Operating manual
For 3D printers
from 2PrintBeta

Version 1.1
Date: 19/06/13

Table of Contents

1 Important Notes

Please make sure to read and follow all notes in this manual carefully before you use the product.

1.1 Warning notices

Safety notices (Warning/ Caution)

- Any operating conditions, such as use in hazardous areas, use in safety-related systems and applications in which the malfunction may pose a special hazard to life and limbs, is prohibited with this product.

- Always position the device so, that nothing can prevent heat removal, and the device is well ventilated.

- Do not place flammable materials and substances near the device.

- Make sure no metallic items touch any electronic components.

- Use matching plugs and wires for all interfaces.

- Only use components within the given specifications.

- Do not place liquids on or into the device.

- Operate the unit in a clean, dry and dust free environment.

- If the device is contaminated, disconnect the device from the mains (remove the power plug and disconnect from the power supply) and clean it carefully.

- Do not use the device unsupervised.

- This kit is designed for assembly and construction by the user. The finished device can produce high-frequency signals. Operation of the product, depending on how it was assembled, may interfere with or disrupt the operation of other electronic devices. You are hereby expressly advised of this risk and should take the necessary measures to prevent this type of interference. Each completed unit must be tested for compliance with the applicable limits.

- The responsibility for the security of a completed system lies with the builder and operator of the system.

1.2 General precautions

General operating conditions:

- Operate the device only in the completed and audited form with the proposed hardware and sufficient cooling.

- This device is intended for use in a laboratory environment and is sensitive to mechanical and physical interferences.

- Turn off the power supply and remove the power cord before you make changes to the installation and wiring of the device. Failure to do so may damage your electronics and the connected equipment.

- The device may only be used for the intended use within the limits of the electrical specification.

- While operating, the hotend and the heated print bed might be hot. Do not touch the extruder or the heated print bed while the device is switched on.

- The 3D printer has open drive systems. Do not touch the device while operating.

1.3 Important notices

Protection against electrostatic discharges

- The device is equipped with the exposed electronic components that can easily be destroyed by electrostatic discharge. Take the usual precautions to protect the components, while you install, handling or use the device.

- Not suitable for children

- The device is not suitable for toddlers and children. Make sure that all small items are kept safe, and that the device is protected against contact when children may be in the vicinity of the device. Also make sure that your required tools are not in the reach of children or infants.

Disposal and recycling

- The product contains substances which, based on Directive 2002/96/EC waste electrical and electronic equipment, must not be disposed of with household waste. Please dispose this product at a designated waste disposal location. Improper handling of parts whose components consist of potentially dangerous or harmful substances can cause damage to the environment and the exposure of humans and animals. Dispose of this product properly and apply to the effective use of natural resources. Your dealer will certainly help with disposing when buying a new device. Also, you can return the device to an authorized collection site for recycling waste electrical and electronic equipment. For information on waste disposal for recycling, please contact your city office, the public waste authority, or an approved site for the disposal of waste electrical and electronic equipment.

2 Introduction

Congratulations on your purchase of a 3D printer from 2PrintBeta. Please read the relevant instructions carefully before putting the unit into operation. Also make note of the general safety instructions.

This device enables you to make real, 3 dimensional, touchable, plastic objects out of your digital 3D models. There are different plastic types available, for example PLA or ABS and those plastics are available in different colours.

The digital models are thereby decomposed by means of software into thin horizontal layers and for each layer the software calculates a corresponding path for the print head. As soon as the printing process is started, the print head begins to follow the computed tracks and prints a thin layer of plastic to the appropriate places. Over time the printer builds up the model layer by layer.

Your digital 3D models can be created in any CAD program, but need to be converted before printing into the STL (stereolithography) format. Most CAD programs offer those capabilites to do so. After exporting the models you need to ensure that the model was completely exported without holes and is a valid STL file.

A good software for it is the free version of netfabb (www.netfabb.com). With this software you can take a look at the object, repair the exported object and move it to the zero position.

You can also download 3D models from the Internet, a good platform for this is Thingiverse (www.thingiverse.com). Again, you should consider using netfabb to make sure the model is correct and possibly move it to the zero position.

After you have your model available as a STL file, you need to compute the necessary print paths for it. For this you need a software like SFACT (http://reprap.org/wiki/Sfact) or Skeinforge (http://reprap.org/wiki/Skeinforge). Before you install SFACT or Skeinforge, you need to make sure you have the python environment installed which is needed for this programs. SFACT or Skeinforge need to be calibrated and adjusted to your machine before you can use them. After the calibration the software will compute the necessary print paths for your model for you. The output of this will be saved to a GCODE file, which will be used to control your printer. This file is a text file and contains machine commands which you might know from a common CNC machine.

To control you 3D printer and execute a generated GCODE file, you also need a corresponding control software. We recommend Printrun (http://reprap.org/wiki/Printrun) for this. This software has a nice GUI to move your print head in all three dimensions, control the extruder and of course allows you to execute a generated GCODE file.

3 Commissioning

3.1 Inspection and Setup

After you have set up the 3D printer using the assembly instructions, you need to configure and set up the printer and the firmware.

Before you connect the printer to mains and switch it on, check the following points again.

- All axes are well lubricated, freely moveable and do not jam.

- If you push the axes completely to one side, the end stop flag moves easily into the end stop.

- All cables are connected with the correct polarity and have a tight and secure fit.

- Check that the polarity of the connected power supply matches with the descriptions on the electronics.

The last two points should be checked by an expert.

When everything is correct, you can connect your 3D printer with mains, connect the USB cable with your PC, and switch it on.

As a first step you need to flash the firmware onto the electronics and configure it. Copy the Arduino development environment, the Sprinter firmware and the Printrun control software from the USB stick to your PC.
You can also download and unzip the current Arduino development environment from http://www.arduino.cc, the newest sprinter firmware from http://reprap.org/wiki/Sprinter and the Printrun control software from http://reprap.org/wiki/Printrun.
Start the Arduino environment and open the "Sprinter.ino" file with it. It should look like the following picture.
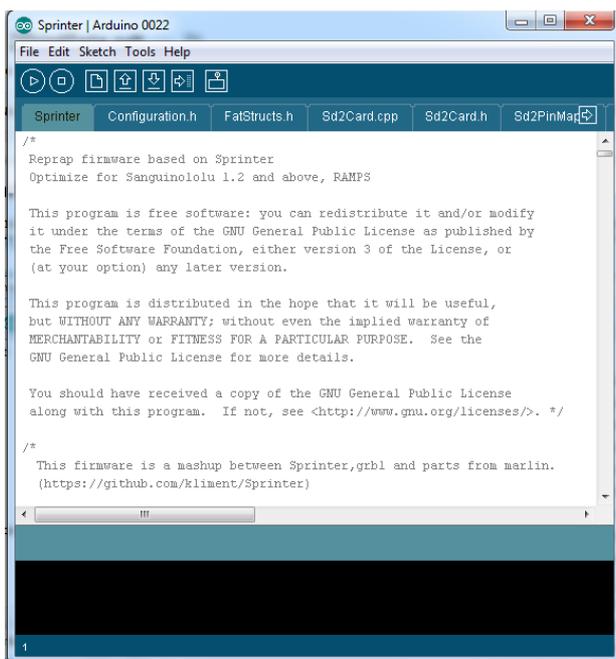
In the Arduino environment select the entry "Arduino Mega (Atmega 1280)" or in older versions "Arduino Mega 2560" from the Tools->Board menu and choose the correct COM port from the Tools->Serial Port menu.

You can now flash the supplied firmware with the Upload button (arrow to the right) onto the electronic. If errors occur, check if you selected the correct board and COM port.

If you don't use the supplied firmware version from the USB stick, you need to adjust a few variables in the firmware. To do this, change to to "Configuration.h" tab where you can configure you firmware. For our electronics, you should set the "MOTHERBOARD" entry to 33 and all "ENDSTOP_INVERT" entries to true.
Then you can flash the firmware for a first test to the electronic as described above.

After the first upload, you should start the previous mentioned Printrun software (http://reprap.org/wiki/Printrun) and carefully move your 3D printer for the first time. To do this, unpack the Printrun software and start the file "pronterface.py". You might have to install python (http://www.python.org/getit/) first, which you can also find on our USB stick. Also you might need wxPython, pySerial and pyReadline, which are all included on our USB stick.

After you started the software, you should see the following window:
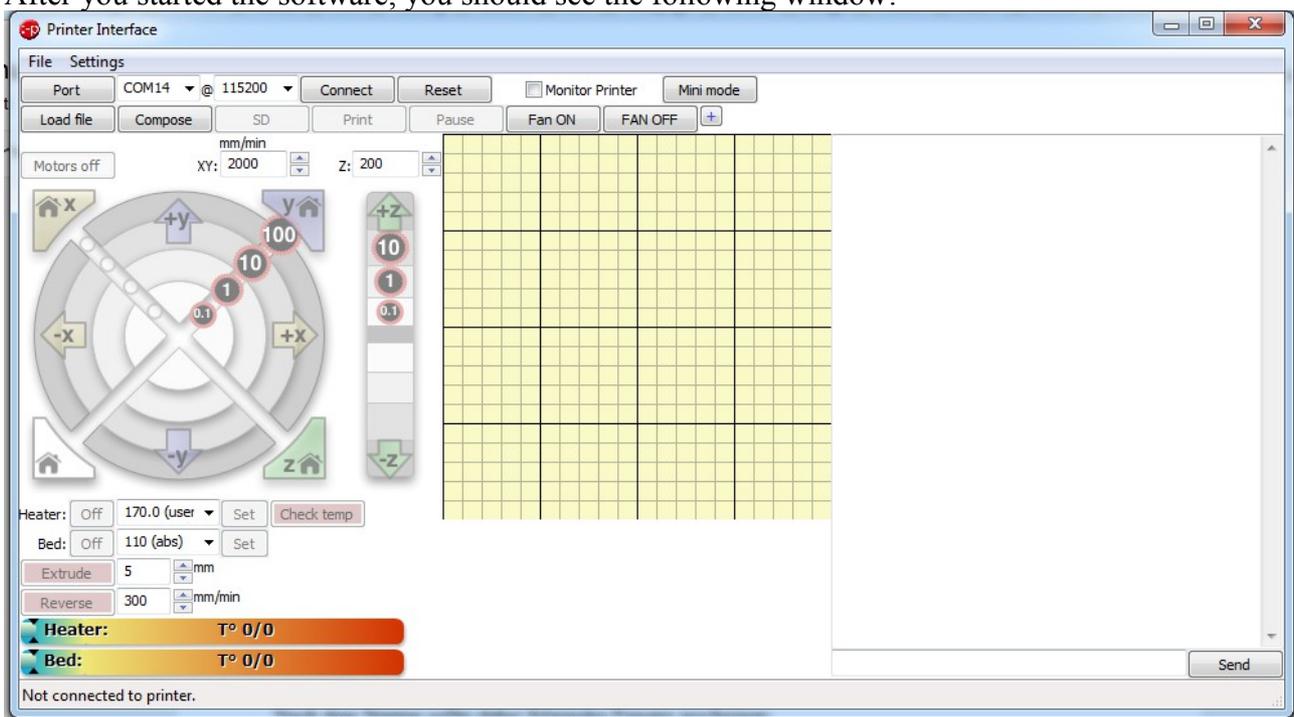
Select the correct COM port and press connect. If everything goes right, you should see some text in the right window, which shows that the software could connect correctly to your 3D printer.

With the controls on the left side you can now carefully move your 3D printer. Only use small steps in the beginning and prepare yourself to switch off your 3D printer, if anything goes wrong or the printer drives into the limits.

Now check if the motor directions are correct, which means to check if the print head really moves to, for example, the X+ direction if you command it so.
Should any axis not move in the correct directions, or if any axis is switched with another check the cabling again. Do not forget to disconnect all power connections before doing any modifications.

Should one axis still move into the wrong direction, you can correct that with the "INVERT_X_DIR", "INVERT_Y_DIR" and "INVERT_Z_DIR" entries in the firmware.
When you want to flash a newly corrected firmware, don't forget to first click the "Disconnect" button in the Printrun software.

Once all three main axes move correctly, the end stops should be checked. For this, move carefully and slow into the corresponding end stops and check with the leds that the end stops really trigger. Also check if the end stops really block movements or if you can move further.

Make sure to not hit the print bed with the extruder while doing this.

Now check if the extruder heats correctly, by entering a sensible temperature into the "Heater" input and pressing "Set". You can then check the current temperature by clicking "Check Temp" or by enabling the "Monitor Printer" function. Check if the extruder gets to 180°C without any problems. If it doesn't, check the cabling. Let the extruder run a little bit without inserted plastic and check in which direction the extruder drive screw turns. It should turn so, that it would pull in an inserted plastic filament downwards. If the extruder turns into the wrong direction, you can correct that by changing the "INVERT_E_DIR" entry in the "Configuration.h" file.
Switch the extruder off after this tests.

If you have a heated print bed installed in your 3D printer, you should also make the same checks as for the extruder. For this press the "Set" button beneath the "Bed" label and check the temperature with the "Check Temp" button or the "Monitor Printer" function. Good temperature values for PLA are around 60°C and 110°C for ABS. If the temperatures doesn't rise, check the cabling.

3.2 Calibration

First you should adjust the print bed with the 4 screw so, that it is completely level and the distance between the print bed and the extruder stays the same over the complete print area.

Make sure you don't crash the nozzle into the print bed while doing this. The Z axis end stop should be adjusted until it triggers when the nozzle is very near the printbed. If you plan to cover the print bed with tape for a better adhesion and removability, you should adjust the distance between the print bed and the nozzle after covering the print bed.

For this adjust the print bed with the M3 nyloc nuts on the bottom until it's level. Move the print bed and extruder to one side, so the nozzle is in one corner of the print bed. Now adjust the Z end stop until you can just barely put a paper between the print bed and the nozzle. Lift the Z axis a bit and move the Y axis to the other side. Now adjust the height of this print bed corner until you again can barely put a paper between the print bed and the nozzle when the Z end stop triggers. Repeat this steps for the other two corners of the print bed.

Now you need to check the steps per millimetre in the firmware and possibly adjust it. This is needed so the print head moves exactly as far as commanded by software. The supplied firmware on the USB stick should need no corrections.

For this, move the print head to one side of the print bed and mark its current position. Then command the print head to move for several centimetres in one axis and compare the commanded distance with the real distance the print head moved.
If the distance is not exactly the same, you can correct that with the "_AXIS_STEP_PER_UNIT" entries in the firmware and upload the corrected firmware. Repeat this steps until all main axes move the correct distance.

After the main axes are calibrated, you need to adjust the extruder. Heat up the extruder with the Printrun software. Good values for PLA are 180°C and 220°C for ABS.
After the extruder has reached the desired temperature, insert your plastic filament and extrude with the "Extrude" button until some plastic comes out of the nozzle.

When the Extruder correctly extrudes some plastic, you should mark a 3 cm piece of filament on the top of the extruder. Then check if with the "Extrude" button and the corresponding length inputs if the extruder really consumes 3cm of plastic when you command it to extrude 3cm. If the lengths are not correct, you can change it with the "_AXIS_STEP_PER_UNIT" entries in the "Configuration.h" file.

After you finished this steps, your 3D printer is ready for the first test prints.

4 Operation

4.1 Filament changing

Inserting filament:

Switch the device off.
Open the M4 x 50 mm screws, which press the Wades Idler onto the Extruder, until the springs can be moved by hand. Take one end of the filament and push it into the extruder to the drive screw. Make sure the filament is pretty straight and cut off nicely. Move the filament deeper into the extruder by moving the springs and turning the big gear. Push the filament into the extruder until you feel a bigger resistance. Now fasten the M4 x 50 mm screws again, until the filament is clamped securely.

Make sure to clean the nozzle before the change from one plastic type to another.

Switch the printer on and start the Printrun control software. Dial in the desired temperatures (180°C for PLA and 220°C for ABS) and wait until the temperature is reached.

Let the extruder extrude a few millimetres and check if the drive screw really grips the filament. If needed fasten the M4 x 50 mm screw until the drives screw doesn't slip again. Now let it extrude a few centimetres and check if the filament flows steadily out of the nozzle. If needed fasten the screws more or change the temperature a few degrees up or down.

Filament changing:

Switch the printer on and start the Printrun control software. Dial in the needed temperature for the current filament (180°C for PLA and 220°C for ABS) and wait until the temperature is reached.

Switch the device off. For the following steps be careful, the nozzle is hot. Use gloves to prevent burns. Open the M4 x 50 mm screws, which press the Wades Idler onto the extruder, until the springs can be moved by hand. Carefully pull on the filament on top of the extruder. If the filament isn't moveable by hand, fasten the screws again, switch on the printer and reheat the nozzle to the target temperature. When the temperature is reached, let the extruder turn backwards fast, until the end of the filament comes out of the extruder.

Carefully remove any leftover plastic resides on the nozzle (For example with a tooth pick or similar). And then continue as described under "Inserting filament".

4.2 Converting digital 3D models into GCODE

Before a digital 3D model can be printed on your 3D printer, the appropriate model must be converted to the required printing trajectories for the 3D printer.

For this we recommend the previously mentioned SFACT (http://reprap.org/wiki/Sfact) software. Unzip this software to your pc and start the "sfact.py" software. You might have to install python (http://www.python.org/getit/) first, which you can also find on our USB stick. The following window should pop if everything has gone correctly.
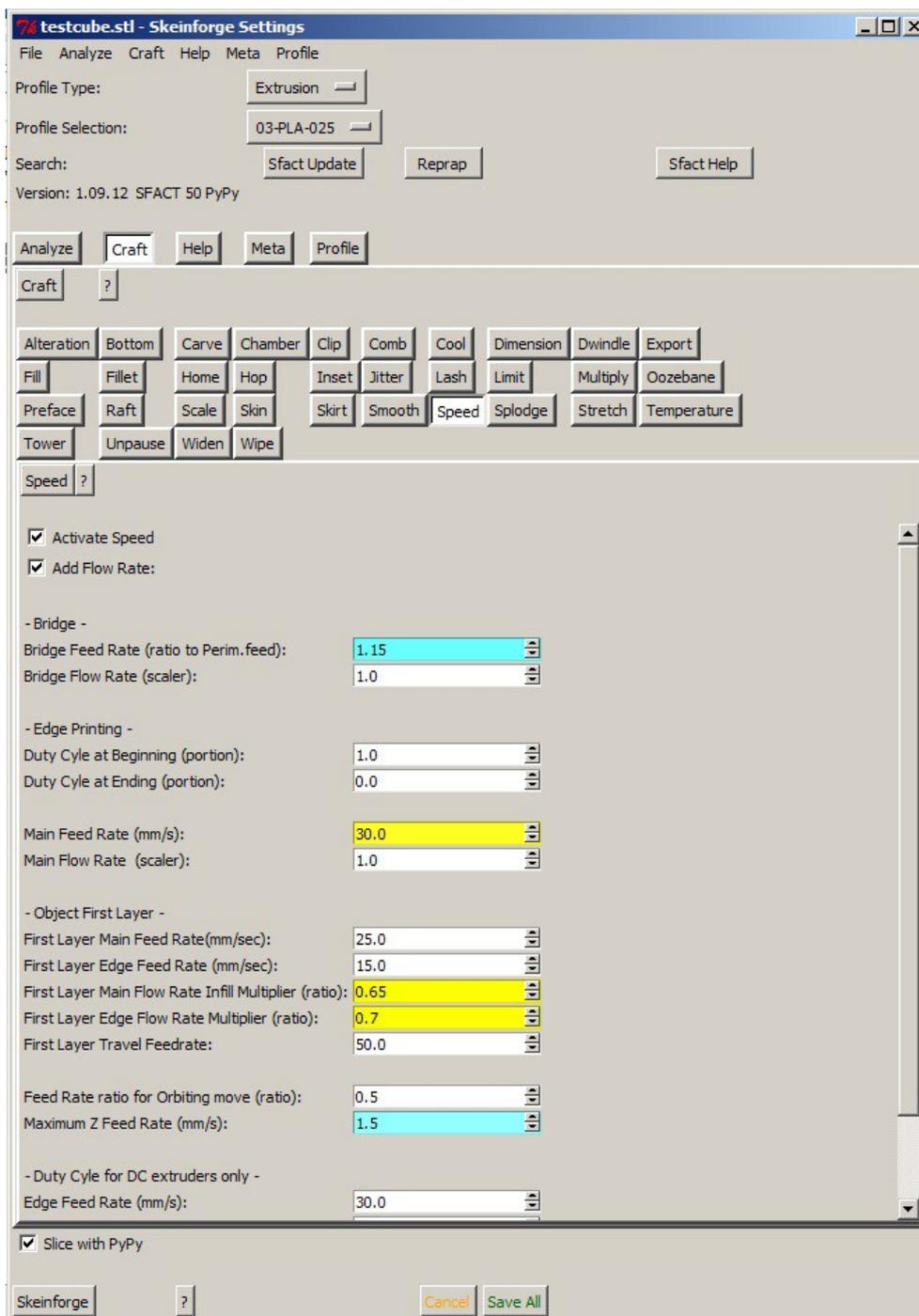
This software offers a variety of settings with which you can influence how your object is printed. You can control how high the resolution of the print should be (layer height), which fill strategie to use, which temperatures are used for printing and much more.

The most important settings for the beginning are:

- "Layer Height" under CARVE, this control the printed resolution in Z direction. For the beginning values between 0.4 and 0.25mm are recommended.
- "Altitude" under BOTTOM, this controls the distance between the zero position of the z-axis and the first layer. This value should be set, so that the first layer is pressed a bit into the print bed but not too much. If the value is too high, the first layer wont stick to the print bed.
- "Filament Diameter" under DIMENSION, this value should be adjusted to the diameter of your filament.
- "Infill Solidity" under FILL. This controls how much percent of your object is filled. For most object a value of 0.2 which means 20% infill is a good value.
- "Temperature". Here you can control the used temperatures for the different areas of the print. It's also possible to deactivate this completely and instead control the print temperature with the Printrun software.

After you have adjusted all settings you can convert a STL file with the "craft" button. Depending on object size, this can take a while.

4.3 Printing a prepared GCODE file

Before printing an object, you should have prepared a corresponding GCODE file like it is described in the previous chapter.

Open the Printrun software and connect it to your printer with the "Connect" button. Move all axes into their end stops and heat the extruder and, if available, the heated bed to the desired temperature.

When everything is prepared, you can load your prepared GCODE file with the "Load File" button and start the print with the "Print" button.

Especially at the beginning, make sure the printer starts at the desired location. If anything seems to go wrong, disconnect the printer from mains to prevent the printer from damage.

If the settings from the previous chapter are good, and the plastic of the first layer sticks to the print bed, you can now watch you printer create you object layer by layer out of plastic.

5 Maintenance

Generally before any change you should make sure that the power of the printer is completely shut off and no danger can emerge from this.

Also mind that after shutting down the extruder nozzle may be still hot for a while.

Wear safety equipment during maintenance.

5.1 Maintenance of the axes

The axes of the 3D printer should be lubricated in frequent intervals. Use technical lubricant which does not corrode the plastic parts.

When lubricating the axes, pay attention that enough lubricant gets into the bearings and that it is distributed evenly over the axes. Especially the driving rod of the z-axis needs thoroughly lubrication at regular intervals. After lubricating, repeatedly move all axes over their full range to ensure the lubricant gets distributed evenly.

Recommended lubrication intervals are between one and two months, depending on the use of the 3D printer.

5.2 Maintenance of the extruder

The extruder should be cleaned in regular intervals. External contaminations on the hot-end can be easily removed by heating the extruder to around 100°C and using a suitable tool to pull off the contaminations. Wear gloves while doing this.

After around one year of use, the PTFE tube inside the hot-end should be replaced. For this you need to unmount and disassemble the extruder. Make sure the hot-end is cold when doing this and any remaining filament has been previously removed.

6 Transport

Please take care of the dimensions and weight while transporting the device.

Weight: 10.5 kg

Dimensions (X x Y x Z): 470 mm x 445 mm x 395 mm

Before transporting, secure all axes with tape against unwanted movements. Use a sturdy cardboard box with adequate padding. Make sure that the electronics are protected against unwanted contact with other parts.

7 Troubleshooting

Problem:
The extruder turns in the right direction, but no plastic is extruded out of the nozzle.
Resolution:
Check the screws which press the filament against the drive screw. If they are too loose, the drive screw will slip. If they are fastened too much, the motor will skip or carve a hole into the filament. Also check that the drive gears are secured correctly on their axes.

Problem:
The filament can be pushed a few centimetres into the nozzle, but then blocks.
Resolution:
The opening in the nozzle is exactly 3.2 mm. The filament is often slightly curved because it has been rolled up which causes this blockage. Sharpening the beginning of the filament helps to fit the filament into the opening of the nozzle, even when its curved a litte bit.

Problem:
After a few minutes the nozzle blocks completely or the filament gets extruded irregularly.
Resolution:
The filament melts in the heat sink and gets compressed by this. Use a fan blowing onto the heat sink to prevent this.

Also the tension of the springs which press the filament against the drive screw might be the cause for this. Make sure the screws with the springs aren't fastened too much or too less.

Problem:
The printed layer are slipping or not exactly over each other.
Resolution:
Several causes are possible for this. First, the belt tension might be too high, so that the X axis is pressed against the M8 drive rods. This causes a wave like pattern on the surface of the printed objects. Reduce the belt tension to prevent this pattern. If the belt tension is too low, the carriage might skip steps and cause slipped layers.

Another cause could be a not well enough secured gear. When the gears are not well enough secured on the motor, they can move around and cause slipped layer. To prevent this, we recommend to file a flat onto the motor shafts. The screw in the gear will then grip perfectly onto this flat and prevent any accidental movement.

Problem:
The printed plastic does not stick onto the print surface.

Resolution:

One cause could be, that the plastic isn't pressed hard enough onto the print surface. To counter this, reduce the distance between the print head and the print surface.

Another cause might be the print surface itself. PLA sticks well onto NOPI tape, on warm (50°C) and cold print surfaces. ABS sticks too well on NOPI tape, here we recommend Kapton tape on a, at least 100°C war print surface.

We also recommend to apply UHU Photostick onto the print surface, this cause the plastic to stick very well, and the glue can be washed away later with water.

Problem:
The motors only jerk and don't turn steadily.
Resolution:

The stepper drivers are not adjusted correctly. Adjust the potentiometer on the stepper drivers until the motors turn evenly and steadily. If you turn clockwise, the power to the motors will be reduces. Make sure to not cause any short circuits when using a screwdriver on those potentiometers.

If the motors only squeak or bleep, you might be trying to drive too fast. Reduce the travel and printspeed in SFACT.

Problem:
Not plastic is extruded from the nozzle.
Resolution:

Check the temperature and the settings for the NTC. Maybe the firmware uses the wrong thermistor table for this NTC.

Also check if you are using the correct temperature for the selected print material. PLA needs around 190°C, soft PLA atleast 200°C and ABS between 220°C and 230°C.

If the filament has a notch on it after you remove it from the extruder, either the nozzle is too cold, blocked or not assembled correctly.

Check if you can extrude filament manually by opening the clamp on the extruder and pushing in the filament by hand.

If this doesn't work, you need to disassemble the hotend and clean it. When assembling it again, make sure the PTFE inliner is secured correctly and can not move up and down.

Problem:
The endstop don't activate or the axis drive into the endstops but don't stop.
Resolution:

Check if the endstops are connected correctly. The LEDs on the endstops should light up if you put something into the endstop.

Adjust the firmware settings for the endstops to inverted, or change the motor direction in the

configuration.h of the firmware.

Problem:
The Z axis jams, or nuts in the X-ends causes the Z axis to not point straight up.
Resolution:
Widen the hole for the Z axis a little bit with a file or a drill. But be careful as the wall is pretty thin.

Problem:
The x carriage jams or drives irregularly.
Resolution:

Loosen the cable ties a bit and check if the axis runs better. If that doesn't help, try to only use 3 linear bearings. You can also try to loose the clamps for the rods a bit.

Problem:
The Y-carriage jams or drives irregularly.
Resolution:

Loosen the cable ties a bit and check if the axis runs better. If that doesn't help, try to use only 3 linear bearings on the carriage.

Problem:
The Z zero-point calibration is not always on the same height. I.e. the nozzle has a different height over the print bed on every homing run.
Resolution:

Fix the M8 nut for the Z Axis with some glue to the X-Ends and maybe add another M8 nut from the top with a slightly tensioned spring.