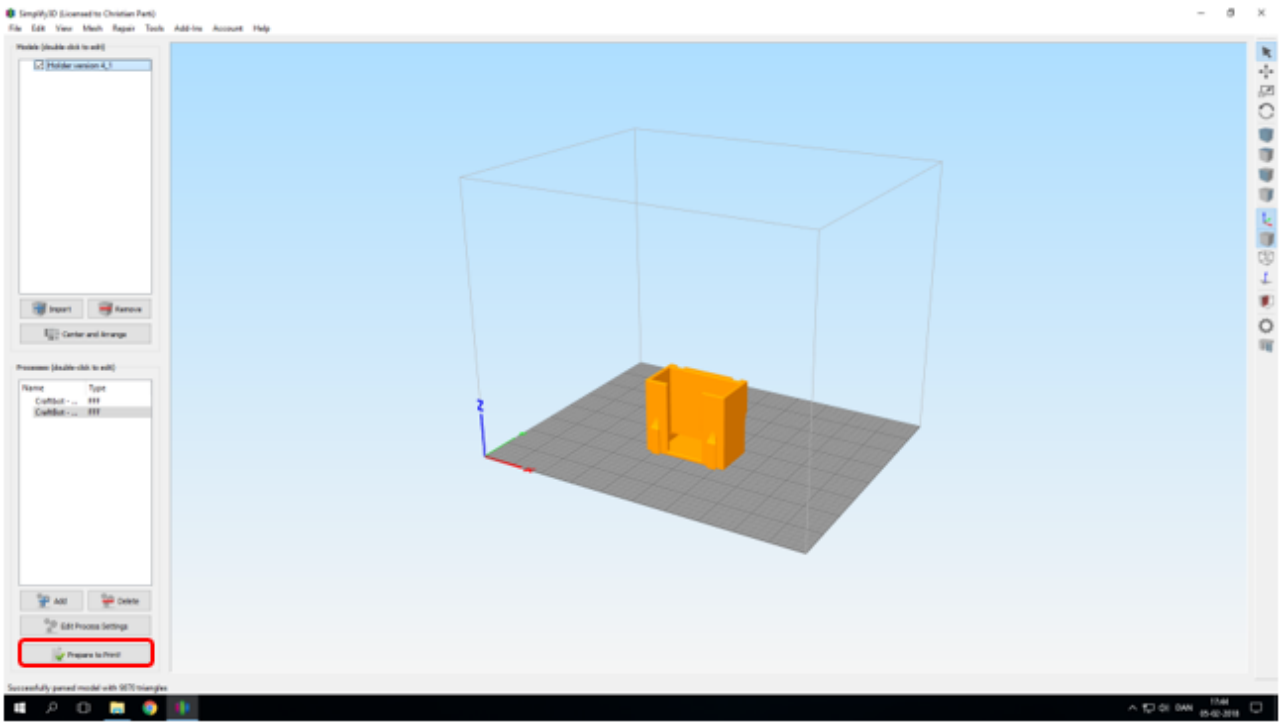
When your file has been successfully imported, expect to see something alike to the picture above.

Orientation

Should the part not be oriented correctly as seen below, double-click the part-name in the upper left corner, whereby rotation and maneuvering of the part can be done using the pane found to the right.



When part is oriented correctly (Make sure Z-offset is at 0), press “Prepare to Print!”



Choose the right filament:

PLA and PETG filament are the standard available printing material. They are compared below:

Material	PLA	PETG
Strength	+	+++
Temperature tolerance	+	+++
Hardness	+	++
Flexibility	+	++
Stiffness	++ (E = 3 - 4 GPa)	+ (E ≈ 2.2 GPa)
Outdoor use	+	++
Color matching	+++	++
Finish	+++	++
Postprocessing	++	+
Density (g/cm^3)	1,25	1,27
Printability	+++	++
Smell and gas	None	None

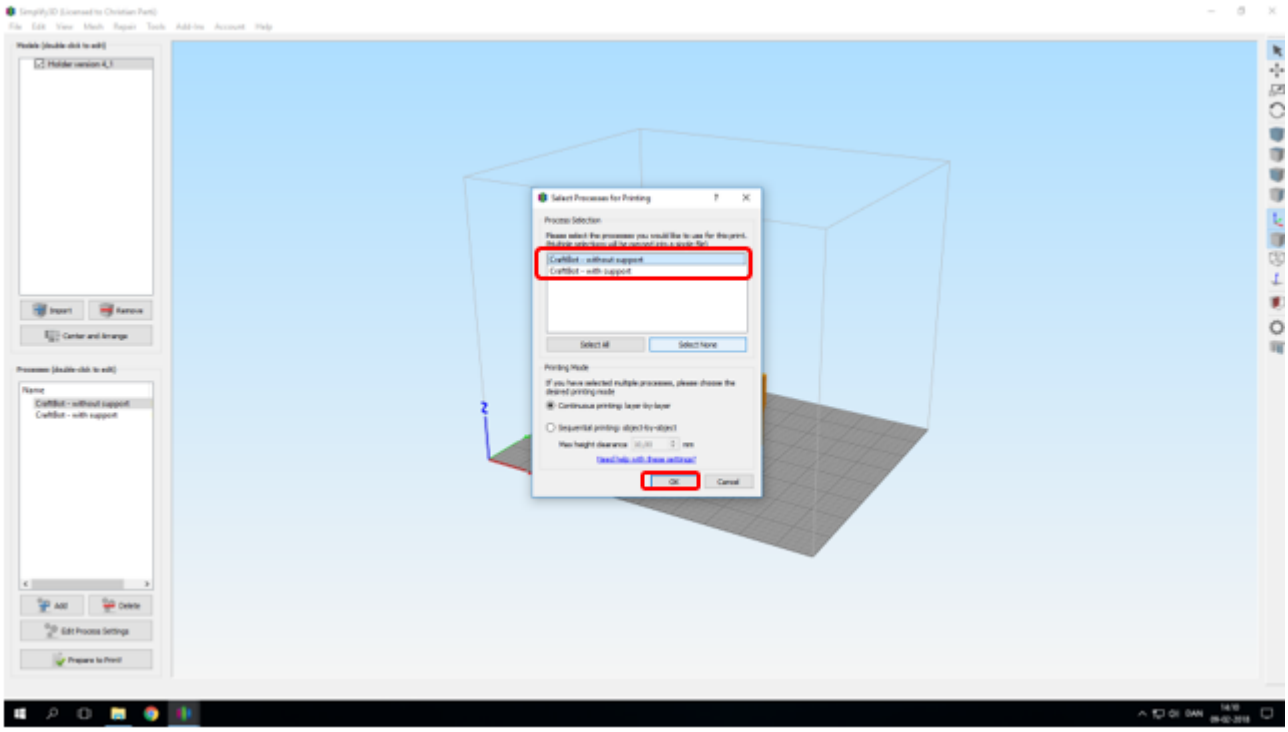
PLA is the easiest material to print, and should be used for most prints. However, make sure the filament properties matches the desired quality of the print.

Special filaments:

If you need more special kinds of filament, contact the Navitas Pilots. We can procure all sorts of filaments for your projects, including: ABS, Carbonfiber, Woodfilament, stonefilament, flexible, crystal, and so forth. Also, should you need different colours in your filaments (Color recognition in robotics), we can find it for you.

Preparing to print

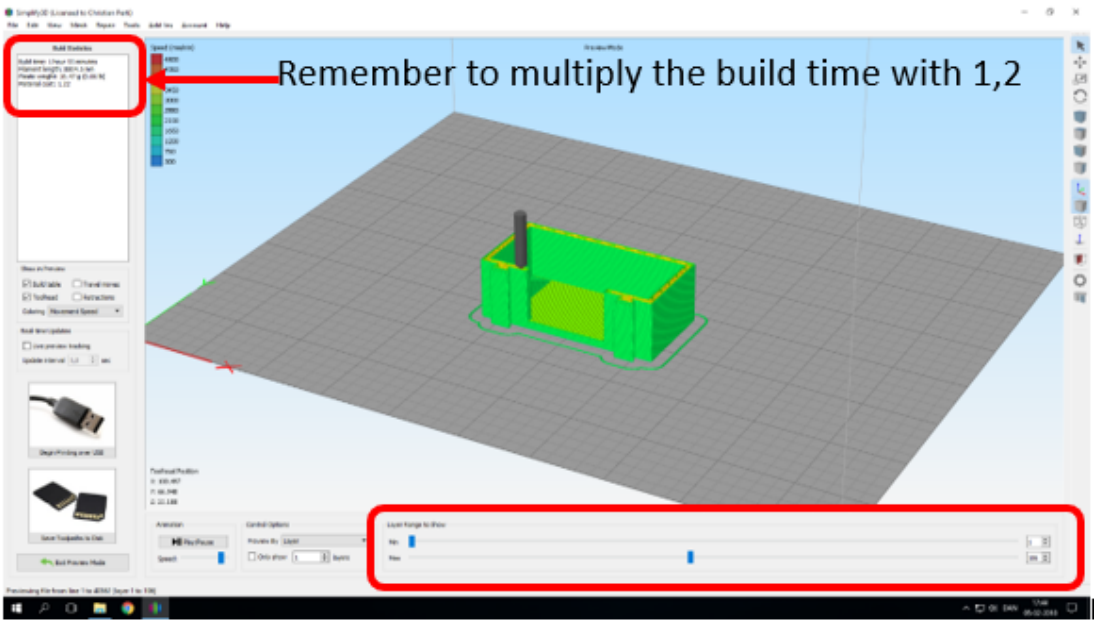
After having pressed “Prepare to print” a process has to picked. Do **ONLY** use the predefined processes, ”**PLA, PLA Support, PETG, PETG Support, XL PLA, XL PLA Support, XL PETG & XL PETG Support!**”. Choose the one best suited for your project, depending on whether your part needs support or not. Make sure to only select one process and not both (highlighted = chosen). Also, make sure that the **process matches the filament attached to the printer** (read the label on the spool).



It is under no circumstances allowed to edit the predefined processes or use others than the ones mentioned above. Should the processes be altered or gone, or your project have special needs, feel free to contact the Navitas Pilots at X-Lab.

Estimate of time, material usage and cost

When the appropriate process has been chosen, the screen will switch to one similar to the one seen below:

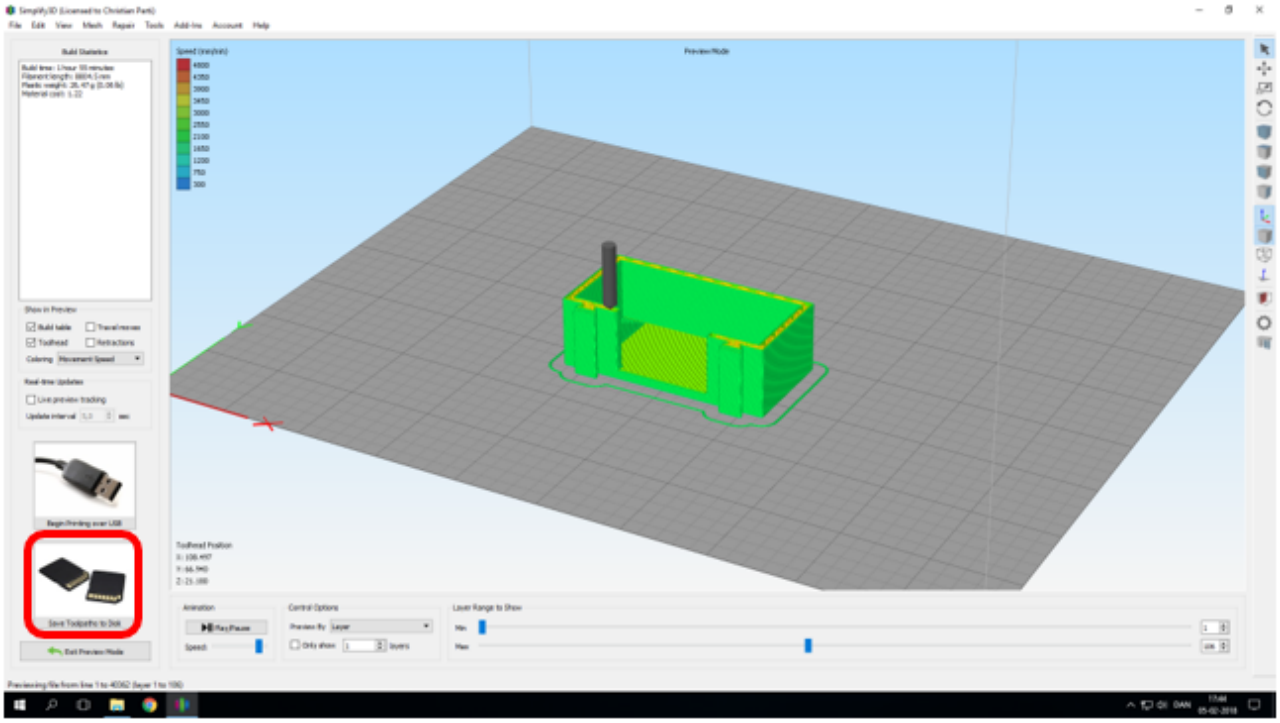


In the upper right left corner an estimated print-time, material usage and material cost are shown. These are only guidlines, which is why the time should be multiplied by 1,2 when determining time needed to print your part(s). Use the multiplicated time-estimate for booking, adding the time necessary for slicing, starting the printer etc.

Inspection of g-code and transfer to CraftBot

In the lower right corner, a slider is displayed, which display the individual layers of the printing of the part. Check the layers of the print, and make sure there are no abnormalities, sudden eaves or missing layers. Make sure that all layers are correct before moving on.

When you've made sure the g-code is working as intended, remove the USB from the printer you've booked and place it into the screen connected to the desktop. From here press "Save Toolpaths to Disk" (see below) and save the g-code to the USB-stick. The name of the part should match the name of the booking of the printer.



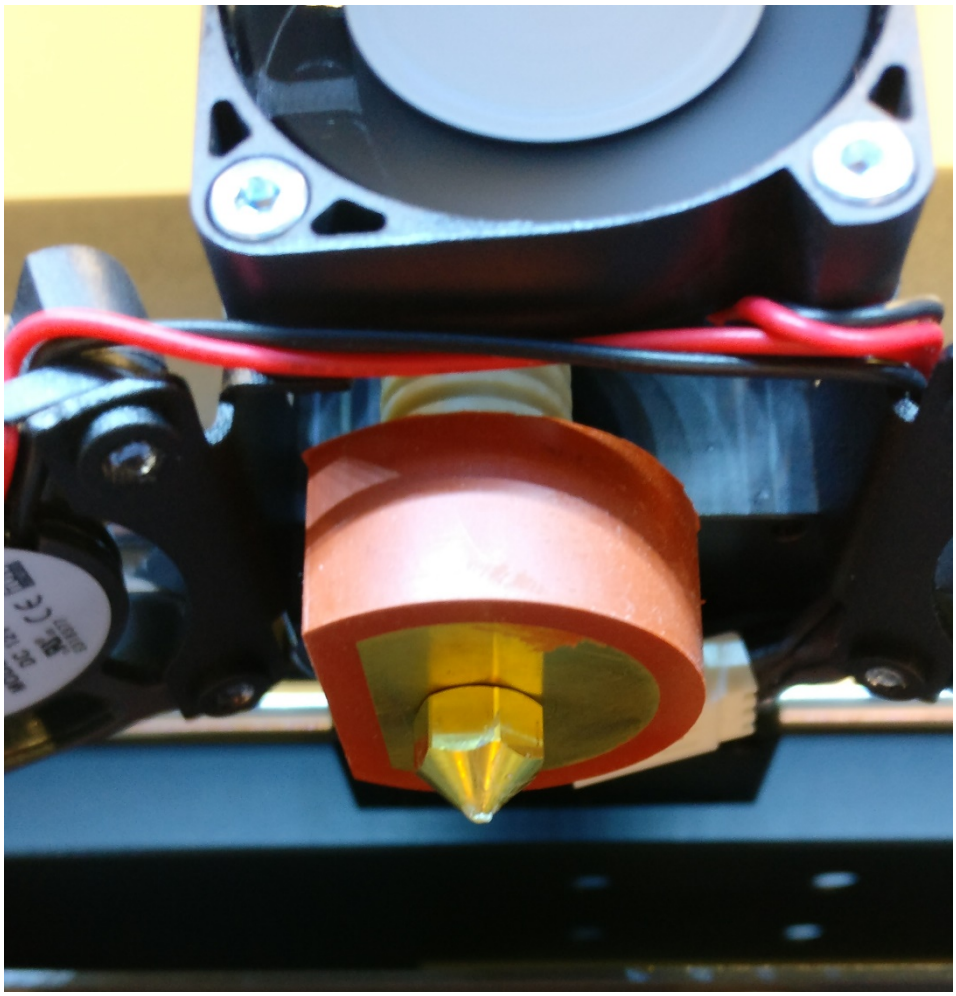
Print

Before the print is started the printer needs to be checked.

Check the following:

- Is the printer clean
    - The printbed is clean
    - No loose filament in the printer
    - General cleanliness
- Is the nozzle clean
- Is there enough filament to finish the print
    - If in doubt, weigh the spool and compare with the estimate of Simplify3D (the software is pretty precise with regards to material usage)
    - The printer won't stop when out of filament and will continue to print air.
- Make sure the bed is unmovable, so it doesn't shift during the printing

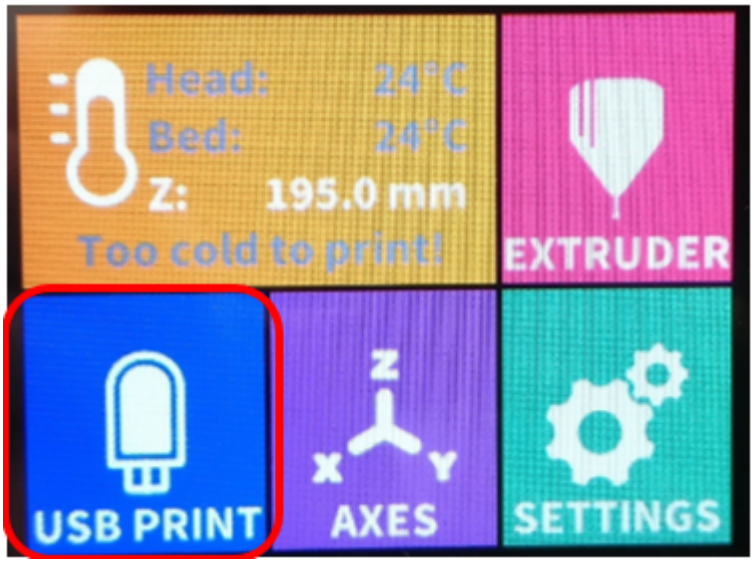
A clean nozzle is seen below:



The 3D-printer is switched on using the switch found on the back of the machine

The USB-drive is inserted back into the 3D-printer in the input above the screen

You are now ready to print. Using the printers interface the printing is started.



Press “USB PRINT”

Choose your file and press print as seen above

The display will now show an interface, where among other information the temperature of the bed and nozzle is shown. The temperature of the bed and nozzle will rise to the temperature shown to the right, and the print will start. Keep your fingers out of the printer while it's printing.

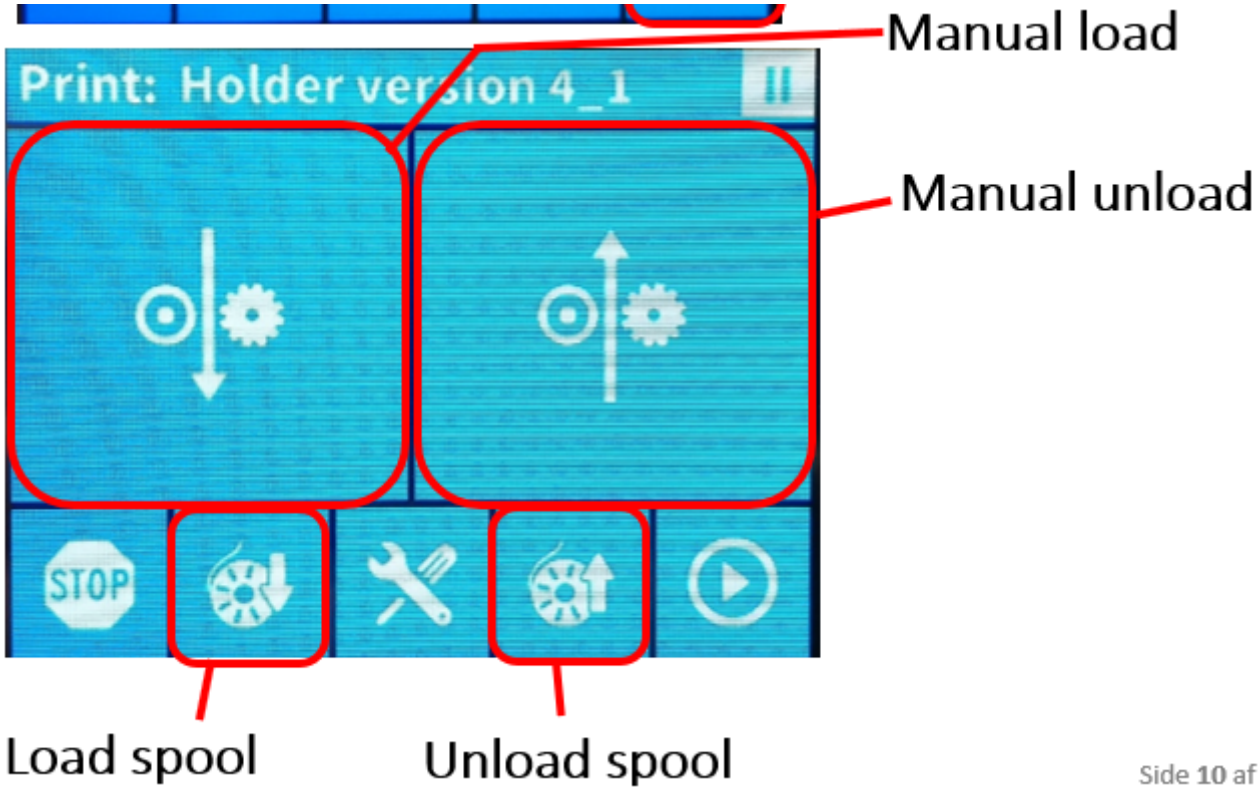
Make sure the fan at the nozzle is always running at temperature above the 150 degrees Celsius.

It is compulsory that you stay for the first 2 layers of the print to make sure the print attaches properly to the print bed.

Changing filament

If you have to change filament during your print, press the pause button:

The following menu will show:

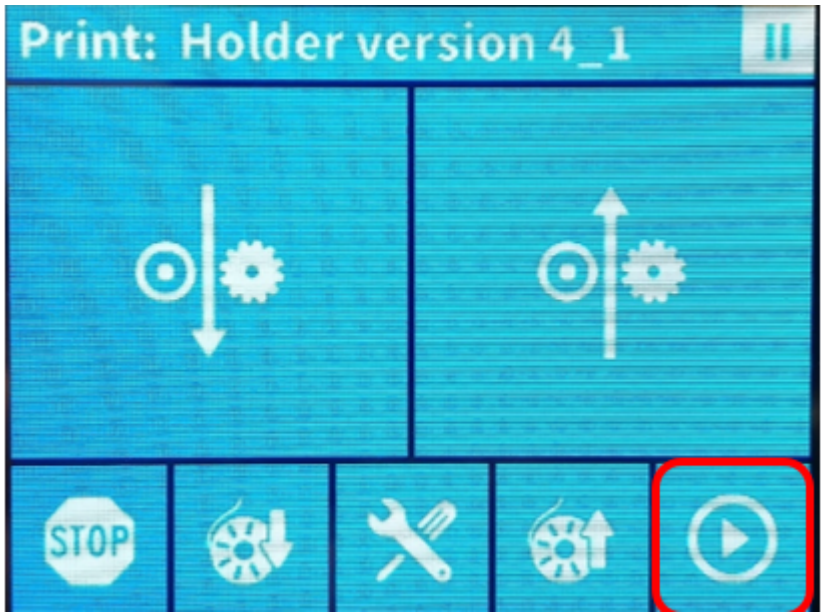


Side 10 af

From this menu it is possible to manually load and unload the spool. You can also let the machine do it.

After pressing “Unload Spool” remove the rest of the filament and the spool is changed. The filament from the new spool is pulled through the plastic tubing towards the nozzle and “Load spool” is pressed. The printer will now purge filament through the nozzle, until the rest of the previous filament is removed.

Press the play button to resume the print, see below:



Finishing up

When the print is done the print bed temperature has to be lower than 40 degrees Celsius before the print is removed. This is to prevent damage to the surface of the bed.

Remove the part(s) without using violence and clean the print bed from any residue that may have been left. This is in almost all cases possible using your fingers.

Clean the entire printer of any residue, so the printer is ready for the next user, and lastly turn off the machine.

Post processing

If the part is printed with support it is often possible to remove using your hands. If tools are necessary, contact the pilots of X-Lab.

Typical mistakes

The part doesn't bind properly to the bed	The bed might not be level, contact the pilots
The part is damaged because of overhang or bridging	Use the support process
The printer won't start the print, when the print-symbol is pressed	Make sure the name of the part doesn't contain æ, ø, å, Æ, Ø or Å
The bed is loose	Tighten the two white screws beneath the bed using your fingers – don't overdo it (no tools!)
The bed isn't level	Contact the pilots

If any other problem is observed or you have any questions, feel free to contact the pilots.