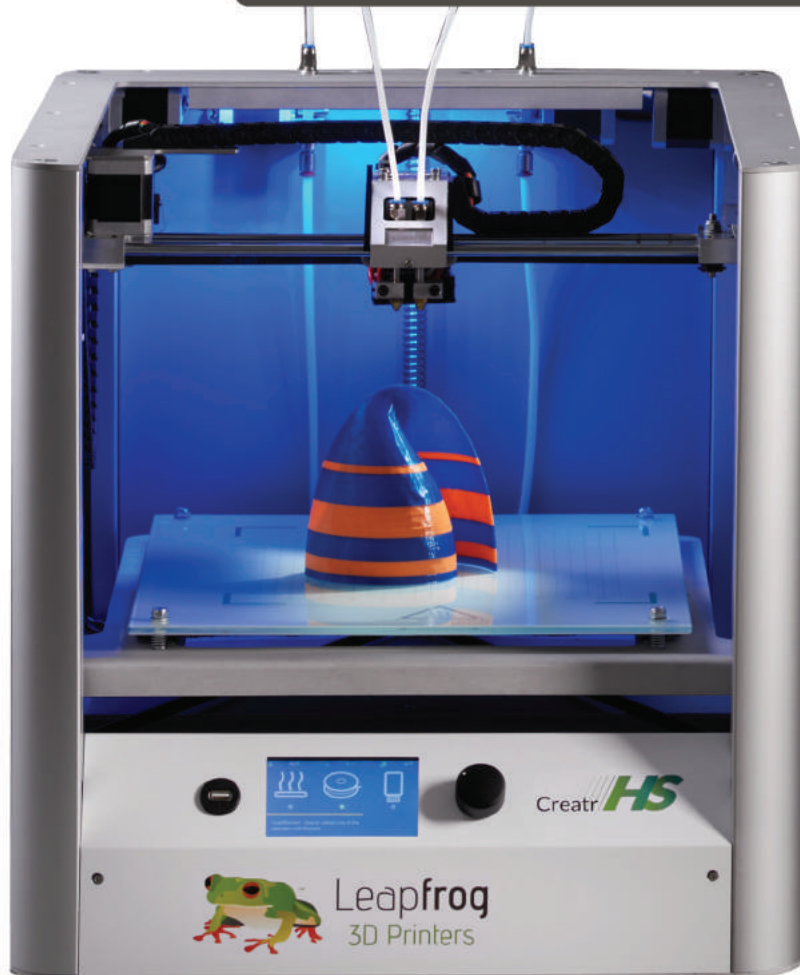




Leapfrog
3D Printers

Manual Creatr *HS*

Create the Future



lpfrq.com

Follow us online!



facebook.com/lpfrg



twitter.com/Leapfrog_3D



instagram.com/leapfrog_3d

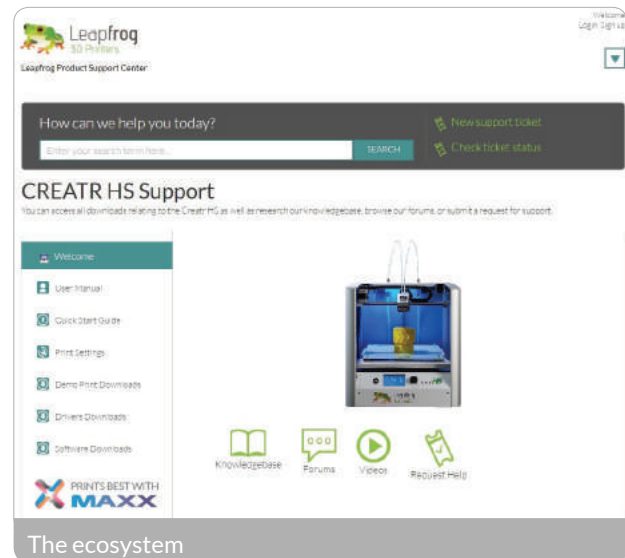


Sign up for our newsletter www.lpfrg.com

Dear Customer,

Congratulations with your very own Leapfrog Creatr HS! In this document we will help you get started with your first print and with all the prints to come after that. Also, we would like to help you get acquainted with the Leapfrog 3D Printers Ecosystem. We will be by your side when you are setting up your printer and help you with any questions you might have during printing. We will also cheer you on when you are making your amazing and value-added prints.

The Ecosystem: tips and tricks, installation video's, and support



You are now officially part of The Ecosystem: the place where all Leapfrog 3D printer users gather to help each other and to interact with the Leapfrog 3D Printers support team. Here are a few valuable tips on what you are able to find there.

The Ecosystem can be entered through creatrhs.lpfrg.com. The Ecosystem is constantly growing in content and users. Here are some of the most important things you can find:

- Installation and support videos, as well as the manual for all the printers
- The latest software settings for all materials (posted under “knowledgebase”)
- Solutions for the most common issues
- Our forum, where users help each other and where you can post your amazing prints (every once in a while, we will highlight a few in our newsletter and on our website, we will always mention you of course!)
- The Leapfrog 3D Printers support team: if you have any questions you can just submit a ticket online (“request help”) and we will get you on your way.

Are you all set to start?

To get started you need the following items:

- Your Creatr HS
- At least one filament
- Print stickers
- Your Simplify3d license key: this is either sent to you in a separate email coming from Simplify3d, or is included on a card in your box.

We will keep you up to date on our social media platforms (left page) about print settings and other Leapfrog 3D Printers news. You are also invited to share your prints on our pages.

Now let's start,
Happy printing!

The Leapfrog team

Table of content

1. Making your very first 3D print – Quick Print Guide

1.1	Unpacking your Creatr HS and finding the right place for it	4
1.2	Getting to know your printer	5
1.3	Plugging in your Creatr HS	6
1.4	Preheating the printer	6
1.5	Loading filament	6
1.6	Checking and adjusting the bed	7
1.7	Printing	8

2. Preparing for your next print

2.1	Removing your print and replacing your print sticker	9
2.2	Changing filament	10

3. Creating your own printable file: from STL to gcode

3.1	The 3D printing workflow – from idea to print	11
3.2	Installing Simplify3d	11
3.3	Simplify3d software overview – the slicing process	12
3.4	Simplify3d workflow	12
3.5	Printing with dual extrusion and support materials	14
3.6	Material guide	18
3.7	Overview advanced print settings	19
3.8	Controlling your printer with Simplify3d – printing while connected to your computer	19
3.9	Recalibrating your printer using Simplify3D	21

4. Frequently Asked Questions

4.1	How can I solve the issue of the print not sticking to the bed?	22
4.2	What should I do when I cannot get my filament to go through the filament guiding tube?	23
4.3	What can I do if my filament does not come out of the extruder?	23
4.4	My print surface is very rough, how can I solve this?	23
4.5	Where do I go with my other questions?	23

5. Glossary of 3D printing vocabulary

5.1	3D printing vocabulary	24
5.2	Getting to know your printer	26

1. Making your very first 3D print – Quick Print Guide

We know you cannot wait to start 3D printing! In this guide we will take you on the easiest journey from box to print.

This section contains the following information:

- 1.1 Unpacking your Creatr HS and finding the right place for it
- 1.2 Getting to know your printer
- 1.3 Plugging in your Creatr HS
- 1.4 Preheating the printer
- 1.5 Loading filament
- 1.6 Checking and adjusting the bed
- 1.7 Printing

1.1. Unpacking your Creatr HS and finding the right place for it

Unpacking your Creatr HS carefully, according to the following steps, ensures that the machine stays calibrated (although in some cases decalibration might occur during transport). These steps you only have to follow when your brand new Creatr HS arrives at your doorstep for the first time.

Before you start, make sure that you have cleared a spot where you want to place the printer. We advise you to find a place at room temperature, where there is no draft (for example next to a window) and where the printer cannot get wet. Choose a surface which is sturdy (so the printer cannot move) and supports the weight of the machine. It is useful to have another set of hands to help out, the machine is quite heavy.

STEP 1: Cut the tie cords and lift the top of the box from where the handles are.



STEP 1: Cut the tie cords and lift the top box.



Leapfrog™
3D Printers

STEP 2: Pull off the tape from the plastic bag wrap on the top of the printer and push the plastic down and away from the printer.

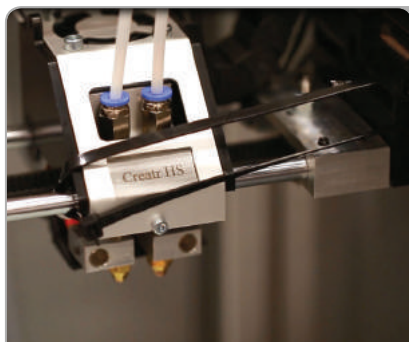


STEP 2: Remove plastic wrap

NOTE: Do not grab the Creatr HS by the top cover as this can damage it. Lift the machine from the bottom.

STEP 3: Place the printer in its new location and make sure it sits firmly on all four rubber feet. If necessary adjust their length by unscrewing the feet slightly.

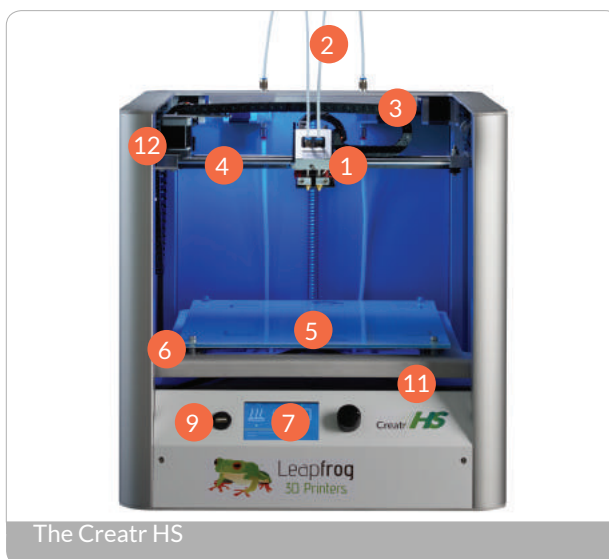
STEP 4: Remove three sets of tie-wraps with a cutting tool: at the front of the carriage, in the corner of the carriage and in the back of the printer.



STEP 4: Cut the tie wraps at the front and back of the carriage and in the back of the printer

1.2 Getting to know your printer

1. Printing head and extruders
2. Filament guiding tube
3. Filament drive unit
4. Axes
5. Print bed
6. Print bed levelling knobs
7. Display
8. Quick release knob (back)
9. USB port for USB stick
10. USB port for connecting to the computer (back)
11. Belt (inside the printer)
12. Z-sensor adjustment knob (only to be used when bed levelling cannot be achieved using the print bed knobs!)



The Creatr HS

Large image including close ups on page 26.

1.3 Plugging in your Creatr HS

OPTIONAL: If you want to use the printer connected to your computer, use the USB cable supplied with the printer, insert the USB-A (flat rectangular connector) cable in your computer and insert the USB-B (square connector) cable in the backside of the Creatr HS.

In this Quick Guide we will use the stand-alone modus, therefore the cable does not have to be plugged in.



NOTE: If your printer has a USB-A (flat rectangular) backside connection, make sure you only use the USB-A cable with the built in FTDI chip that is included in the box. Any other USB might interfere with the communication.



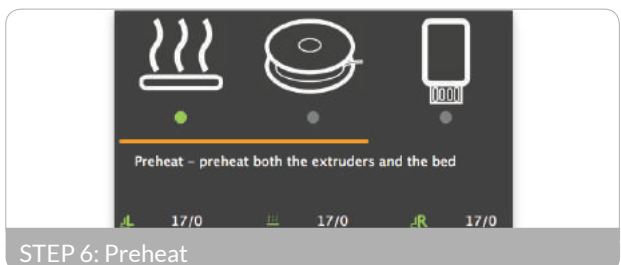
OPTIONAL: connect USB cable to port

STEP 5: Plug the power cord in the outlet and turn on your Creatr HS.

1.4. Preheating the printer

Before you start loading your filament, it is necessary to preheat your printer. Heating your extruders will make sure that your filament will melt and feed through. You are also preheating your print bed. This will make sure your print will stick to your bed.

STEP 6: To preheat, scroll to the preheat icon on your screen, and push the button. If the colour of the icon changes, your nozzles and bed are preheating.



STEP 6: Preheat

1.5. Loading filament

The Creatr HS is compatible with a wide range of 1.75 mm filaments. All filaments provided by Leapfrog 3D Printers are of high quality and are thoroughly tested by us. For every filament, we also provide standard settings for Simplify3d (to be found in The Ecosystem).

STEP 7: Unpack your role of filament and cut the end into a sharp point.



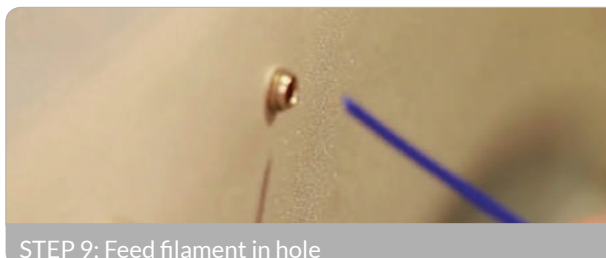
STEP 7: Cut to a sharp point

STEP 8: Home the printer so you can reach the bottom where the filament has to be fed through. To do this, select the home icon on your screen and press the button to start homing.



STEP 8: Home the printer

STEP 9: Make sure your nozzles are preheated to at least 180 degrees Celsius. Place your role of filament in the bottom of the printer. Start feeding the filament through the hole.



STEP 9: Feed filament in hole

STEP 10: Once you reach the filament drive unit, pull out the quick release knob on the back, and feed the filament through further until they reach the printer head.



STEP 10: Pull the quick release knob



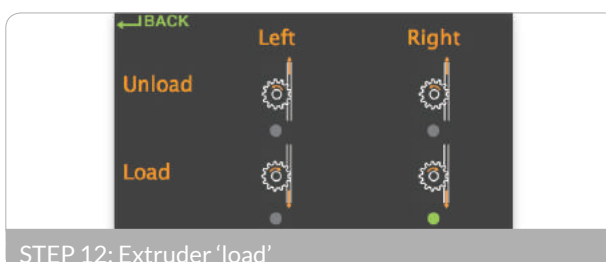
TIP: If you have problems with feeding your filament, there is a possibility that your filament has trouble to pass the filament guiding tubes connection. Take the top connections out by turning them counter-clockwise. Manually feed some filament while pulling the quick release knob, and put the connections back. Put the filament guiding tubes back in place.



STEP 11: Select filament icon

STEP 11: On your screen, select the filament icon and push the button.

STEP 12: Select your extruder and press "Load". After a few seconds, filament should be flowing out of the extruder. Remove the printed filament once the extrusion is finished.



STEP 12: Extruder 'load'



NOTE: Do not touch the bed or the tip of the extruder with your fingers, this is very hot.

STEP 13: To go back to the main menu turn the button counter-clockwise.

1.6. Checking and adjusting the bed

Each Creatr HS is carefully calibrated before it leaves Leapfrog 3D Printers' headquarters. However, during transport, calibration might get shifted. Calibration means that your extruder is at the correct distance from the bed to print smoothly, and that your bed is exactly levelled. Here is explained how you can slightly adjust the calibration during printing. If a full recalibration is necessary, the steps described in Chapter 3.9 should be followed.

In the next paragraph you are going to start printing. During this print you can manually adjust the bed while printing. Check if the bed is levelled during the first 2 or 3 layers of the print and verify if the filament is adhering well to the bed.

- If a corner of the bed is too far from the nozzle you observe filament being extruded too loosely in this area of the print bed, causing your print not to stick.
- If a corner of the bed is too close to the nozzle the extruder scratches through the print sticker, or (almost) no filament comes out of the nozzle.

You can manually adjust the four corners of the bed where necessary during printing by very carefully turning the bed levelling knobs. Only turn it slightly! Usually not more than a quarter of a turn is necessary. Looking from the top, slightly turn the knob clockwise in the corner where no filament is flowing or where the nozzle is scratching the print sticker. Turn the knob counter-clockwise in the corner where the filament is not pressed enough onto the bed and comes loose easily.

If these steps are not sufficient to successfully calibrate the printer, it can be recalibrated using the computer, which is explained in Section 3.



1.7 Printing

STEP 14: Surf to creatrhs.lpfrg.com and click on 'Demo Print Downloads'. Select the single extruder print.

STEP 15: Insert your USB stick in the front USB port of your printer.

STEP 16: On your screen, select the USB stick icon and press the button.

STEP 17: Find your file, select it and press the button. It can take a few seconds for the files to be displayed if there are many files on the USB stick. The printer will now start to heat up to the required heat encrypted in the file. Once it hits that temperature, it will start printing!



TIP: The first layers of a print are always the most difficult and it is best to carefully observe them. Slight adjustments to the level of the bed during the first print layers might be necessary to obtain a successful print.



2. Preparing for your next print

In this chapter we will take you through the processes of preparing your printer for your next print. We will take you through the processes of replacing your print sticker, exchanging your filament and we will offer you to methods of calibrating your machine: manually and through using Simplify3d.

- 2.1. Removing your print and replacing your print sticker
- 2.2. Changing filament

2.1 Removing your print and replacing your print sticker

The print bed of the Creatr HS is made out of glass to ensure that it is as flat as possible. However, it is sometimes difficult to keep your print stuck to it. To make sure that your print sticks to the bed, you need to apply a print sticker. You can re-use your sticker for prints over and over again as long as it is not damaged. Even when it is damaged, you could opt to place your print on another section of the bed where the sticker is not damaged yet (in order to learn how to do this, revert to Chapter 4 where we cover the Simplify3d software).



TIP: Removing your print from the bed: You can use a putty knife to make it easier to remove your printed object. If the print still sticks on the print bed, heating up the print bed to 40 degrees Celsius (see quick start) can make it easier to remove prints.

STEP 1: Check whether your print sticker is damaged or not. If it is damaged in one place, you may also position your print on a different, undamaged area of the bed (you can do this while slicing your print in Simplify3d).

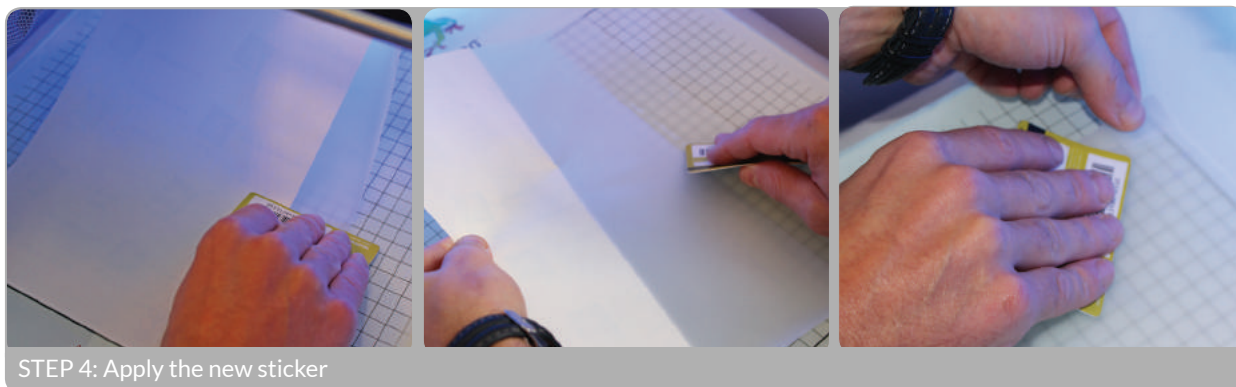
STEP 2: If the print sticker is damaged and you cannot position your print on an undamaged spot, remove the sticker by lifting one of the corners of the sticker and peeling it off. If you remove it too quickly, the sticker is more likely to tear.

STEP 3: Remove the sticker glue by applying sticker remover or glass cleaner on the print bed. Be sure to remove the whole sticker and all of the sticker glue so you will end up with a clean glass plate. If residue is left behind, this could affect your print quality.



TIP: Use some abrasive soap with your putty knife on a cold (non-heated) print bed.

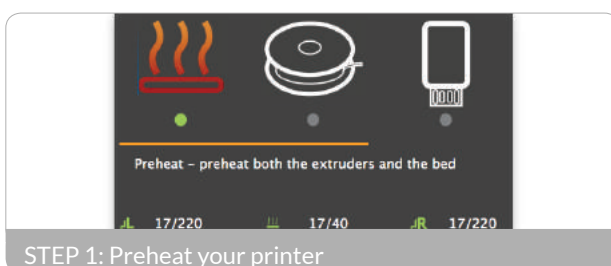
STEP 4: Apply the new sticker by placing the sticker on your print bed. If you are satisfied with the position, take off the small part of the sticker. Use a plastic card to apply the sticker, work from the inside to the outside and avoid air bubbles. Take off the big part slowly and use a plastic card to apply the sticker, work from the inside to the outside.



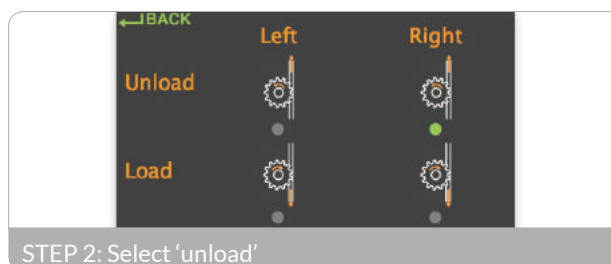
2.2 Changing filament

If you are ready to use a different role of filament for your next print, follow these steps to remove your current role of filament.

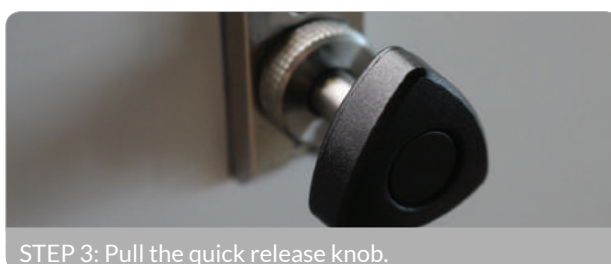
STEP 1: Preheat your printer by selecting the preheat icon in your screen and pushing the button. Make sure the nozzle which contains the filament you are about to exchange heats up to at least 180 degrees Celsius.



STEP 2: Select the filament icon in your screen and enter this menu by pushing the button. Select the extruder from which you want to remove the filament and select 'unload'. The drive unit will retract the filament.



STEP 3: Pull the quick release knob on the back and pull the filament out all the way from the feeding hole on the bottom. Make sure you roll up your role of filament carefully and that the end cannot get loose. Tangled filament can affect the print the next time you use it.



3. Creating your own printable file: from STL to gcode

In this chapter we will teach you how to 'slice' (create your own 3D printable file named 'gcode') using your 3D model (or 'stl') as input. The slicing software we will be using to do so is Simplify3d.

In the gcode, all movements of your printer and characteristics of the different layers are encrypted. With Simplify3d, you have the possibility to print with either standard print settings that we prefabricated for you, or (as you get more advanced) you can tweak your own settings for your print.

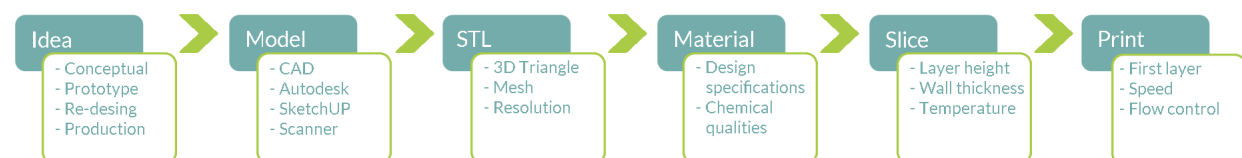
Among the many options that Simplify3d offers, there are options to change the temperature, the layer height, the infill (how massive or how hollow you want your object to be), as well as options for support structures (for structures with an overhang of more than 45 degrees). In Section 4.6. we will go over the most important options. If you want more information on advanced printing options, or support for your software, please revert to the website of Simplify3d: simplify3d.com/support/.

We will go over the following:

- 3.1 The 3D printing workflow – from idea to print
- 3.2 Installing Simplify3d
- 3.3 Simplify3d software overview
- 3.4 Simplify3d workflow
- 3.5 Printing with dual extrusion and support structures
- 3.6 Material guide
- 3.7 Overview advanced print settings
- 3.8 Controlling your printer with Simplify3d – printing while connected to your computer
- 3.9 Recalibrating your printer using Simplify3D

3.1 The 3D printing workflow – from idea to print

Below is a schematic workflow on how you get from an idea to a print. In this chapter, we will take you through the last four steps: from STL to print.



3.2 Installing Simplify3d

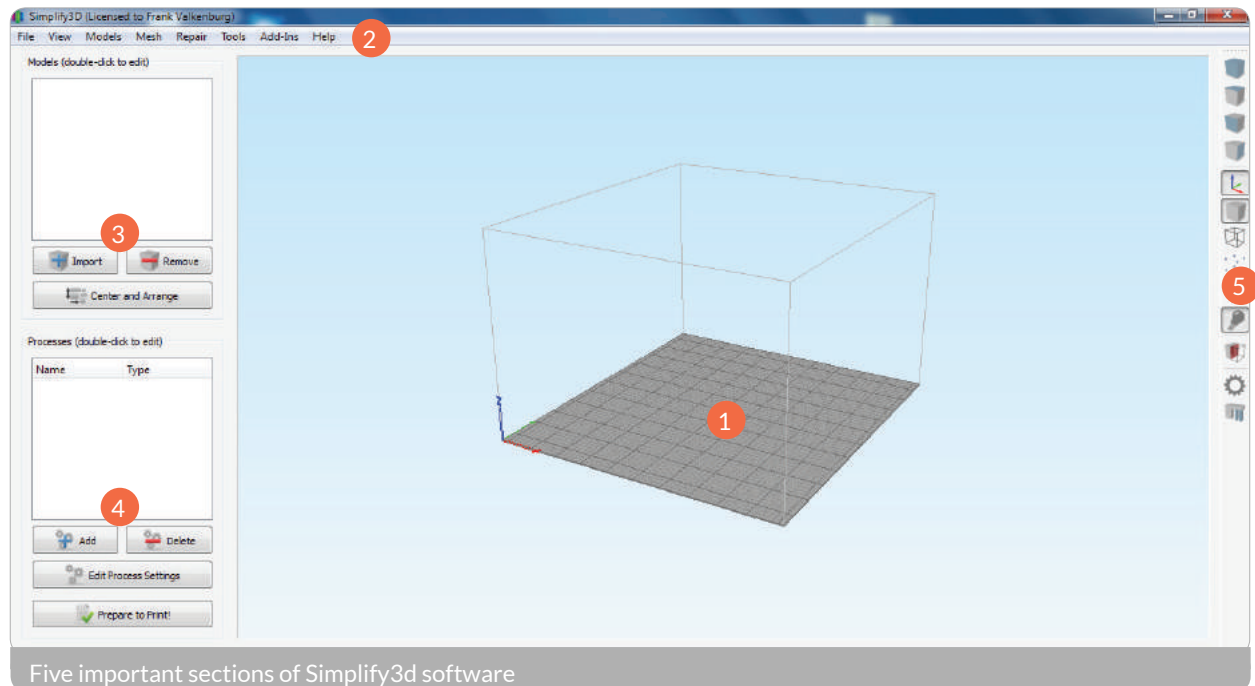
STEP 1: Download and install the Simplify3D software for your Mac or PC. The download key is either on a card in the box with the printer, or it is sent to your email address straight from Simplify3d. Instructions on how to download Simplify3d can be find on the software card in your box or in your email.



TIP: Simplify3d is regularly updated for new functions which are accessible for free for existing customers.

3.3 Simplify3d software overview

Here we will familiarize you with the Simplify3d software. When you open the Simplify3d software, you see the following screen. There are 5 sections in this screen that are important to get familiar with:



1. Build table: the build size you see in your screen corresponds to the build volume of your Creatr HS. Here, you can scale and position your model.
2. Menu bar: under 'tools' you can find the "machine control panel" that you will use if you are using the printer while connected to your computer.
3. Models: here you can import your stl file. If you double click on your model you can scale and position it.
4. Processes: here the different slicing processes are listed. Slicing is the process during which your stl is transformed into a gcode.
5. Toolbar: here you can change the way your model is displayed, and can find tabs to the control panel and to manually define support.

3.4 Simplify3d workflow – the slicing process

In this section we will guide you through the slicing process. We will start with the import of the latest print settings and take you through all steps to transform your 3D file into a printable gcode. Leapfrog 3D Printers provides standard print settings to print with the filaments we provide. These standard settings can be uploaded in Simplify3d and are used to slice your 3D file into a printable gcode. To make sure you always have the latest and improved settings for your materials, revert to The Ecosystem online to download the latest settings and import them in your slicing software.

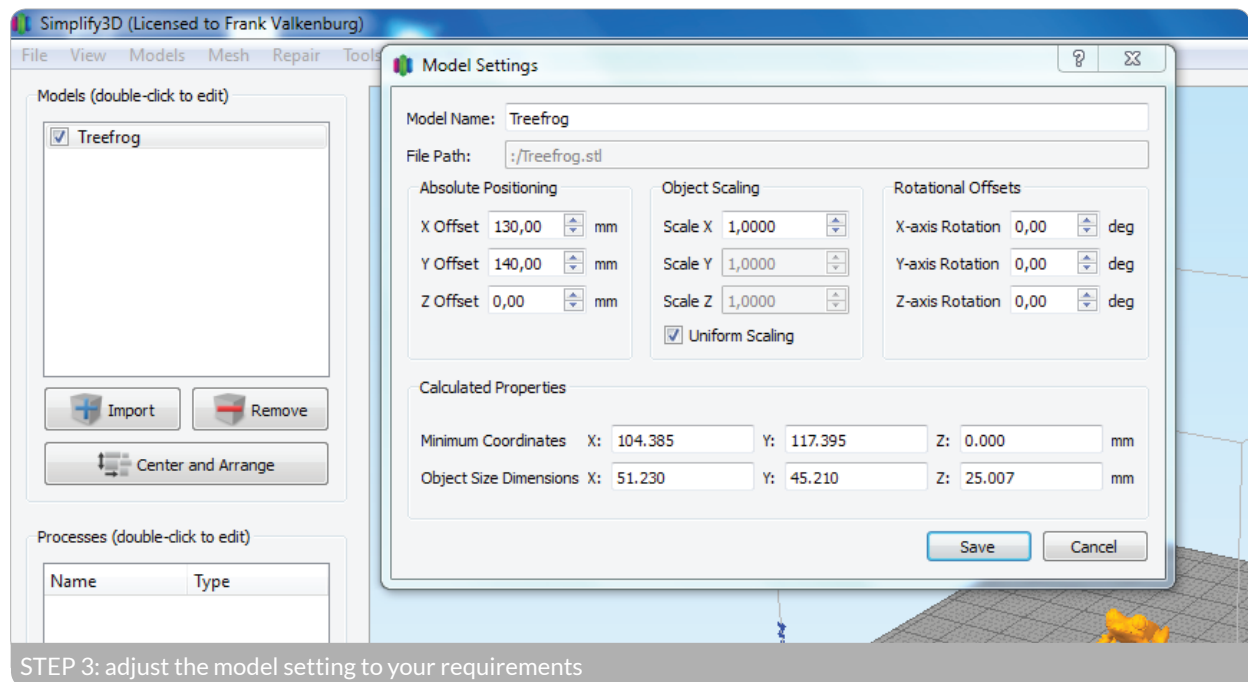
STEP 1: Download the latest material settings from lpfgr.com/printersettings and save them on your computer.



TIP: The standard print settings for each material are divided in categories: there is a trade-off between quality and speed of getting a part. Before printing you need to ask yourself what is of more importance to you: would you like to receive your part as fast as possible with a rougher finish? Choose fast printing. If you want to have a very detailed print with a nice finish, you can choose high quality printing. If you want the best of both worlds, use medium printing.

STEP 2: Make sure the 3D model that you are using is saved as file format .stl. Import your stl file by clicking on “Import” in the models section.

STEP 3: Double click on the model name to scale and position your model according to your requirements.



TIP: for successful printing, be sure to position the print on a point where the sticker is not damaged. To move a print, hold the control and left mouse button down.

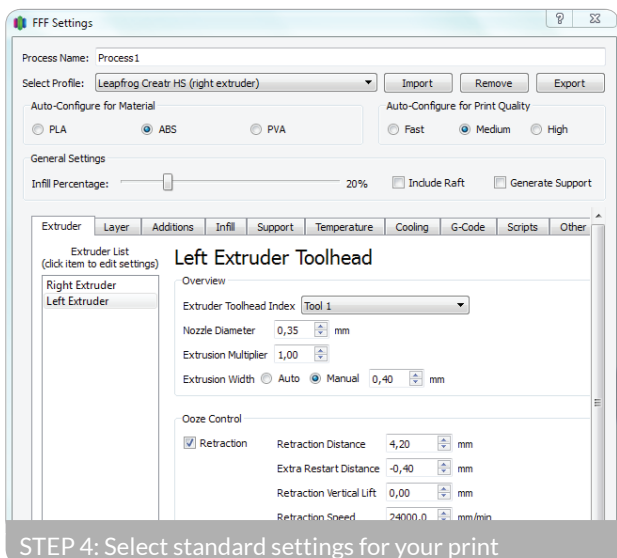
TIP: If your stl model has overhang angles of more than 45 degrees, the printer will not be able to print it without a support structure (simply because the printer cannot lay down layers of plastic in mid-air). If you want to learn how to add support structures, revert to section 3.5.

STEP 4: From the downloaded standard settings you now need to determine which proper profile you want to use. To do this the following choices must be made:

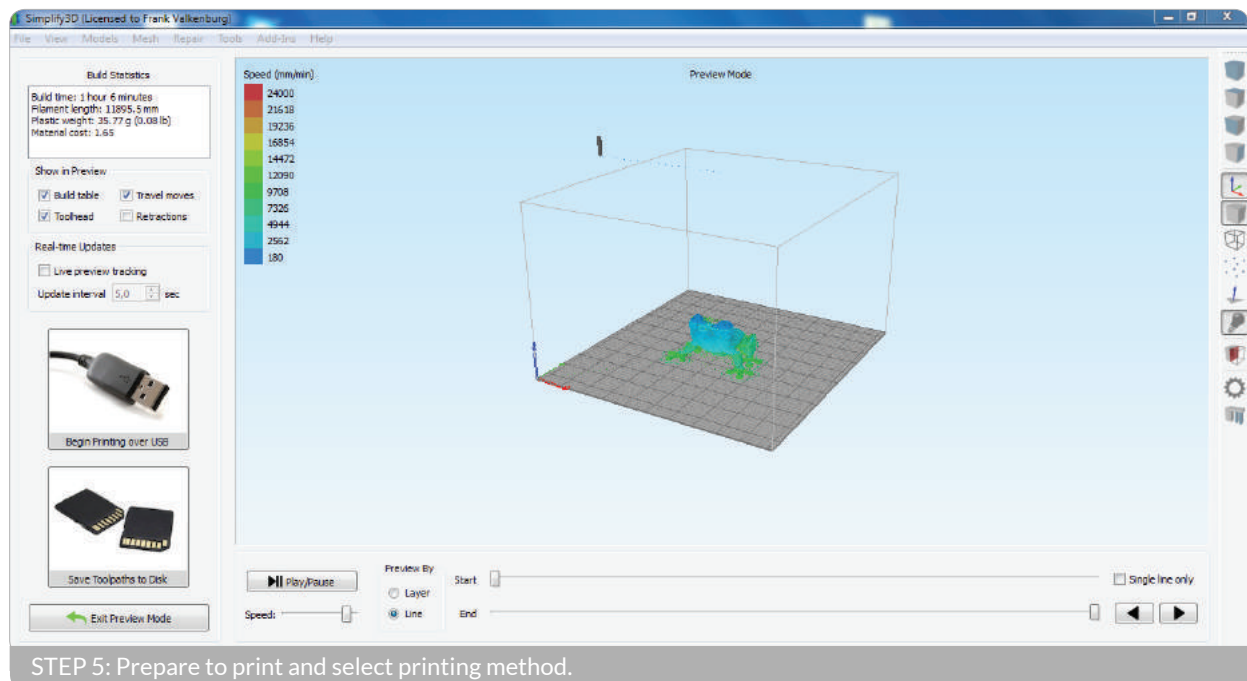
- Which filament material is used in your right and in your left extruder
- Which (part of the) model should be printed with the right or left material
- Which extruder should print support if this is needed
- Should the print be fast, medium, or should it be detailed

For a simple single colour print without support, click on “add” in “processes” and select the profile with the correct material, extruder, and print detail (for instance, a fast print using PLA from the right extruder). For more complicated prints see Section 3.5.

STEP 5: Click on ‘prepare to print’. In the next view you can see all the movements your printer is going to make while printing. If you are satisfied, click on either ‘begin printing over USB’ (if your printer is connected to your computer through USB) or ‘save toolpaths on disk’ (to save your gcode to an USB stick to be printed stand-alone).



STEP 4: Select standard settings for your print



STEP 5: Prepare to print and select printing method.

3.5 Printing with dual extrusion and support materials

Your Creatr HS has two extruders, which means that you can print with two materials in one print. For some users this might mean two different colours of the same material, for others this means printing with two different types of materials. Here are some ideas on the latter option:

- Printing complex shapes with overhanging parts (angles of more than 45 degrees, which the printer is not able to print) with one plastic such as PLA and ABS, while using a soluble material such as PVA or HIPS as a support structure (to dissolve in water or D-Limonene after printing).
- Printing prints with flexible and solid parts (combining Flex (available from Leapfrog soon) with PLA for example)

3.5.1 Breakaway support versus soluble support

Support material is added during the printing process in order to support the overhanging parts of your design. Both the objects in the image are not printable without support (as some of the angles are more than 45 degrees). If you would print these designs without support material, the print ma-

material would just fall on the bed. You can build two types of support structures; breakaway support and soluble support.

Breakaway support is a support type that adds support material in the same material and color as the material you use to print your object. In this image the support material is printed with the second extruder. After the print is finished you have to breakaway the support structure manually. Using breakaway support simplifies the slicing and printing process, however, it is not suitable when the support is in places you can not reach. It will be difficult to remove if your print is small.



Structure with support overhang

Soluble support is only possible if you have two extruders (with your Creatr HS you are the lucky owner of a dual extrusion 3D printer). The advantage of soluble support is that you can dissolve your support structure after printing, and it hardly leaves any marks on the surface of your print. At this moment, we support two types of support material:

- PVA, to be used to support a PLA print. PVA is solvable in ordinary tap water. The soluble support material needs to be inserted in the second extruder. PVA is the safest material to dissolve since you only need water.
- HIPS, to be used to support an ABS print. HIPS can be solved using D-Limonene.



TIP: If you take care of a few things before or after using PVA or HIPS, you will ensure the highest quality of your prints. Before using PVA or HIPS to print, make sure you extrude the PVA or HIPS generously to flush away any remaining material in the nozzle.

TIP: After using PVA or HIPS, make sure you unload the filament after the print. Also, clean out the nozzle thoroughly by loading another material (PLA or ABS for example) through the nozzle and having it extrude. This will minimize the amount of residue left over in your nozzle.

After your Creatr HS is finished printing, you need to remove your print [including the support material] from the bed. Place your object together with the support material in the correct solvent for your support material and the support structures will dissolve. You do need some patience for this, as it might take up to a few hours to dissolve.



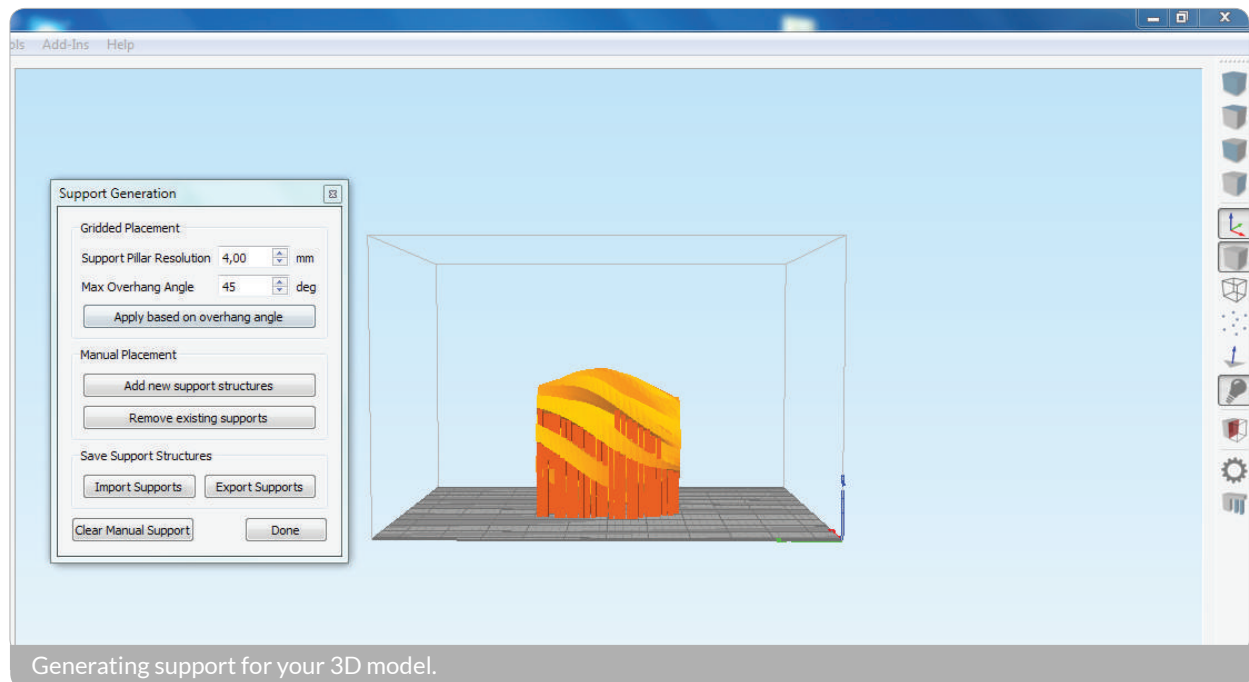
NOTE: we sometimes get feedback from clients that want to speed up the dissolving of PVA by using hotter water. Hot water might damage your print.

3.5.2 Applying support structures to your stl model

You can either define the support structures manually or you can have Simplify3d place support for you on all parts where there is a 45 degrees overhang. It is beneficial to use the minimum amount of support structures that are required as this will keep the surface of your print as smooth as possible.

You always apply support structures to your stl model, not to your gcode (so you have to do this before or during the slicing procedure).

To add support, click on the “manually define support for model” button in your toolbar or in the menu under “tools”. If you want to manually define support, choose “add new support structures” and place the support structures where you see fit by left-clicking locations underneath the model. The other option is to have Simplify3d place support for you automatically by selecting the option “apply based on overhang angle”.



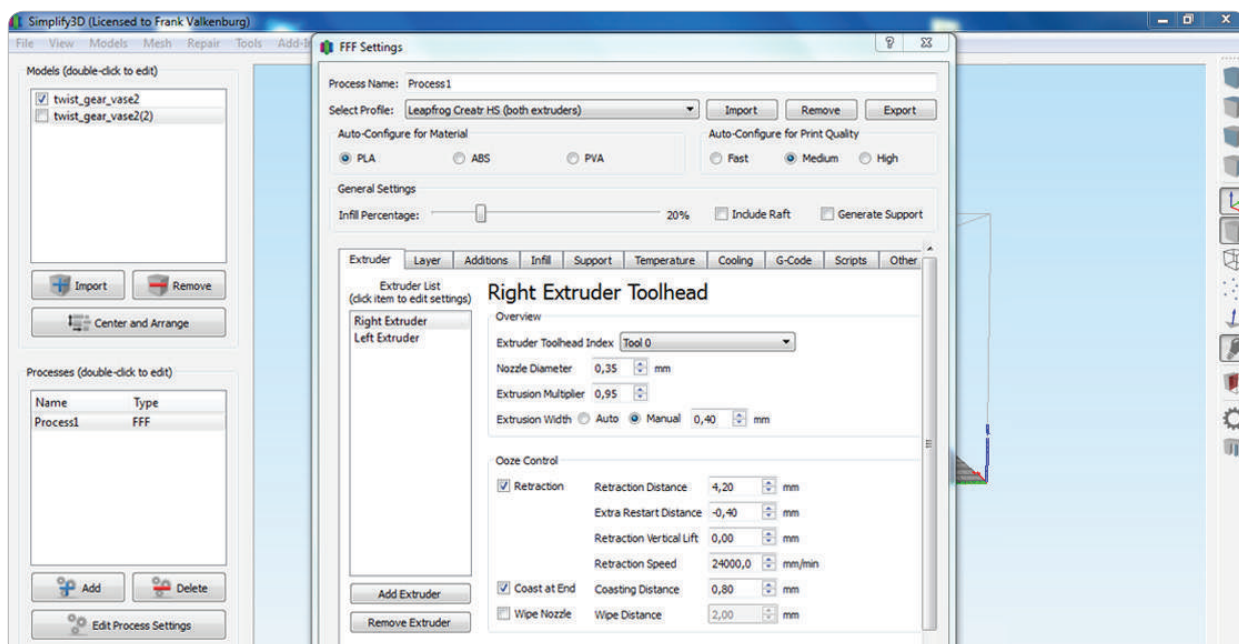
To define the proper process, click ‘add process’ and select a process for ‘Dual Support’, with your desired material and detail properties (fast, medium, or detailed). This loads the profile in which the right extruder is primary and the left extruder will generate the support material.

3.5.3 How to print with dual extrusion

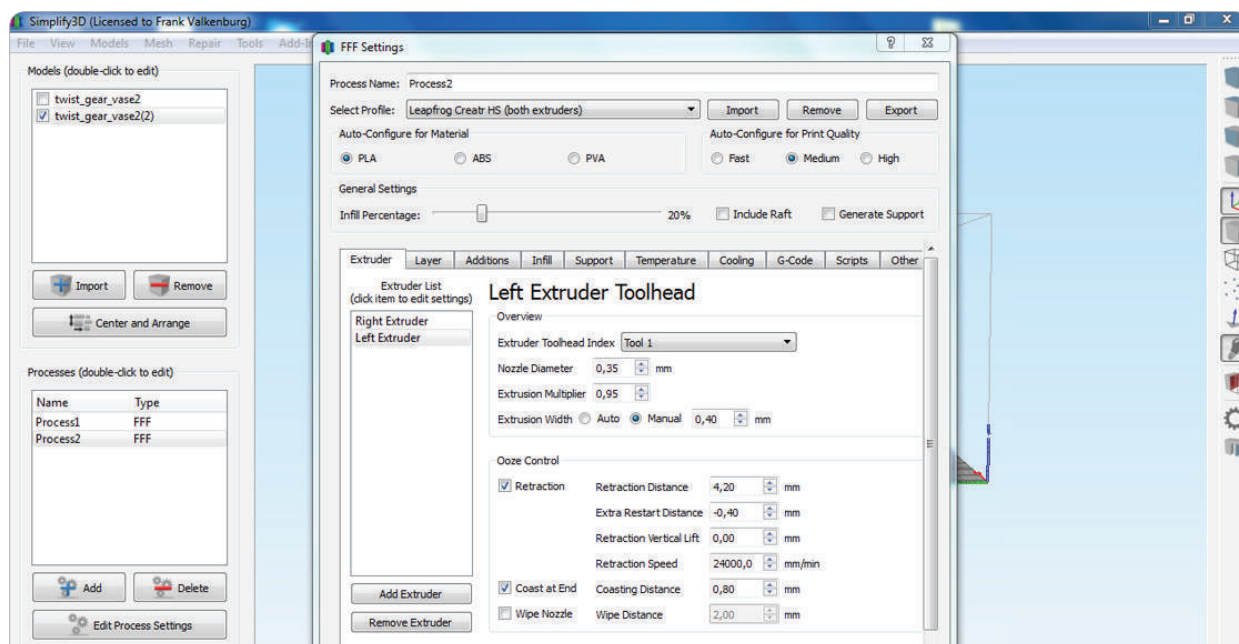
When you would like to print with dual extrusion, you need to follow the general slicing steps as in paragraph 4.2. If you use two types of materials, you need to make two different slicing processes and connect the right process to the correct extruder as shown on page 17.



Leapfrog™
3D Printers



Printing with dual extrusion: allocating the right process to the correct extruder .



Printing with dual extrusion: allocating the right process to the correct extruder .

The key to printing in two different colours is that you need two different STL files; one for each color. So in your CAD program, you have to save your file in two parts (you will get two stl files). In Simplify3d, you can place both stl files over each other to create one object again.

You will also have to make two different slicing processes; one process with the right extruder and one with the left.

STEP 1: click 'add process' and select a process for 'Dual Support', with your desired material and detail properties (fast, medium, or detailed). This loads the profile in which the right extruder is primary and the left extruder prints support.

STEP 2: Double-click on the process and click on the 'select models' tab. Select the models that you want to be printed by the right extruder.

STEP 3: click 'add process' and select another process for 'Dual Support'. However, this one must be

configured for the left extruder.

STEP 4: Double-click on the process and click on the 'layer' tab. In the tab 'primary extruder' now select 'left extruder'. Then select the models that you want to be printed by the left extruder.

STEP 5: In the default setting any support present will be printed by the left extruder. To change this, select the proper extruder for both processes in the 'support extruder' tab under 'support'.

3.6 Material guide

In this section we will discuss the materials that Leapfrog 3D Printers offer and give you a general idea on how to use them.

Material	Description
PLA	Poly(lactic acid) or polylactide (PLA) is a thermoplastic aliphatic polyester derived from renewable sources, such as corn starch, tapioca products or sugarcane. After printing, the surface is instantly smooth. There are no toxic fumes coming from heated PLA, so printing with this filament is safe (although it is always advisable to print in a ventilated area).
PLA PRO	PLA PRO is a PLA variety which provides more intense colouring as well as UV-, draught- and heat resistance. This makes it very suitable for architects and industrial designers.
ABS	Acrylonitrile butadiene styrene (ABS) is a common thermoplastic. This is the same material as LEGO bricks are made of. After printing you can smooth the surface of ABS using sand paper or acetone. ABS is commonly used in engineering applications, since it can handle much more stress than PLA. When printing with ABS, a very small amount of toxic fumes may get into the air. It is advisable to print in a ventilated environment or with a (fume) cover.
PVA	Polyvinyl alcohol (PVOH, PVA, or PVAL) is a water-soluble synthetic polymer. It is used as a support material for PLA in 3D printing because it can be easily removed.
HIPS	High Impact Polystyrene (HIPS) is used as a support material for ABS in 3D printing. HIPS can be dissolved using D-Limonene



Hybrid	'Hybrid' is high in strength and toughness, extremely suitable for printing moving parts since it has an improved temperature resistance. This also means Hybrid has to be printed on a higher temperature to reach the right viscosity. It is a high strength engineering plastic which is also food safe (FDA approved). There are no toxic fumes vaporizing when printing with hybrid.
Nylon	Nylon is a generic designation for a family of synthetic polymers known generically as aliphatic polyamides. Nylon is one of the most commonly used polymers. It has self-lubricating properties which can be useful in for example bearings. The material is very tough and light. It is also able to withstand higher stress than most other printable plastics making it more suitable for engineering applications.
Brick	Brick Filament contains small brick fibers, which makes your 3D print look like it is made from sandstone. Architects make beautiful brick-like structures with this.

3.7 Overview advanced print settings

If you are ready to start printing with advanced settings, consult the website of Simplify3d. On <https://www.simplify3d.com/support/> you can find manuals and support regarding the software.

3.8 Controlling your printer with Simplify3d – printing while connected to your computer

While the Creatr HS is a standalone 3D printer, which can print with the input from a USB stick, you can also opt to use it connected to your computer. If you want to calibrate your printer with your computer, connecting is necessary. Other benefits of printing while you are connected to your printer is that you can override the gcode settings while printing. For example, if you see too little filament coming out of the print at one point (under-extrusion), you can increase the extrusion flow.

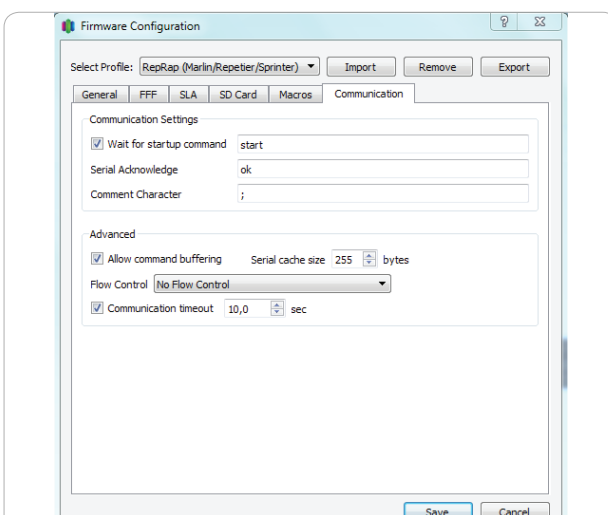
To print with the computer attached, you need to follow these steps:

STEP 1: Connect the computer to the backside of the Creatr HS using the provided USB cable.

STEP 2: In the 'Tools' tab, select 'Firmware configuration'. In the 'Communication' tab select set the serial cache size to 255 bytes. Press save.

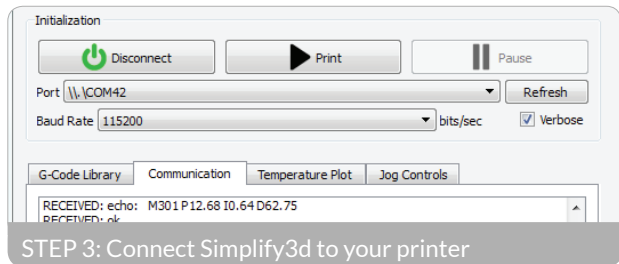


STEP 1: Connect your printer



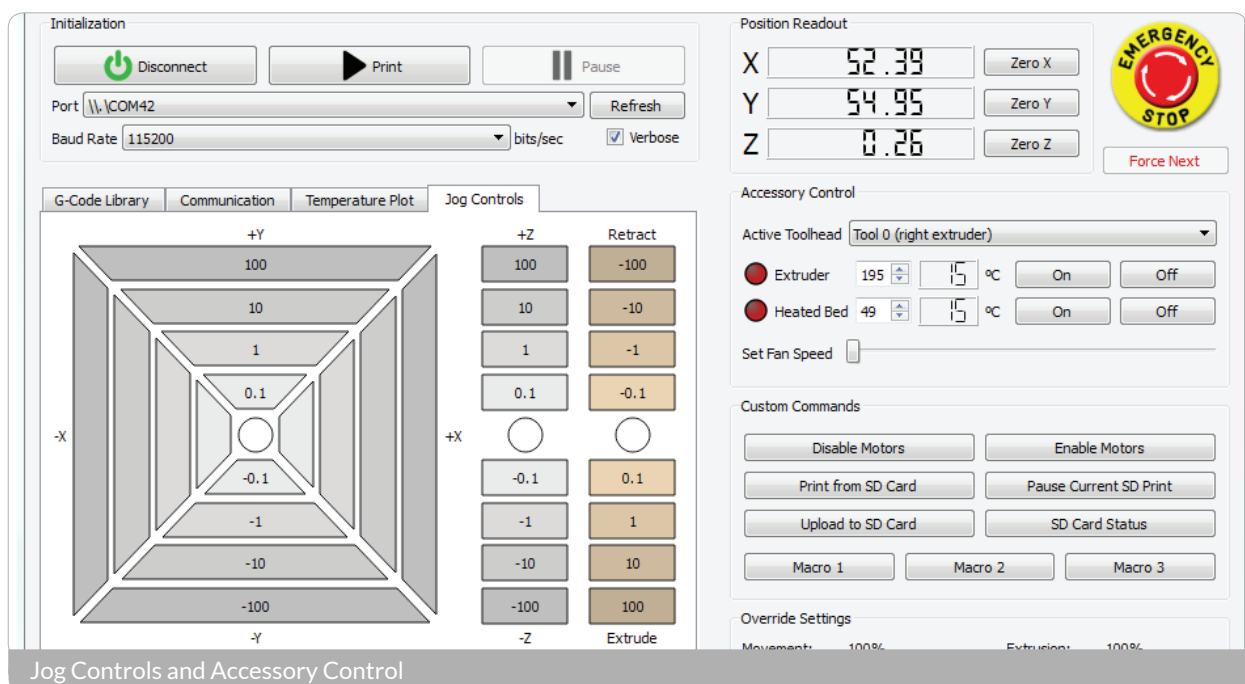
STEP 2: Firmware configuration

STEP 3: In the Machine Control Panel you can see which COM port auto selected. Verify that the baud rate is set to 115200. Click on the “Connect” button. If the connect button turns from a red “Connect” to a green “Disconnect” you are connected.



The following are some functionalities of controlling your printer directly with Simplify3d:
In the “machine control panel” (to be accessed below “tools” in the menu bar) you can:

- Move your print bed and nozzles over the three axes using ‘jog controls’
- Extrude filament using the extrude buttons in ‘jog controls’
- Heat up your nozzles and bed “accessory control”



3.9 Recalibrating your printer using Simplify3D

If small manual adjustments during the first layers of a print are not sufficient to calibrate the bed, it is best to recalibrate the bed using Simplify3D.

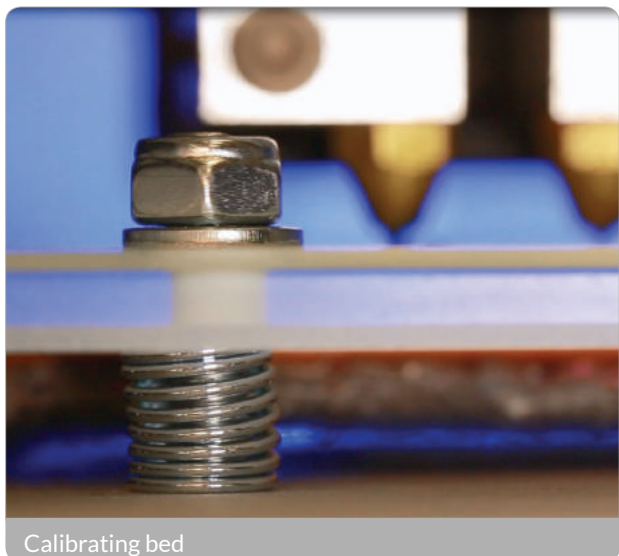
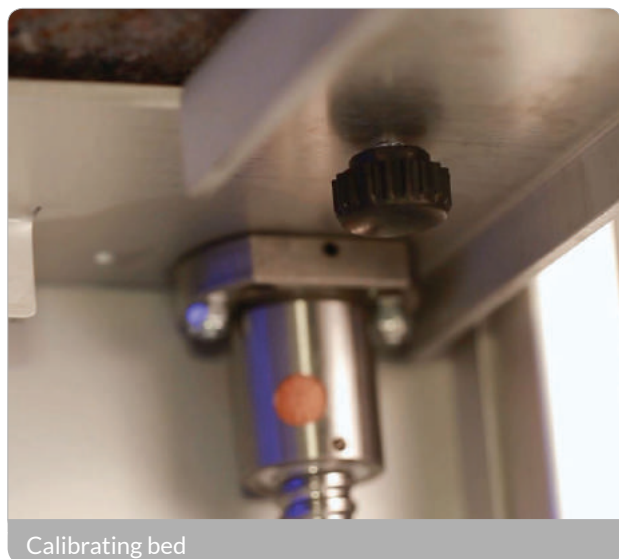
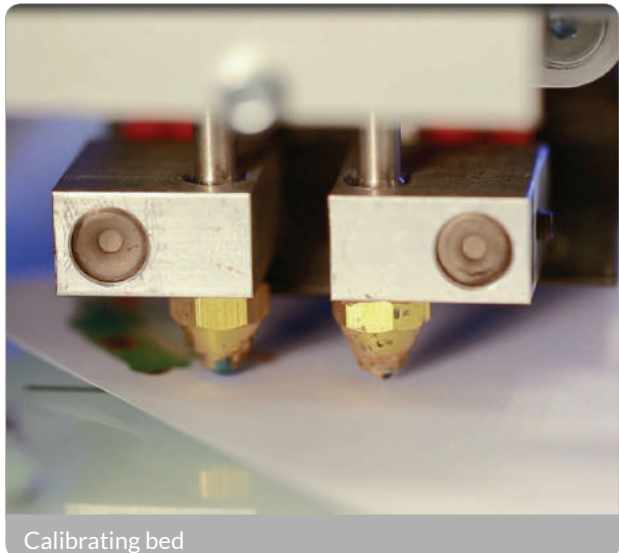
STEP 1: Connect your printer to Simplify3D as described in Section 3.7 and go to 'jog controls' in the "machine control panel"

STEP 2: Select 'Home All'. This moves the print bed up as close as possible to the extruders, and the head to the left-front.

STEP 3: Grab a thin business card and slide it between the right extruder and the bed. Adjust the bed levelling knob in the left-front corner of the bed until the card is slightly pinched between the bed and the extruder (i.e. the moment it can no longer move freely).

STEP 4: Manually move the print head to the far right and repeat the procedure with the business card in the front right corner.

STEP 5: Do this for all 4 corners.



4. Frequently Asked Questions

Although our instructions in this manual are of course highly brilliant ;-) we can imagine that you run into some questions while you are printing. This section of the manual is devoted to that. Note that you can find much more help and support in The Ecosystem, which you can access through creatrhs.lpfrg.com/.

This section contains the answers to the following frequently asked questions.

- 4.1 How can I solve the issue of the print not sticking to the bed?
- 4.2 What should I do when I cannot get my filament to go through the filament guiding tube?
- 4.3 What can I do if my filament does not come out of the extruder?
- 4.4 My print surface is very rough, how can I solve this?
- 4.5 Where do I go with my other questions?

4.1 How can I solve the issue of the print not sticking to the bed?

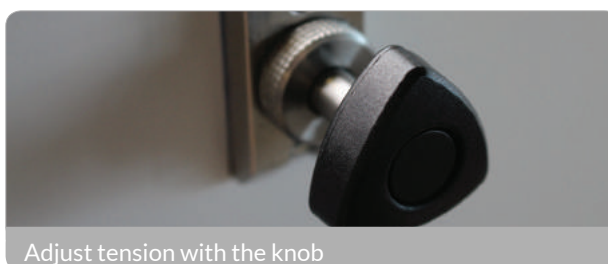
If your prints do not stick to the printing glass there are several steps you have to check on:

- The distance between the extruder and the printing bed: If the distance is too big the layers will not stick and when the distance is too small the extruder will scratch the previous layer causing it to come off.
- The temperature of the bed and the nozzle: Make sure they are at the right temperature for your material.

Material	Bed Temperature	Nozzle Temperature
PLA	40-45 C° (when using the printing sticker). Turn off your bed after layer 5	210-220 C° Depending on printing speed. The lower the speed the lower the temperature.
ABS	75-80 C° (Keep your bed heat turned on for the entire print)	230-240 C° Depending on printing speed. The lower the speed the lower the temperature.

(For all recommended settings, please visit: creatrhs.lpfrg.com)

- The tension of the filament drive on the filament: The filament drive is located at the back of the printer. Too much tension of the drive gear on the filament causes small cuts on the filament. If there is too little tension the drive gear wheel will slip and insufficient material reaches the extruder. You can adjust the tension by loosening or tightening the knob.



Adjust tension with the knob

If the problem persists even after you level the extruders please try to calibrate the printing glass as well (as explained in section 1.6.)

4.2 What should I do when I cannot get my filament to go through the filament guiding tube?

If you cannot get the filament to go through the guiding tube remove the push in-coupling at the top of the drive unit push the filament through and then screw back the push in coupling and push the filament all the way to the printhead.

4.3 What can I do if my filament does not come out of the extruder?

The first thing to check is if the drive wheel is slipping and not pushing the filament through. If the pressure on the filament is ok, remove the guiding tube from the print head and heat up the extruder. Then use a 1.5 mm thin steel tool to push through the extruder tube to remove any remaining filament out of the extruder. Then reload the filament and extrude.

4.4 My print surface is very rough, how can I solve this?

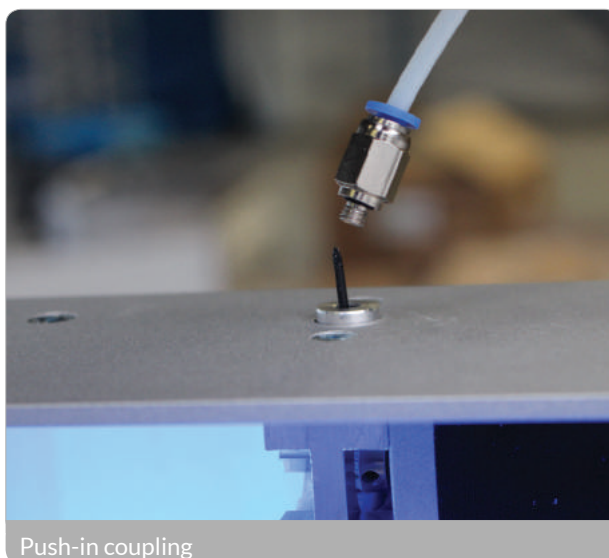
Are you using the right settings for your material (are you printing with PLA with PLA settings?) And are you using the latest settings for your material? The latest settings for your material can be found here: creatrhs.lpfrg.com/. In section 3.4 you can read the instructions on how to import these into the Simplify3d slicing software.

4.5 Where do I go with my other questions?

As a Leapfrog 3D printer owner, you are now officially part of The Ecosystem: the place where all Leapfrog 3D printer users gather to help each other and to interact with the Leapfrog 3D Printers support team. Here are a few valuable tips on what you are able to find there.

The Ecosystem can be entered through creatrhs.lpfrg.com/. The Ecosystem is constantly growing in content and users. Here are some of the most important things you can find there:

- Installation and support videos, as well as the latest version of the manual for all the printers
- The latest software settings for all materials (posted under “knowledgebase”)
- Solutions for the most common issues
- Our forum, where users help each other and where you can post your amazing prints (every once in a while, we will highlight a few in our newsletter and on our website, we will always mention you of course!)
- The Leapfrog 3D Printers support team: if you have any questions you can just submit a ticket online (“request help”) and we will get you on your way.



Push-in coupling



1.5mm steel tool in nozzle

5. Glossary of 3D printing vocabulary

5.1 3D printing vocabulary

Slicing what?! We know that the vocabulary of 3D printing may be quite new to you. That's why we are proving you with a short glossary below.

Extruder

The extruders of your Creatr HS can be found in your printhead. They handle the feeding and extruding of the filaments. They consist of two assemblies: a cold end to pull and feed the thermoplastic from the spool, and a hot end (nozzle and hot end) that melts and extrudes the thermoplastic. The Creatr HS is fitted with a dual extrusion system allowing it to use two plastics in the same print.

Extruding

Extruding is the term for the process during which filament feeds through the nozzle. It is the opposite of retracting.

Extruder handle

The extruder handles are the two parts at the back of your Creatr HS that you pull out to feed the filament through.

FDM or FFF

Fused Deposition Modelling (FDM) or Fused Filament Fabrication (FFF) is the 3D printing technique that is used in your Creatr HS. FDM/FFF works on an additive principle by laying down material in layers; a plastic filament is unwound from a coil and supplies material to produce a part.

Filament

Filament is the material that is used by the 3D printer to build the 3D object. The Creatr HS uses spools of filament with a thickness of 1.75mm of a variety of plastics and composites. For an overview of different filaments and their characteristics, revert to page 18.

Filament drive unit

This is the part at the back of the printer with the quick release knob which is designed for easy feeding of the filament.

Filament guiding tube

These are the white tubes that guide your filament from the feeding hole in the bottom all the way to the extruder.

Gcode

The computer language that the Creatr HS understands and receives its instructions from. The Simplify3d software is used to create a gcode out of a stl file. The instructions encrypted within a gcode can range from moving the printing head in X and Y direction to setting the temperature of the hot-end.

Hot end



The heated portion of the extruder mechanism, which gets hot enough to melt plastic (or potentially other materials). The hot end used in the Creatr HS can withstand temperatures of up to about 270 °C.

(Heated) Print bed

A build surface that is warmed in order to keep the base of an extruded part from cooling (and shrinking) too quickly. If the base layers of a print shrink too quickly, this will lead to so-called 'warping': the most common result is corners of parts lifting off the build surface. Heated beds usually yield higher quality of finished prints. You can heat the bed to a maximum of 90 degrees.

Nozzle

The nozzle is the part of the extruder where the filaments exits from: The default nozzle orifice diameter of the Creatr HS is 0.35mm, but this part can be replaced with different sizes to create a larger or smaller flow of filament.

Print head

The print head is the part of the printer that moves along the x- and y-axes to build up the print. The print head contains the nozzle, extruder and the hot end.

Print Sticker

Since the print bed is made out of glass (to keep it as straight as possible) , you need to place a print sticker on the bed before printing, to make sure your print sticks to the bed.

Retracting

Retracting means that the filament is pulled out of the extruder. It is the opposite of extruding.

Slicing

Slicing is the process through which the stl file (or the 3D model) is transferred into a gcode (printable file). Simplify3d is the slicing software that comes with your Creatr HS.

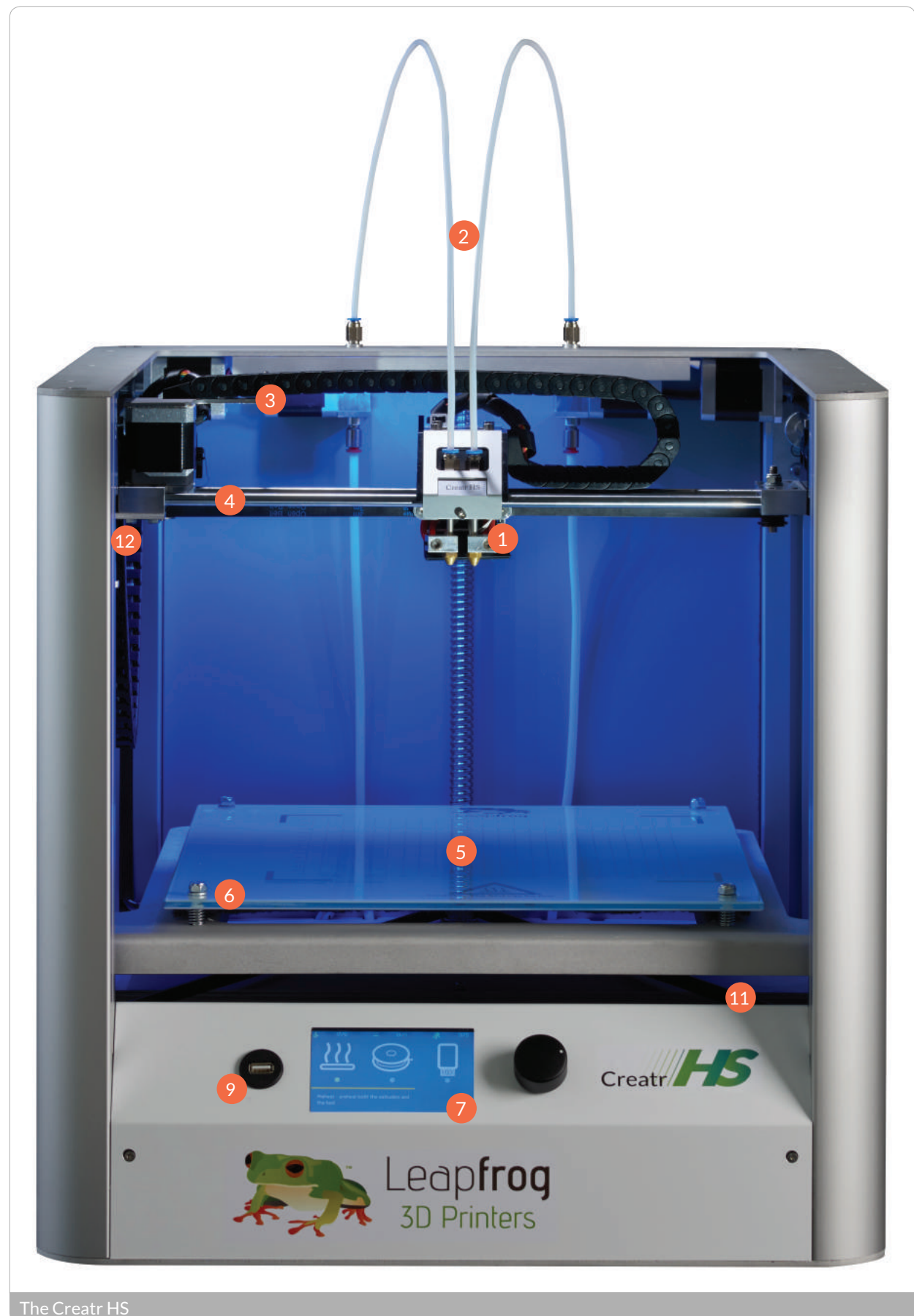
STL

STL is a file format in which you have to save your 3D model in order to be able to convert it to a printable file. STL files describe only the surface geometry of a three dimensional object without any representation of color, texture or other common CAD model attributes.

Support material

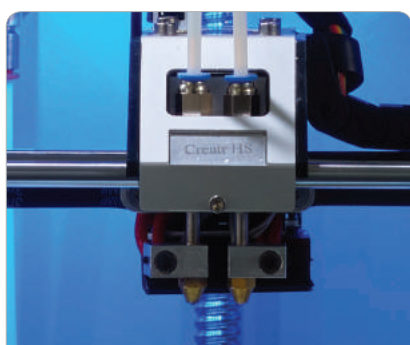
Due to the printing technique used, the Creatr HS has limitations in printing objects with more than 45 degrees overhang. To overcome this problem the printer can print a support structure which literally supports your actual print.

5.2 Getting to know your printer

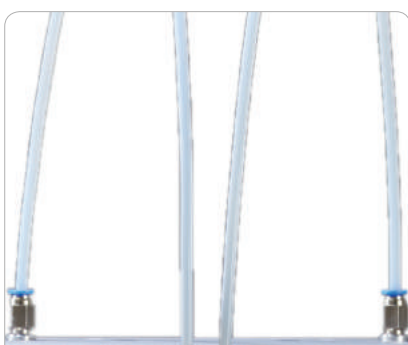


The Creatr HS

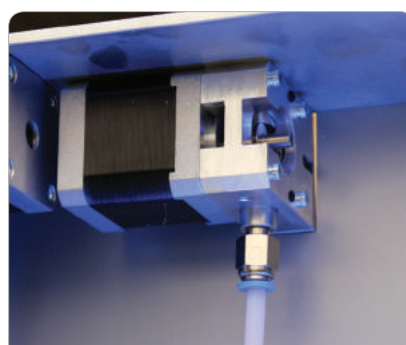
8 10 positioned at the back of the printer



1. Printing head and extruders



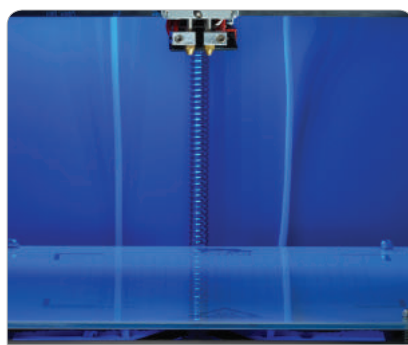
2. Filament guiding tubes



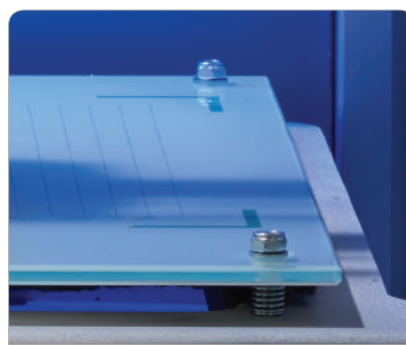
3. Filament drive unit



4. Axes



5. Print bed



6. Print bed levelling knobs



7. Display



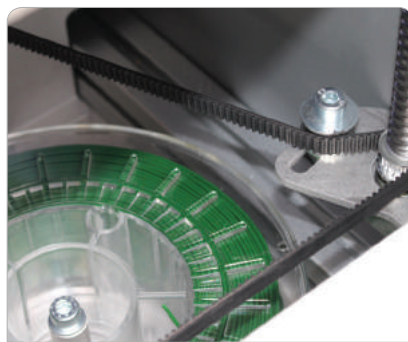
8. Quick release knob



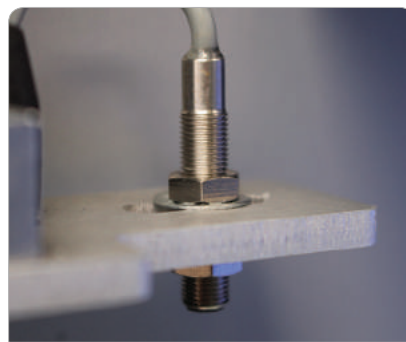
9. USB port for USB stick (front)



10. USB port for computer (back)



11. Belt



12. Z-sensor adjustment knob*

*(only to be used when bed levelling cannot be achieved using the print bed knobs!)

