

www.massportal.com

Portal Delta Pro 3D printer

User's Manual - v1.0



Welcome to the world of 3D printing!

Thank you for your purchase of the Mass Portal 3D printer – we hope that it will serve you well and even exceed your expectations. Mass Portal strives to deliver quality products, which will ease your work and make your life even more interesting. Please take your time to study this guide thoroughly to enable the most flawless experience with your new 3D printer. We can assure you that it will make your setup and work much more easy. Have fun!

Please note that the pictures in this manual may slightly differ from the product or software you have, because we are constantly improving and updating our products.

Table of contents

1. Introduction	3
1.1. User manual	3
1.2. Warranty	3
1.3. Safety	3
1.4. Protection	3
2. General description of the printer	4
2.1. Visual	4
2.2. Main characteristics	5
2.3. Package contents	5
2.4. Other optionally needed but not included tools	5
3. Usage	6
3.1. Setup	6
3.2. Configuration	6
3.2.1. Driver	6
3.2.2. Repetier-Host software	7
3.2.3. Windows	9
3.3. Insertion and retraction of plastic filament	10
3.4. Calibration	11
3.5. Repetier-Host usage	12
3.5.1. Object placement	12
3.5.2. Object slicer	13
3.5.2.1. Slic3r slicer	13
3.5.3. G-Code Editor	22
3.5.4. Manual Control	23
3.5.5. Main menu	24
3.6. LCD control panel	26
3.7. Maintenance	27

1. Introduction

1.1. User manual

This user manual includes such chapters as: Introduction, General description, Usage and Printing of 3D objects. Please study this manual carefully before connecting, starting and using your printer and use your device according to these instructions. Save this manual for future reference and use whenever needed. Whenever you need additional clarification or information regarding the use of your Mass Portal 3D printer, feel free to browse our support website http://massportal.com/en/support/ or contact us by sending an e-mail to: sales@massportal.com.

1.2. Warranty

For individual users warranty time is 1 to 2 years (depending on country, but not less than the minimum required by the state's law), for juridical entities (companies) the warranty is 1 year from the date of purchase. The warranty does not cover the [open-source] software. You need to carefully follow the instructions and recommendations from the User's manual and you must be able to present a proof of purchase upon request to be eligible for warranty service.

1.3. Safety

To avoid burns and physical damage do not touch the printer's head, heating plate or moving parts unless the printer has been completely turned off and the heating parts (extruder head, heating plate) has been cooled enough (to room temperature or less). The printer, filament and cabling must be placed so that it cannot be unintentionally touched or hooked by clothing. If you need to clean the heating bed or the extruder use a clean paper wipe or kitchen towel.

Warning! Printing surface is made from glass which can crack rather easily so be very careful with it. The heating bed can be sprayed lightly with non-flammable consumer plastics or glass cleaner and then wiped off with a clean wipe. The screws which hold the heating bed must not be overtightened because it may crack when it heats up. Avoid impacts and putting heavy objects on the heating bed or other parts.

Warning! When printing especially with ABS plastic it will lightly emit fumes and smell so the premises where the printer is located must be well ventilated, but must avoid draught. If the plastic is being cooled too rapidly it may warp. If the plastic is burned it will emit toxic fumes.

Warning! The printer must not be left unattended with children. This printer is not really a toy (for children) and therefore children must always be accompanied and supervised by an adult when using this product. Do not leave the printed objects, other plastic parts and tools unattended in the reach of small children to avoid harm and choking hazard. Be careful when tooling printed objects.

Warning! Do not touch printer's moving or heated parts when the printer is turned on or hasn't cooled enough.

I.e. please use common sense when working with the printer and always make sure that your actions are well justified. In case of doubt or further questions please contact us by e-mail <u>sales@massportal.com</u>.

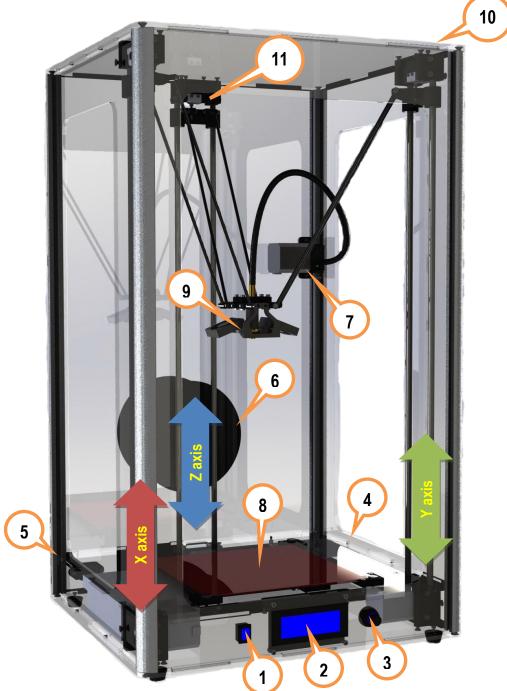
1.4. Protection

The printer is intended to be used only indoors. The ambient temperature must be between 15°C/59F and 30°C/86F (including). When working outside this temperature range the printed object may not print so well. The Mass Portal 3D printer under no circumstances must be subjected to high levels of humidity, excess heat, water, frost or other condition which may damage the electronics or mechanical parts.

To avoid clogging of extruder it must always reside at least 0.2mm above the printing surface. The extruder is intended to be heated up to 250°C/482F. Prolonged exposure at or above the maximum temperature may damage the printer's head.

2. General description of the printer

2.1. Visual



- 1) On/Off switch;
- 2) LCD display for standalone operation;
- 3) Rotary switch for controlling the printer;
- 4) USB port, MicroSD slot, RESET button;
- 5) Power supply AC 110-240V;
- 6) Plastic filament reel and holder;
- 7) Plastic filament supply motor;
- 8) Printing surface heated bed;
- 9) Printing head with fans for cooling of PLA plastic;
- 10) Timing belt adjustment screws (two on each axis);
- 11) Adjustment screws for the printer head (one on each axis).

2.2. Main characteristics

Printing technology	Fused Deposition Modeling
Mechanical technology	Delta robot
External dimensions	~420x400x720 mm (width x depth x height)
Weight	~13,5 kg
Case	Aluminium and plexiglass
Case color	Black, transparent
Printing dimensions	210x210x260 mm
Printing surface	Glass, heated up to 120°C
Printing head	Up to 250°C
Printing materials (filament)	PLA, ABS, PET, PVA plastic
Filament diameter	1.75 mm
Printing layer height	From 0.05 to 0.3 mm
Extruder	0.4 mm
Max speed	400 mm/s
Tolerance at max speed	+/- 0.3 mm
Power supply	110-240 VAC to 12VDC 200W
Stepper motors	NEMA 17
Timing belts	GT2.0
Control board	Megatronics v2.0 with USB connection, MicroSD card reader and LCD display
Software	Repetier-Host and Repetier-Firmware (http://www.repetier.com)

2.3. Package contents

- User's manual (this one);
- USB cable 1.8 m;
- AC power cable 220V 1.8 m;
- One reel with PLA filament 1 kg;
- Reel holder;
- MicroSD card;
- Kapton tape (50 mm wide) for the printing surface (heated bed).

2.4. Other optionally needed but not included tools

- Computer for controlling the printer and generating the G-code;
- Knife, file, sandpaper for tooling and trimming the 3D printed objects;
- Paper tissues for cleaning the printing surface and extruder;
- Glass or plastic cleaner for cleaning the printing surface;
- Screwdriver with H2.5 bit for removing the heating bed and calibrating the printer.

3. Usage

3.1. Setup

After removing the printer from packaging, remove the safety straps and fastenings (where applicable). **NB! By powering on the printer with the safety fastenings on you will most likely damage the printer!** Printer then must be placed on flat and stable surface. (Preferably out of reach of small children and pets). Make sure that the printer is not wobbling and is completely stable.

Insert the reel holder in the slot on the back of the printer. Place the filament reel on the holder clock-wise (with the filament thread on the left side pointing upwards). Put the filament thread in the hole from the back and feed it in the plastic filament supply motor as shown in the picture in chapter 3.3.

Connect the printer with computer using the supplied USB cable. Connect the mains power supply cable accordingly and press the On/Off button on the front panel to turn the printer on. The button, printer's inner lighting and LCD backlight should turn on automatically when the printer is turned on.

When the extruder is heating up you will hear cooling fans turn on.

3.2. Configuration

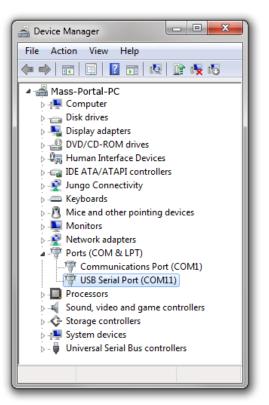
3.2.1. Driver

For the first time when the printer is connected to a computer running Windows Vista or newer Windows operating system it will automatically find and install correct drivers and make a connection to the printer. To make sure that the drivers are correctly connected, open **Device Manager** by clicking on the **Start Menu**, type in **devmgmt.msc** and press *Enter*. You can run the same command from the Windows Command Prompt.

In the **Device Manager** click on **Ports (COM & LPT)** and there you should find an entry called **USB Serial Port (COMX)** Where "X" represent your current port number assigned for the printer.

If you do not see the entry and are sure that physical connection between the printer and a computer has been made, you will have to install the drivers manually. Please visit the following website <u>http://www.ftdichip.com/Drivers/VCP.htm</u> and download the correct driver for your operating system. Please note that for simplified installation you can download *setup executable* from the website's Comment section.

After verifying that the correct drivers have been installed, double-click on the corresponding entry USB Serial Port (COMX) then click the Port Settings tab and change Bits per second: 9600 to the maximum 921600 and then click OK. This will allow you to send data from your computer to printer and back at maximum speed.



USB Serial Port (COM11) Properties				
General Port Settings Driver Details				
Bits per second: 921600				
Data bits: 8				
Parity: None				
Stop bits: 1				
Flow control: None				
Advanced Restore Defaults				
OK Cancel				

3.2.2. Repetier-Host software

Download the newest Repetier-Host software version from <u>http://www.repetier.com/download/</u> for your operating system, install and run it.

Follow the following steps to set the software up with your new Mass Portal 3D printer. For supplemental information regarding installation and configuration please visit <u>http://www.repetier.com/documentation/repetier-host/rh-installation-and-configuration/</u>.

Repetier-Host V0.95D	
File View Config Temperature Printer Tools Help	ow Filament Hide Travel
3D View Temperature Curve	Object Placement Slicer G-Code Editor Manual Control □ ● Pa ■ ♦ ↓ ∴ 1
	Name Mesh Collision
	Translation X Y Z Scale X Y Z Rotation X Y Z Cut Objects Cut Objects Cut Objects
rate	Position Inclination Azimuth
Show in Log: Commands Infos Wamings Errors ACK 15:38:49.723 OpenGL version:4.3.0 OpenGL extensions:GL_AMD_multi_draw_indirect 15:38:49.723 OpenGL extensions:GL_AMD_multi_draw_indirect 15:38:49.723 OpenGL renderer:GeForce GTX 560 Ti/PCIe/SSE 15:38:49.723 Using fast VBOs for rendering is possible	Auto Scroll Clear Log Copy Ct GL_ARB_arrays_of_arrays GL_ARB_base_instance GL_ARB_blend
Disconnected - Idle 1753 FPS	

First choose your language from the *Config->Language* menu.
 You need to configure *Printer settings* to connect to your printer. Please make the following changes as shown in the pictures. Press Apply after making changes in each of the tabs.

3) Press the *Connect* button to connect to your printer. If it becomes green it means that the connection is successful and the printer is ready for work.

Printer Settings		Printer Settings		
Printer: Portal Del	ta Pro 🔹 💼	Printer: Portal Delta Pro		▼ 💼
Connection Printer Print	er Shape Advanced	Connection Printer Printer Shape	Advanced	
Connector: Serial (Connection	Travel Feed Rate:	4000	[mm/min]
D _1	COM11	Z-Axis Feed Rate:	4000	[mm/min]
Port: Baud Rate:	Therear Toka	Default Extruder Temperature:	210	°
Transfer Protocol	115200 -	Default Heated Bed Temperature:	70	°C
	Autodetect	Number of Extruder:	1	
Reset on Connect	DTR low->high->low	Check Extruder & Bed Temperatu	ure	
Reset on Emergency	Send emergency command and reconnect	Remove temperature requests from the second seco	om Log	
Receive Cache Size:	127	Check every 3 seconds.		
	From Arduino 1 on the receiving cache was reduced from 127 to 63 bytes!	Park Position: X: 0	Y: 0	Z-Min -2 [mm]
Use Ping-Pong Com	imunication (Send only after ok)	Send ETA to printer display		
The printer settings alw are stored with every O	ays correspond to the selected printer at the top. They K or apply. To create a new printer, just enter a new	Go to Park Position after Job/Kill		
	apply. The new printer starts with the last settings	Disable Extruder after Job/Kill		
aciected.		Disable Heated Bed after Job/Kill		
		Disable Motors after Job/Kill		
		Add to comp. Printing Time 35	[%]	
	OK Apply Cancel		ОК	Apply Cancel

OK Cancel Apply

Printer Settings			Printer Settings
Printer: Portal D	Delta Pro	▼ 💼	Printer: Portal Delta Pro 🔹 💼
Connection Printer Printer	inter Shape Advanced		Connection Printer Printer Shape Advanced
Printer Type: C	Classic Printer	_	Post Slice Filter
Home X: 0	✓ Home Y: 0	✓ Home Z: Max ✓	Filter Path and Parameter: vourFilter #in #out
X Min -105	X Max 105	Bed Left: -105	Run Filter after every Slice
Y Min -105	Y Max 105	Bed Front: -105	You can run a filter program after each slicing action. The filter will be run on the G-Code produced by the slicer. Use #in and #out to insert the input and output filenames as parameter.
Print Area Width:	210	mm	
Print Area Depth:	210	mm	
Print Area Height:	290	mm E	
can be negative and out	side the print bed. Bed left/from changing the min/max values y	Atruder coordinates. These coordinates it define the coordinates where the ou can even move the origin in the Apply Cancel	OK Apply Cancel

3.2.3. Windows

Most of the modern operating systems have so-called *Sleep mode*. The process of 3D printing is not blazingly fast and may take several hours. Therefore it is recommended to disable this mode so that the computer stays awake when printing.

0	🔍 🗢 😼 « All Control Panel	Items 🕨 Power Options 🔹 🍫	Search Control Panel
	Control Panel Home	Select a power plan	
	Require a password on wakeup Choose what the power buttons do	Power plans can help you maximize your compute plan active by selecting it, or choose a plan and cu <u>Tell me more about power plans</u>	
	Create a power plan	Preferred plans	
Ŀ	Choose when to turn off the display	Balanced (recommended) Automatically balances performance with end Automatically balances performance with end Automatically balances performance	Change plan settings nergy consumption on capable hardware.
۲	Change when the computer sleeps	eigh performance Favors performance, but may use more energy	Change plan settings rgy.
	See also Personalization User Accounts	Hide additional plans Power saver Saves energy by reducing your computer's	Power Options Advanced settings Select the power plan that you want to customize, and then choose settings that reflect how you want your computer to manage power. High performance High performance
		S Windows you can do it by opening pe powercfg.cpl and press <i>Enter)</i> 9	 → Hard disk → Turn off hard disk after → Eetting (Minutes): 0 → ○ Internet Explorer ○ Desktop backtop backtop constraints ○ Wireless Adapter Settings ○ Sleep ○ USB settings ○ Power buttons and lid → Brit Evencer

3.3.

Change from **Balanced** to **High performance** and click **Change plan settings**. Click **Change advanced power** settings in the Edit Plan Settings window.

2) Find the Hard disk entry, open the Turn off hard disk after sub-entry and change the setting to 0 and press OK.

2 3

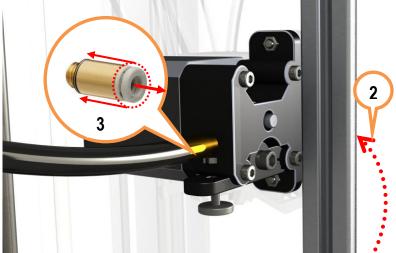
least 170°C to avoid motor or tensioner damage.

To insert the plastic filament follow steps below.

1) Filament tensioner screw. Always make sure that the tensioner is released by turning the screw with your fingers outwards (counter-clockwise) before inserting or removing filament. After inserting the filament adjust the tensioner by turning the screw clockwise until it feels tight, but do not over tighten it. It must move evenly when the following commands from Repetier-Host Manual Control tab are run: Extrude or Retract. Please run the above commands only when the head has been warmed to at

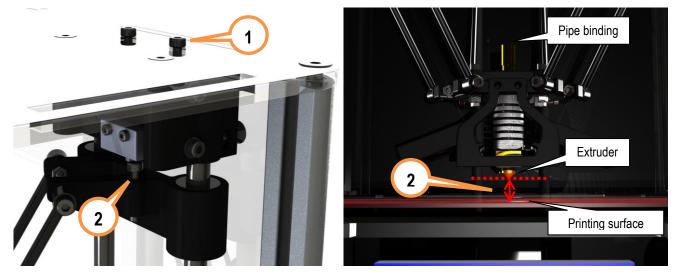
2) Plastic filament. We use plastic filament with 1.75 mm in diameter. Manual insertion and retraction is done from the back of the printer in the dedicated hole.

3) Pipe binding. In case the tip of the filament has melted so wide that it is unable to remove it, please detach the binding from the pipe and snip the melted filament tip using pliers. After that you will be able to remove the filament. The pipe can be removed from the binding only when the base of the pipe is squeezed together and then pulled outwards.



Insertion and retraction of plastic filament

3.4. Calibration



1) *Timing belt adjustment*. Use the two screws at the top to adjust the timing belt. By turning the screws clockwise the timing belt is tightened, but by turning them counter-clockwise the timing belt is released. The screws must both be turned equal amount of turns. You can use a marker to mark the position of screws. The timing belts on each axis must be checked by hand – it must be able to push them lightly together, but they must not be too tight or too loose.

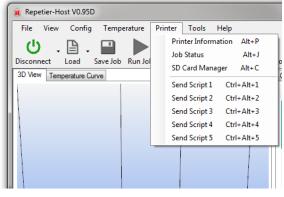
2) Extruder gap height in relation to printing surface. In case the printing head's extruder is not at equal height from every position against the printing surface, it must be calibrated. One of the symptoms is uneven adhesion of the first extruded layers when printing a 3D object. Use the screws on top of each axis sliders (refer to illustration in chapter 2.1 for axis labels). By turning the screw **clockwise** the gap between extruder and printing surface is **extended**, by turning it counter-clockwise the gap is reduced. It is rather likely that the calibration must be made after changing the extruder or after transportation.

To ease the calibration use *Scripts* from *G-Code Editor* in the Repetier-Host software:

Script 1 G1 X-100 Y-100 Z0.1 F4000; place near the X axis column *Script 2* G1 X100 Y-100 Z0.1 F4000; place near the Y axis column *Script 3* G1 X0 Y100 Z0.1 F4000; place near the Z axis column *Script 4* G1 X0 Y0 Z0.1 F4000; the center of the printing surface

agle Log	Show Filament Hide Travel	Printer Settings Emergency Stop
	Object Placement Slicer G-Code E	Editor Manual Control
	DEXABOC	Script 1 🔹
	2 3 4 5	G-Code Start Code An on Kill Run on Pause Script 1 Script 2 Script 3 Script 5
		* 4 III
	Visualization Help Show complete Code Show	v Single Layer 💿 Show Layer Range

You can run these scripts by pressing *Ctrl+Alt*+[Num] where [Num] is the script number.



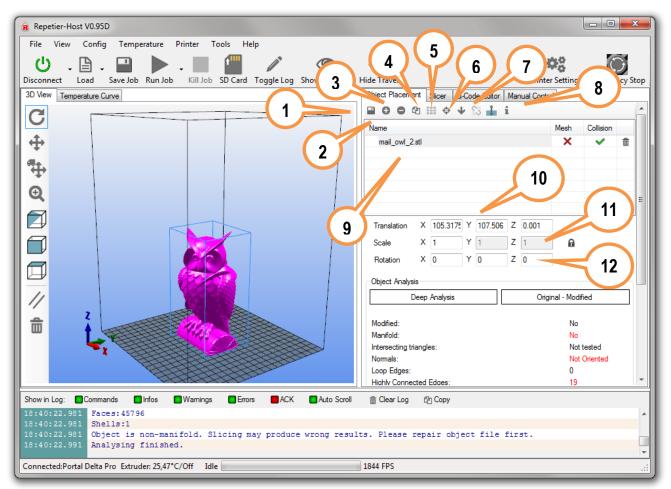
exactly 1:1.

By sending each script command you can check the gap at the corresponding head placement. It must be 0.1 mm. You can check the height by placing a piece of common plain paper between the extruder and a printing surface (before sending the command). Move the paper around and notice the resistance – if it is unable to move it means that the gap is too tight, if it is too easy to move the paper under the extruder then the gap is too big. Adjust the screws accordingly. Note that one full turn of a screw moves the slider by 0.5 mm, but the ratio between the turns of a screw and the extruder's gap is not

NB! Keep your hands and fingers clear from the printing surface before sending script commands or moving head manually. Make sure that it is unable to move the head accidently when calibrating or checking status of the printer's inner parts to avoid damage!

3.5. Repetier-Host usage

3.5.1. Object placement



- 1) Export saves the changed object as an STL or OBJ file;
- 2) Add object add new 3D objects. You can add one by one or even several at a time;
- 3) *Remove object* remove each selected object or use *Delete* button on keyboard;
- 4) Copy object(s) make several copies of the selected object;
- 5) Autoposition positions objects on the surface. Works only when there are more than one object;
- 6) Center object center each selected object in the center of the plate;
- 7) Drop object automatically position each selected object on the Z axis ground level;

8) **Object information** – show detailed information about the selected object(s) like volume, dimensions, etc.;

- 9) **Objects** lists all objects added to the project as filenames. Select individual filename to perform actions with it;
 - 10) **Coordinates** move the object on the printing surface by changing $\pm X$, $\pm Y$ and $\pm Z$ values;

11) **Scale** – change the object's size by adjusting the value in the X field. The value 1 sets the object to its original size;

12) *Rotation* – rotate each object according to your needs by degrees (0 - 359)

3.5.2. Object slicer

Repetier-Host V0.95D		
File View Config Temperature Printer Tools Help		
U - C - C - C - C - C - C - C - C - C -	ilament Hide Travel	Setting _ mergency Stop
3D View Temperature Curve	Object Placement Slicer G-Code Editor Manual Control	
	Slice with Slic3r	Kill Slicing
	Slicer: Slic3r	 Open Manager
Q	Print St 1 kc_slic3r_2013-12-27	Configure
	Printer Settings: Portal Delta	
	Filament settings:	
	Extruder 1: Portal Delta	
	Extruder 2: Portal Delta	
	Extruder 3: Portal Delta	
	Override Slic3r Settings	Created ans saved
	Copy Print Settings to Override	profiles
J HILL	Enable Support	
	Layer Height: 0.2 mm	-
· Show in Log: Commands Infos Warnings Errors ACK AL	to Scroll 💼 Clear Log 🖓 Copy	
18:56:08.610 Faces:642 18:56:08.610 Shells:1		•
18:56:08.610 Object is manifold. 18:56:08.610 Analysing finished.		
		•
Connected:Portal Delta Pro Extruder: 25,35°C/Off Idle	1428 FPS	

- 1) Slicer select your preferred 3D object slicer;
- 2) **Slice with [Slic3r]** slice the object(s) with predefined profiles using the slicer of your choice ("Slic3r" by default). *Slicing* means converting 3D object to G-Code which is read by the printer;
- 3) Kill Slicing stop the slicing process using this button;
- 4) Configure opens selected slicer's configuration window.

3.5.2.1.Slic3r slicer

Slic3r is one of the *slicer* programs which converts a 3D objects in layers and generates G-Code which contains coordinates and settings for the printer on how to exactly print a 3D object. The quality of the printed object greatly depends on these settings.

For each Settings type (Print Settings, Filament Settings, Printer Settings) you can create and change as many profiles as you like and select them in the **Slicer** tab from main window of Repetier-Host.

Note: You can view description of each function just by hovering cursor over it.

💋 Slic3r		
<u>File Plater Window H</u> elp Plater Print Settings Filament Setti	Create a	and save many profiles
0.2 Layers and perimeters Layers and perimeters Infill Speed Skirt and brim Support material Notes Output options Multiple Extruders Advanced	Layer height Layer height: First layer height: Vertical shells Perimeters (minimum): Spiral vase: Horizontal shells Solid layers:	0.2 mm 0.2 mm or % 2 ▼ ■ Top: 3 ▲ Bottom: 3 ▼
	Quality (slower slicing) Extra perimeters if needee Avoid crossing perimeter Start perimeters at: Detect thin walls: Detect bridging perimeter Advanced Randomize starting point External perimeters first:	rs: Concave points: Non-overhang points: Concave points: Non-overhang points:

- Layer height: the height of each layer for the 3D printed object, it should be between 0.05-0.3 mm;
- *First layer height:* height of the first layer set it to at least 0.2 mm to get better grip to the printing surface;
- **Perimeters (minimum):** how many lines should be made at the 3D object's perimeter making a shell. Usually 2-3 is enough;
- **Solid layers:** number of solid layers at the top and bottom of the object. If set to 0 layers will be empty. Usually 2 3 at 0.2 0.3 mm layer height, for 0.05 0.1 mm layer height use ca. 6 8;
- **Avoid crossing perimeters:** Optimizes printing path to almost avoid crossing the perimeter. Usually not used as it prolongs the printing and G-Code generation time.
- **Randomize starting points:** select to have the starting point of each layer at a different place instead of one;
- External perimeters first: usually not used;

💋 Slic3r		
<u>File Plater W</u> indow <u>H</u> elp		
Plater Print Settings Filament Settin	ngs Printer Settings	
0.2 🔻 🗒 🥥	Infill	
Layers and perimeters	Fill density:	0.3
Infill	Fill pattern:	rectilinear 🔻
 Speed Skirt and brim 	Top/bottom fill pattern:	rectilinear 🔹
🚊 Support material 🥃 Notes	Reducing printing time	
🙀 Output options	Combine infill every:	1 ayers
Multiple Extruders Advanced	Only infill where needed:	
	Advanced	
	Solid infill every:	0 A layers
	Fill angle:	45 <u>•</u> •
	Solid infill threshold area:	70 mm ²
	Only retract when crossing perimeters:	
	Infill before perimeters:	

- *Fill density*: how much should the object be filled up. 1 means 100% and gives a solid object, but it takes much more time to make one. For partially filled objects use values between 0.1 and 0.9 which corresponds to 10% and 90%. Use 0 to get empty object with just the outer shell;
- *Fill pattern*: choose a pattern for the object to be filled up with;
- Top/bottom fill pattern: you can select different pattern for top and bottom layers;
- **Combine infill every**: how many layers to combine together. By setting the value above 1 you can save time and increase printing speed;
- Only infill where needed: automatically calculate where the infill is needed;
- Solid infill every: usually not used, but can increase structural strength for larger, partially filled objects;
- Fill angle: choose an angle at which the infill walls are created;
- **Solid infill threshold area**: 3D objects will have solid infill if their cross section is smaller than the given value (e.g. 7x10mm);
- **Only retract when crossing perimeters**: does not waste time on retracting the filament, if the plastic thread will not be visible. (i.e. in the inner layers of the object) ;
- Infill before perimeters: Usually not used as it creates the perimeters last and therefore does not guarantee smooth outer surface;

💋 Slic3r				J
<u>File Plater Window H</u> elp				
Plater Print Settings Filament Settin	ngs Printer Settings			
0.2 🔻 🗎 🥥	Speed for print moves			
Layers and perimeters	Perimeters:	60	mm/s	
Infill	Small perimeters:	60%	mm/s or %	
Speed Skirt and brim	External perimeters:	80%	mm/s or %	
L Support material	Infill:	80	mm/s	ŀ
🔲 Notes	Solid infill:	80%	mm/s or %	
Output options	Top solid infill:	60%	mm/s or %	
Multiple Extruders Advanced	Support material:	50	mm/s	
/ Autoneca	Bridges:	50	mm/s	
	Gap fill:	30	mm/s	
	Speed for non-print moves		=	
	Travel:	350	mm/s	
	Modifiers			
	First layer speed:	40%	mm/s or %	
	Acceleration control (advance	d)		
	Perimeters:	0	mm/s ²	
	Infill:	0	mm/s ²	
	Bridge:	0	mm/s ²	
	First layer:	0	mm/s ²	
	Default:	0	mm/s²	

- Perimeters: printing speed for normal perimeters usually about 50–100 mm/s;
- **Small perimeters**: If the radius is <=6.5 mm, decrease the *Perimeters* speed usually to 50-80%;
- **External perimeters**: printing speed for outer perimeter, recommended as above 50-80% from *Perimeters*;
- Infill: how fast to print the infill. Can be set to 60-120 mm/s as it mostly won't be visible;
- **Solid infill**: Printing speed for solid layers should be less and therefore more accurate. Recommended 50-80% from *Infill*;
- Top solid infill: same as the previous setting, but for the top layer. Also should be 50-80% from Infill;
- Support material: recommended 30-60 mm/s for the support material printing;
- Bridges: recommended 30-60 mm/s for the bridge printing;
- Gap fill: smaller gaps should be filled a bit slower 20-40 mm/s;
- Travel: traveling speed of the printer head, 200-400 mm/s is recommended;
- *First layer speed*: print the first layer at about 30-40% of the general speed to get better adhesion to the printing surface.

NOTE! By using too fast printing speeds the quality of printed 3D objects may decrease significantly. Please use the recommended values or lower ones to get better printing results

💋 Slic3r			
<u>File Plater Window H</u> elp			
Plater Print Settings Filament Sett	ings Printer Settings		
0.2 🔻 🗒 🥥	Skirt		
Layers and perimeters	Loops:	1	
Infill	Distance from object:	5 mm	
© Speed	Skirt height:	1 🚔 layers	
Skirt and brim	Minimum extrusion length:	0 mm	
Notes Output options	Brim		
Multiple Extruders	Brim width:	0 mm	

- **Loops**: how many loops to print around the object before printing itself. This allows to get an even flow of filament. Usually 1 to 3 is enough;
- Distance from object: how far should these loops be printed. 5-6 mm is usually optimal;
- Skirt height: loop height in layers. One layer is OK most of the time;
- *Minimum extrusion length*: print so many layers to extrude the specified minimum of filament in millimeters;
- **Brim width**: addition of filament on the first layer around each printable 3D object to get a better grip. 3-5 mm usually is enough when needed.

💋 Slic3r		
<u>File Plater Window H</u> elp		
Plater Print Settings Filament Settin	gs Printer Settings	
0.2	Support material]
Layers and perimeters	Generate support material:	
Infill	Overhang threshold:	0 * *
 Speed Skirt and brim 	Enforce support for the first:	0 ayers
 Support material Notes 	Raft	
Output options	Raft layers:	0 ayers
🤌 Advanced	Options for support material and raft	
	Pattern:	rectilinear 🔻
	Pattern spacing:	2 mm
	Pattern angle:	45 °
	Interface layers:	0 ayers
	Interface pattern spacing:	0 mm

- Generate support material: enable or disable generation of support material usually for objects with
 relatively large overhangs and outward slopes which are at more than 45 degree angle/ as calculated by
 the program/ or specified by the next setting;
- **Overhang threshold**: 0 for automatic calculation. Set 45 for optimal manual value.
- Enforce support for the first: number of layers from bottom to be extra-enforced (regardless of the settings above). Good for smaller objects which are standing on one small leg or similar – to get better adhesion to the printing surface;
- Raft layers: "push" the printable object upwards by generating specified amount of raft layers;
- **Pattern**: pattern of choice for raft and support materials;
- Pattern spacing: use lower value to get better support for flat surfaces;

- **Pattern angle**: adjust the support material pattern angle;
- Interface layers: add extra layers where supports will be attached;
- Interface pattern spacing: lower value gives denser interface filling.

💋 Slic3r			
<u>File Plater Window H</u> elp			
Plater Print Settings Filament Setti	ngs Printer Settings		
0.2 • 🗎 🥥	Sequential printing		
Layers and perimeters	Complete individual objects:	\checkmark	
Infill	Extruder clearance (mm):	Radius: 20 Height: 2	20
🕑 Speed			
Skirt and brim	Output file		
🛄 Support material 🔲 Notes	Verbose G-code:		
Output options	Output filename format:	[input_filename_base].gco	de
Multiple Extruders	output mename format.	[input_mename_base].geo	
dvanced	Post-processing scripts		

• **Complete individual objects:** use when printing multiple objects simultaneously. Check to complete each object's layer before going to work on another's.

💋 Slic3r			
<u>File Plater Window H</u> elp			
Plater Print Settings Filament Setti	ngs Printer Settings		
0.2 • 🖹 🥥	Extrusion width		
Layers and perimeters	Default extrusion width:	0.4	mm or % (leave 0 for auto)
Infill	First layer:	0	mm or % (leave 0 for default)
Speed	Perimeters:	0	mm or % (leave 0 for default)
Skirt and brim Support material	Infill:	0	mm or % (leave 0 for default)
Notes	Solid infill:	0	mm or % (leave 0 for default)
Gutput options	Top solid infill:	0	mm or % (leave 0 for default)
Multiple Extruders	Support material:	0	mm or % (leave 0 for default)
	Flow		
	Bridge flow ratio:	1	
	Other		
	Threads:	2	(more speed but more memory usage)
	Resolution:	0	mm

- **Default extrusion width**: the width of one extruded and printed layer's line. It may differ a bit from layer to layer. Also depends on the width of extruder (0.4 mm). Set to 0 for automatic calculation.
- **Bridge flow ratio:** change only when experimenting/needed to less or more than 1. Increases or decreases the flow of filament when building bridges.
- **Threads**: set this accordingly to your computer's CPU threads to get better performance but larger memory consumption for object slicing.
- **Resolution**: set the value above 0 mm to get faster printing but less details/texture.

💋 Slic3r			
<u>File Plater Window H</u> elp			
Plater Print Settings Filament Setti	ngs Printer Settings		
PLA 🔻 🗎 🥥	Filament		
Filament	Diameter:	1.75	mm
Z Cooling	Extrusion multiplier:	1	
	Temperature (°C)		
	Extruder:	First layer:	
	Bed:	First layer:	70 Other layers: 70

- **Diameter**: the width of filament diameter. May differ a bit in reality for lower quality filaments. Set 1.75 mm for Mass Portal printers;
- Extrusion multiplier: normally set to 1, but can be adjusted if the extrusion flow is not as needed;
- **Extruder**: change the extruder's temperature for first and all other layers. Depending on printing speed and material sett 180-230°C for PLA and 220-240°C for ABS. The faster you print the more heating will be needed. Remember that the maximum temperature for the extruder is 250°C, do not use it at or above maximum temperature for prolonged periods;
- **Bed**: temperature of heating bed (printing surface) for the first and rest of the layers. Use 60-70°C for PLA and 100-110°C for ABS. Remember that maximum for the heated bed is 120°C.

💋 Slic3r		
<u>File Plater Window H</u> elp		
Plater Print Settings Filament Setti	ngs Printer Settings	
PLA 👻 📄 🥥	Enable	
Filament	Keep fan always on: Image: Construction of the second	
	will never be reduced below 20mm/s). If estimated layer time is greater, but still below ~60s, fan will run at a proportionally decreasing speed between 60% and 40%. During the other layers, fan will be turned off.	
	Fan settings Fan speed: Min: 40 Win: 60 Bridges fan speed: 60 0 V Disable fan for the first: 2	
	Cooling thresholdsEnable fan if layer print time is below:60Slow down if layer print time is below:5Min print speed:20	

- Keep fan always on: may be useful for printing with PLA, but not recommended for ABS as it will always cool the object when printing;
- Enable auto cooling: uses the cooling fan(s) automatically under 3D printing recommended to use this option. When printing with ABS set *Fan speed* and *Bridges fan speed* to zero.

💋 Slic3r		
<u>File Plater Window H</u> elp		
Plater Print Settings Filament Setti	ngs Printer Settings	
Portal Delta Pro 🔹 📄 🥥	Size and coordinates	
General	Bed size:	x: 210 y: 210 mm
🔅 Custom G-code	Print center:	x: 0 y: 0 mm
🖗 Extruder 1	Z offset:	0 mm
	Firmware	
	G-code flavor:	RepRap (Marlin/Sprinter/Repetier)
	Use relative E distances:	
	Capabilities	
	Extruders:	1
	Advanced	
	Use firmware retraction:	
	Vibration limit:	0 Hz

- Bed size: size for the heated bed. 210x210 mm for Portal Delta Pro and 150x150 mm for Portal Delta;
- **Print center**: depending on printer but usually 0;
- **Z offset**: use this in case you need to adjust the gap between extruder and printing surface without calibration. Use negative value to decrease the gap, but positive value in millimeters to increase the gap;
- Extruders: number of extruders, depending on printer. Currently set to 1;

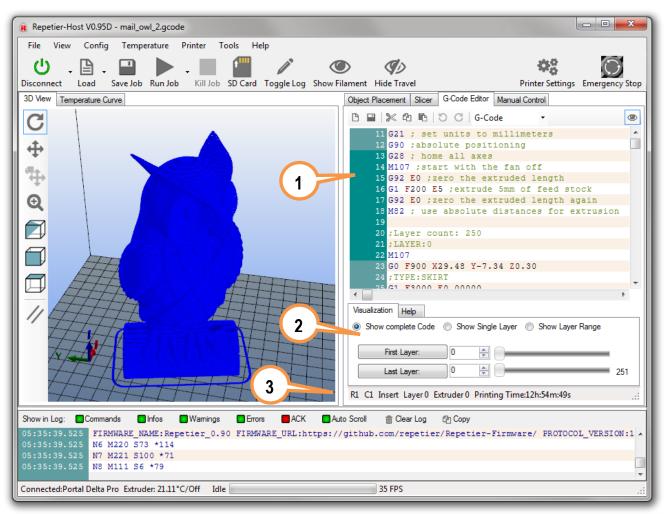
💋 Slic3r		_ D X
<u>File Plater Window H</u> elp		
Plater Print Settings Filament Settin	gs Printer Settings	
Portal Delta Pro 🔻 📄 🥥	Start G-code	^
General Custom G-code Fxtruder 1	G28 ; home all axes	*
	End G-code G28 ; home all axes	* E
	M104 S0 ; turn off hotend M140 S0 ; turn off heated bed M107 ; turn off fan M84 ;disable motors	

You can use custom G-Code commands to add at the beginning and end of generated G-Code for the object.

💋 Slic3r			
<u>F</u> ile <u>P</u> later <u>W</u> indow <u>H</u> elp			
Plater Print Settings Filament Setting	ngs Printer Settings		
Portal Delta Pro 🔹 📄 🥥	Size		Â
General	Nozzle diameter:	0.4 mm	
Custom G-code Factor Extruder	Position (for multi-extruder printers	;)	
	Extruder offset:	x: 0 y: 0 mm	
	Retraction		
	Length:	4.0 mm (zero to disable)	
	Lift Z:	1.0 mm	_
	Speed:	40 mm/s	
	Extra length on restart:	0 mm	
	Minimum travel after retraction:	2 mm	
	Retract on layer change:		
	Wipe while retracting:		
	Retraction when tool is disabled (ad	dvanced settings for multi-extruder setups)	
	Length:	3 mm (zero to disable)	
	Extra length on restart:	0 mm	

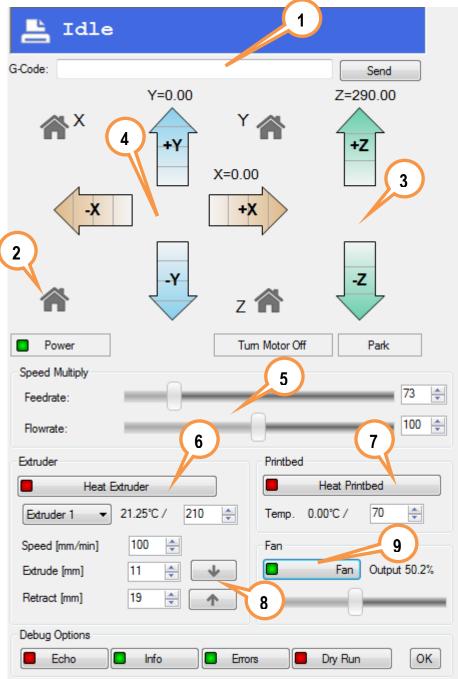
- **Nozzle diameter**: diameter in millimeters for the extruder's nozzle. Normally 0.4 mm;
- [Retraction] Length: set the length of raw filament to be retracted when retraction command is sent. 4 5 mm is optimal;
- Lift Z: how high should the extruder be lifted up above the object(s). Normally 1 1.5 mm;
- Speed: the retraction speed of the filament. 30 50 mm/s is recommended;
- Extra length on restart: rarely needed. Use when there is no or very little filament extruded after continuing the layer on different spot;
- Minimum travel after retraction: do not retract the filament if the head has to move less than the specified value;
- Retract on layer change: usually needed to get prettier perimeters;
- Wipe before retract: tries to minimize blobs whenever retracting. Usually not needed.

3.5.3. G-Code Editor



- G-Code Here are all the generated G-Code commands. You can make corrections, copy, paste and save. Depending on the slicer program used, you may find the calculated length and volume of the filament to be extruded for the object
- 2) Visualization View the whole sliced object, each layer or layer by layer;
- 3) Stats view layer count and estimated printing time.

3.5.4. Manual Control



- 1) **G-Code** *input field* here you can manually type and send G-Code commands. You can find more about G-Code at <u>http://reprap.org/wiki/G-code</u>;
- Home position printer head resets to the home position which is at the top of the printer. It should always be at Home position before printing objects. It is not recommended to press it (and other movement control buttons) under printing;
- 3) **Z** axis movement use this to move printing head up or down; Each arrow has 3 or 4 separate blocks which define the amount in millimeters to move the printing head;
- 4) *Movement of X and Y axis* move the printing head to left or right, front or back;
- Feedrate and Flowrate these settings can be changed under printing. E.g. use the federate to slow down the printing process (for better quality or more time for quality inspection) Flow rate will change the amount of filament being extruded;
- Heat Extruder to speed up the process you can heat up the extruder in case you plan to print soon, but it will heat up automatically once the printing job has started. Printing temperature can be adjusted under printing;

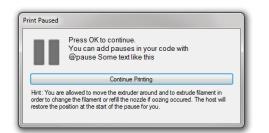
- 7) *Heat Printbed* same as with the extruder can be heated up in advance; otherwise it will heat up automatically if specified in G-Code. Can be adjusted under printing;
- 8) Extrude and Retract NOTE: it is recommended to check the flow of filament (and if it is not clogged) before starting print to get good adhesion once the printing has started. Heat up the extruder to at least 170°C before using these buttons. The filament must flow out evenly, without interruptions to ensure good start of print. Be careful when removing the extruded test-filament though it cools rather quickly, it may still be hot enough to burn skin. Use caution!
- 9) *Fan* turn on or off the fan(s) at the printing head. Mostly used under PLA printing to improve the quality of print. Can be adjusted as needed.

3.5.5. Main menu

Repetier-Host V0.95D - mail_owl_2.gcode		9 - 0 %
File View Config Temperature Printer Tools Help	8	
 ・ B · E · E 		*:
		ter Settings Emergency Stop
3D View Tempreture Curve	Object Placement Slicer G-Code Editor Manual Control	
	6 7	Mesh Collision
	Translation X 0.317498 Y 2.506 Z 0.001	
	Scale X 1 Y 1 Z 1	G
	Rotation X 0 Y 0 Z 0	
and the second sec	Cut Objects	
Z	Position	
₩ X 2000	Inclination	
	Azimuth	
Show in Log: Commands Infos Warnings Frors	ACK Auto Scroll 💼 Clear Log 🖓 Copy	
05:58:12.271 N460 G1 Y-50 F4000 *57		
05:58:12.271 N461 G90 *51		Î
05:58:19.031 N464 G28 Y0 *124		
05:58:29.331 N469 M84 *52		Ŧ
Connected:Portal Delta Pro Extruder: 22.22°C/Off Idle	24 FPS	.::

Include "Start" an	d "End" Code		
Include Job Finish	ed Commands		
Save in Binary For	mat (for Repetier-Fin	mware)	
This save function is in run from a sd card. For code along with the re during a direct print for commands get removing your editor, use the sa	or that reason it can a egular finish commar om the host. All comm ed! If you only want	add the start ids like they ments and h	and end are executed ost
	Save	Cano	cel

- 1) Load file allows to load 3D object files in the following formats: *.gcode, *.gco, *.stl; *.obj, *.3ds;
- 2) Save Job by pressing this button to save G-Code:
 - Using the options as in the picture on left will save the object as a simple, text-based G-Code.
 - If you select the third option in the dialog (Save in Binary Format) you won't be able to make any further manual corrections in the file, but it can easily be used to print objects directly from MicroSD card.



- Run (Pause) job you can start the printing by pressing this button. If the printing has already been started, it will change to Pause job. You can pause printing if you need to change filament and even move the head around. It will resume where you left when clicking on Continue Printing;
- 4) *Kill job* completely stops the printing and the head returns to home position;
- 5) **SD Card** by pressing this button you will see a dialog which will allow you to manage the MicroSD in the printer;

d SD Card Manager G File Name PORTAL B C e f f	
Close	
Print Paused	

- a) Upload file Upload G-Code currently loaded in the Repetier-Host software or browse the computer for pre-generated G-Code file. Due to the limited baud rate the upload by this method will take quite a while. Recommended alternative is to unmount the MicroSD card from the printer put it in a card reader attached to computer and upload sliced object file directly;
- b) Delete file remove files from the memory card;
- c) New folder Make a new folder to save the data;
- d) Run (Continue) job using this button you can print selected file from the memory card ;
- e) Pause / Stop stop current print from the memory card;
- f) Mound SD Card push this button after inserting memory card in the printer (optionally use the LCD control panel directly on the printer);
- g) Unmount SD Card use this before removing the memory card from printer.
- 6) Toggle Log show or hide log entries at the bottom of the screen;

- 7) **Show/Hide filament** show or hide filament being printed in the real time. May not be smooth on older computers;
- 8) **Show/Hide travel** show or hide the movements of the printer head;
- 9) *Emergency Stop* use this button to immediately stop whatever printer is doing. Useful if you notice unusual, erratic or incorrect movements of the printing head.

3.6. LCD control panel

You can use the front panel on the printer to check current status of the printer, control various parameters and settings, print files standalone – without the need of external computer, do software calibration and other tasks.

Use the **Rotary button** to navigate your way through menus. You can rotate the button clockwise or counter-clockwise to move the cursor. Press in the button to select a menu. Be careful to not turn the button while pressing it in as it may switch to other setting than wanted.

For newer firmware and 3D printer versions the LCD menus may slightly differ and therefore following images and descriptions does not have to 100% accurately represent your current LCD panel setup. Use this as a general guide.

Main screen.

You can switch between main screens by rotating the button. It shows current/set temperature, extruder position, and status.

@125.3/200°B: 59/		
	N 2 m m 2 m m	E1:125.3/200°C> 100%
Mul:100 Buf:30	Z: 15.00 mm	E2: 25.1/ 20°C> 0% B: 59.0/ 60°C> 90%
Idle	Idle	Idle

Main menu

>Back ^	
Quick settings	
Position	
Extruder	
Fan_speed	
SD Card	
Debugging	
Configuration	

Quick settings

Position - allows to manually move printing head

Extruder - turn on or off the heating for extruder, extrude filament and adjust temperature

Fan speed - Change the fan speed from 0 to full speed

SD Card - Mount/unmount , access the contents of memory card and print selected file

Debugging - Print without actually using filament (Dry run) or turn on various info

messages

Configuration - configure advanced printer settings like Acceleration, Feedrate, Extruder, Calibration and others.

WARNING: Incorrect settings in either software or firmware may damage the printer and even void the warranty! Use with caution!

Main menu > Quick settings

>Back ^	Home all – put the printing head in the home position (center, at the top)
Home all	Preheat PLA – heats the head up to 220 C° and heatbed up to 70 C° for printing with
Preheat PLA	PLA plastics
Preheat ABS Speed Mul.:100%	Preheat ABS - heats the head up to 240 C° and heatbed up to 110 C° for printing with
Flow Mul. :100%	ABS plastics
Cooldown	Speed Mul. :100% - adjust printing speed
Set to origin	Flow Mul. :100% - Adjust extrusion flow rate of the filament
Disable stepper	Cooldown – stop heating both heating bed and extruder
	Disable stepper – cut power to stepper motors

MASS PORTAL

Main menu > Position

>Back ^	
Home all	
Home X	
Home Y	
Home Z	
X Pos. Fast	>
X Position	>
Y Pos. Fast	>
Y Pos. Fast Y Position Z Pos. Fast	>
Z Pos. Fast	~ ~ ~ ~ ~ ~
Estr. Position	>

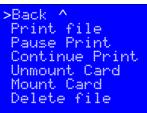
Main menu > Extruder

>Back ^
Bed Temp.: 60°C
Temp. 0 :200°C
Temp. 1 : 30°C
Extruder 0 OFF
Extruder 1 OFF
o Select Extr.0
o Select Extr.1
Extr.position \rightarrow
Set origin

Main menu > Fan speed

>Back	\wedge		
Turn	Far	0ff	
Set F			
Set F			
Set F			
Set F	-an	Full	

Main menu > SD Card



3.7. Maintenance

Please do regular maintenance to allow the printer to serve you well and long. Maintenance includes:

- Cleaning the glass printing surface do every time before printing to assure good adhesion of 3D objects. Clean with glass or plastics cleaner and kitchen towel. If plastic has melted hard on the surface, you may use a scraper or blade, but be very careful not to cut yourself or damage the printing surface!
- External extruder cleaning do every time before printing or as needed. Heat the extruder to at least 200°C, and use kitchen towel folded in many layers to wipe off excess plastic from the extruder. As always be VERY careful and don't hold the kitchen towel on the extruder for too long and avoid touching it with fingers. Use heat-protective gloves if needed;
- Cleaning and oiling of steel rods for each axis do once in 2-3 months. Clean the rods carefully
 with dry paper towel from dirt and grease. Oil the rods lightly in full length. You can use mineral oil,
 special bearing oil or oil for bicycles;
- Calibration as needed. Detailed description can be found in chapter 3.4.

Home all - put the printing head in the home position (center, at the top) Home X – level the printing head at the home position of X axis Home Y - level the printing head at the home position of Y axis Home Z - level the printing head at the home position of Z axis X/Y/Z Position – move the head on the corresponding axis Extr. Position > - extrude or retract filament. Note that the extruded must be warm enough for this operation.

Bed Temp.: $60 C^{\circ}$ - Set the heating bed temperature Temp. 0 :200 C° - Set the extruder temperature Extruder 0 OFF - Status of the extruder

Extr. Position \rightarrow - extrude or retract filament. Note that the extruded must be warm enough for this operation.