



Mass Portal® Pharaoh® XD Series Desktop 3D Printer

User Manual



Mass Portal® Pharaoh® XD

Desktop 3D Printer

User manual

English version 4.0

If you have any comments about this user manual
please let us know: support@massportal.com

REFERENCE VERSIONS

Model Number: MP20, MP30, MP40

Pharaoh® XD Hardware ver. 4.0DV, 4.0MV

Pharaoh Touch™ 1.0

Simplify3D® ver. 3.0.2

Repetier Firmware ver. 203-ebd8c58

*The firmware and Pharaoh Touch version may change due to updates.

Contents

Contents.	3
Safety	4

Getting started

Welcome	6
Printer Setup	8
My First Print	10
Printing process overview	12

Printer operation & software

Print Bed Coating and Heating	14
Print Bed Leveling / Calibration	15
Ventilation System	16
Standalone Printing	18
Using Pharaoh Touch™	19
Software: Simplify3D	28
Printing with Dual Extruder	31

Printing materials

Printing Materials Overview	36
Post Processing of Prints	38

Maintenance

Printer maintenance	39
-------------------------------	----

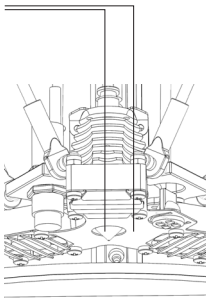
Troubleshooting

Troubleshooting	42
---------------------------	----

Safety

Mass Portal® 3D printer is an electro-mechanical robotic device that has hot and moving parts when operating. You should always operate the printer with caution and follow the instructions in this manual.

WARNING The nozzle and the glass plate below the print head are extremely hot when printing. Do not touch.



TIP Use paper towel to clean the nozzle from excess material before printing. This must be done when the nozzle is heated, please use extreme caution.

INDOOR USE ONLY

The printer is intended to be used only indoors — ambient temperature must be between 15°C and 30°C. The Mass Portal 3D printer under no circumstances should be subjected to high levels of humidity, excess heat, water, frost or other condition which may damage the electronics or mechanical parts. When transported or unpacked, it must be left to warm up to room temperature before operation.

SAFE PLACEMENT

The printer, filament and power cable must be placed so that it cannot be unintentionally touched or hooked by clothing. The printer is considerably heavy, extra care must be taken when moving. Do not use unnecessary force.

HOT SURFACES AND PARTS

To avoid burns and physical damage do not touch the printer's head and heated print bed unless they have cooled down to 40°C or less. These temperatures are displayed on the built in touch screen display and in the printing software interface. If in doubt, double check. Keep in mind that the readings could be incorrect.

HOT PRINT BED

The glass print bed can be heated up to 120°C. Remember that adjacent metal surfaces can be warm. When at room temperature, print bed can be sprayed lightly with non-flammable consumer plastics or glass cleaner and then wiped off with a clean wipe.

HOT PRINT HEAD

Print head heating block and nozzle can be heated up to 300°C. Keep in mind that surrounding parts can be hot.

HOT MELTED PLASTIC

The melted plastic extruded from the nozzle can reach near 300°C. Care and caution must be taken when operating.

WARM PARTS

The printer frame and outer parts can be warm during operation, reaching temperatures up to 60°C after extended use.

CLEANING

DO NOT USE FLAMMABLE LIQUIDS TO CLEAN HOT PARTS.

Use dry paper towel to clean print bed and print head. If more thorough cleaning is needed, the printer must be turned off and cooled down to room temperature.

MOVING PARTS

The printer mechanics consist of moving parts while operational. Care must be taken not to touch them during operation.

ELECTRICAL SAFETY

Do not place the printer where it could come in contact with moisture, especially near open windows. Do not leave the printer unattended while it is operating. ALWAYS USE GOOD QUALITY CABLE WITHOUT ANY SIGNS OF DAMAGE OR MALFUNCTION. DO NOT USE WORN ELECTRICAL SOCKETS. PLEASE ENSURE THAT THE OUTLET IS PROPERLY GROUNDED

STATIC ELECTRICITY

Release your body's static electricity by touching a grounded object before any repair or service operation. POWER MUST BE OFF AND CABLE UNPLUGGED BEFORE ANY WORK.

TOXIC FUMES

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. The odour, toxicity and intensity of fumes vary with the type of material used.

NOT A TOY FOR CHILDREN

The printer must not be left unattended with children. Children must always be accompanied and supervised by an adult when using or observing this product. Do not leave the printed objects, waste filament and tools unattended in the reach of small children to avoid harm and choking hazard. Be careful when tooling printed objects.

Welcome

Thank you for your purchase of the Mass Portal® Pharaoh® XD® 3D printer. We hope that it will serve you well and help you create many beautiful and useful things.

Getting started

Please take your time and read through the Getting started chapter of this manual – it will introduce you to some of the terms you will stumble upon when printing and generally give you a good overview of the printing process and printer operation. Basically, it's a bird's eye view on how it all works and at the same time a step by step guide to your first print.

Safety

Please read the safety chapter. Mass Portal printers are designed to be safe to use, but you need to have a general understanding of it's safe operation.

User manual

Please familiarize yourself with this manual before connecting the printer to your computer, installing the Simplify3D software and tinkering with settings.

Successful prints

The manual is designed to guide your attention to the important steps and settings first, so you can get great results faster. It should cover most common use cases and settings, and enable to get reasonable quality results when printing common objects with standard materials. If that works for you, leave the advanced printing for later experimentation.

Difficult prints

If you are about to print complex objects or have special requirements for the printed object properties, you will most likely have to experiment a bit until you find the perfect settings for your objects and the particular material. You may need to consider using a particular print material for your object that will deliver the best results.

Experimenting

Once you start printing, you will want to tweak this or that. Of course, 3D printing is all about experimentation and going where no one has gone before, but the possibilities can be intimidating at first. This manual is NOT designed to cover all possible printing situations, but rather to help you start along the path. Discussing every last software setting is beyond the scope of this manual as well — the best places for that are online forums, local FabLabs or 3D printing meet-ups.

Printing Material — 1,75mm filament

Mass Portal 3D printers are designed to work with print material from any vendor — 1,75 mm thermoplastic filament — as long as it's a good quality product, with uniform diameter. We try to find and recommend the best materials on the market, since the materials play the most important role for quality printouts.

Software

Mass Portal 3D printers ship with Simplify3D — a computer program that you use to prepare models for printing and control the printer when printing from a computer. Simplify3D can be used with Windows, Mac and Linux systems. Simplify3D is licensed commercial software, that is registered to your e-mail address. You can also use it with another 3D printer you own.

Maintenance

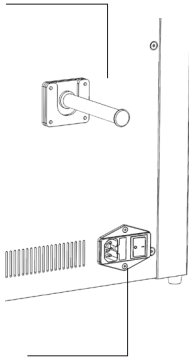
Mass Portal 3D printers are known to work for extended durations without requiring attention and are actually designed and built for just that. Please follow the advice in the chapter on printer maintenance to keep your printer in top shape.

Support

Whenever you need support for your Mass Portal 3D printer, please send an email to **support@massportal.com**. We do our best to answer promptly.

Printer Setup

INFO Filament spool holder is designed to be compatible with different form factory filament spools from various manufacturers.



INFO Power switch, power cord connector and fuse panel.

1. Unbox the printer - using the included instructions

Be careful, the printer is heavy..

2. Let the printer warm up to room temperature

This is important — especially in cold weather. In winter, it may take a long time, don't rush it.

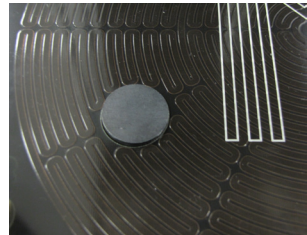
While waiting unbox and inspect everything — the power cord, the USB cord, filament spool holder, sample filament roll.

WARNING If the printer does not warm up to room temperature before operation there could be electrical issues due to condensation or there could be calibration issues

3. Install glass print bed

Align the thumbscrews with the gaps on the glass plate. Gently push down the bed and turn the glass plate clockwise.

WARNING When installing the glass bed, for MP20, make sure that the metal sensor discs for calibration are in their proper position. If they are not in the proper position, the automatic calibration process will not work properly. If there are any questions, please contact support.



4. Attach the filament spool holder

It simply slides in to the filament spool holder bracket on the back side of the printer.

5. Browse the manual and read the Getting Started chapter

6. Connect the power cord to the printer, plug it in and turn on the printer.

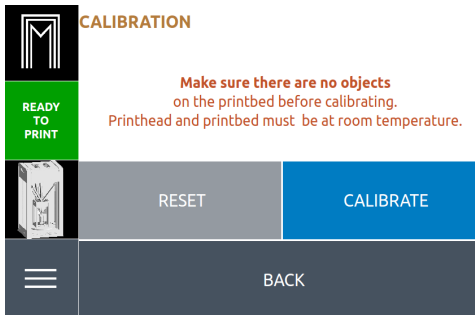
The power switch is on the back of the printer.

7. Calibrate the printer.

After turning on the printer it is necessary to calibrate the printer to make sure that the bed is level. (Before starting make sure the print bed is clear of any objects)

Calibration process for MP20

1. From the home screen select “**Settings**”
2. Select the “**Calibration**” option
3. Select “**Calibrate**” and the printer will go through the automatic calibration process
4. The printer is calibrated and ready for printing!

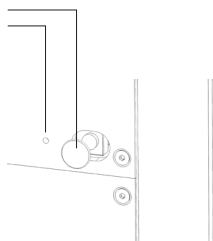


Calibration process for MP30 and MP40

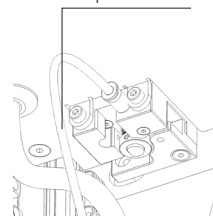
1. From the home screen select “**Settings**”
2. Select the “**Calibration**” option
3. Make sure that before starting the calibration process that you have the calibration coin prepared.
4. Select “**Calibrate**”
5. When the calibration process begins, the print head will slowly lower to three locations on the print bed. When the head starts going down, place the coin on the build plate under the inductive sensor for the printer to take a measurement.
6. The printer will run a verification process so you will need to repeat step 5 one more time.
7. The printer is calibrated and ready for printing!

My First Print

INFO Insert filament into this hole on the back side of the printer. You need to firmly press and hold the filament release lever to do so.



TIP You should see the filament moving down through the bowden tube when inserting it. Firmly push the filament in until it stops.



The printer comes with G-codes already loaded to the internal memory, so you can print a test print without connecting to a computer.

1. Prepare the printer

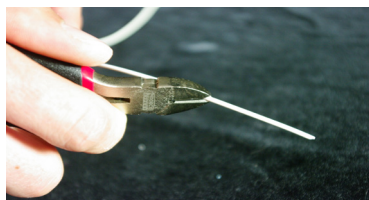
Make sure there are no objects on the print bed. You may keep the door closed or open for the first test print. As the printer is shipped with PLA, the side walls should be removed from the printer.

2. Place the sample filament spool on the holder

- Unbox the bundled filament
- Place the spool on the holder on the backside of the printer.

3. Prepare the printbed for printing

-Place the BuildTak on the printbed. Slowly removing the white film on the bottom and gradually stick the BuildTak on the bed trying to avoid making air bubbles underneath.



TIP It is easier to insert the filament if you cut the end of the filament at an angle using the clippers included with the printer (see image)

4. Insert the filament into the extruder

- Firmly push the filament release lever on the back side of the printer
- Insert filament into the filament hole besides the release lever. (You should see filament moving through the bowden tube.)
- Push in the filament until it stops.

5. Start printing:

- Load a gcode to the printer using a USB flash drive or IP/TCP or Wifi connection
- Select the **"Ready to Print"** icon on the right side of the screen
- Select **"Object to Print"** and find the file you loaded to the printer
- The material should be set to PLA and the bed coating set to BuildTak
- Select **"Start Printing"**
- The printer will heat up and start to print when ready.

6. Watch it print

DO NOT touch the moving parts or print bed while printing, both print bed and print head will be hot when printing.

7. Let it cool down for a while after printing

Wait until the lights below print bed are yellow or white.

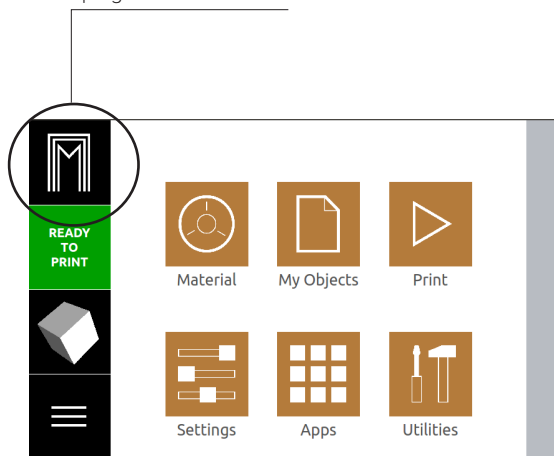
You can also see the temperatures on the screen.

You can remove the filament while the print head is still hot.

8. Remove the print from the print bed

Use the supplied tool to gently remove the object.

TIP If you ever want to return to the main screen press the Mass Portal logo in the top right corner of the screen.



TIP To remove or change the filament, the print head must be heated or still hot after printing.

TIP You don't need to remove the filament after each print.

TIP If you are having problems removing the print, you can remove the glass print bed to help remove the print

Printing process overview

Mass Portal® 3D printers use process called Fused Filament Fabrication or FFF — printing objects out of melted plastic layer by layer.

How it works

Thermoplastic printing material (the filament) is melted in the 3D printers printing head, and pushed through a nozzle. This process is called extrusion. The nozzle moves according to the geometry of printable object, depositing the melted plastic and drawing the object layer by layer. When a layer is finished, printing head moves up a single layer and draws the next layer of melted plastic upon the previous, that has now somewhat hardened, until the object is finished. The layers of plastic bond or melt together, forming a single finished object.

Slicing and G-code

The movements of the nozzle, the temperature settings of the printing head and of the printing bed are controlled by computer instructions called G-code files. G-code is generated from 3D models imported to software programs called slicers. The process of generating the G-codes is usually called slicing. Mass Portal uses the Simplify3D program for slicing and printer control.

Although G-code is used to control many 3D printers, it generally must be created for each particular printer. You can store the generated G-code files for later use on the computer or in the internal memory of the printer.

Standalone printing and host software

When using the printer in standalone mode, printing G-code files prepared earlier and in the printers internal memory or USB flash, no computer is needed to operate the printer. The printing is controlled by the using built-in software, Pharaoh Touch™. This can be useful for longer prints when you don't want to leave your computer connected to the printer, or when what you do on the computer might interfere with the printing.

In other cases it makes sense to control the printing process from a computer using so called host software. It also allows control of the printing process, displays printing progress visualization, temperature graphs and more. The printer is shipped with Simplify3D, but the printer works with other programs as well.

STL or OBJ files

G-code is usually generated from 3D models, that are saved in the STL or OBJ format. Most 3D modelling software today can export to these formats, and printable files on the Internet are also usually found in these formats.

TIP Test and sample files for the bundled material are provided on the printers internal memory

3D models

You can generally print 3 kinds of models:

- Printable 3D models downloaded from the Internet — either free or commercial
- Objects, that are scanned on a 3D scanner and prepared for 3D printing
- Your own 3D models, designed in any 3D drawing, modeling or CAD software, that can export to a printable 3D format such as STL or OBJ

In all cases it is important that the 3D object geometry forms a solid, dimensional object, that would be possible to print — you cannot print objects like infinitely thin walls for example, although you can design such objects on the computer. Sometimes the STL or OBJ files to be printed can be repaired with the built-in tools or host or slicing software, and there are also software applications just for that purpose.

Infill and perimeters of walls

You can choose whether the printed object will be hollow, filled or partially filled when preparing the model for print with slicing software. Some software allows for smart infill, placing it only where necessary. Wall thickness is also set before printing.

Support

It is impossible to reasonably print material, that is “hanging in the air” — not at least partially supported by the underlying layer. You can generally print slopes (overhangs) about 45°, when the underlying layer supports some of the next layer’s plastic. The bundled Simplify3D software allows to place supports manually and to remove automatically generated supports.

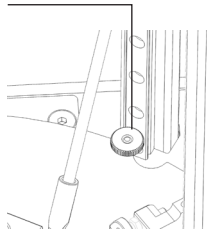
Print Bed Coating and Heating

WARNING The correct bed coating must be selected on the printer, so it “knows” how to print the first layer, because accounts for the thickness of the coating. (see page 26)

TIP The print bed must be kept clean. If prints do not stick after multiple prints on the same coating, try cleaning or replacing the coating.

TIP If you need to adjust the setting for the BuildTak™, use the “Custom” option and set the thickness to 0.4 - 0.55 mm.

TIP The print bed is replaceable. The glass surface can be removed or installed, by gently pressing it down and turning the glass plate so it can be lifted. You do not need to undo the thumbscrews holding the glass in place to remove or install it.



WARNING Do not remove the print bed while it's hot. Let it cool down.

Mass Portal printers are equipped with a glass print bed. Although you can print directly to the glass surface, some materials work better on a different coating. Work better means the material will adhere firmly or “stick” to the print bed while printing, but still come off easily after printing.

Even surface

Glass is the most reasonable choice for print surface, because it's straight and level, even when heated.

BuildTak™

BuildTak™ is the recommended coating and commonly used at Mass Portal. You received some BuildTak™ with your printer. It works with most materials with very good results.

Kapton and PET tape

Kapton and PET tape are quite durable, and can be cleaned between prints.

Blue painters tape (masking paper tape)

Blue painters tape is the material of choice for printing PLA, especially when printing without heated bed. Sometimes, material may stick too well to it and the objects may be difficult to remove.

Other coatings

Some materials can be used with different coatings. There's a Custom menu option for that.

Glue stick

Glue can be used to get difficult prints to stick — for example, usual office glue stick. You should apply the glue before heating up the print bed. Using a glue stick will require extra cleaning and maintenance.

Print Bed Calibration

Bed coating will be taken into account when calibrating the print bed — as the different coatings have different thicknesses, the correct coating must be selected. This can be done before or after the calibration. You can change the bed coating setting as often as needed. There's no need to recalibrate after changing the bed coating if the first layer prints correctly.

Print Bed Leveling / Calibration

The print bed on Mass Portal printers is held in place with three thumbscrews. The slant and height of the print bed can be adjusted by turning the thumbscrews. As a general rule, the print bed should not be slanted — the thumbscrews are used to keep it level. Normally the thumbscrews would not need adjusting by more than 1mm or 2mm at max.

Mass Portal printers have a built in sensor, that can measure and adjust to the print bed accordingly. This is done to ensure that first layer of prints is correctly printed — that the nozzle is not too high (material not sticking to the print bed) or too low (nozzle too close to the print bed, material “smeared” into print bed).

TIP The print bed can be leveled manually by adjusting the three thumbscrews that hold the glass print bed. This can be done after calibration, if minor adjustments are needed.

WARNING The thumbscrews **MUST NOT** be screwed in till lowest possible position, but it is normal for the screw to stick up a little bit (see picture). The print bed must have some play, so the springs below it can compress — you must be able to move print bed down a bit when pressing on it. This is a safety measure in case if the print head moves too low.

TIP It is not necessary to use calibrate the print bed before each print, or even regularly. If the first layer of the print is correctly printing, there is no need to recalibrate the printer. The printer can go for a long time before needing to be recalibrated.

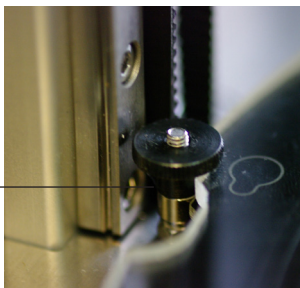
Calibration Process

Please go to the chapter about ‘Printer Setup’ (Page 8) to see the calibration process for your printer.

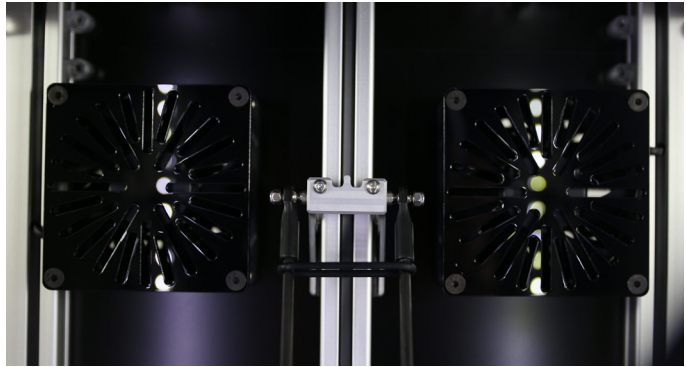
TIP The sensor plates in the print bed are not attached, so it is normal for the sensor plates in print bed to move.

WARNING To ensure correct measurements, the calibration process works **ONLY** when the printer is at room temperature. If you attempt using the auto calibration when the printer is hot, you’ll see a warning and the calibration will not be done.

WARNING The print head will move down when calibrating to make the measurements. Make sure there are no objects, hands, or previous prints on the print bed before performing the calibration process, otherwise the print head can damage them.



Chamber Ventilation



The chamber ventilation system has been created for safe, high quality printing. With safety as the system's main purpose, the chamber ventilation will make your printing experience worry free.

Safety

When printing using the chamber ventilation system, you can always leave the windows and door on the printer. This will help prevent any foreign objects from going into the printing area. Also it will help the printer to be safe around any age group as children will not be able to put their hands in the printing chamber while the printer is in operation.

The fans also include filters to clean any fumes from the print chamber. Studies have shown that some printing materials, such as ABS, can release toxic fumes while printing. With the use of the Mass Portal ventilation system, these fumes are filtered from the air creating a safe work environment.

Material

Though it is not suggested to print some materials with an enclosed chamber, such as PLA, the ventilation system has different settings to allow the printer to print any material in an enclosed chamber.

Constant Chamber Temperature

While 3D printing, it is also important to keep the printing environment at a constant temperature. With the heated bed the chamber can be warm, which is useful for printing materials such as ABS, Nylon, or Polycarbonate. (See table)

Chamber Temperature		
*Approximate temperatures depending on which ventilation option is used		
MP20	MP30	MP40
50°C	30°C	25°C

Ventilation Settings

To use the ventilation system properly, there are few different options available to use for multiple purposes

Circulation

Circulation is used for materials the generally do not need a warm print chamber. Material such as PLA, PLA composite materials, XT, flexible materials, etc. The circulation creates an airflow through the print chamber so that the chamber will not become too warm for the printed part.

Filtration

Filtration is used for materials that will possibly emit toxic fumes while printing.

Slow Extraction

The slow option should be used while printing to filter the air during the print without creating too much of a cooling effect in the chamber

Fast Extraction

The fast option should be used when the print is finished so that all of the air in the chamber can quickly be filtered. Once the air is filtered, you can open the front door and remove the finished print.

Off

Use the "Off" option when the material you are using would benefit from a heated chamber, materials such as PC. Can also use the filtration options for such materials as well.

Standalone Printing



Pharaoh Touch™ has been designed to make the printing process more convenient to you. You are able to control your printer easily with the use of the touch screen.

WARNING The SD card should **NOT** be removed from the printer unless instructed to by Mass Portal support. To load gcodes to the printer use a USB flash, TCP/IP, USB cable, or Internet connection.

When to print in standalone mode

- When printing takes a long time for larger objects, independent from the computer
- When you repeatedly print the same object, when you know that all the settings in your G-code are just fine
- To save electricity by not having the computer turned on
- When taking the printer to another location like shows or meetings

Power management

The printer will cool down when print is finished (depending on settings in g-code file or after an hour of inactivity), but it will not turn itself off.

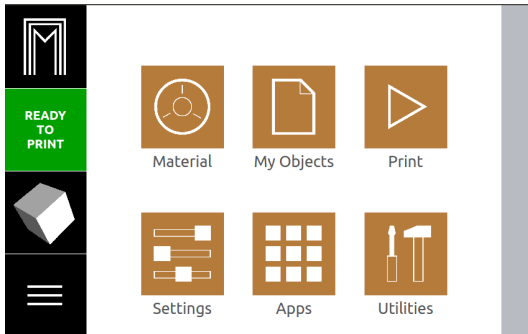
Standalone printing process

- Select the “Ready to Print” icon on the right side of the screen
- Select “Object to Print” -> Locate the file you want to print -> Select for Printing”
- Select the material you will be printing with
- Select the proper bed coating that is currently on the printer
- Select “Start Printing”
- The printer will heat up and start to print when ready.

Pharaoh Touch™

Main Screen

Every time you select the Mass Portal Logo in the top left corner of the screen it will take you to the Main Screen. From the Main Screen you will be able to navigate to any desired menu from the main screen.



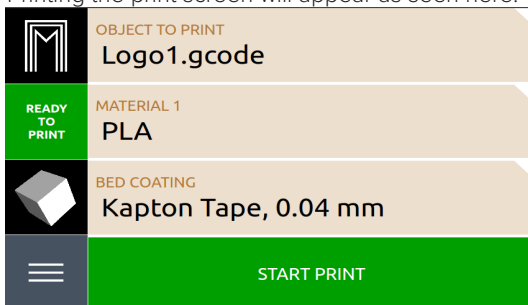
INFO The context menu, located in the bottom left corner of the screen, will give you extra options, if available, for the current screen.



Main Screen Context - Reboot or Shutdown the printer

Print Screen

You can get to this screen by pressing “Ready to Print” on the left side, or the “Print” tile on the main screen
Before Printing the print screen will appear as seen here:



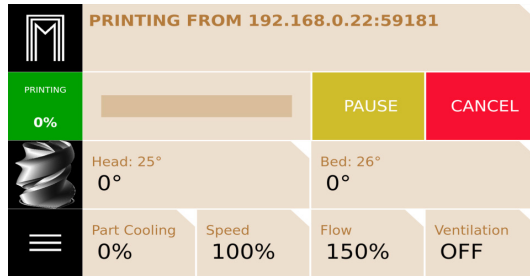
Object to Print- Select the object you would like to print

Material - Select the material with which you are printing

Bed Coating - Set the current bed coating that is on the print bed

Start Print - When everything is ready, start the print

TIP Fan speed percentage is estimated, and fans will not work all the way down to 0 percent.



The print screen will look like this while the printer is in operation:

Printing: The model you are printing

% Complete: Percent of how much of the object has been printed

Head: Big numbers: Set temperature
Small numbers: Real temperature

Bed: Big numbers: Set temperature
Small numbers: Real temperature

Fan %: Can adjust the fan speed

Speed %: Can adjust the speed of the print head

Flow %: Can adjust the flow of the material

Ventilation: Can adjust the ventilation process depending on what material you are using.

Pause: Can pause the print

When the pause button is pushed the option “Continue Printing” will appear when you are ready to continue the print.

Cancel: To cancel the print if needed. When pushed, you will be asked to confirm the cancel or go back, so you do not accidentally stop the print.



Material

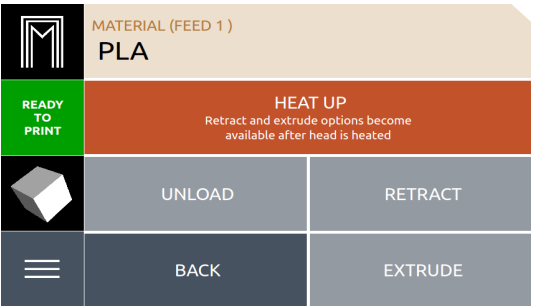
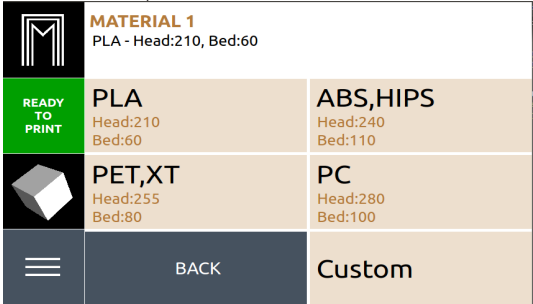
You will be directed from the Print screen to this screen to select the material with which you would like to print. From the Main Screen you After choosing the desired material, you will brought to the material control menu to prepare the nozzle and material for printing.

Heat up: Heat up the nozzle to prime the extruder

Retract: Pulls the material away from the nozzle

Extrude: Pushes the material into the nozzle

Back: Returns to the previous screen

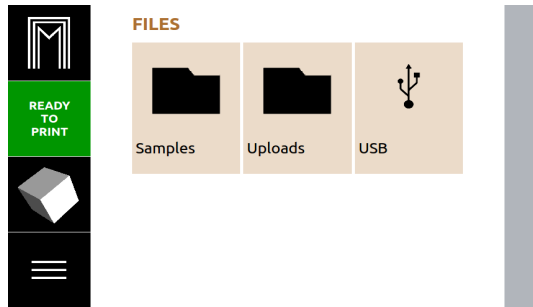


My Objects

In the Objects menu you will find all of the G-codes that you have prepared for printing and loaded to the printer or USB drive. On this screen you will select the file that you would like to print. If the G-code is located in a folder, you will select the folder where the G-code is located and then select the G-code you would like to print. If the G-code is located in the USB drive, select the USB icon, then select the G-code you would like to print.

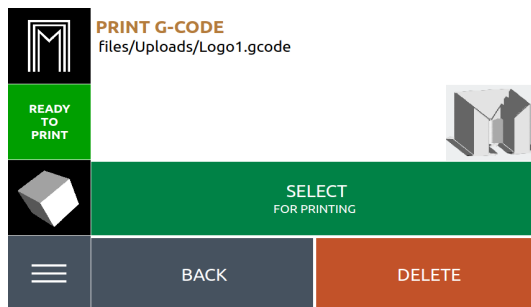
TIP Print head must be heated to remove / change filament. You need to heat up to the corresponding temperature to remove a particular kind of filament. Example: If you have been printing PET and need to remove it to change to PLA, first heat up for the PET (higher temp) to remove it, then change to the lower printing temperature for the PLA. You usually do not need to fully reach the target temperature to remove the filament.

TIP Printer hotend temperature is measured in the metal heating block. The actual temperature of melted filament in the nozzle is about 10–20°C lower, however, this is the temperature that is commonly shown on the filament package (print temperature, usually a range). Make sure you add the difference to the print head temperature — for example, if the printing specification for the material is 200°C, the print head should be set at about 215°C.



Select Object

When choosing a file, you will be directed to the select object screen to verify your selection. If it is not the object you wanted you can press the back button to return to the previous screen. Or if you would like to delete the object, you can do that as well.



Objects - Context

Unmount All: Unmount the USB from the printer to safely remove the USB drive

View as tile: View folders and files in tile format

View as list: View folders and files in list format

Delete this folder: Deletes the current folder you are using

Re-render all file thumbnails: Re-renders all the G-codes in the printer internal memory

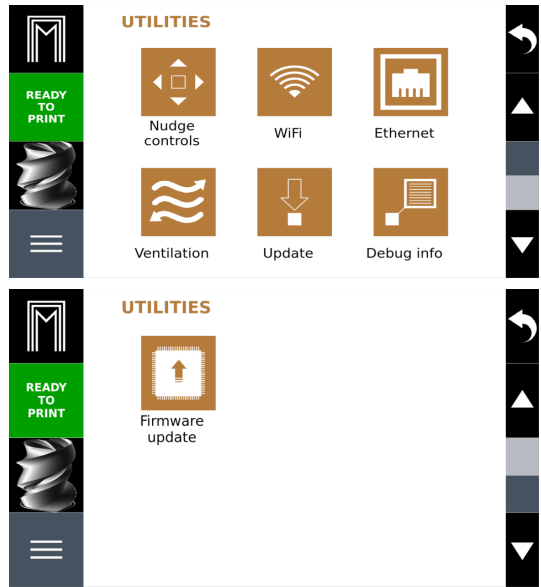
Render new file thumbnails: Renders images for only the newly uploaded files

Apps

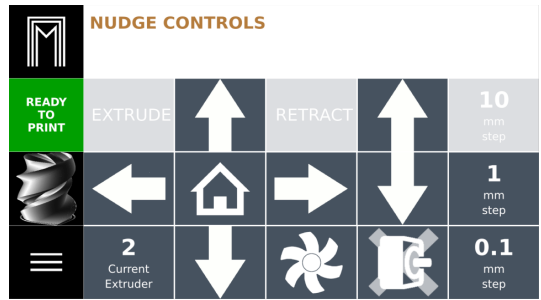
Currently there are no apps, but they will soon be there to help improve your printing experience.

Utilities

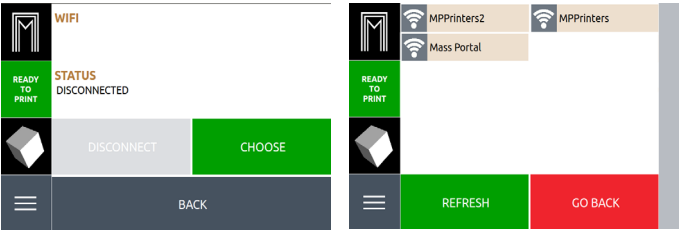
The utilities menu offers different tools to help improve the use of your printer.



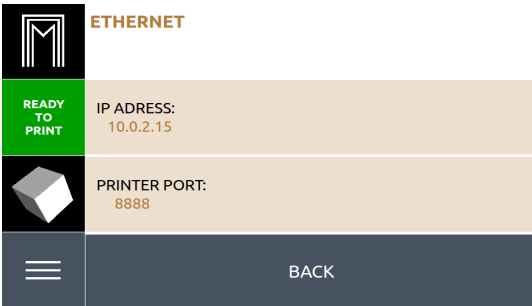
Nudge control: With the nudge control panel you can move the print head in the X, Y, and Z direction in steps of 10, 1 and 0.1 mm, enable or disable stepper motors and fans, and extrude or retract with all extruders.



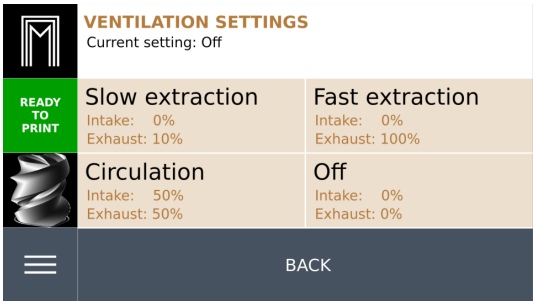
Wifi: Connect to the local wifi network. You will be directed to choose the desired network and type in the password, if needed.



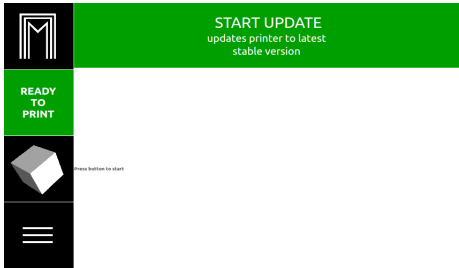
Ethernet: If connected to an ethernet cable, the screen shows the IP address so you can connect to the printer using TCP connection



Ventilation: Change the different ventilation options depending on what material you are using (See chapter on Ventilation)



Update: Update the Pharaoh Touch™ software to make sure that your printer is up to date with the latest Pharaoh Touch™ software



INFO Please update your printer at least once a month.

WARNING: In the “Update” context menu there is an option “Nightly Update.” DO NOT select this option unless instructed by Mass Portal support, It could cause problems with your printer.

Nightly Update

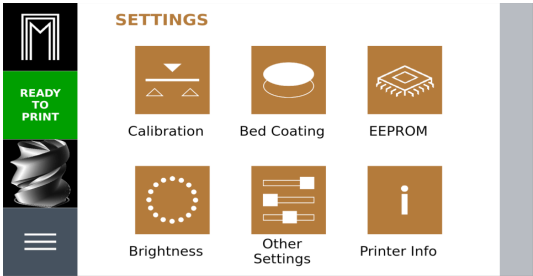
Debug: Shows communication information during the print. You will not need to go to this screen, unless instructed to by Mass Portal support.

Firmware Update: The Firmware update will automatically happen with a new software update, when available. The Firmware update screen should not be used unless instructed by Mass Portal support.

WARNING If you do view the Debug info, pressing the “Exit App” button will reboot the printer. If you would like to exit the Debug Info screen, press the Mass Portal logo.

Settings

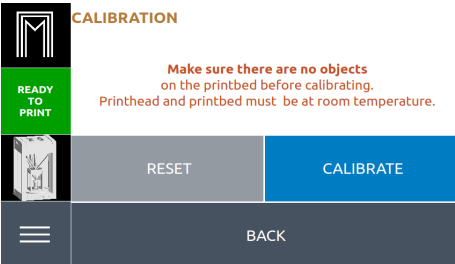
With the settings menu you can adjust the printer settings so that it will print the best quality possible.



Calibration:

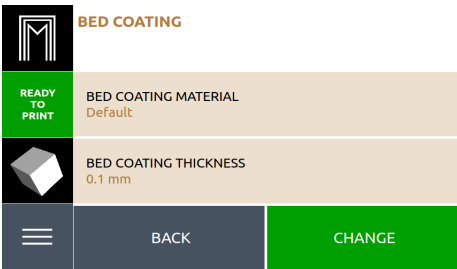
Reset: If there was some problem with the previous calibration, notice that the bed was slanted, you can reset the calibration information

Calibrate: Start the calibration for your printer



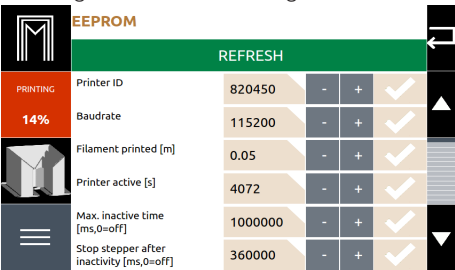
Bed coating:

Can set the bed coating to the presets using the “Set bed coating material” or can set a custom thickness by selecting the “Thickness” option.

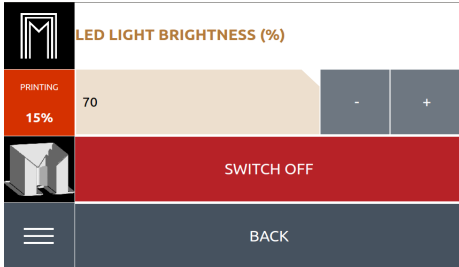


NOTE Be sure to click the “Save” button after changing setting in the EEPROM Data. The check mark will turn to a Save button when something is changed.

EEPROM: You no longer need to connect to a computer to edit the EEPROM settings. Here you are able to edit all EEPROM settings using the +/- buttons or by selecting the number and typing in the desired value. You can use the arrows on the right side to scroll up and down. The button in the top right of the screen is used to go back to the settings menu.

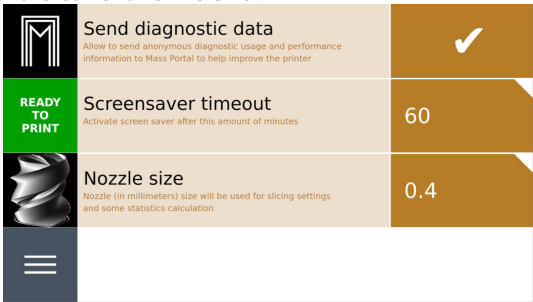


Brightness: You can change the brightness of the LED lights by using the +/- button. When the value is changed, there will be a small pause and then the bed will display how bright the lights will be in different colors. You can also turn the lights off and on using the On/Off button. The “Back” button will take you back to the settings menu.

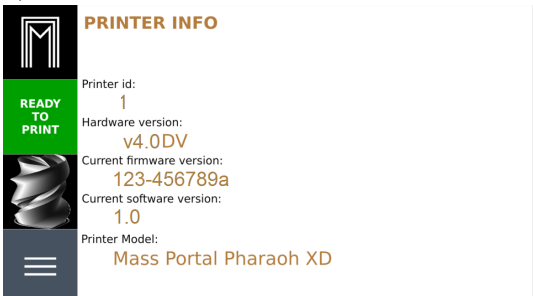


WARNING Do not change the LED lights while printing, it could cause the printer to pause for a moment affecting the surface quality.

Other Settings: Can choose to send data to Mass Portal to improve printing experience, adjust when the screen saver will be activated, and change the nozzle size so that the printer “knows” the current nozzle size.



Info: Contains all basic information about the printer. If you are writing support about an issue, please include the ID, Hardware, firmware, and Software versions.





Software: Simplify3D

Downloading process for Simplify3D:

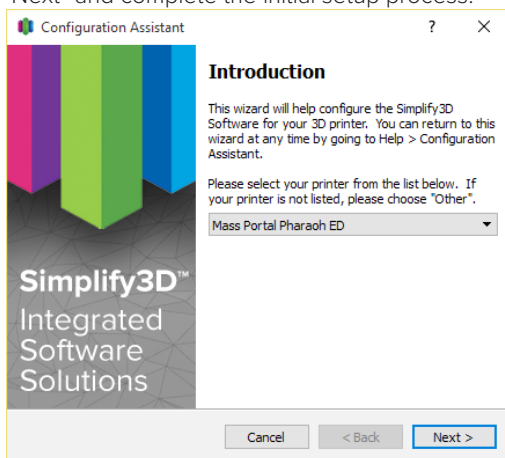
1. Find the Redemption Card you received with your printer
2. Go to the website found inside the card, Simplify3D.com/redeem
3. Fill in the information and redeem the code
4. Follow the downloading instructions

Setup

1. After you complete the installation of Simplify3D, please choose the “Mass Portal Pharaoh ED” from the printer models pop-up menu.
2. Click “Next” and complete the initial setup process.

WARNING The “Bed Leveling Wizard” does not work with the Pharaoh® printer. Do not try using it, it could damage the printer.

INFO The settings in Simplify3D for the Pharaoh ED will work for the Pharaoh XD as well.



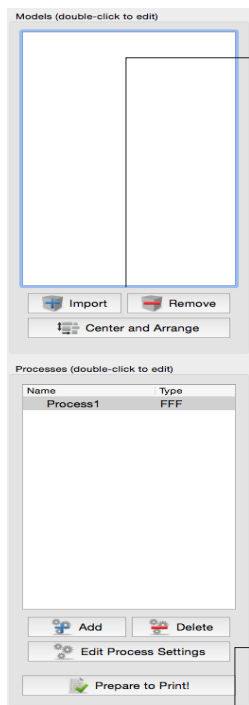
3. For MP30 and MP40, when Simplify3D is open select File>Preferences> Machine Tab and input the following information for your printer:

For MP30:

Machine type	Delta robot (cylindrical build volume)		
	X-Axis	Y-Axis	Z-Axis
Build volume	210,00	210,00	300,00 mm
Origin offset	105,00	105,00	0,00 mm
Homing direction	Center	Center	Max
Flip build table axis	<input type="checkbox"/> X	<input checked="" type="checkbox"/> Y	<input type="checkbox"/> Z
Toolhead offsets	Tool 0	X 0,00	Y 0,00

- For MP40:** X and Y Build volume: 280, Z Build Volume: 400
X and Y Origin offset: 140, Z Orgin offset: 0
4. Click “Ok” and the setup is complete

The Printing Process with Simplify3D



1. Importing models

1. After completing a 3D model and saving it as an STL or OBJ file, or downloading an STL or OBJ file from the internet, you need to import it to Simplify3D.
2. Click the "Import" button
3. Find the desired file, and click Open
4. You can import more than one model to print multiple objects

2. Manipulating your models

1. Your model(s) will be centered when imported into Simplify3D, but sometimes will not be arranged how you would like
2. To manipulate your model(s) you can double click on your model and you will see the "Model settings" window.
3. Here you can rotate, scale, and relocate your object very precisely to you desired arrangement.

TIP You can also drag you files into the printing area instead of using the import button.

TIP After your rotate your model(s), for example, they might be above or below the bed. Click the "Center and Arrange" button and it will move it back to the middle.

TIP Here are some short cuts for manipulating your models:

Move Model:

Ctrl-left-click and drag

Scale Model:

Ctrl-right-click and drag up or down

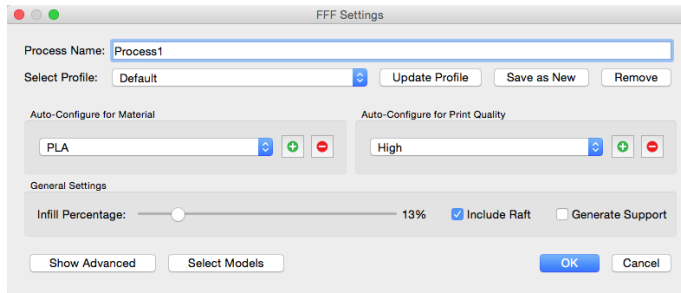
Rotate Model:

Alt-left-click and drag right or left

3. "Edit Process Settings"

1. Before you can slice your model, you need to give Simplify3D the desired settings you want for your print.
2. Simplify3D comes with some basic preset settings which are very easy to use,
3. Choose the material you are using
4. Choose the desired quality (Fast - higher speed lower quality, High - slower speed better quality)
5. Choose infill percentage (How "solid" you want your print. 0% hollow, 100% is solid. 20%-50% is a normal range of use)
6. Check Raft if desired. A raft is a structure that will be printed to help ensure your print will "stick" to the print bed.
7. Do you need supports? As mentioned in the printing process overview, you cannot print well "in air" without adding supports. If you have any overhangs it is suggested to use supports.
8. You can now click OK and the process is ready.

TIP Typically, a 45° or greater overhang will benefit from support material underneath it. With Simplify3D you can customize your supports, with add and delete support buttons.



TIP If you want more information about any particular setting, hover the mouse cursor over that element to display a tool tip describing the functionality.

Preparing to print

1. Now that your model is ready, you can click the “Prepare to Print!” button in the bottom left corner
2. Now you will see Simplify3D in “Preview mode”
3. You will see different “Build Statistics” that will tell the estimate of how long your print will take, amount of material used, or how much the part will cost to make with a certain material.
4. If you are connected to your printer with a USB cord, then you can click “Begin printing over USB” when you are ready to print.
5. If you would rather use the SD card, you can save your G-code to the disk and print in standalone mode.

Resources within the software

Once you start using the advanced settings, or just need some help, visit Simplify3D.com where you will find a wide array of support materials.

- Tutorials
- Hardware Setup Guides
- User Forum
- FAQ

Printing with Dual Extruder

Material

When printing with your dual extruder, you need to make sure that the materials you are printing can print at the same temperature. The Mass Portal dual extruder only uses one nozzle, so will be operating a one temperature. You can print with two different colors, standard material with support material, or even different materials if they use similar temperatures (Such as ABS and HIPS).

Preparing Material for printing

When preparing the printer to print, using the dual extrusion capabilities, you need to make sure that the material is properly aligned at the beginning of the print. **When inserting the filament, watch it as it goes down the bowden tube and align it approximately 5 mm above the grey coupling. (see Picture).**



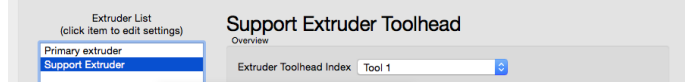
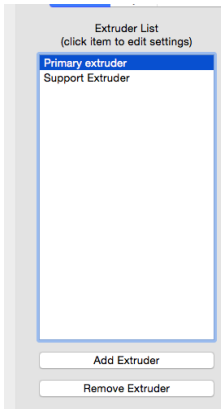
NOTE This is only needed for the first time or if you have changed the materials. If you are printing with the same materials then you can simply start the new G-code.

Simplify3D Settings

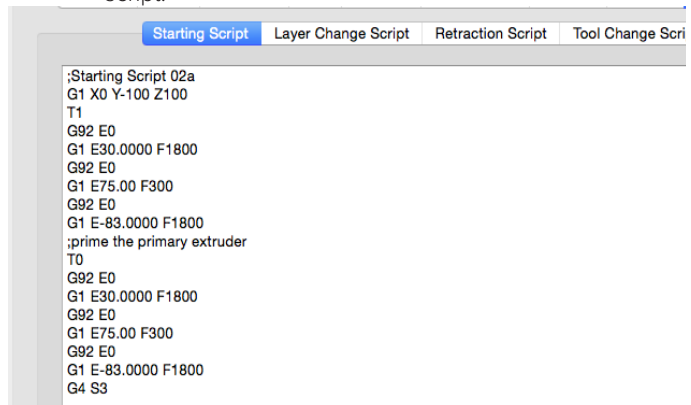
We are working with Simplify3D to get a default dual extrusion profile prepared for more convenient printing, but until that time, please follow the following steps.

Dual Extrusion for Support:

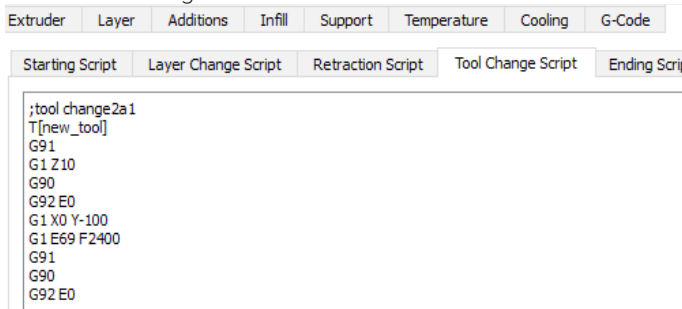
1. Import the model you would like to use
2. Open the “Process Settings” window to edit the settings and show advanced settings
3. Add another extruder in the extruder tab for supports. Make the Primary extruder “Tool 0” and Support extruder “Tool 1”



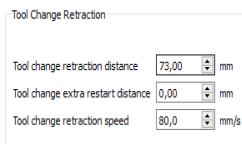
4. Go to the scripts tab:
 - a) In the “Starting Script” tab, write the following script:



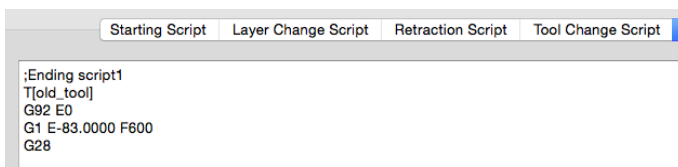
b) In the “Tool Change Script” write the following



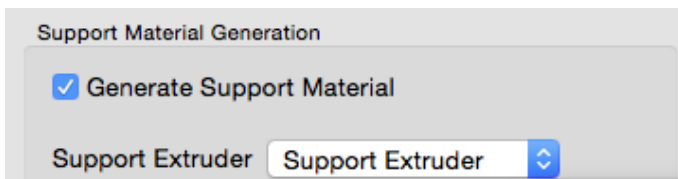
INFO You will also need to go to the “Advanced” tab to make sure the “Tool Change Retraction” is also set to the proper value. (see image)



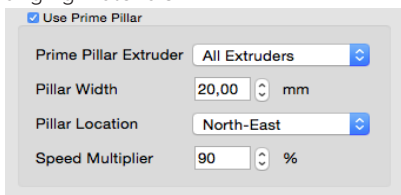
c) In the “Ending Script” tab write the following



5) In the “Support” tab make sure that the support extruder is properly selected



6) In the “Additions” tab select “Prime Pillar.” This will be used when changing materials



INFO You will need to adjust the “Pillar Width” size depending on the colors you are using. For some colors it will need to be larger and others can be smaller.

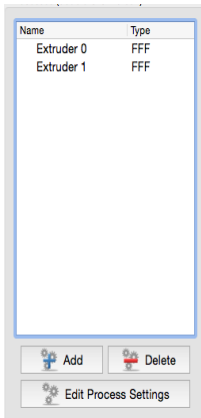
7) Prepare the rest of your settings according to the object and click “Prepare to print”

Dual extrusion for Multiple Colors:

Printing with two colors will be similar to printing with support, but when printing with two colors, you will need to use two processes.

1) Add a new Process on the left side of the screen.

2) When editing Process 1, make the “Extruder Toolhead Index” Tool 0 and for Process 2, Tool 1



3) Create the Scripts as shown previously

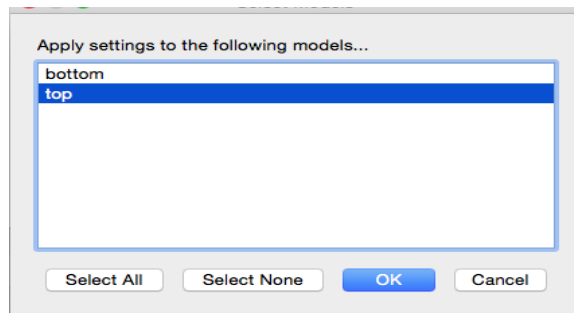
4) Select “Prime Pillar” in the Additions Tab and make sure that the “Prime Pillar Extruder” is set to “All Extruders.”

5) For each process you will need to select which model you would like printed with that process.

a) In “Process settings” click “Select Models”

b) Select the model you would like with that process

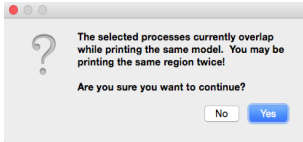
6) Prepare other settings for your object as needed



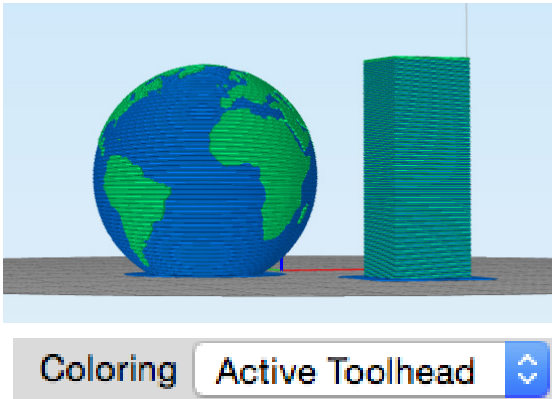
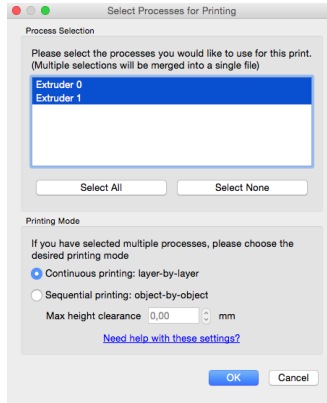
TIP You can rename the processes if you would like. First they will come up as Process 1 and Process 2

7) Click “Prepare to Print” and make sure that both processes are selected

INFO If there is a conflict with your settings or selection of processes with the part, Simplify3D will give a warning



8) To make sure that the G-code is prepared as you would like it to be, in “Preview Mode” on the left side of the screen there will be an option “Coloring.” Select “Active Toolhead” and it will show you which extruder is printing which part.



If you have any further questions about the dual extrusion set-up or process, please feel free to contact **support@massportal.com**

Printing Materials Overview

TIP If you're new to 3D printing, PLA Filament is a good material to start with because it's easy to use and performs well on most prints.

NOTE Materials from different manufacturers may differ. For example, what prints very well for one material might not print so well with a different material

NB. Printer hotend temperature is measured in the metal heating block. The actual temperature of melted filament in the nozzle is about 10–20°C lower, however, this is the temperature that is commonly shown on the filament package (print temperature, usually a range). Make sure you add the difference to the print head temperature — for example, if the printing specification for the material is 200°C, the print head should be set at about 215°C.

Each material has different properties that make it useful for different tasks. For example, nylon is both strong and non-reactive to many substances. Other plastics can have excellent optical clarity or can be easy to work with. PLA is considered the default 3D printing material for general purpose use, although ABS has been a preferred choice for prototyping and for usable printed parts.

PLA

PLA (Poly-lactic acid) is a bio-plastic material that has become the de-facto standard of 3D printing materials. It is bio-degradable, mostly easy to print, and has great aesthetic properties. Choose it for general printing of 3D objects. The post-processing of other than coloring is not that great though, because the material is rather hard. It will not last too long outside, and may deform in temperatures as low as 50°C. When printing, emits light pancake-like odor as it is made of corn. Can be printed without heated bed on blue painter's tape.

ABS

ABS is still the prototyping material of choice. It is the same plastic likely later used for injection molding, and the prints can be easily post-processed: sanded, painted, polished. ABS can be treated with acetone vapour to obtain very polished look. ABS emits unpleasant fume when printing. When printing any significant quantities, good ventilation is a must.

PET

PET based materials are among the most transparent available. PET is very tough when solid, and more flexible than PLA and ABS. PET objects can have some of the most shiny, intense surface color. PET is generally very predictable when printing, when you play by it's rules. Some PET materials are certified food-safe. To get the most transparent prints, maximum layer height for the respective nozzle should be used and relatively slow printing speed.

Nylon

Known for it's toughness, Nylon and Nylon-based materials are mostly used for parts that will be actually used for prototype devices. Another important property is transparency, allowing use of nylon for non-destructive evaluation.

HIPS

HIPS is a High Impact Polystyrol, the surface can bit or a lot more matte than ABS. Useful for medical visualizations, architecture and renovation work. Common applications of HIPS include toys and product casings. HIPS is usually injection molded in production. HIPS has very similar requirements to ABS for printer settings.

Flexible materials

Flexible materials can be a bit challenging to print with a bowden based system like the Mass Portal printer uses — but the Mass Portal Pharaoh is known to print even pretty flexible materials like thermoplastic elastomers, thermoplastic polyurethane and various kinds of flexibility-modified PLA.

Composite materials

Plastic filament mixed with metal (brass, copper, steel), chalk, wood, carbon fibre and other materials. Usually these filaments are based on PLA, but there are filaments based on PET and ABS also. Depending on the percentage of powder added, the physical and chemical properties of the materials change. These materials can wear out the nozzle quicker, are more challenging to print and have to be considered as “advanced” printing.

The most typical use for the composite materials is to imitate the look and surface feel of other materials. Polished prints of metal composites look like they are made from real metal and also are considerably heavier than regular plastic prints. Chalk infilled materials tend to look like sandstone or concrete, carbon fibre infill adds stiffness and lightness. The physical properties of the printed objects vary greatly.

Post Processing of Prints

These articles describes how to clean up and otherwise post-process 3D prints made with a 3D printer.

Prints may have imperfections, e.g. small parts that stick out and that you will have to remove. You can do this mechanically or by using fire, heat or a solvent.

Removing Support Material

Once the part is removed from the platform, gently remove the bulk of the support material used to build the part. Break away as much support material as possible with your hands or pliers. You can also use dental picks and brushes to remove excess material.



TIP A simple nail filer works very well (see picture)

Sanding, grinding, polishing

For example you can use “multi tool” for carving, grinding, cleaning, sanding, etc. Typical accessories are: Carving tools (to carve holes that you can’t print) Grinding/sanding tools, to remove imperfections or irregular bottom layers. Although for the latter, using a file is recommended.

Use cleaning/polishing tools, to remove small imperfections.

Using solvent

Vapor smoothing is a technique for improving the surface finish of 3D-printed parts. It uses acetone vapor to smooth the surfaces of ABS prints, removing the layer lines that result from the printing process.

Gluing

Glues for ABS

- Super glue

Glues for PLA

- UHU® POR, an “elastic” glue that is transparent and not very visible

Printer maintenance

To ensure long service life and correct operation of the printer, simple maintenance must be done when appropriate:

- after long periods of printing (approx. 100 printing hours);
- after long periods of inactive use or storage;
- once every three months.

Cleaning the print bed

The glass print bed is commonly coated with BuildTak printing surface, Kapton tape or Blue painter's tape when printing. Although these usually leave no traces, the print bed must be as even as possible. You should carefully clean it when suspecting that material buildup, from the self-gluing tapes or other coating materials, has happened. Especially if you print directly to the glass printing bed.

Use non-flammable cleaning spray and paper wipes. Plastic buildup can require mechanical removal, but care must be taken not to scratch the print bed or injure yourself.

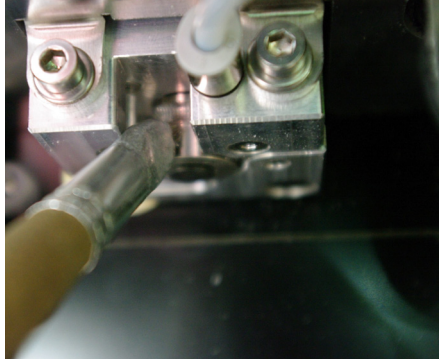
Cleaning outer side of the nozzle

Removing excess plastic from the nozzle should be done before each print. Cleaning must be performed when the nozzle is heated to about 200°C or more, depending on material. Do it quickly and carefully by using paper towel, folded a few times. Use no cleaning agents. Remember that the nozzle, the surrounding parts and especially the melted plastics are very hot.

Cleaning the printing chamber

The bottom of printing chamber must be cleaned from parts of printed objects and wasted filament before each print. Care must be taken so bits of plastic do not fall into drive belts. If that has happened, remove the plastic before operating the printer.

Cleaning the extruder drive gear - without removing drive gear



The extruder is located inside the printer chamber, upper left corner. The design of the extruder allows cleaning of the drive gear without disassembling it. The drive gear teeth can sometimes become filled with plastic parts from filament, when the print head has been clogged and the drive gear has “dug” into the filament.

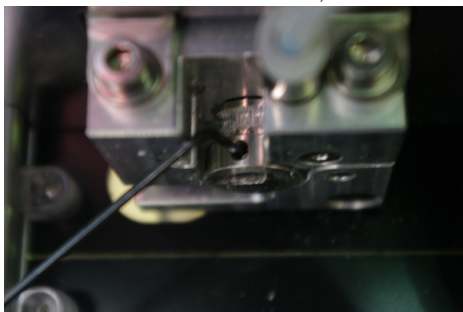
To clean the extruder

1. Remove the filament from the printer
2. Use hard brush or plastic tip to remove any stuck plastic from the drive gear.
3. When the area you are cleaning is clear from material, use the extrude on the control panel (see page 24) or using the printer control panel from your computer.
4. Continue turning and cleaning until the extruder is clean

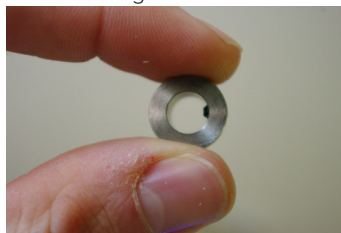
Cleaning extruder drive gear / replacing extruder drive gear

Removing the drive gear

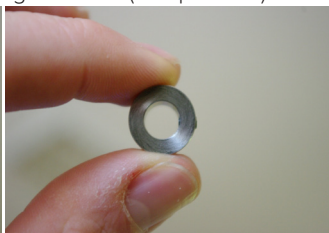
1. First, remove the filament.
2. Then turn the gear by hand so the fixing screw is visible
3. With a 1.5 mm hex head screwdriver, loosen it.



4. Push the black center button and the gear should come off.
5. When the drive gear is clean and ready to put back, make sure that the fixing screw is not blocking the hole. (see pictures)

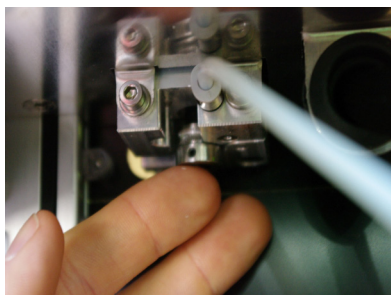


Incorrect



Correct

6. Replace the driver in the same position and tighten with the 1.5 mm hex head screwdriver.



Troubleshooting

Support contact

Technical support is available via e-mail support@massportal.com

Feedrate and Flowrate – these settings can be changed during printing. E.g. use the feedrate 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.

Extrusion problems

No or not enough plastic is released from nozzle.

Usually the most common reason that nozzle gets clogged is because of inappropriate print settings such as print speed, retraction speed and distance, temperature etc..

Problem:

Filament is caught or ended.

Solution:

Check the filament spool and also watch if the filament isn't twisted before extruder.

Problem:

Print head isn't heated up properly.

Solution:

Make sure that the temperature is set right for particular material. If the heater or thermocouple wires are decoupled, the LCD screen will display an error message, in that case contact customer support.

Problem:

Print head's cooling fan isn't spinning when print head is heated up (Note: it only spins when printhead's temperature is more than 50°C).

Solution:

Check if fan is free of dust and nothing physically blocks it from spinning, for example, heater wire. If fan is broken, contact customer support.

Problem:

Extruder wheel is full of filament dirt

Solution:

Clean it with hard brush and spin it around to clean it all (Note: Remove the filament before cleaning the extruder wheel). See page 36

Problem:

Nozzle is too low to print bed and that cause filament blockage, because of high back pressure in print head.

Solution:

Level the print bed with auto calibration or three screws and/or change the settings to the correct bed coating.

After leveling remove filament cut the end at angle and insert it again.



Problem:

Material is low quality.

Solution:

Try to print with different material.

Problem:

There is some dust inside the nozzle which blocks filament extruding.

Solution:

Heat up print head ~30C above normal material printing temperature and try to extrude filament manually by hand. Repeat this few times. If that doesn't help there is need of nozzle change.

Print not sticking to bed

Things to make part adhesion better:

- Use right bed coating, as a universal solution we suggest to use BuildTak surface, also remember to change "Bed coating" type in printer menu
- Print first layer slower (15-25mm/s)
- Use brim or raft
- Turn part cooling fans off for first layer
- Increase print bed temperature
- Rotate model on print bed to get the largest possible surface to print bed
- Level print bed with auto calibration function or with 3 screws
- Move nozzle little bit lower to print bed ("Bed coating"->"Custom"->reduce the number little bit)

Removing filament from the bowden tube

There is sometimes a case when filament breaks near feed hole and if the printer is still printing filament end goes after drive gear in bowden tube then it is impossible to push it out.

To remove filament from bowden tube:

1. Heat up print head
2. Push with two fingers bowden tube's mount button at the extruder,
3. Then remove tube firstly pushing it in then pulling out.

The same bowden tube mount is also on other tube's end by the extruder.

WARNING Do not use excessive force. If bowden tube mount button is pressed the tube should come out freely. If repeatedly mounted and unmounted, the tube ends can become worn or damaged — please replace the bowden tube.

Press the grey coupling to release filament tube, then first push and then pull the tube out while coupling is pressed

