TRACTUS3D

A PROUD DUTCH COMPANY

HEART
SOUL
&
DUTCH
PERFECTION

Inside every 3D Printer

PRO series

Installation and user manual

Original manual v3.0

TRACTUS3D

Table of Contents

Disclaimer

Please read this installation and user manual carefully before using this product. Misuse or failure to follow warnings and instructions may result in personal injuries, inferior results or damages to the 3D printer. Users and purchasers of this product must know and understand the contents of the manual for correct use and care of the Tractus3D Printer.

The conditions or methods necessary for assembling, care, storage, use or removal of the product is not within our control or knowledge. Therefor we do not take any responsibility and explicitly disclaim liability for injuries, loss, damages, or costs ensued from or connected with the assembly, care, storage, use or removal of the Tractus3D Printer.

The contents of this document is provided by Tractus3D and without warranties of any kind, either expressed or implied in relation to the propriety, reliability or accuracy or otherwise of these contents.

Intended use of our 3D printers

Tractus3D printers are engineered and manufactured for use within a commercial environment and for FFF materials. Tractus3D printers are ideal for functional prototypes, concept models and series production on a small scale. With the usage of Simplify3D, we have reached a very high level in the production of 3D models. However the user is responsible for qualifying and validating the utilization of the printed object, in particular if these are vital in highly restricted circumstances, like medical or automotive applications.

Our printers are compatible with 3rd party filaments but we do recommend the use of Tractus3D materials for the best results. The properties of these materials are fully matched with our printer settings.

Preface

This is the installation and user manual for your Tractus3D printer.
This manual contains chapters about the installation and use of the 3D printer.

The manual contains important information and instructions on safety, installation and use. Please read all information and follow the instructions and guidelines in this manual carefully. This ensures that you will obtain great quality prints and that possible accidents and injuries are prevented. Make sure that everyone who uses the Tractus3D printer has access to this manual.

Every effort has been made to make this manual as accurate and complete as possible. The information is believed to be correct but does not aim to be all inclusive and shall be used only as a guide. Should you discover any errors or omissions, please bring this to our attention, so that we can make amendments.
This will enable us to improve our documentation and service to you.

1. Safety and compliance

Always work safe with your Tratus3D printer and follow all warnings and instructions to prevent any possible personal injuries or accidents.

1.1 Safety Notices

This manual includes warnings and safety notices.
Provides additional information that is helpful to do a task or to avoid problems.
Warns of a situation that may cause material damage or injuries if one does not follow the safety instructions.

General safety information

The Tractus3D printer has moving parts that will get hot and generate high temperatures. This may cause injuries. When in operation, never reach inside of the 3Dprinter. Control the 3Dprinter with the touchscreen, positioned on the front or with the power switch at the front. Let the Tractus3D printer cool down for at least 5 minutes before reaching inside.
Do not leave or store items inside the 3D printer.

Without explicit instructions from Tractus3D, please do not adjust or change anything on the Tractus3D printer. Persons with reduced physical and/or mental capabilities that are unable them to act accurately in case of emergency or persons with lack of experience and knowledge, should not use the Tractus3D Printer. Only when given instructions concerning the use of the appliance, or under supervision by a person responsible for their safety.

Children should always be under supervision when using the 3D printer.

1.2 Hazards

Electromagnetic compatibility (EMC)

This product has been tested and found to comply with the limits for a digital device Class A, according to part 15 of the FCC Rules. These limits are designed to provide reasonable protection against harmful interference when the 3D printer is operated in a commercial environment. This Tractus3D printer generates, uses, and can radiate radio frequency energy and, if not installed and used in accordance with the instruction manual, may cause harmful interference to radio communications.

Electrical safety

Every printer has passed testing before shipment. This test guarantees the right level of insulation against electrical shock. An earthed mains socket must be used. Be sure that the building installation has dedicated

means for over-current and short-circuiting.
The printer is powered by a power adapter or mains voltage, which is hazardous when touched. This has been marked with a high-voltage symbol.

Always unplug the printer before doing maintenance. Never turn off the printer with a print head above 50 degrees Celsius!

Mechanical safety

The Tractus3D printer complies to the Machine Directive 2006/42/EU. The Tractus3D printer contains moving parts. No damage to the user will be expected from the drive belts, arms or slides. Always unplug the printer before doing maintenance.

Risk of burns

Do not touch the heated bed or printhead with your bare hands.
The print head can reach temperatures up to 450 °C and the heated bed can reach temperatures of more than 100 °C. This means there is a potential risk of burns!

Let the 3D printer cool down for 30 minutes before you perform any maintenance. Never turn off printer with a print head above 50 degrees Celsius!

Health and safety

Our printers are compatible with all brands of 3D print filament, but we do recommend the use of Tractus3D materials for the best results. The properties of these materials are fully matched with our printer settings. These materials can be printed safely if the recommended temperatures and settings are used. Printing in a well-ventilated area is always recommended. Please check the SDS of each specific material for more information.

Third-party materials may release VOCs* that are above regulated threshold levels when processed in Tractus3D printers (not covered by warranty). These substances may cause headaches, fatigue, dizziness, confusion, drowsiness, malaise, difficulty in concentration, and feelings of intoxication. We recommend the use of a fume. Check the SDS of these materials for more information.

*Volatile Organic Compounds

2. Introduction

About the possibilities of your new Tractus3D printer

2.1 Main components

- Print head(s)
- Heated build plate
- Spool holder
- Touchscreen
- Extruder
- Bowden tubes
- Polycarbonate panels
- Power socket with cord or adapter
- Ethernet port
- Power Switch

2.2 Specifications T850P

Machine properties

Printing technology	Fused filament fabrication (FFF) - Fused deposition modeling (FDM)
Build width	280 mm diameter (11”)
Build height	400 mm (15.7”)
Speed	450 mm/s
XYZ resolution	10, 10, 10 micron
Layer resolution	0.4 mm nozzle: 300 - 10 micron
	0.6 mm nozzle: 450 - 10 micron
	0.8 mm nozzle: 600 - 10 micron
	1.0 mm nozzle: 800 - 50 micron
Operating sound	+/- 40 dBA
Power rating max.	3100 W
Connectivity	LAN
Display	7” color touchscreen (17.8 cm)
Language support	English, Dutch, French, German
Monitoring	Web interface
Mohawk	Included

TRACTUS3D

Physical dimensions

Dimensions	550 mm x 476 mm x 1255 mm
Net weight	30 kg
Shipping weight	37 kg
Shipping box dimensions	660 mm x 620 mm x 1010 mm

Heated chamber

Heating	Actively heated chamber
Chamber heat up time	< 7 min (from 0 to 60 °C)

Build plate

Build plate material	Heated glass build plate
Build plate temperature	10 - 175 °C
Build plate levelling	Automatic bed levelling
Build plate heat up time	< 2 min (from 20 to 60 °C)

Print head

Feeder type	Hardened steel geared (ready for composite materials)
Nozzle diameter	0.4 mm, 0.6 mm, 0.8 mm, 1.0 mm
Nozzle temperature	Up to 450 °C
Nozzle heat up time	< 90 sec
Max. extrusion volume	< 40 mm3/s

Ambient conditions

Operating ambient temperature	15 - 35 °C, 10 - 90% RH non-condensing

Materials

Supported materials	PEEK, PEI, PLA, PRO1, PET-G, ABS, ASA, Facilan C8, TPU/TPE, Nylon, PC, Fillers
Filament diameter	1.75 mm

Software

Supplied software	Simplify3D with 1 license (usage on 2 computers)
Supported OS	MacOS, Windows and Linux
File types	STL, OBJ, 3MF, JPG, PNG
Printable formats	G, GCODE

2.3 Specifications T650P

Machine properties

Printing technology	Fused filament fabrication (FFF) - Fused deposition modeling (FDM)
Build width	165 mm diameter (6.5”)
Build height	300 mm (11.8”)
Speed	300 mm/s
XYZ resolution	10, 10, 10 micron
Layer resolution	0.4 mm nozzle: 300 - 10 micron 0.6 mm nozzle: 450 - 10 micron 0.8 mm nozzle: 600 - 10 micron 1.0 mm nozzle: 800 - 50 micron
Operating sound	+/- 40 dBA
Power rating max.	480 W
Connectivity	LAN
Display	5” color touchscreen (12.7 cm)
Language support	English, Dutch, French, German
Monitoring	Web interface

Physical dimensions

Dimensions	371 mm x 324 mm x 845 mm
Net weight	9 kg
Shipping weight	20 kg
Shipping box dimensions	430 mm x 495 mm x 905 mm

Build plate

Build plate material	Heated glass build plate
Build plate temperature	10 - 150 °C
Build plate levelling	Automatic bed levelling
Build plate heat up time	< 3 min (from 20 to 60 °C)

Print head

Feeder type	Hardened steel geared (ready for composite materials)
Nozzle diameter	0.4 mm, 0.6 mm, 0.8 mm, 1.0 mm
Nozzle temperature	Up to 450 °C
Nozzle heat up time	< 90 sec
Max. extrusion volume	< 15 mm3/s

Ambient conditions

Operating ambient temperature	15 - 35 °C, 10 - 90% RH non-condensing

Materials

Supported materials	PEEK, PEI, PLA, PRO1, PET-G, ABS, ASA, Facilan C8, TPU/TPE, Nylon, PC, Fillers
Filament diameter	1.75 mm

Software

Supplied software	Simplify3D with 1 license (usage on 2 computers)
Supported OS	MacOS, Windows and Linux
File types	STL, OBJ, 3MF, JPG, PNG
Printable formats	G, GCODE

3. Unboxing

Unpack your Tractus3D printer carefully and set up the hardware according to the instructions on the quick start manual supplied with the 3D printer.

Make sure when you install the printer on the location, you will use the swivel feet to adjust the 3D printer to be level as much as possible in all directions.

Don't forget to remove all yellow clips and zip-ties before turning on the 3D printer!

4. Setup for first use

You need to set up the printer for the first use, after installing. This chapter will explain how to set up network settings, load materials, update the firmware, install Simplify3D and prepare the build plate.

Turn on the printer with the power switch at the front, to perform the setup steps.

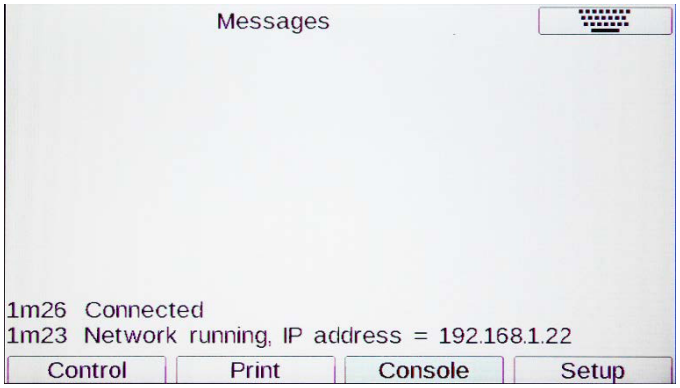
4.1 Set up network

After turning on the printer you need to configure the printer to be used in your network. We prefer to have static IP address for the printer or a DHCP reservation made by the network administrator.

Preparation needed for network settings if you want to use static IP:

A free IP address
Subnet mask
Gateway IP address

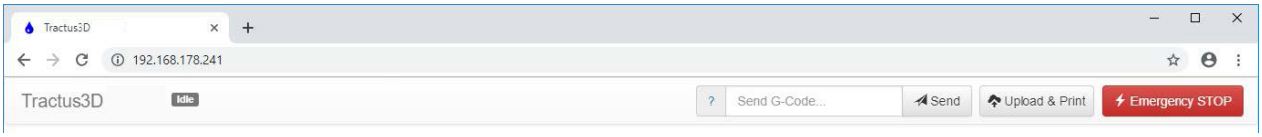
When you turn on the printer for the first time the IP address will be assigned by your DHCP server in the network. To view the IP address, go to the console tab on the LCD screen.



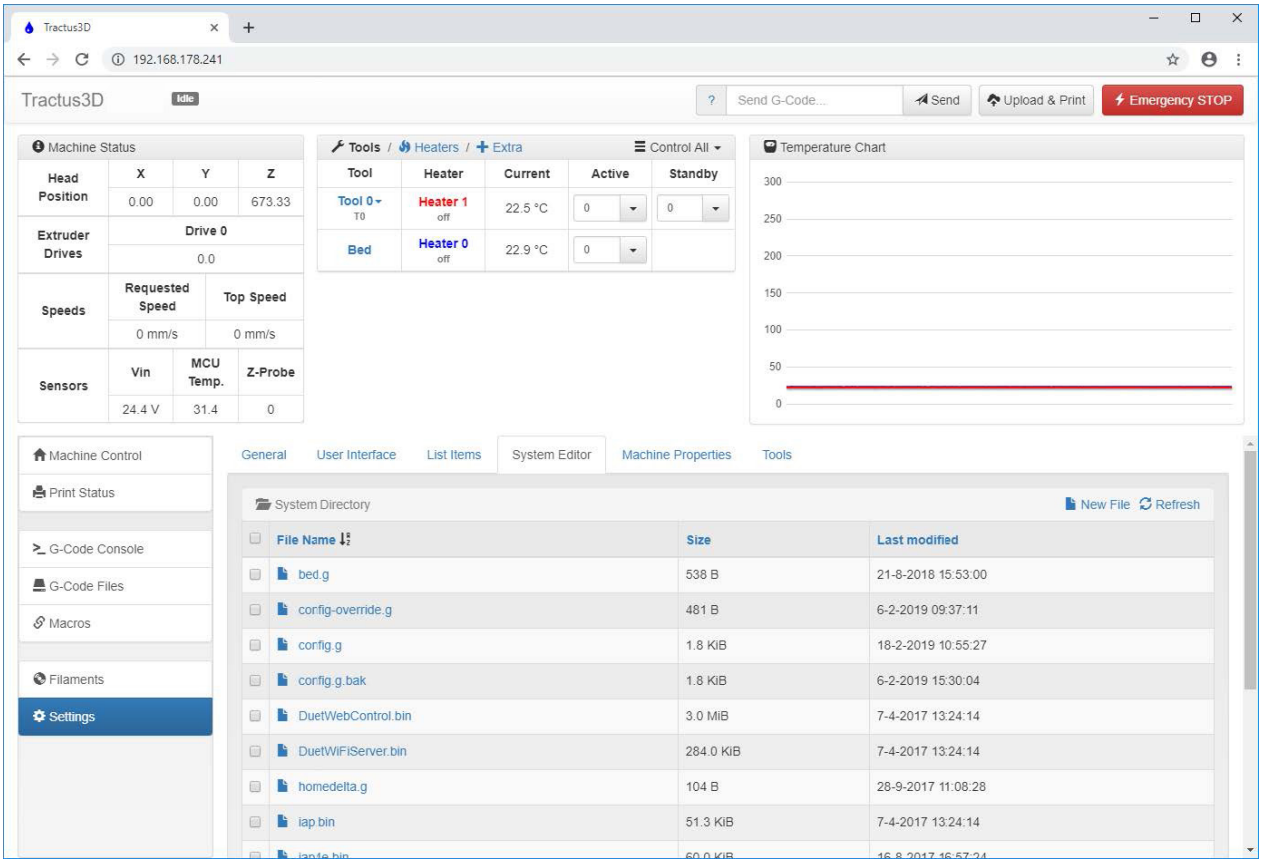
Change to Static IP address

To set a static IP address please use the following instructions in the Web Interface of the printer.

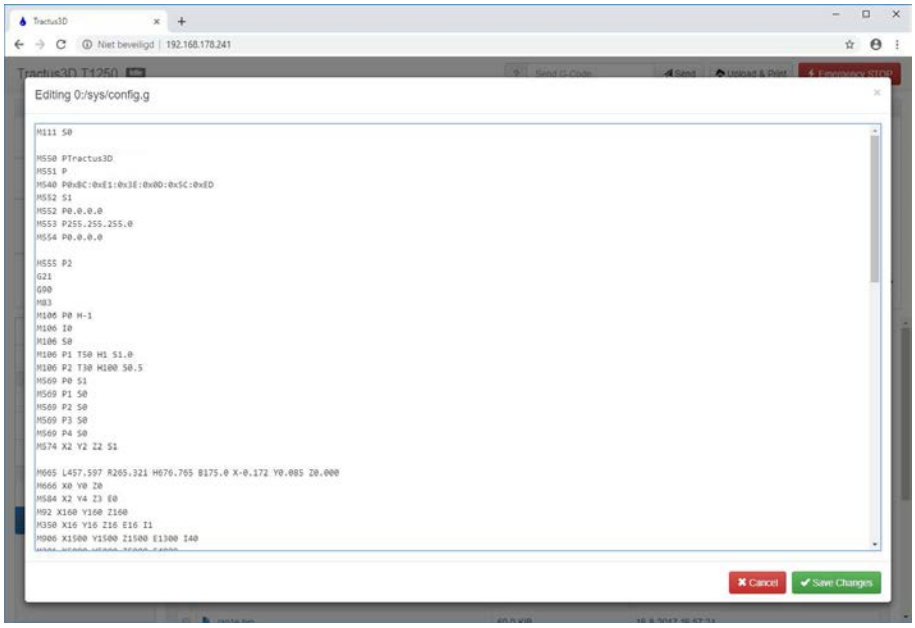
To access the web interface, open a web browser (prefer Chrome) and surf to the IP address which you found on the console tab of the LCD screen.



Go to Settings and select the System Editor



Open the config.g file via a left mouse click on the name config.g



Search in this file for the line:

M552 P0.0.0.0

Change this line to:

M552 Pxxx.xxx.xxx.xxx (Where xxx.xxx.xxx.xxx is your IP address, for example: M552 P192.168.1.10)

Next search for the line:

M553 P255.255.255.0

This is your subnet mask. If you have to change it, do this accordingly your details of the subnet mask in the same way as you changed the IP address

Next search for the line:

M554 P0.0.0.0

Here you set your Gateway IP address in the same way as you changed the IP address.
After changing this detail, please click on **Save Changes** in the bottom of the editor.

Now you only have to reboot the machine to make the changes active. At the question, if you want to restart the printer please click **Yes.**

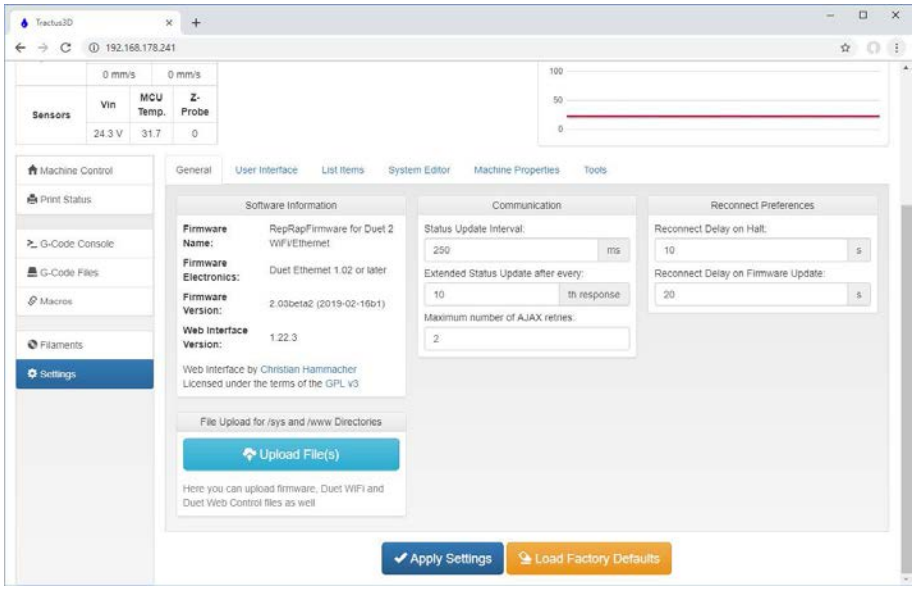
Change to DHCP IP address

If you want to revert back to DHCP mode, please change the IP address to 0.0.0.0 and follow the chapter about **Change to Static IP address.** You don't have to change the gateway to put the machine in DHCP mode.

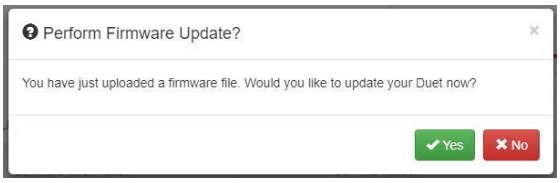
4.2 Update the firmware

All printers will be shipped with the latest firmware onboard. In some rare situations it's possible that while shipping your 3D printer a new firmware is released. In this case, follow these steps to update the firmware. Make sure you did the setup procedure for the web interface described in chapter **4.1 Set Up Network.**

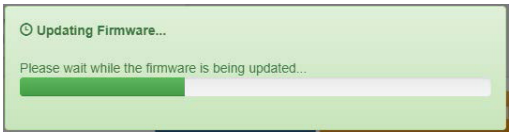
In the web interface go to **Settings** and tab **General.**



Click on the blue button Upload File(s) and select the firmware you've received from Tractus3D.
The interface will give you a notification that you uploaded a new firmware.



Click **YES** to update your firmware.

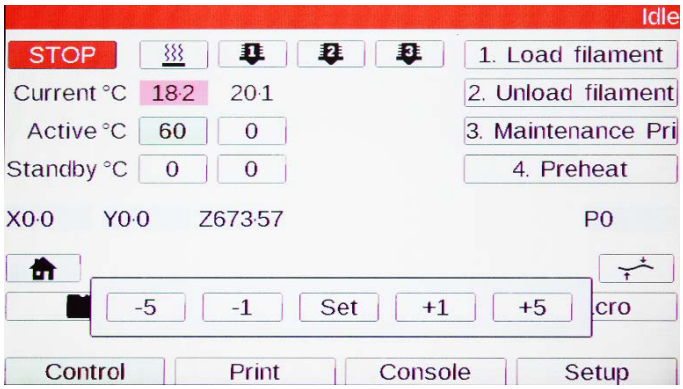


Check the Firmware Version number to see if the update is loaded successfully.

4.3 Calibrating the print bed

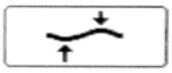
When you install the 3D printer for the first time or you have moved your printer to another location please always calibrate the print bed.

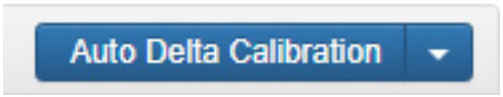
Make sure the print bed is empty and cleaned. The nozzle is clean without any residue of filament.
Heat up the bed to 60 degrees by clicking the LCD screen on the **BED heater icon.**



Click on **the active temperature box** below this heater and set the temperature to 60 (select temperature by clicking + until desired temperature is reached and click **SET** afterwards). You can also use the web interface to set the bed temperature to 60 degrees.

Use the LCD screen on the **Bed Level icon** or in the web interface on the **Auto Delta Calibration** button. Please make sure the 3D printer has reached the bed temperature and the bed is clean!





The printhead will go down and perform some calibration points on the bed. This will take around 1 minute to complete. When the calibration is done the printhead will go up and calibration is done.

To make the calibration more accurate you can redo this step multiple times. The machine will try to get better each try.

If you want to view the results you can click on the console button and look for the results.

4.4 Installing slicing software

Default we supply Simplify3D with our 3D printers as slicing software. As we do recommend this software it's not the only one that you can use for the 3D printer. Tractus3D is a reseller of Simplify3D and does give basic support for this and their printers but most of the support will be found online at the Simplify3D website.

https://www.simplify3d.com/support/

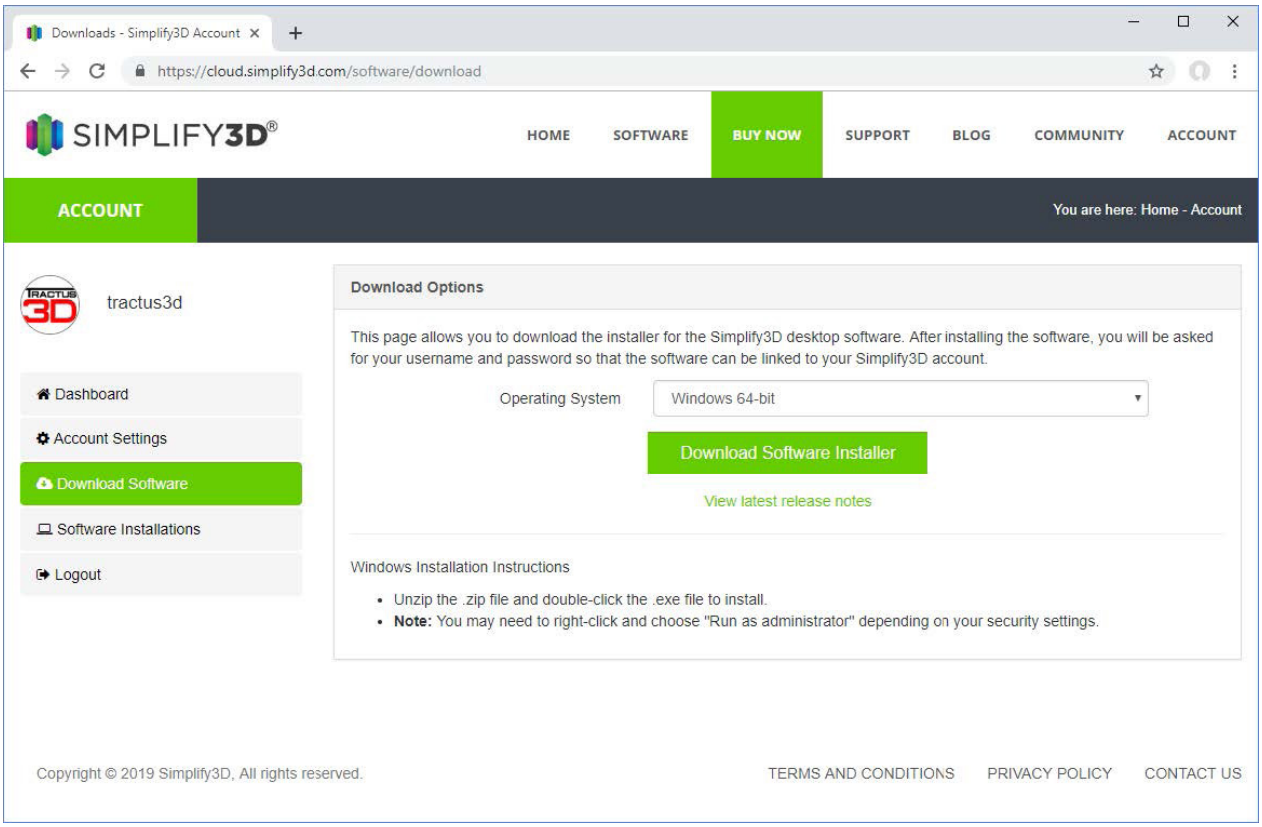
To get the Simplify3D license with your 3D printer please fill in the Request Simplify3D license on our website. Please use a general e-mail address for your request as this will be used company wide.

https://tractus3d.com/getsimplify3d

Fill in this form and you will receive an e-mail with your license.

You'll have to make a login which we advise to keep it simple and general (like the company name) because also this will be used company-wide on all Simplify3D installations.

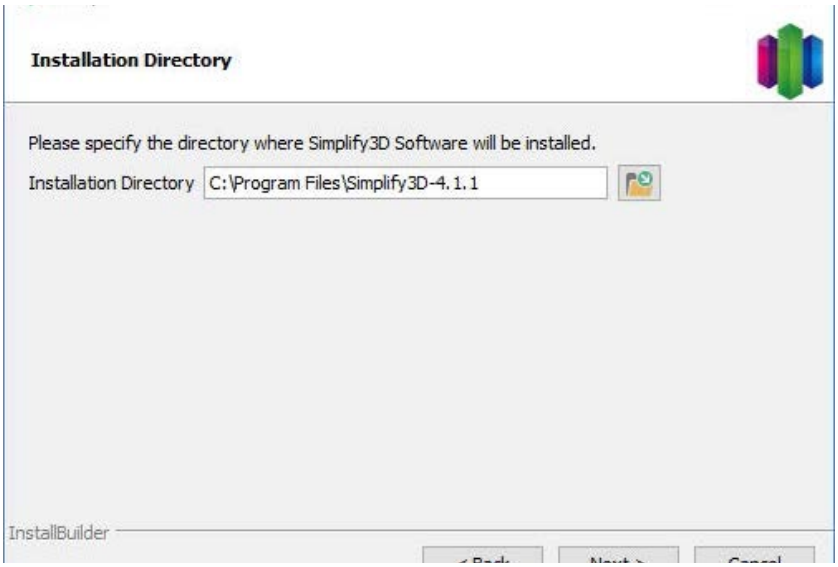
When you login on the Simplify3D website click on **Download Software** and choose your operating system.



See the Windows Installation Instructions on the website how to open the installer.

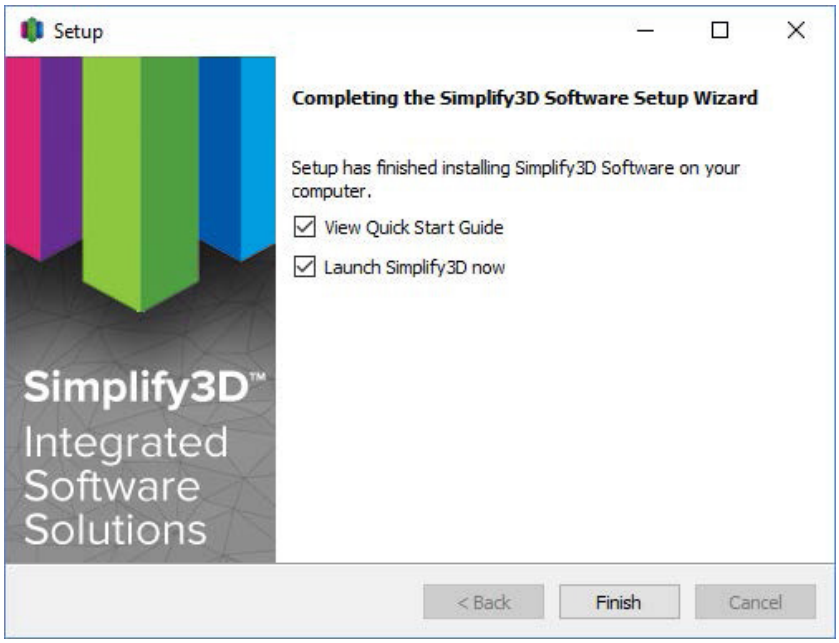
Select your installation language and click OK.

Select Next > read the Simplify3D software End User License Agreement and **Accept** the agreement to continue. Choose **I accept the agreement** and click **Next >**



In the next window enter the installation directory. Click **Next** >

After this the installation will start by clicking one more time on **Next** >



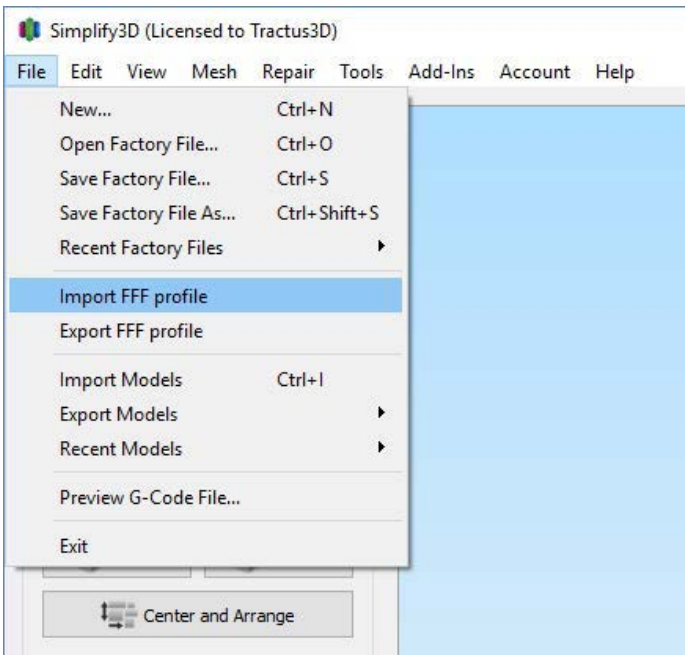
Click on Finish to start Simplify3D for the first time.

On the printer installation screen click **Cancel**. Simplify3D will now open in default settings.

Importing Tractus3D settings in Simplify3D

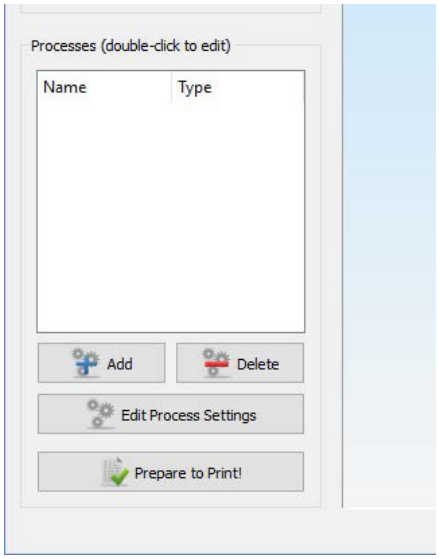
Now it's time to setup your printer in Simplify3D by the latest settings delivered by Tractus3D.
When you requested your license you also received a mail from Tractus3D with a .FFF file. (settings file for Simplify3D).

In Simplify3D go to **File > Import FFF profile**



Select the .FFF file you received from Tractus3D and click **Open**

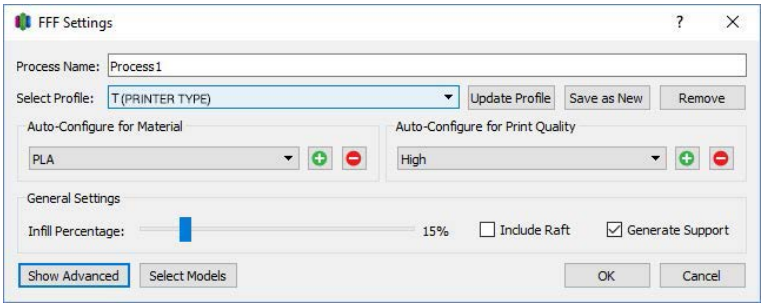
You will get a message which name the software gave the imported settings file.



Next go at the left side of the screen to **Processes** and click the **Add** button.

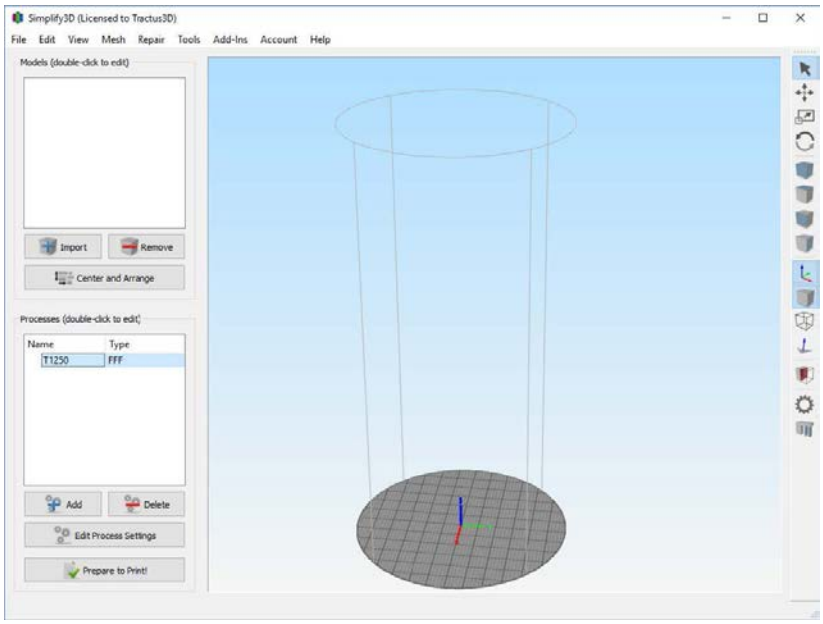
Select the imported settings under **Select Profile:**

Give the process profile you are making a name by entering a recognizable name in **Process Name:**



To see more details of the process, click on **Show Advanced**

Click on **OK** to close this window, your process profile is set up now



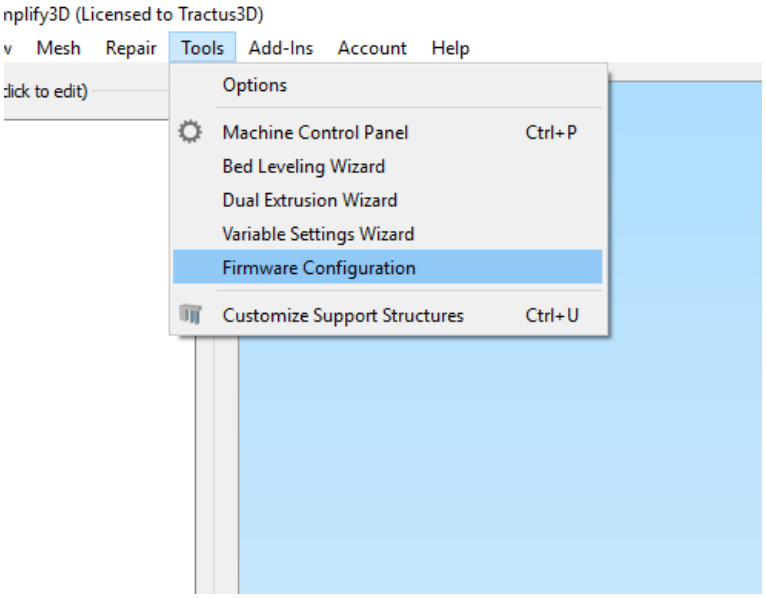
Click on **Import** at the **Models** section to import a STL or **OBJ** file which you want to print.
For more information how to get the best results with Simplify3D we recommend reading the support page on their website.

https://www.simplify3d.com/support/

Updating firmware settings in Simplify3D

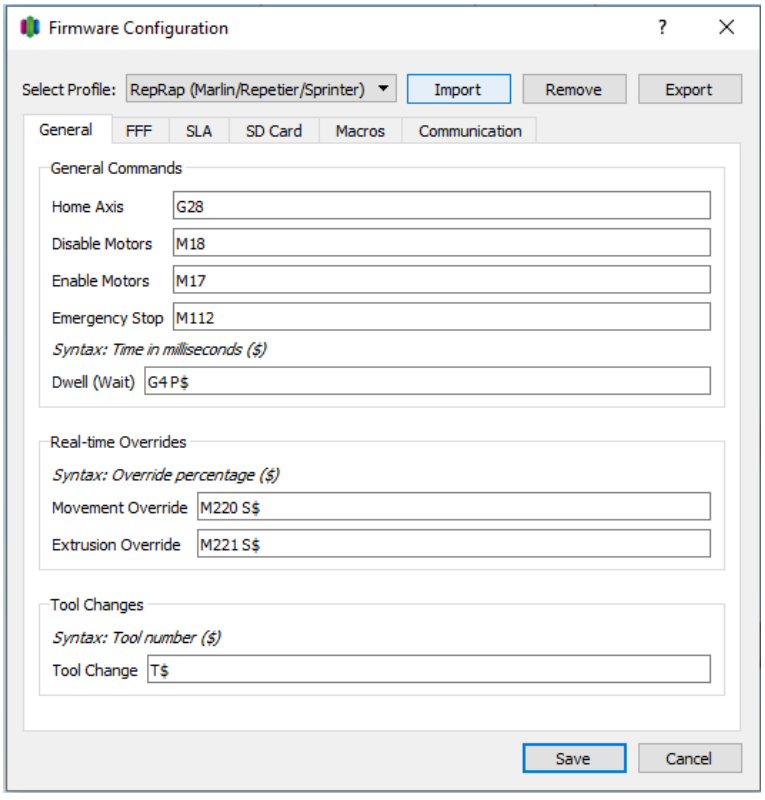
Next to configuring the profiles, the firmware file also has to be imported. The file is send out together with the printer profile. Follow the steps below to update the firmware settings in Simplify3D

In Simplify3D go to **Tools > Firmware configuration**



In this window choose Import and navigate to the location where you placed the firmware file.

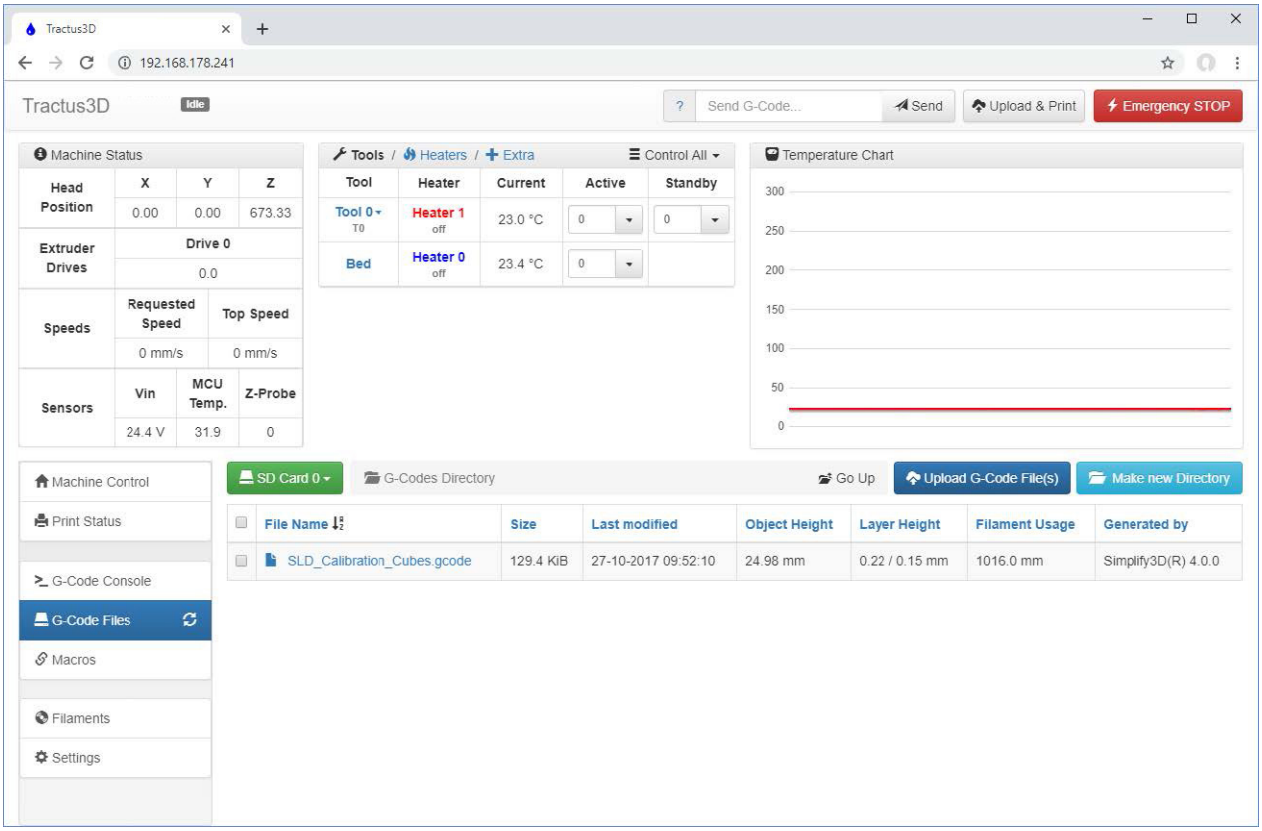
Please be aware that this file has to be loaded on overy boot of Simplify3D. So best practice is to store the file in a location that is always accessible.



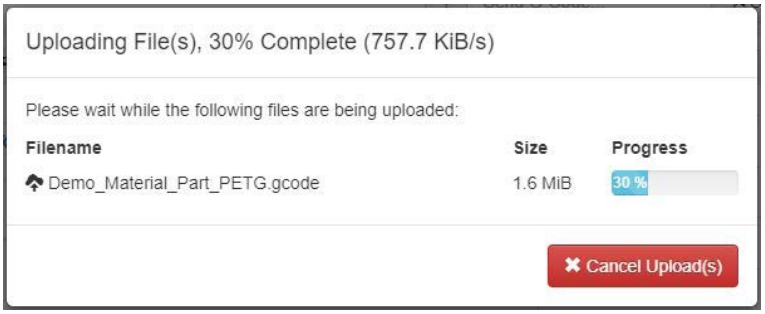
4.5 Uploading files to 3D printer

When you have created a file with your slicer software and you stored the G-Code file on your computer it's only a matter of uploading this to the 3D printer to get started.

To upload the file please open the web interface.



Go to **G-Code Files** and click the **Upload G-Code File(s)** button.



Select your file(s) from your computer. To upload click **Open.**
Now you will see an upload box, wait for it to complete.

When the upload is complete there will be a green CLOSE button. Now you can start printing your files!

5. Operation

After installation, it's time to start printing with your 3D printer. This chapter provides information about the web interface, LCD touchscreen, cleaning the bed, starting a print, removing the print and support material.

5.1 Web interface

With our web interface you can control the 3D printer remotely. In this chapter we will explain the main web interface buttons and actions takes or NOT to be taken while printing.

Machine Control

The web interface contains a lot of technical information which gives you heads up about your printer and status.

Basic information about the position of the printhead, speeds achieved and hardware monitoring such as input voltage and mainboard temperature.

Machine Status

Head Position	X	Y	Z
	0.00	0.00	673.33
Extruder Drives	Drive 0		
	0.0		
Speeds	Requested Speed	Top Speed	
	0 mm/s	0 mm/s	
Sensors	Vin	MCU Temp.	Z-Probe
	24.4 V	32	0

Here you can see the actual temperature of your bed and nozzle (Tool 0). You can also set a new active temperature.

Tools / Heaters / Extra — Control All

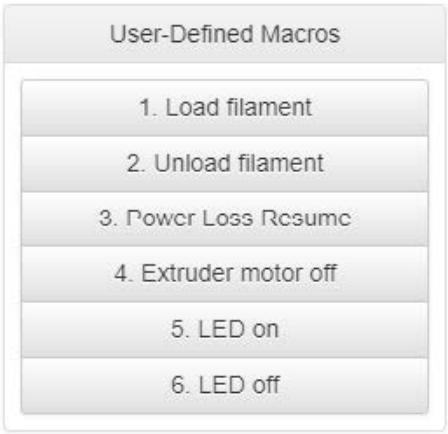
Tool	Heater	Current	Active	Standby
Tool 0 T0	Heater 1 off	23.0 °C	0	0
Bed	Heater 0 off	23.4 °C	0	

When you set a Tool temperature the tool isn't switched on default, you'll have to click the **Heater 1** to enable the heater.

Control your printhead when the printer is idle with these buttons. **Home All** will bring the printer to its starting position. **Auto Delta Calibration** is used for calibration of the machine **(see chapter 4.3 Calibrating the bed)**

NOTE: Please make sure you don't use these buttons while the printer is printing an object!

Here you can find pre-defined shortcuts or make your own shortcuts (macro's).
A macro is a sequence of g-code commands which you are using more often.



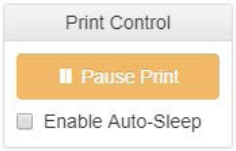
When you want to extrude manually use this control. The buttons **Retract** and **Extrude** will be only available when the nozzle (Tool 0) is at temperature for at least 170 degrees Celsius.

For quick loading use the feed rate of 15mm/sec or higher but for extrusion of filament only use 5mm/s or 1mm/s.

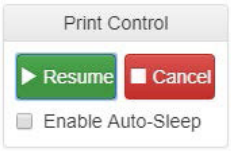
Print Status

This part of the interface will be only working when a print is running.

When the print is running and you want to pause the print job click on Pause Print.

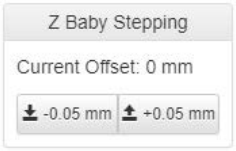


When you want to cancel the print first Pause the job and press **Cancel** to stop the job. Press **Resume** if you want to continue the print.

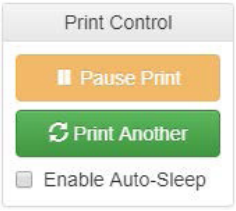


When printing the first layer your print head/nozzle can be too high or too low from the bed.

With "Baby Stepping" you can correct this for a better result., because your first layer is very important. To correct some slight offset you can use the baby stepping to get the print head closer to the bed **(-0.05mm)** or get the print head further away from the bed **(+0.05mm).**



When the print is ready you can click **Print Another** to make the same print again. Please make sure before you click this button you cleaned the bed as mentioned in chapter **Cleaning your bed.**



Here you can override the settings of your print file.

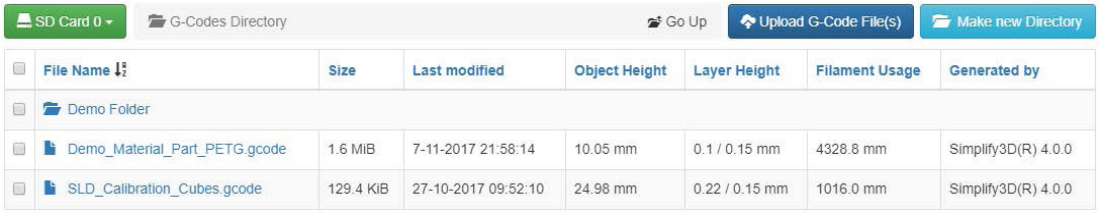
Control your Tool Fan (Material Fan) more or less than setup in the slicing software.

Control the speed of the printer

Control the extrusion multiplier of the printer

G-Code Files

Here you can upload your files or start a print. It's also possible to create folders to get your files organised.



Start a print

Click on a filename in the G-Code Files list to start printing this file.

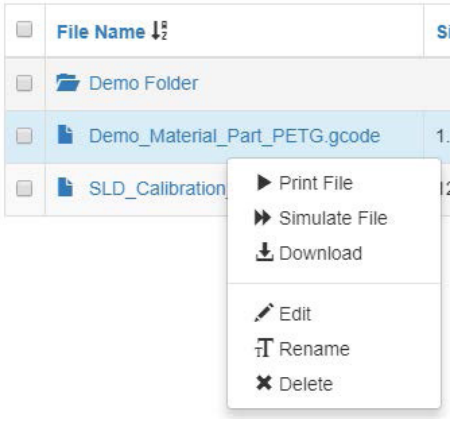
If you press **Yes**, the 3D printer will start this file. Most of the time (depending of the start up settings part of your slicing software) the 3D printer will start warming up the bed first.

Extra file options

Right mouse click (windows users) on the file will give you more options.

The **Simulate File** option will start a print (simulated) directly without visible motion on the 3D printer.

With this option you can calculate a really accurate time of printing and usage of the material.



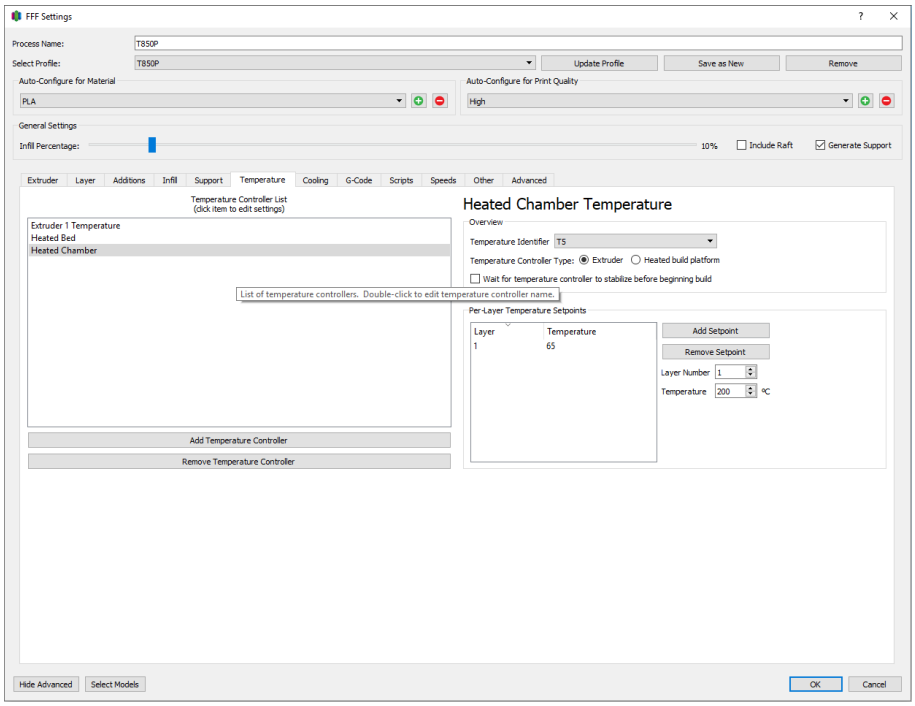
When the simulation is finished you will get a message on the web interface:

File Demo_Material_Part_PETG.gcode will print in 0h 38m plus heating time

With the more option list, you can also **rename**, **delete** a file or **download** the file to the computer. We don't advise to use the Edit if you have an editor like notepad++ on your computer as this is a slow method compared to an external editor.

5.2 Heated Chamber

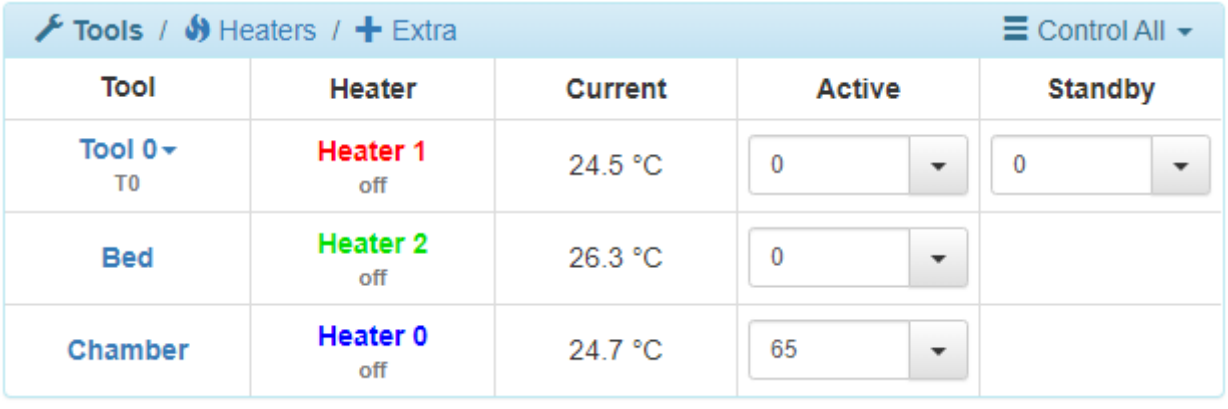
With the Tractus T850P 3d printer there is an option to heat up the build chamber. The air temperature around the printed model will remain stable throughout the entire print job to help avoid warping. This option can be set in Simplify3D on the temperature tab. Please note that we have unchecked the box "Wait for temperature controller to stabilize" which means the print will start, no matter what the chamber temperature is at that point.



The chamber temperature can also be set via the webinterface, or via gcode on the lcd. Please use the following gcode in the console tab of the lcd:

"M141 S65" to set the temperature to 65 degrees Celcius.

Currently the active chamber temperature is only displayed on the webinterface. Please keep in mind that when a file has been cancelled or has finished printing, the chamber heater will deactivate.



6. Materials

6.1 Material compatibility

The Tractus3D 3D printers are all with an open material support. This means you can use every material on the machine, but Tractus3D does not guarantee a successful print.

We know there are a lot of brands and material types available on the market. We do love experimenting with new materials if you know the risks and if you're not worried about a failed print to learn.

If you have troubles with the machine, we will always advise to go back to Tractus3D supported PLA with a clean (prefer new) nozzle.

Tractus3D supports the following materials (Tractus3D preferred filament)

- PEEK (CF and normal)
- PEI
- PLA
- PETG
- ABS
- Facilan C8
- PRO1

Known "experimental" materials can be used (not supported):

- ASA
- TPU (all shore values)
- PC (PolyCarbonate)
- POM
- PA (Nylon)
- PP
- CF-PET
- Carbon, Wood, Brass, Bronze, Copper Fillers
- HIPS

If you are using Glow in the Dark or Filler materials (like Carbon, Wood, Brass, etc.) please make sure to use a special nozzle (hardened, ruby or stainless steel) which you can order at Tractus3D.
You will have to investigate which bed temperature will be the best for which material. We can't supply a complete list of all materials available. In this manual you will find a list with our supported materials and print recommendations as a starting guide. You're advised to fine tune those settings to your needs.

Our support desk can't always have an answer on questions about experimental materials!

6.2 Print recommendations

Each material requires different settings for optimal results. If you use Simplify3D with our profiles, the settings are a nice start for fine tuning. For all materials that are supported, you have to use a clean glass plate with adhesion products like 3DLac or glue sticks. The overview below shows the recommended settings per supported (Tractus3D preferred) material.

Bed: **Default as delivered**
Nozzle: **Default 0.4mm brass nozzle**

PLA

- Bed temperature between 55 and 65 degrees Celsius, first layer 65 degrees Celsius after first layer back to 55 degrees Celsius.
- Nozzle temperature 205 degrees Celsius
- Print speed 60mm/s
- Full cooling with material fan after first layers are printed.

PETG

- Bed temperature between 70 and 80 degrees Celsius, first layer 80 degrees Celsius after first layer back to 70 degrees Celsius.
- Nozzle temperature 230 degrees Celsius
- Print speed 80mm/s
- Full cooling with material fan after first layers are printed.

ABS

- Bed temperature between 90 and 100 degrees Celsius, first layer 95 degrees Celsius after first layer back to 90 degrees Celsius.
- Nozzle temperature 250 degrees Celsius
- Print speed 70mm/s
- Max 50% cooling with material fan after first layers are printed.

Facilan C8

- Bed temperature between 35 and 40 degrees Celsius, first layer 40 degrees Celsius after first layer back to 35 degrees Celsius.
- Nozzle temperature 200 degrees Celsius
- Print speed 60mm/s
- Full cooling with material fan after first layers are printed.

PRO1

- Bed temperature between 60 and 65 degrees Celsius, first layer 65 degrees Celsius after first layer back to 60 degrees Celsius.
- Nozzle temperature 225 degrees Celsius
- Print speed 120mm/s
- Full cooling with material fan after first layers are printed.

PEEK

- Bed temperature between 120 and 130 degrees. First layer 130 degrees Celsius after first layer back to 125 degree Celsius.
- Nozzle temperature 390 degrees Cescius
- Print speed 60mm/s
- Max 20% cooling with material fan after first layers are printed (for T650P remove fanduct)

PEI

- Bed temperature between 135 and 150 degrees. First layer 135 degrees Celsius after first layer back to 130 degree Celsius.
- Nozzle temperature 325 degrees Celsius
- Print speed 30mm/s
- Max 20% cooling with material fan after first layers are printed

6.3 Tips for experimental materials

When you are going to use experimental materials always remember to watch the print process as much as possible to change settings while printing.

Tractus3D printers can be adjusted while printing, such as temperature adjustments, fan speed adjustments, speed adjustments and many more options.

Printing with filled materials like Glow in the Dark or wood fill for example will require you to change your nozzle to a hardened or stainless steel nozzle because all fill materials are abrasive and will wear out your default (brass) nozzle really quickly.
When you start printing with experimental materials, we advise you to keep the temperature settings available of the filament manufacturer. But with this in mind we do have some tips:

- Make sure the first layer sticks perfectly to the bed. If needed use adhesion products like glue stick or sprays to fix the print or try a higher/lower bed temperature.

- Watch the model for temperature issues. Most of the time too cold causes extruder problems or separation of layers. Too hot (or not enough cooling with fan) is causing warping at overhangs or the material will degrade in the nozzle which gives a nozzle block after a while.
- Start printing with a low speed and slowly rise the speed. Never try to run before you can walk!

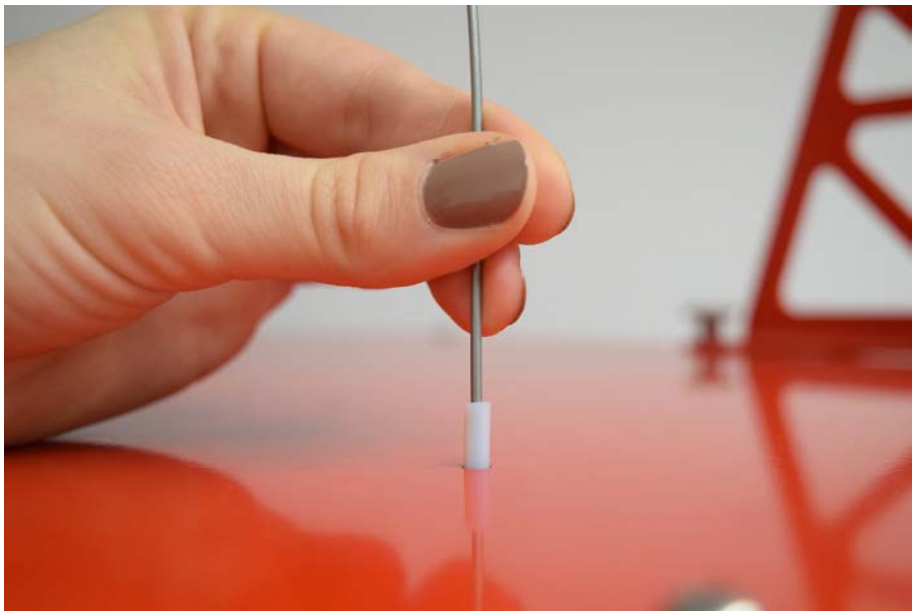
Also remember that every material has its own properties like contraction after cooldown. This means that the dimensional accuracy will be different for every material.

7. Operating the 3D printer

7.1 Load material

You need to load the material into the 3Dprinter, before you can start printing. It is recommended to use the spools of PLA that comes with your 3Dprinter. Take the filament out of the box and place it on the printer like the picture below. Watch the place where the filament wire is located on the spool! (Wire needs to go to printer on top of the spool)

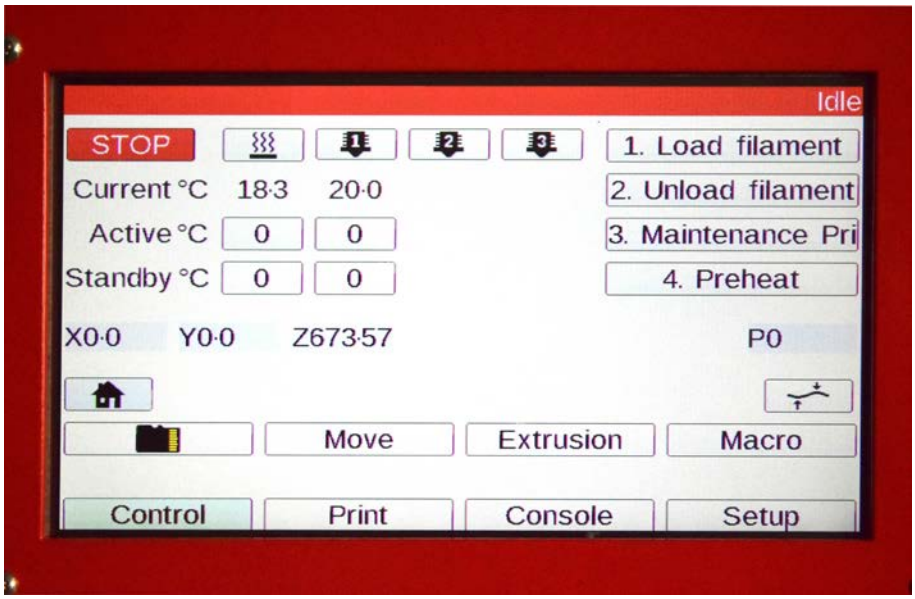
Put the filament wire inside the white tube on top of the printer (see picture below).



Push the black button on top of the printer to release the feeder motor and gently push the filament wire through the feeder unit.

You can only load filament with the commands if the nozzle is on temperature above 170 degrees. This to prevent any cold extrusion damage.

Heat up the nozzle temperature to load the filament. Click on the **nozzle icon** on the LCD screen (it will become **RED** to show you it's powered on). Click on the active selection box below the nozzle to set the temperature at 180 degrees (select temperature by clicking + until desired temperature is reached and click **SET** afterwards). You can also use the web interface to set the nozzle temperature.

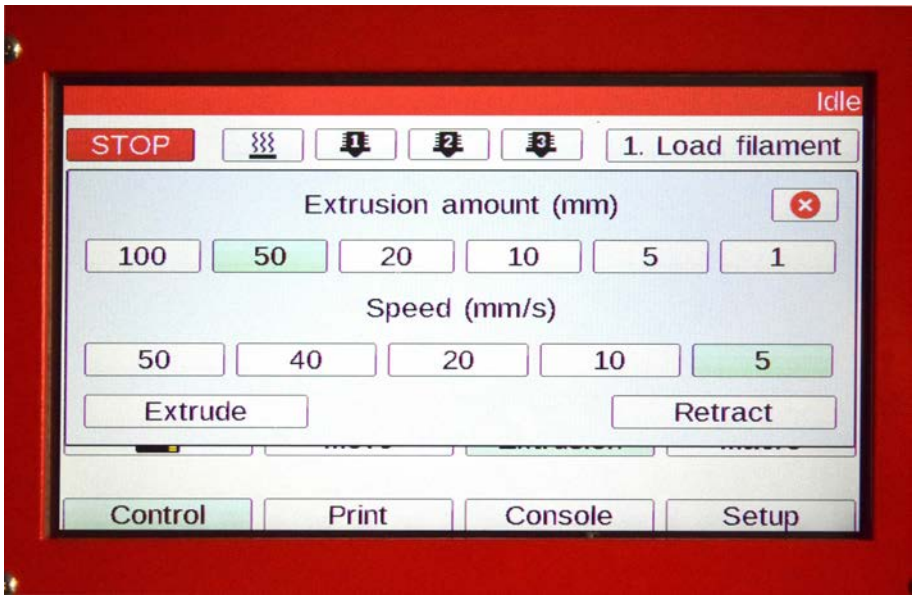


Wait for the nozzle temperature to reach the set temperature.

Click on the button Load Filament on the LCD screen or on your web interface.

Manually extrude a little bit to spool the filament completely through the nozzle. Click on the **Extrude** button and select **50mm** and **5mm/s**.

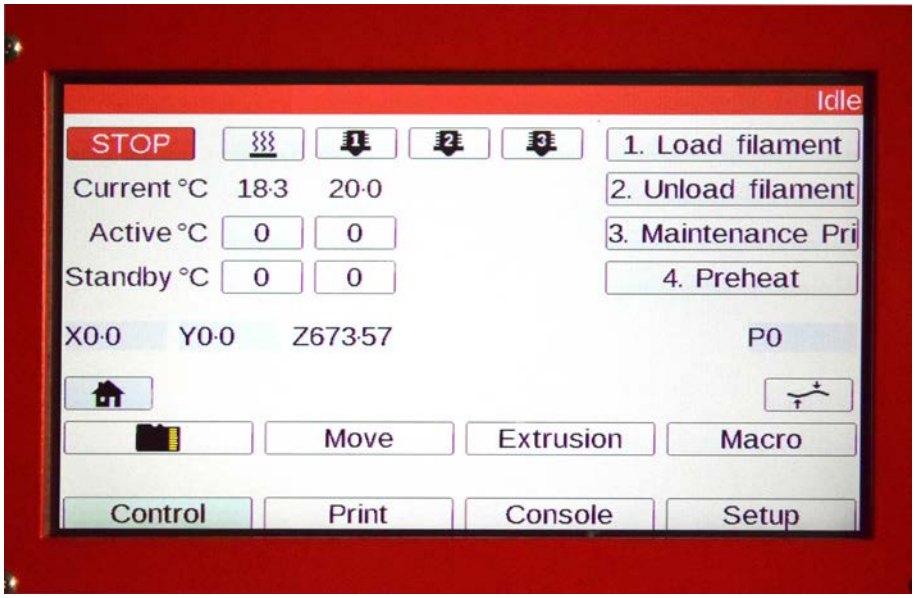
Please don't use more than 5mm/s for manual extrusion!



7.2 Unload material

Heat up the nozzle temperature to the printing temperature of the material which is inside your 3D printer at this moment. Click on the **nozzle icon** on the LCD screen (it will become **RED** to show you it's powered on). Click on the active selection box below the nozzle to set the temperature (select temperature by clicking + until desired temperature is reached and click **SET** afterwards). You can also use the web interface to set the nozzle temperature. **Wait for the nozzle temperature to reach the set temperature.**

Click on the button Unload Filament on the LCD screen or on your web interface or do it manually.

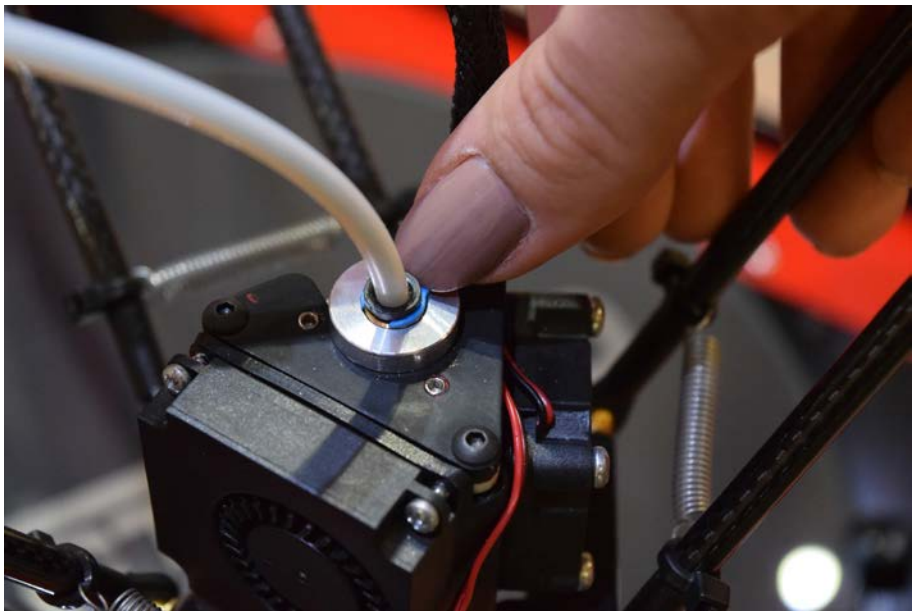


Push the black button on top of the machine to unload the filament by gently pulling the filament wire.

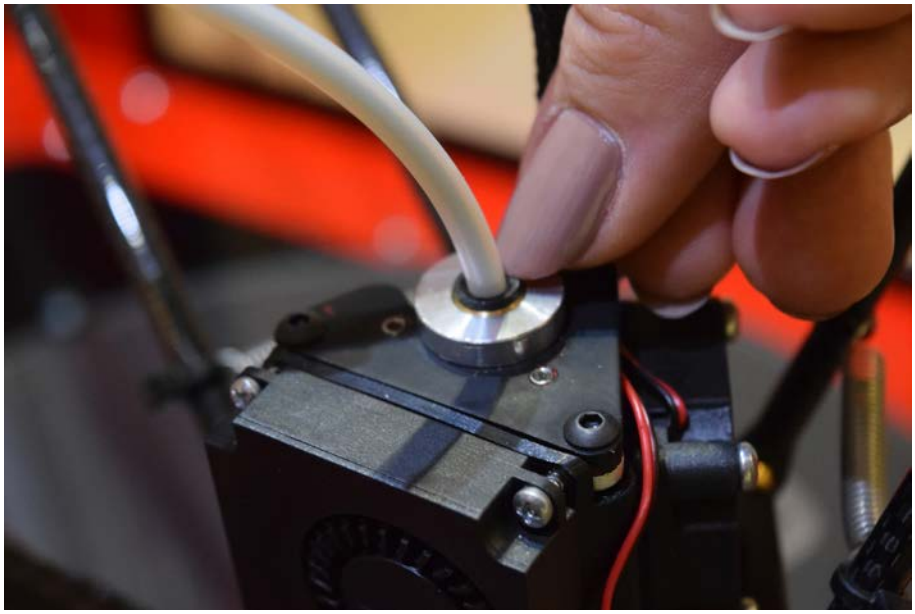
When you feel any obstruction or friction please stop pulling and push the filament back a little. There is a chance you have a blob on your filament.

7.3 Removing the filament with a blob

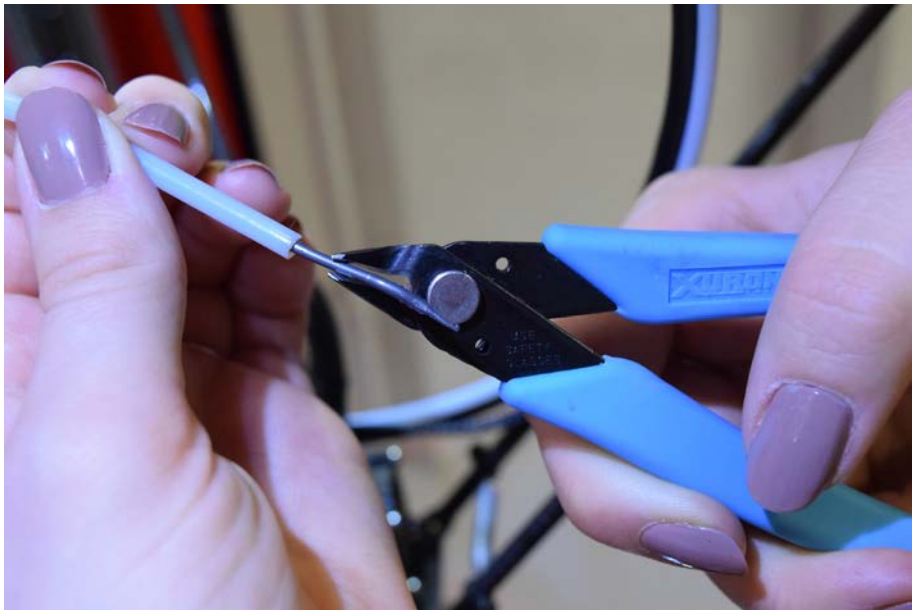
Remove the tube from the feeder by removing the blue clip from the fitting.



Push down the edge of the fitting together with pulling the tube out of the feeder.



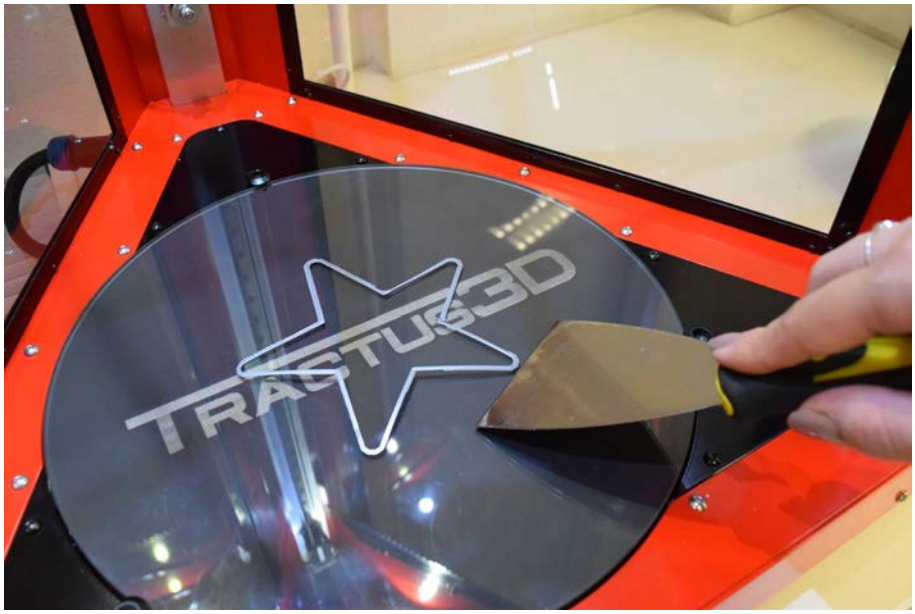
You will see the filament getting out of the feeder with the blob. Get a cutter to cut off the blob of the filament.



Now push back the tube inside the fitting place the blue clip back on the fitting and try to remove the filament again by pushing the black button on top of the printer and pulling on the filament.

7.4 Preparing your print bed

We recommend you clean the plate with a spatula if printing remains are there.



After this, clean the plate with some hand warm water and a towel. In case of some remaining grease, use some Isopropanol. Make sure the bed is completely cleaned and don't touch the plate after cleaning.

We recommend using an adhesion product like spray (3DLac) or hairspray (make sure it's sticky). In some cases, using glue sticks (Elmer's Washable School Glue) will work great too. Make sure you not to use too much of the adhesion product as this will make the adhesion to the bed worse.

7.5 Removing a print from the bed

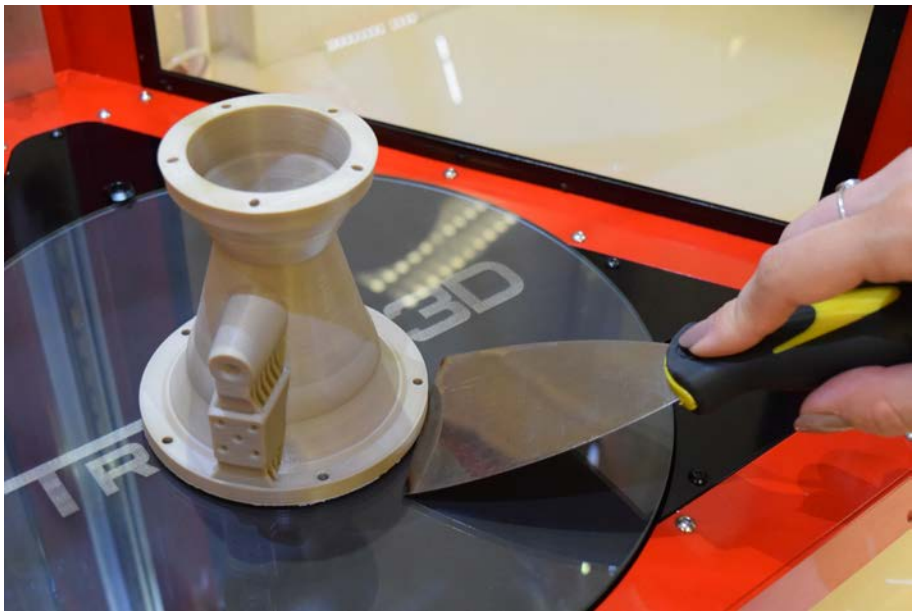
Once your 3D print is finished it must be removed from the build plate.

Always let the print bed cool down to room temperature before removing the print. Otherwise you can damage the bed!

There are several methods to remove the printed object from the bed on room temperature.

- After cooldown is completed, just remove the loosened prints from the bed.

- Use a spatula under the edge of a print, keep the spatula always as flat as possible parallel with the bed to avoid scratches or damage to the bed. Apply a little amount of force to remove the print of the bed.



Still stuck? Try some handwarm water around the model and leave it for a few minutes.

When you print a product with a brim, be aware of the danger of cutting yourself when removing the print from the build plate. A brim can be very sharp!

7.6 Remove support material or brim

Prints which are using (breakaway) support will require post-processing to remove the support structures. You can manually break the support structures from the model you printed. It will require sometimes a tool like pliers or tweezers to break away the support material.

It is advised to wear protective gloves when the support structure contains sharp corners or half broken parts.

After removing most of the support structure, the remaining part(s) of the support structure can be pulled from the model with cutting pliers. Carefully get underneath it and then bend it upwards.

Sometimes a final layer of the support material remains after pulling the support from the model. Try pulling this off with pliers or with tweezers for smaller parts.

8. Maintenance

8.1 Maintenance schedule

To keep your 3D printer in optimal condition we recommend the following maintenance schedule assuming you print 1.500 hours a year.

Every print again
- Check if the print head fan is running after heating up
- Clean the print head and nozzle

Every month
- Clean the 3D printer
- Check all arms
- Clean the slide wheels

Every 3 months
- Check tension and wear of belts
- Check for play on the slides
- Clean the feeder/extruder motor
- Clean the Bowden tube

Every 12 months
- Replace the main Bowden tube

Replace the feeder/extruder gear

If the usage frequency is higher, we recommend performing more frequent maintenance on your printer to ensure optimal printing results.

8.2 Change the print head

The print head can be changed quick on a PRO series 3D printer.

Unload the filament

Make sure you unloaded the filament if possible. (See chapter of unloading filament for help)

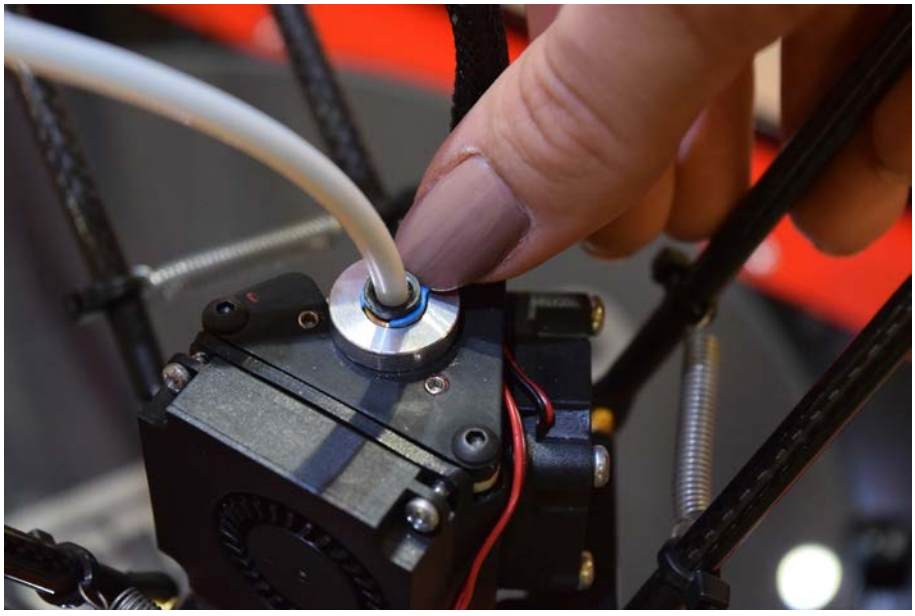
If this is really stuck, continue to next step and cut the remains of filament.

Remove the Bowden tube from the print head

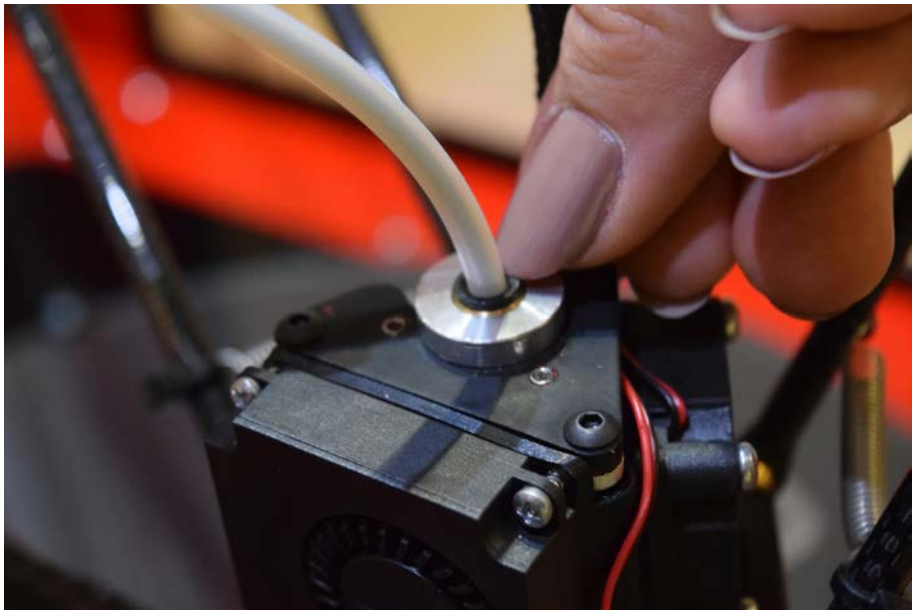
Lower the print head by moving the printer to the maintenance location.

Press Macro > Maintenance

Remove the blue clip (if present) from the print head

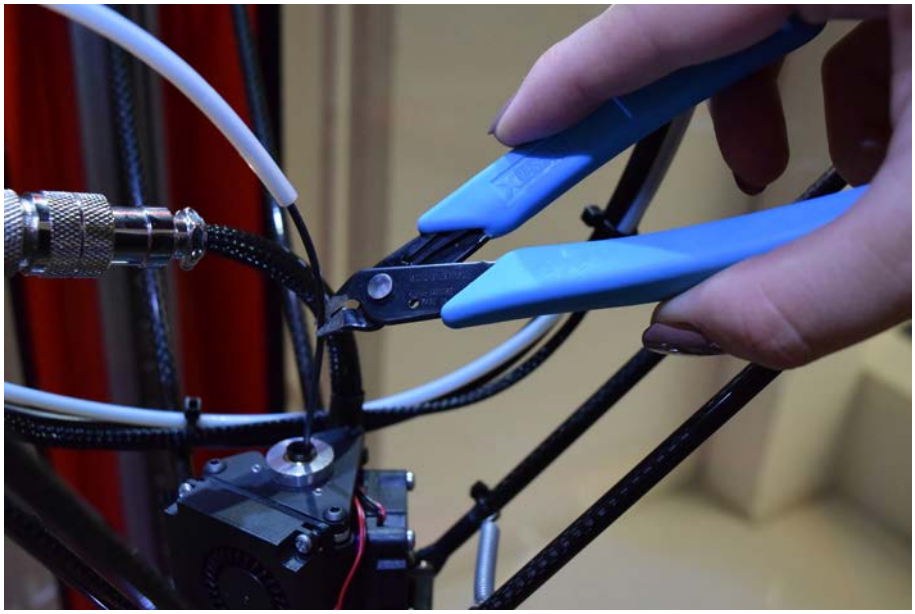


Press the black ring (pneufit) down with your finger and pull the tube upwards away from the print head.



Cutting stuck filament

Sometimes the filament is really stuck inside the print head. In this case you want to change the print head but the filament is holding you for a quick swap.



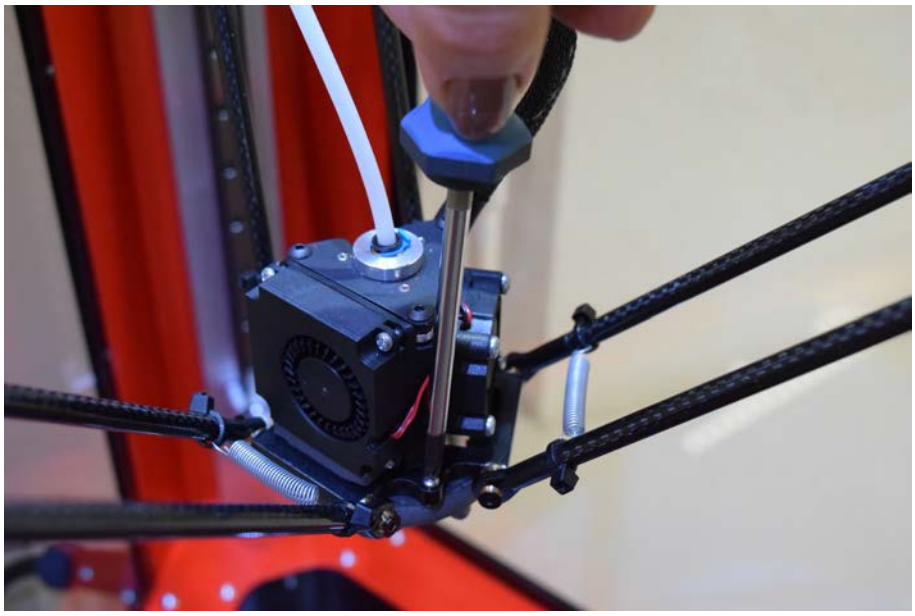
Disconnect the main cable

When the bowden tube is disconnected, unscrew the silver connector of the print head from the main cable

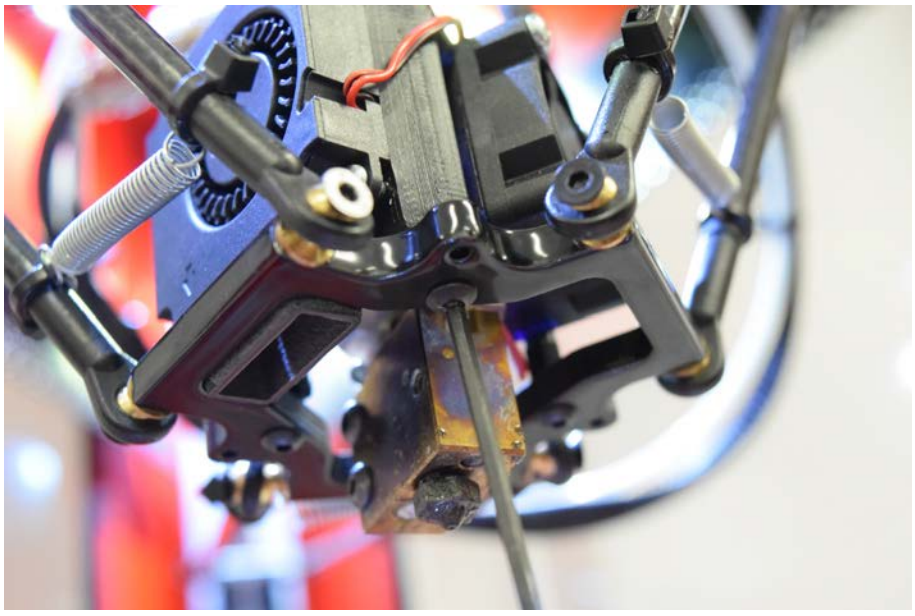
Disconnect the arms from the print head

Now unscrew the small screws between the arms on the hotend unit while holding the fanduct.

Take the fan duct unit off the printhead.

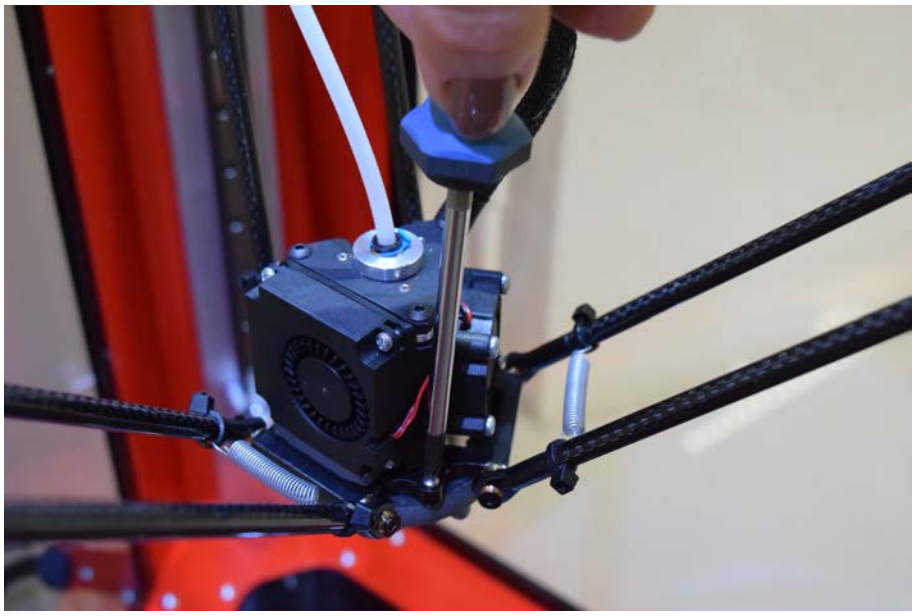


For the T850P unscrew the 3 screws at the bottom of the printhead unit.



Installing new print head

Put back the printhead on top of the 3 holes in the effector (plate) and screw the 3 silver screws back in place.



Secure the main cable to the print head by plugging in the connector and tighten the screw fitting.

Push back in the bowden tube until you can't get in any further. Place the blue clip (if present) back under the black ring of the bowden tube.

Restart the printer by turning it off and on again or pressing the **STOP** button on the LCD screen.

8.3 Change the bed plate

When you change a bed plate please make sure you calibrate the machine after installing a plate.

Turn around (quarter turn) the bed clip with some force (it's spring loaded). Do this for all 3 clips around the bed and remove the glass bed.

When placing a new glass plate back, do it the other way around and carefully turn the clips.

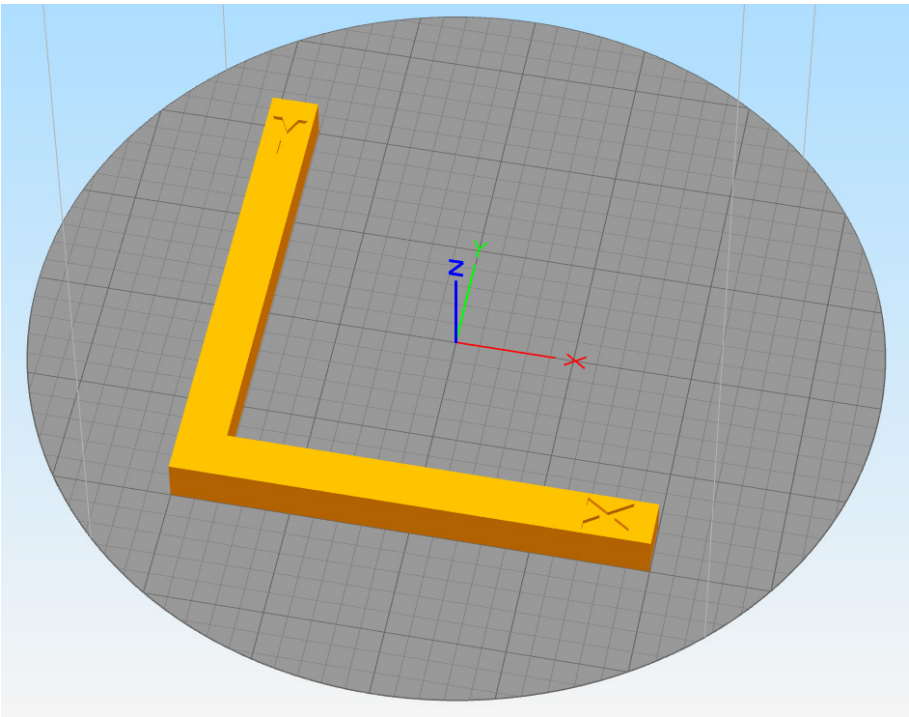
8.4 Dimensional accuracy calibration

When printing objects that really needs a high accuracy on the dimensions you can compensate it by adding a line in the start G-Code of your slicing software. By default, there is a compensation done with PLA in the factory. This setting can be found in the configuration file of the 3D printer.

Every material will need its calibration if you really want to have super high accuracy in your printed dimensions. Some materials will contract more than others.

In the folder Tractus3D on the machine (if not deleted) you will find a file called: **DimensionCalibration.gcode**

Use this file only for PLA on the machine, in combination with a 0.4 nozzle. If you want to calibrate for other materials use the STL file and slice the file to generate a g-code (don't rotate the file on the bed in your slicing software).



If you deleted the folder or need the STL file, please download these from our website or ask our support desk to send you this file.

Print this file with your selected material.

Take a calliper and measure the length of the X and Y part and thickness (Z) of the print and write down the numbers which in this example we call them PrintX, PrintY and PrintZ

The next steps can be a bit theoretical but when you take the time to master this you can hit a high accuracy.

In the console of your machine (LCD or web interface) type the following command:

M579 (press enter or click send)

You'll get the result like this: Axis scale factors: X: 0.997, Y: 1.009, Z: 1.000

Rename this MachineX, MachineY and MachineZ to make the calculation.

If the dimensions are right the PrintX and PrintY side of the print would be 100mm and the PrintZ height would be 10mm. In this example your print as PrintX=99.82mm and PrintY=100.33mm and PrintZ is correct.

Formula for dimension calibration

To calculate the scaling factor, we need to do the following steps:

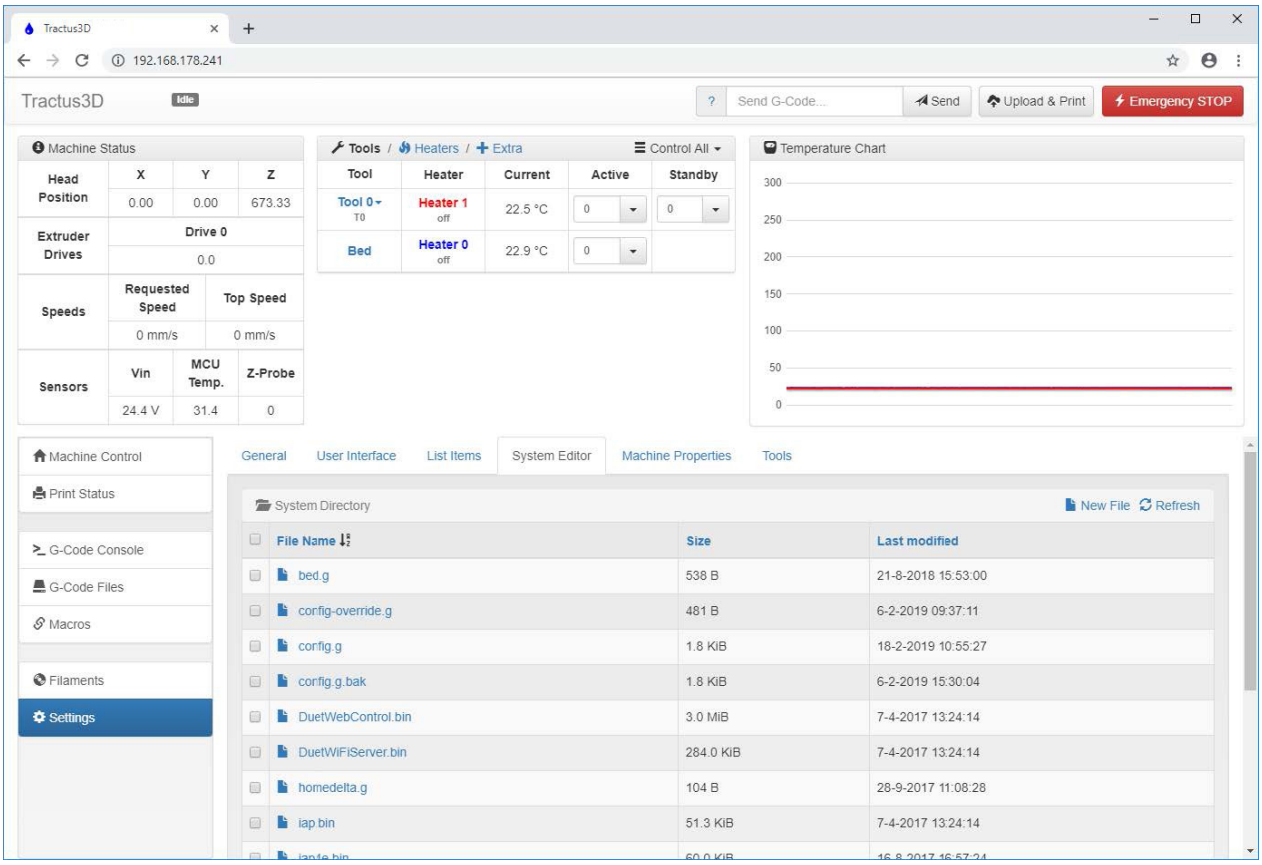
- PrintX / MachineX = RealX (99.82 / 0.997 = 100.12mm)
- PrintY / MachineY = RealY (100.33/1.009 = 99.435mm)

Now we know the real machine values we can calculate the new dimensions:

- 100 / RealX (= 0.9988)
- 100 / RealY (= 1.0057)

This number is our new scale factor, in your slicer would be Scale X to 99.88% and Scale Y to 100.57%

Or find and change this line in your config.g file which you can access through the **Web interface > Settings > System Editor**



Change the M579 X0.997 Y1.009 line to **M579 X0.9988 Y1.0057**

8.5 Material usage and storage

When you have multiple opened spools of filament, you must store them properly.

It may affect its quality and usability, if the filament is not correctly stored.

To preserve the optimal conditions of your filament it is important that you store them:

- Cool (below 30 degrees Celsius) and dry (Humidity below 50%)
- Out of direct sunlight
- In a resealable bag or box

If the filament is exposed to a higher humidity the quality of the filament can be affected.

You can store the material in a resealable bag or box, including some silica gel.

8.6 Clean the 3D printer

For the best print results, it is important to keep the 3D printer clean while using it. It is advised to not use the printer in a room where it can easily be covered with dust and to remove small pieces of material that might be in the printer.

Clean the bed plate

To clean the bed plate please see **Chapter 7.4 Preparing your print bed**

Clean the casing

The transparent part of the casing is made of polycarbonate. The leading cause of unnecessary damage to polycarbonate is the use of improper cleaners or improper cleaning techniques. This is completely avoidable with just a little bit of knowledge. Lucky for you, we're going to tell you exactly how to clean and care for your 3D printer casing.

The first thing you need to know is to never use any kind of cleaners containing ammonia. This is a sure way to cause damage to your polycarbonate and tarnish the look of it. Popular household cleaners such as Windex or 409 will cause damage to polycarbonate sheets and should never be used as a substitute for a proper cleaner. It is also imperative that you never use a dry cloth, or your hand, to wipe clean a piece of polycarbonate. Using a dry cloth to clean your polycarbonate will rub the dirt and dust into your polycarbonate sheet, scratching or causing damage to it. This is the same reason that car washes use damp towels instead of dry ones for drying cars, to avoid unnecessary scratches.

For proper cleaning and care of your polycarbonate sheets, we recommend a soft cloth. The first step to cleaning your polycarbonate is to remove all the dust and dirt from it. In order to do this, you can either blow the debris off the polycarbonate or use water and a soft cloth to float the debris off. After you have removed the dust from your polycarbonate you should clean it with a mild solution of warm water and dish detergent.

Make sure you clean the inside and outside of the casing including transparent parts regularly.

8.7 Check all arms

There is a PTFE layer on the magnets which prevents to much friction between the print head steel balls and the magnets. This layer can wear out and make the friction too high. This will result in bad print results and possible layer shifts.

Take one arm at the time between your fingertips and turn the arm around. If it spins freely it's ok. If you feel it has friction because it stutters when turning around or it's totally not moving at all, please lubricate with PTFE again. You can alternatively also use PTFE grease between the magnet and balls.

8.8 Clean the print head and nozzle

While printing some materials can get stuck to the outside of the nozzle and will degrade. This won't damage your printer, but it can generate fumes and can drip off in your print. It is recommended to keep the nozzle clean in order to achieve the best print results.

Before starting a new print, always check the nozzles. At least once a month, remove the plastic from the outside of the nozzles by taking the following steps:

During this procedure do not touch the nozzles and be careful while cleaning them as they will become hot.

Case of PLA / PETG / ABS

1. Heat up the print head to 200 degrees Celsius
2. Lower the print head by moving the printer to the maintenance location
   Press Macro > Maintenance
3. Wait for the print head to reach this temperature
4. Take a heat resistant towel/cloth and wipe the nozzle block quickly on all 4 sides (don't do this too slowly as your towel/cloth maybe get too hot)
5. Cool down the print head again
6. If there is still a lot of debris on the print head, please contact our support desk

Case of PEEK / PEI

1. Heat up the print head to 340 degrees Celsius
2. Lower the print head by moving the printer to the maintenance location. Press Macro > Maintenance
3. Wait for the print head to reach this temperature
4. Take a brass brush and carefully brush only the lower side of the hotend
   (Never reach the side of the hotend as you can create shortage in the hotend)
5. Cool down the print head again
6. If there is still a lot of debris on the print head, please contact our support desk

8.9 Clean the Bowden tube

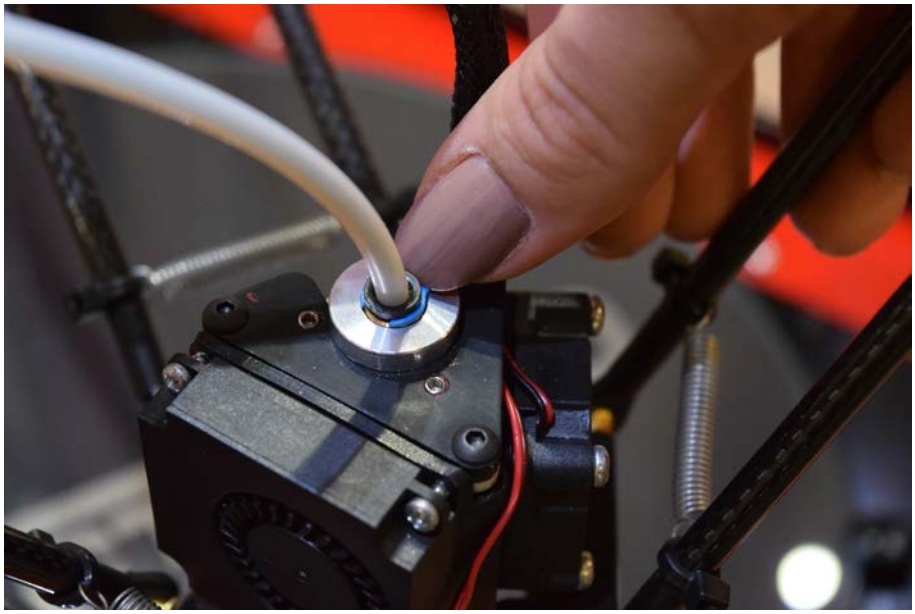
Particles in the Bowden tubes can impede smooth movement of the filament. Clean the Bowden tubes at least once a month, or after experiencing an issue with filament grinding. To clean the Bowden tubes, they must first be removed from the printer.

Unload the filament first (see chapter 7.2 Unload Filament for help)

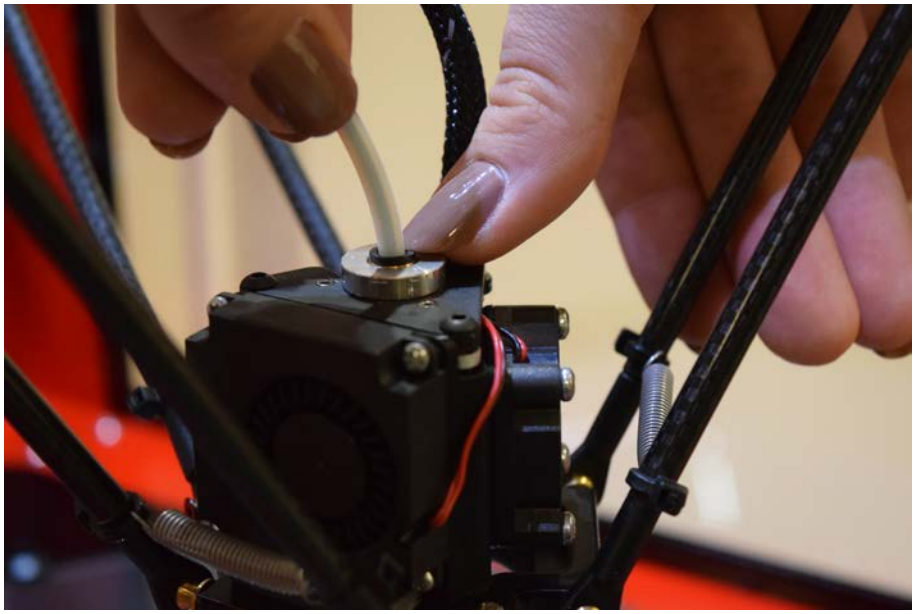
Lower the print head by moving the printer to the maintenance location.
Press Macro > Maintenance

You can alternatively also use PTFE to grease between the magnet and balls.

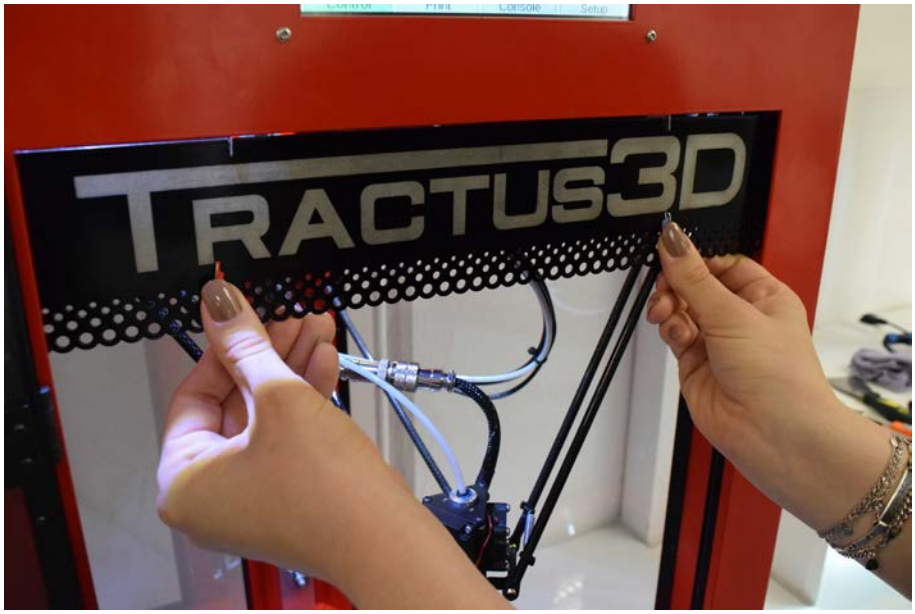
Remove the blue clip (if present) from the print head

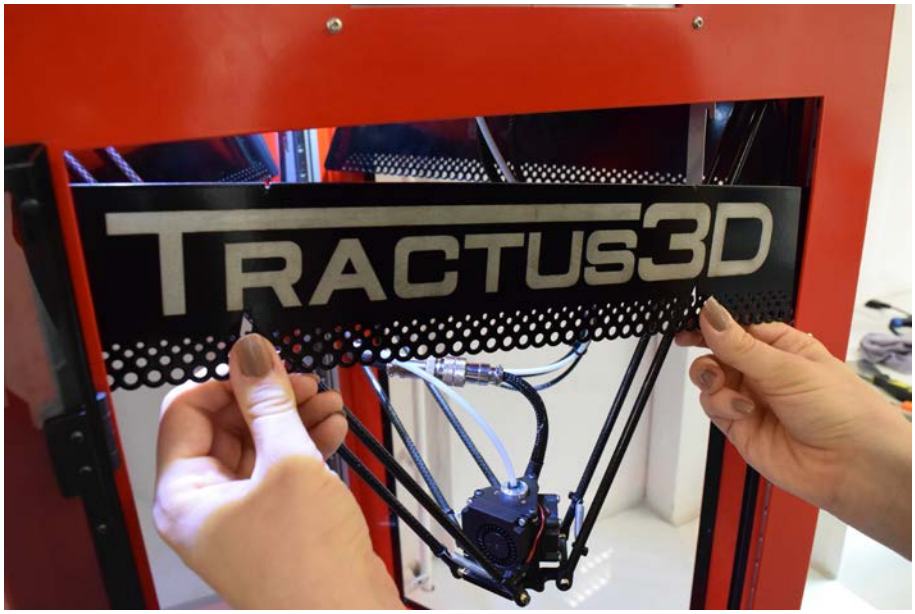


Press the black ring (pneufit) down with your finger and pull the tube upwards away from the print head.



Remove the front air diffuser panel by clicking the clips to the outside.





Press the black ring (pneufit) on the feeder/extruder motor and pull out the tube.

Cut off a small piece of sponge or ball up a piece of tissue.
Insert this into the feeder end of the Bowden tube and push it all the way through the tube with a length of the filament.

Place the tube back in the printer, watch the correct sides of the tube on the correct spots in the printer/print head. (slightly chamfered outside is the print head side of the tube)

8.10 Clean the slide wheels

The linear slide of the printer needs to be clean all time. For an optimal print result clean the linear rails by following these steps:

- Lower the print head by moving the printer to the maintenance location.
  Press Macro > Maintenance

Take a dry cloth/towel to manually clean the linear slide.

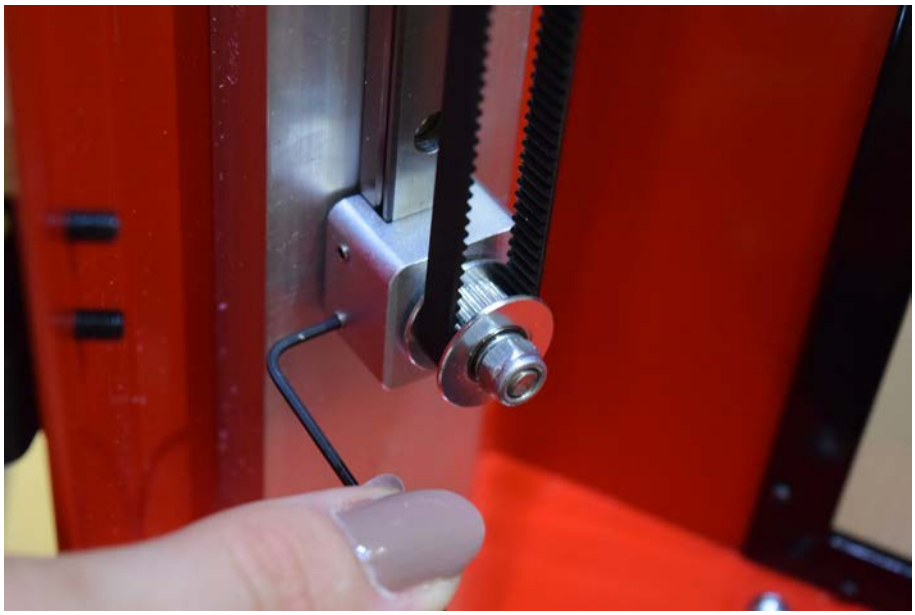
Optional: You can use some thin grease/oil (very lightly) to lubricate the linear rails

8.11 Check tension and wear of belts

Maintaining correct tension on the belts is important to ensure good print quality. The belts transfer the movements of the motors to the slides. If the belts are too loose, print head movement may not be accurate, which can cause print inaccuracies. Over time, the belts may become slack.

It is recommended to check the tension of the belts at least once every three months. Pluck the three belts to check their tension. They should resonate, like a guitar string. Furthermore, the tension of the three belts should be equal. To restore the tension, perform the following steps:

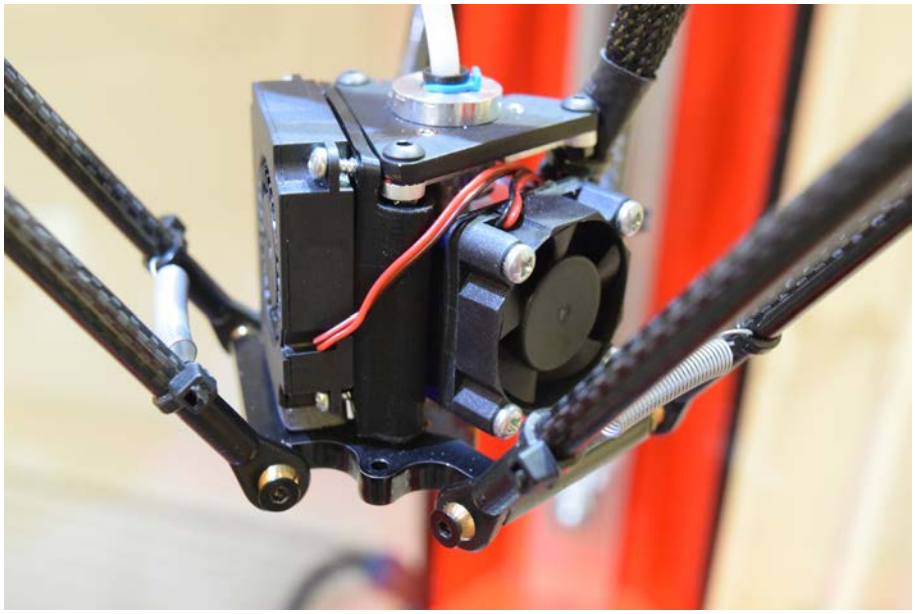
1. Home the printer (make sure the slides are all the way up)
2. Loosen the set screws of the idler pulley with only 2 turns (don't unscrew them completely)



3. Push the idler unit firmly down with your thumb (don't push on the pulley but push on the aluminium block/mount)
4. Tighten the set screws of the idler pulley unit again to fix it.
5. Repeat this for all three belts
6. Make sure all belts are feeling the same in tension

8.12 Check if print head fan is running after heating up

The small fan on the print head cools the print head during a print. This helps to prevent the heat from the nozzle traveling too far upwards. The fan takes in air from the front of the print head and directs it towards the metal cooling ribs of the print head. Sometimes the airflow causes thin strands of filament to be sucked into the fan during a print. If strands accumulate in the fan, they can decrease the effective cooling and eventually obstruct the fan and prevent it from spinning.



If you heat up the print head above 50 degrees Celsius the fan should start spinning. If it's not spinning, please don't heat up the print head anymore and let it cool down. See if there are any obstructions in the fan. If there are any, remove them with some tweezers.

If the fan is still not spinning and there is no obstruction found, the fan should be replaced. Contact our support desk for more information.

8.13 Clean or replace the feeder/extruder motor

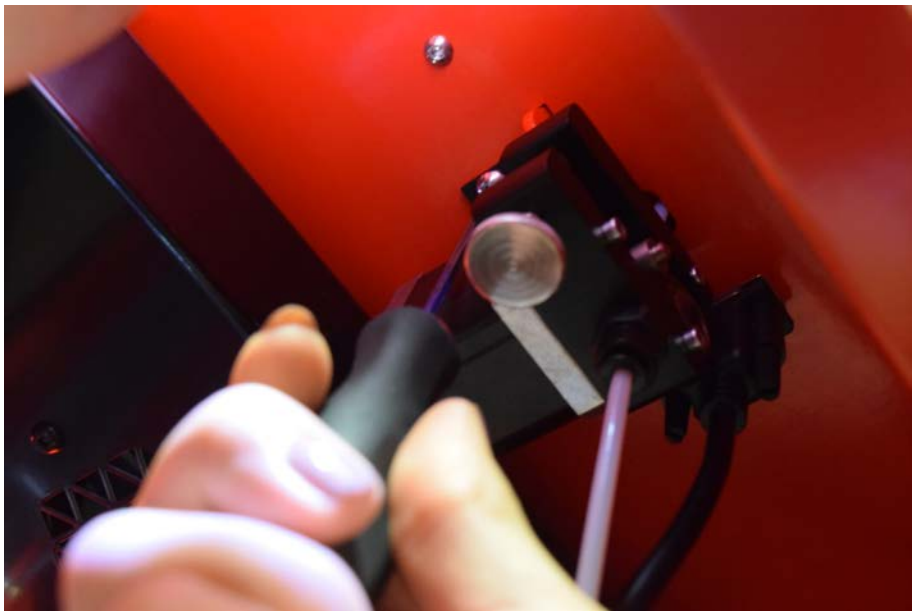
The feeder leads the filament to the print head. To ensure that the exact right amount of filament is extruded, the feeder gear needs to turn turn smoothly.
Small filament particles might remain in the feeder after many hours of printing. We recommended to clean the inside of the feeder after three months of use.
The following steps need to be performed

- Heat up the print head and unload filament **(see chapter 7.2 Unload Filament)**
- After filament is unloaded let the print head cool down. Never turn off the printer with a print head above 50 degrees Celsius!
- Lower the print head by moving the printer to the maintenance location.

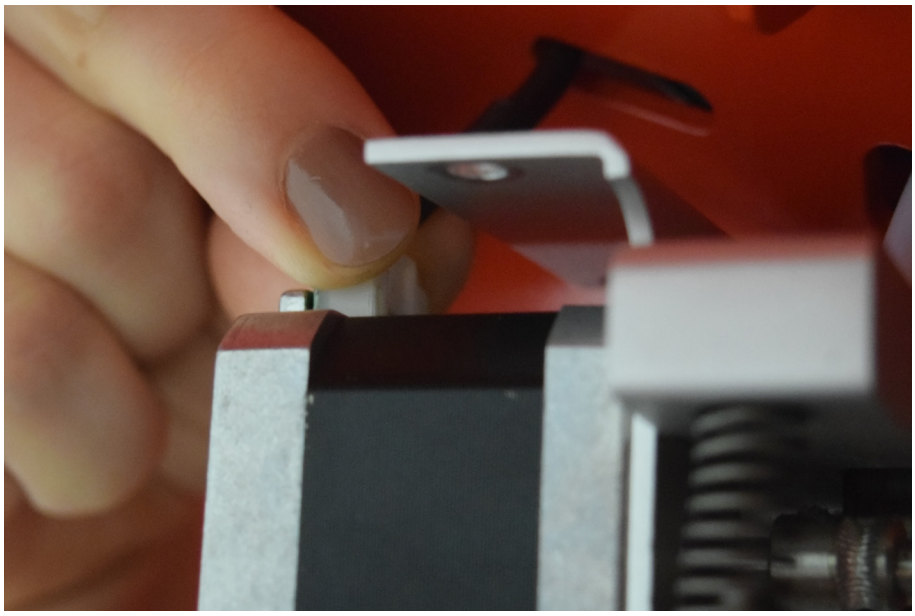
- **Press Macro > Maintenance**
- Turn off the power of the printer with the front switch.
- The feeder (also know as extruder motor) is mounted on the ceiling inside the printer.
  Here you will find a silver/black feeder unit with an extruder motor attached.
- To clean this motor and gear, you need to remove the motor from the ceiling for easy access.
  This can be done with only removing 2 screws and 1 connector.

Please make sure the printer is power off before removing the extruder motor connector!

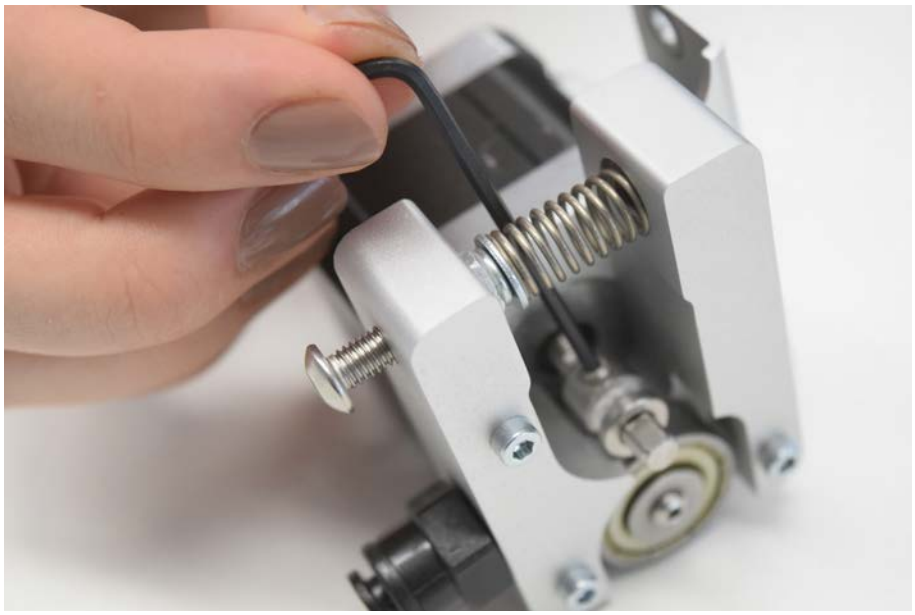
- Unscrew the two hexagon screws to get the unit loose.



Disconnect the white connector gently.



- Now you can clean or replace the gear from the extruder unit. The gear is locked with a set screw. If you replace this gear, make sure the set screw of the new gear is locked with loctite or a similar product and the setscrew is not sticking out of the gear unit.



- To easily clean this gear, you can use a toothbrush or brass brush while slowly spinning the gear with a finger.
- After cleaning or replacing the gear make sure you carefully connect the connector on the motor again. Watch the connector closely as it will fit only in one way. You don't need a lot of pressure to connect the connector.
- Push the small Bowden tube through the hole in the ceiling and roof.
- Screw back the two hexagon screws make sure the extruder is mounted the right way and no cables are between the mount and ceiling.

9. Troubleshooting

A few printer-specific issues may occur when using your 3D printer. You can troubleshoot those yourself using the following information.

9.1 Unclog a print head

Hot Pull

1. Insert the filament (cleaning filament or PLA) into the heated (200 degrees Celsius) print head until you feel some resistance.

2. Hold the filament with pliers and gently apply pressure to the material for ±2 second so that it extrudes from the print head or until it cannot be pushed any further, and directly pull the filament out with a quick, firm pull. Use pliers to prevent injuries to your hands in case the material breaks.

3. Cut off the tip of the filament that you have just pulled out.

Cold Pull

1. Insert the filament (cleaning filament or PLA) into the heated (200 degrees Celsius) print head until you feel some resistance.
2. Hold the filament with pliers and gently apply pressure to extrude some material.

3. Maintain pressure on the filament with the pliers while you switch off the heater of the print head to let it cool down but maintain pressure at all times.

4. When the print head reaches a 90 degrees Celsius pull out the filament with a firm pull.

5. Take a look at the tip of the filament and see if it has a clean, cone-shaped tip like shown on the picture.

If the tip of the filament is not clean, go back to hot pull or cold pull to repeat the cleaning steps.

9.2 Print quality issues

Poor bed plate adhesion

If the adhesion of a print to the build plate is troublesome , the following actions can be undertaken:

- Ensure that the correct material settings and adhesion method were used **(see chapter 6.2)**
- Recalibrate the machine **(see chapter 4.3)**
- Make sure the bed is cleaned correctly **(see chapter 7.4)**
- Use adhesion products, like sprays or glue stick, if necessary for the filament.

Under-extrusion

Under-extrusion is simply when the printer is not able to provide enough filament. You can recognize under extruding when you see very thin layers , missing layers, or layers that have random holes and dots in them. There are several reasons that may cause under-extrusion:

- Low quality filament (inconsistency of the diameter) or using wrong settings
- Incorrect set up of the feeder tension
- Friction in the Bowden tube
- Small filament particles in the feeder or Bowden tube
- A partial obstruction in the print core

If under–extrusion is affecting your 3D printer and mentioned causes do not apply, please contact the support desk.

Warping

When filament shrinkage occurs while printing, the corners of the print will lift and the print will become detached from the bed plate. This is called warping. When printing plastics, the plastic first expand a little, but it contracts during cool down. If the filament contracts too much, it causes the print to bend upwards from the build plate.

When warping occurs, please make sure you have done the following:

- Use the calibration to level the bed
- In case of a glass build plate, apply a thin layer of glue or spray
- Use the right temperature and settings as known for the filament you use
- Modify the shape of your model
- Use a brim
- Choose another material that is less sensitive to warping

For detailed support, please contact us: https://tractus3d.com/support/service/

Phone +31(0)85 130 72 70
Email support@tractus3d.com
Website www.tractus3d.com

Address Bernseweg 14
5324 JW Ammerzoden
The Netherlands

TRACTUS3D