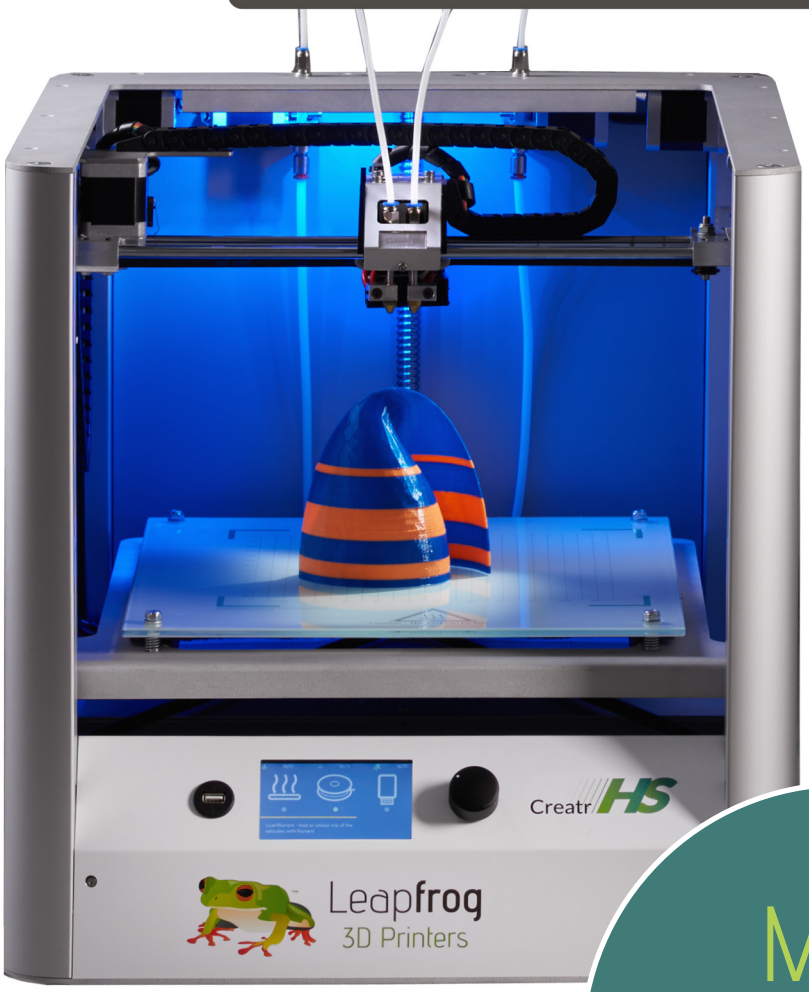
Leapfrog
3D Printers

# Manual CreatrHS

## Create the Future




lpfrg.com

## Mac Version

Full manual available in
EN / NL / DE / FR
from creatrhs.lpfrg.com

**Follow us online!**


facebook.com/lpfrg

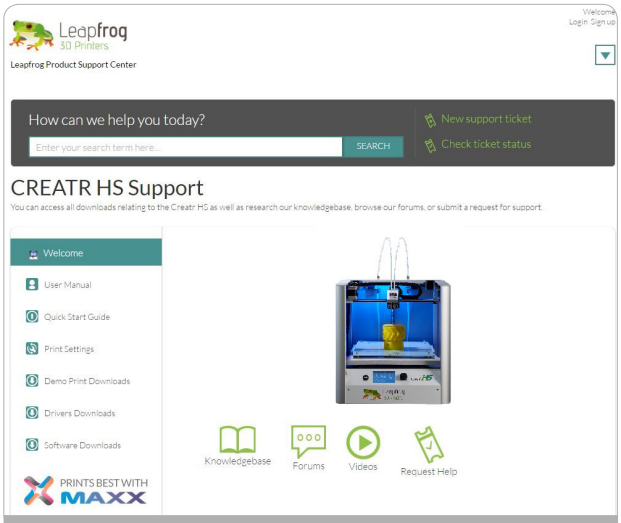

twitter.com/Leapfrog_3D


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

**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 Materialise Creatr license key that you received with the Leapfrog 3D Printer
-

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

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**

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.

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.

OPTIONAL: connect USB cable to port

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.

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

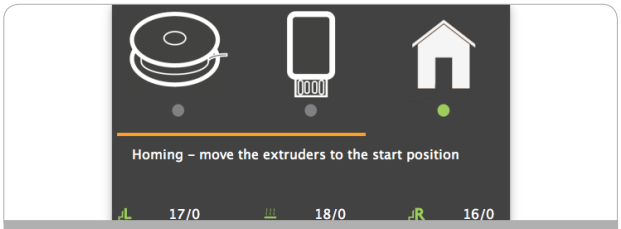
### 1.5. Loading filament

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 profiles for Materialise Creatr Software.

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

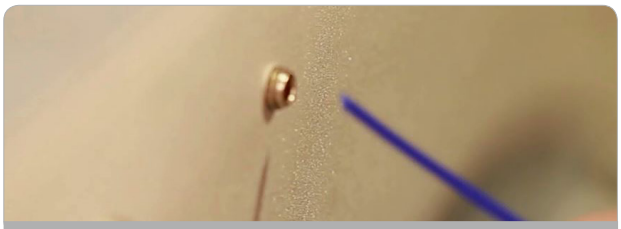
STEP 7: Cut to a sharp point

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.

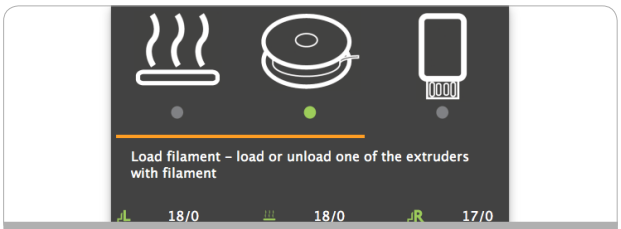

STEP 9: Feed filament in hole

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

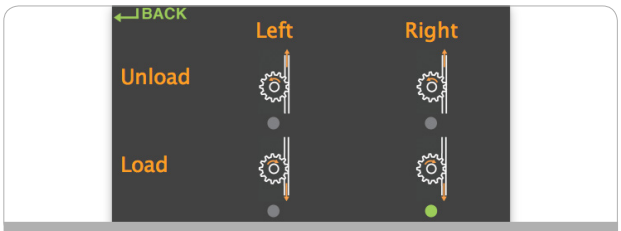
TIP: If you have problems with feeding your filament, there is a possibility that your filament has trouble to pass the filament guiding tubes connection. Take the top connections out by turning them counter-clockwise. Manually feed some filament while pulling the quick release knob, and put the connections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

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



STEP 11: Select filament icon

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.

Calibrating the printer.

**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!


STEP 17: USB port and display in one screen

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 Materialise Creatr software.

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 Materialise Creatr 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.

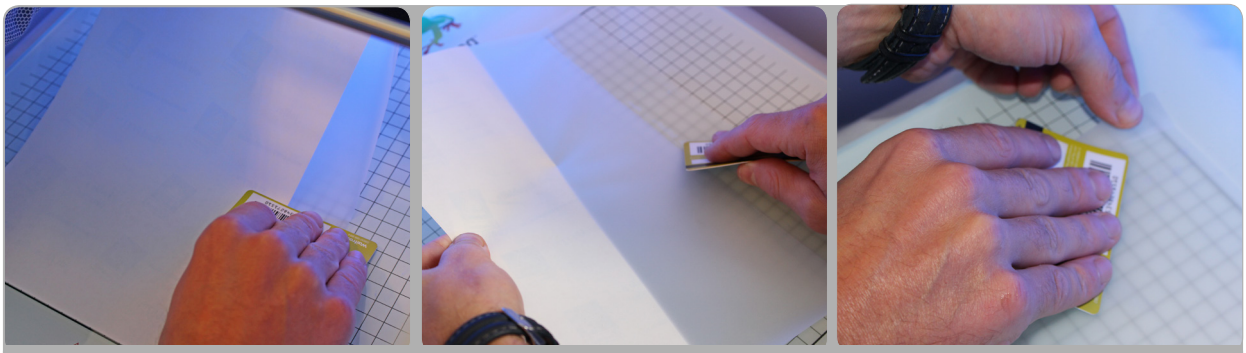
STEP 1: Check whether your print sticker is damaged or not. If it is damaged in one place, you may also position your print on a different, undamaged area of the bed (you can do this while slicing your print in Materialise Creatr).

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 pealing 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 of 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.

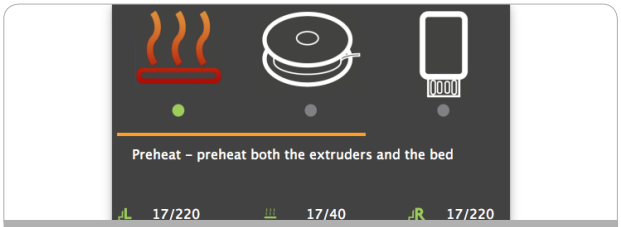


STEP 4: Apply the new sticker

### 2.2 Changing filament

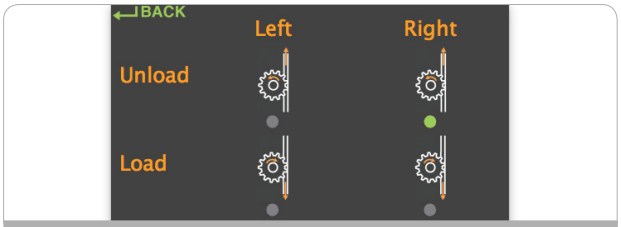
If you are ready to use a different role of filament for your next print, follow these steps to remove your current role of filament.

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 1: Preheat your printer

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 2: Select 'unload'

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.

STEP 3: Pull the quick release knob.

# 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 name 'gcode') using your 3D part (or 'stl') as input. The slicing software we will be using to do so is Materialise Creatr

In the gcode, all movements of your printer and characteristics of the different layers are encrypted. With Materialise Creatr, 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.

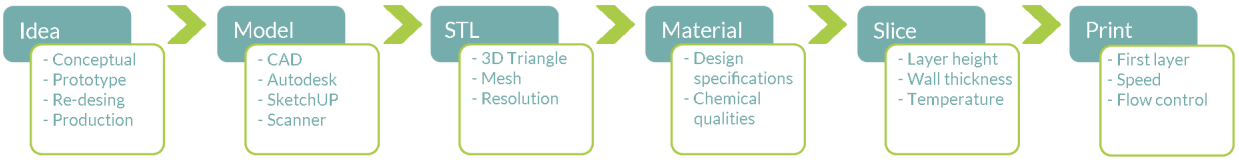
Among the many options that Materialise offers, there are options to change the temperature, the layer height, the infill (how massive or how hollow you want your object to be), as well as options for support structures (for structures with an overhang of more than 45 degrees). In Section 4.6. we will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

We will go over the following:



### 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 Materialise Creatr

The preferred software for your Leapfrog 3D printer is Materialise Creatr. This software package prepares your 3D part into machine instructions for your 3D printer. Materialise Creatr is not sold separately but bundled together with your 3D-Printer. So together with your 3D-Printer, you also received a voucher code. You should use this voucher code to register the software.

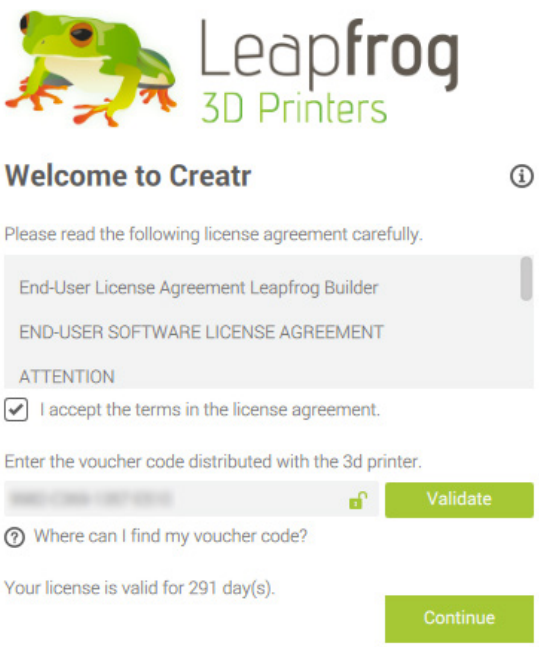
1. Open the installer (.dmg) and drag and drop the application to your Applications folder.

2. After installing Creatr, right click the application and press open.

3. Click Open in the dialog box. If prompted, enter an administrator name and password.

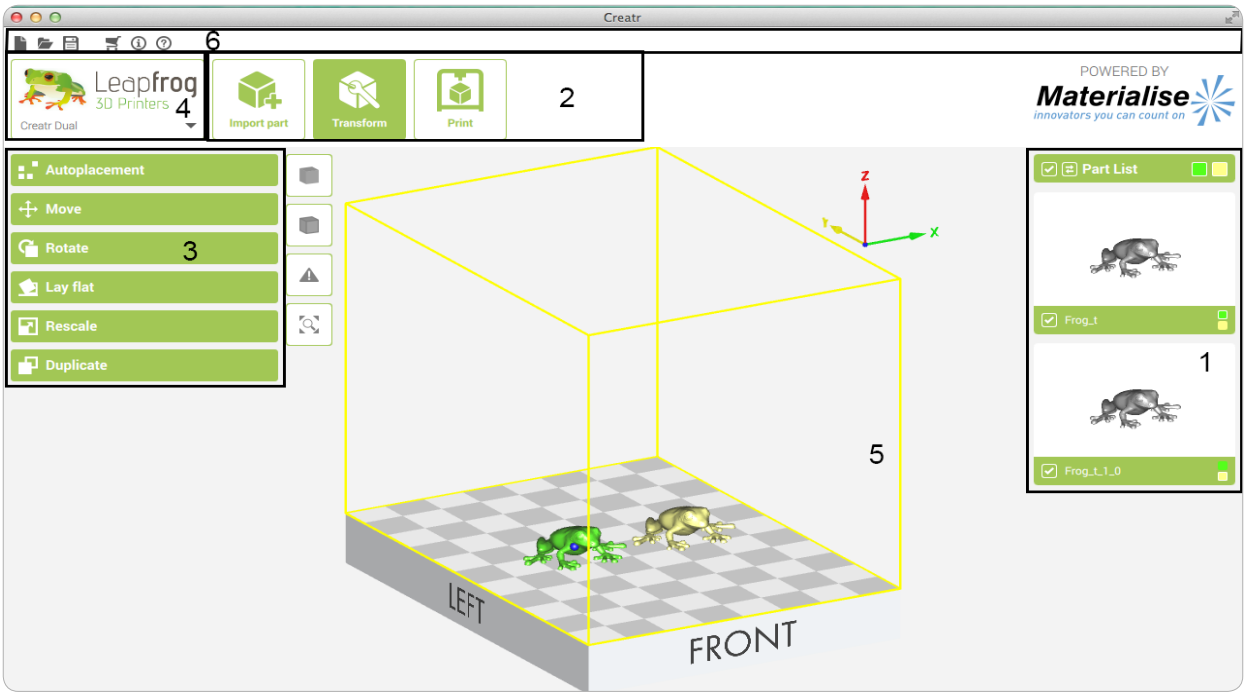
4. Approve the end-user license agreement.

5. Enter the voucher code that you have received with the Leapfrog 3D printer and press validate.
6. The software will be registered and you will get notified of the validity period of the voucher.
7.Press finish to complete the installation.




### 3.3 Creatr software overview

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




1. Part List
2. Flow toolbar
3. Function-toolbar
4. Printer selection
5. Workspace
6. Main toolbar

TIP: For more detailed info, please revert to the Materialise Creatr Manual via

The Flow:

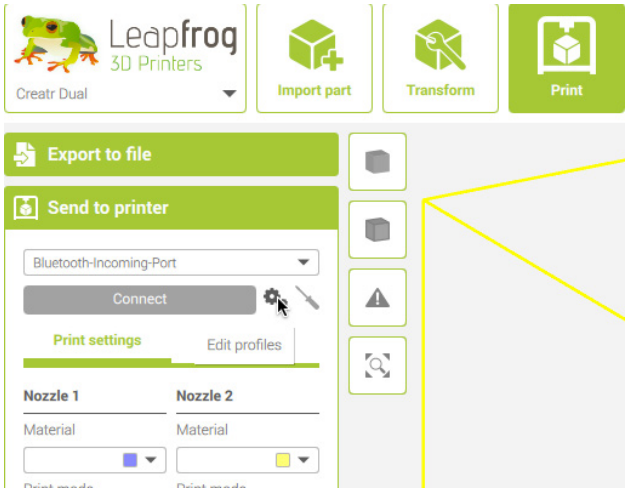
A. Click the Import Part flow button to load your STL-files

B. Click the Transform flow button if you want to translate, rotate, rescale or duplicate part(s)

C. Click the Print button if you want to build your part(s) on your 3D-Printer

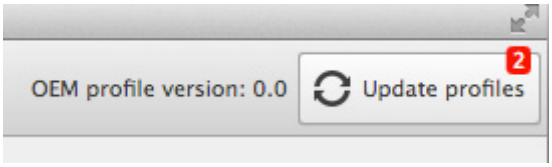

### 3.4 Printer setup

STEP 1: Navigate to print -> Edit profiles

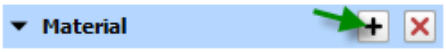



STEP 2: Proflie editor window opens

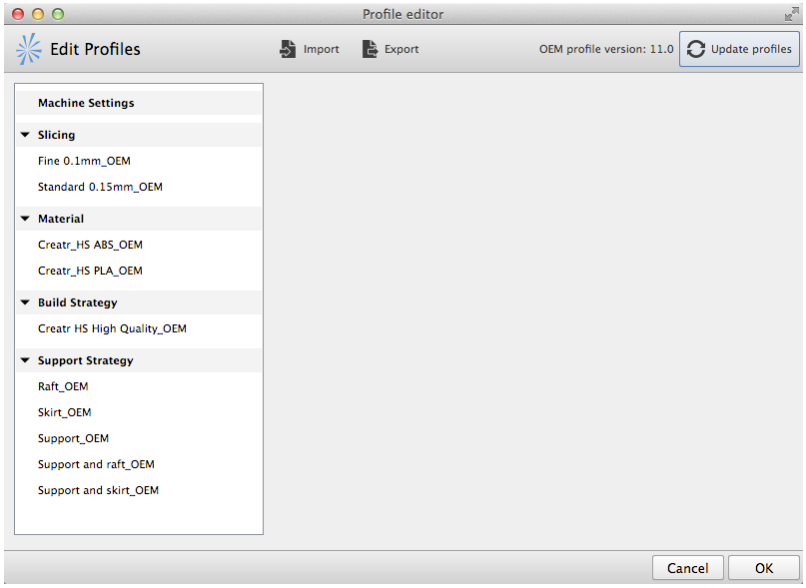
STEP 3: Press up to get the latest Leapfrog profiles (internet is required)




If you don't have access to the internet you can load a default profile by clicking here:




STEP 4: The latest Leapfrog profiles are loaded in the software



### 3.5 Materialise Creatr Workflow

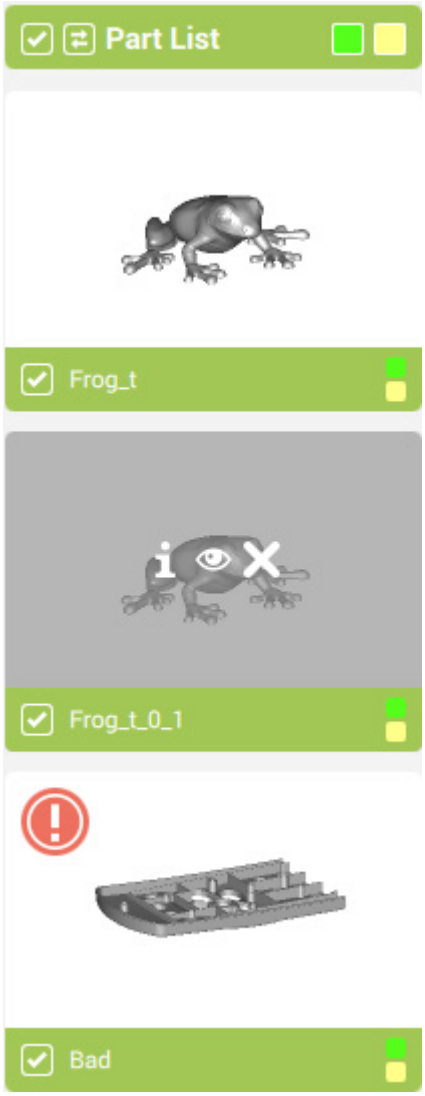
STEP 1: New Project

In the top left corner you can find the 'new project' button. This function will remove all parts and scenes from the current project and generates a new, empty project. The user is asked to save the current project (parts and scenes) before closing. If you want to work in two (or more) parallel Creatr sessions, just open Creatr a second time.

STEP 2: Click on import part and select the part you want to add to the platform.


STEP 3: The part will be added to your part list

STEP 4: You can select/deselect a part by clicking on the selection checkbox in the part list or on the part on the build platform.



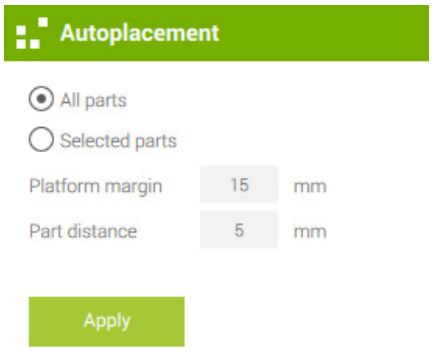

TIP: for successful printing, be sure to position the print on a point where the sticker is not damaged.

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

The Transform tools by pressing on the transform button in the flow toolbar. The transform tools will appear in the function toolbars on the right-hand side of the screen.

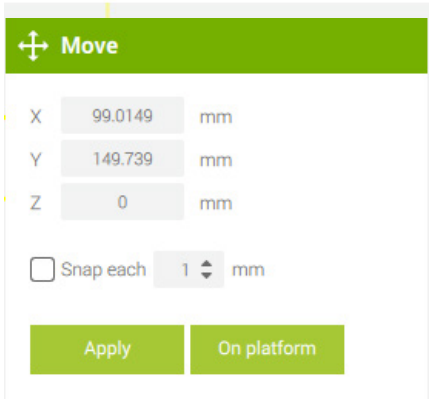
### Autoplacement

This command will automatically place parts on the build platform. The part interval will define the space between the parts when placing the parts automatically. The platform margin will indicate how far the parts need to be placed from the borders of the platform. The parts to nest selection will give the user the possibility to select all parts or just the selected ones.




### Move

The move operation allows you to interactively move a part (or a group of selected parts) to another position.
Grab an Axis or the blue plane on the platform to move the selected part(s) in one or two dimensions while visualizing in 3D view.



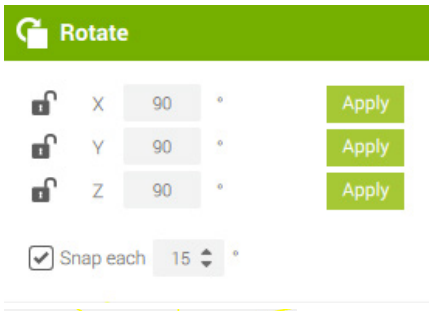

### Rotate

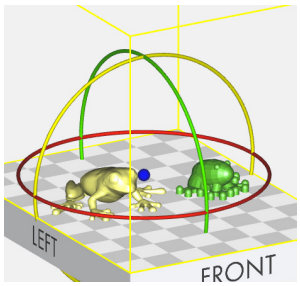
Grab one of the circles to interactively rotate about the chosen axis. You can also change your part selection during operation.

Use the Enable Snapping checkbox to move parts in defined increments. You can specify the step size yourself.

In addition, you can specify a relative rotation, fill in your desired degrees to rotate about the X,Y or Z axis. Press apply to rotate.

Use the lock icons to counter movement about the specified axis.



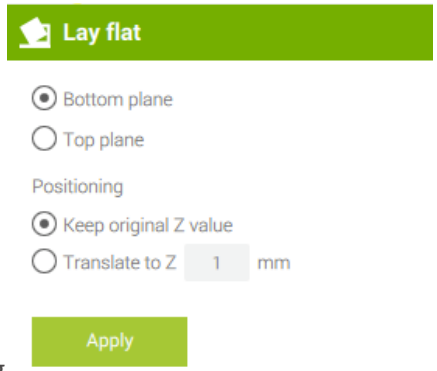




### Rescale

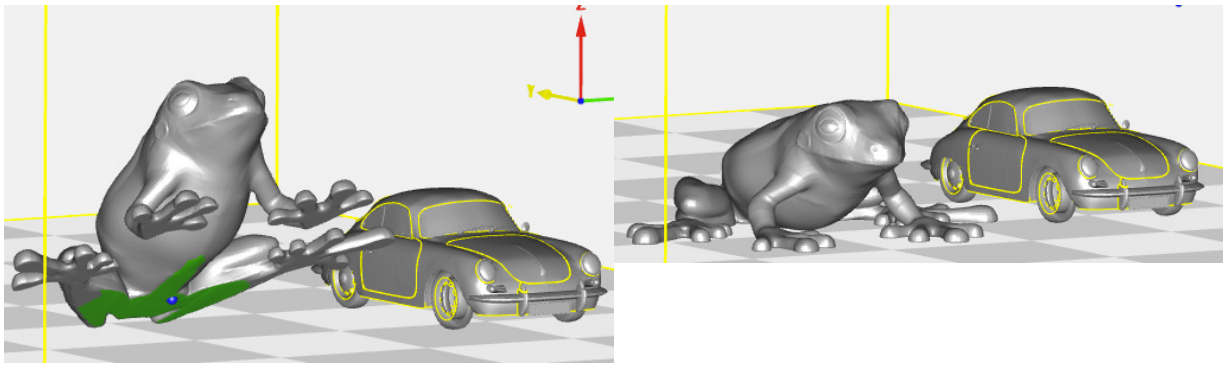
The Rescale Factor is a multiplying percentage for the dimensions in that direction. When the factor is 100%, no rescaling is performed, when the factor is 200%, size is doubled. A factor larger than 100 will enlarge the part, a factor smaller than 100 will shrink the part.
Enable Uniform rescale when you want to use an identical factor in all directions. Disable this option if you want to specify a different value in X, Y and Z.

### Lay flat

The Bottom/Top function allows easy orientation of the selected part by indicating a plane as the bottom/top plane. This plane will automatically be oriented parallel to the platform (i.e. the XY-plane).

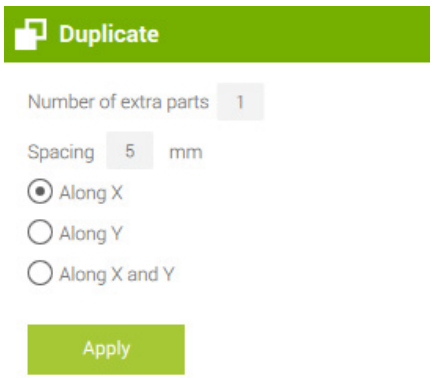



Click the Indicate Plane button to select you reference plane (the selected plane is indicated in green). The selected plane will be orientated parallel to the platform facing the upper or bottom (depending on your selection).



### Duplicate

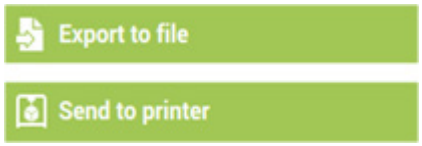
This command automatically duplicates the selected part(s).




You can specify the number of extra parts you like to create, the spacing between the duplicated parts and the direction you like to duplicate in.

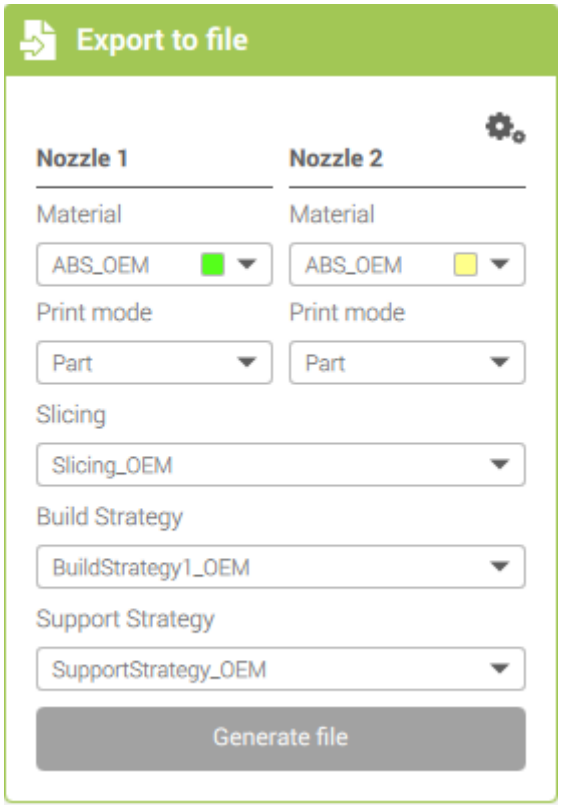
## 3.6 Export to file or print directly

Once you are happy with your part(s) you can generate a file which you can save to your computer or USB drive. However, you can also directly send the part(s) to the printer when it's connected to the computer. In the latter case, additional printer control settings such as temperature, extrusion and calibration are available.



### 3.6.1 Export to file

The export to file functions creates a machine instruction file (.gcode) for your 3D printer. To generate a file that is appropriate for your print job you will have to choose the right profiles and a print mode.
The original manufacturer profiles (OEM) are set as default, but customized profiles can be selected for:



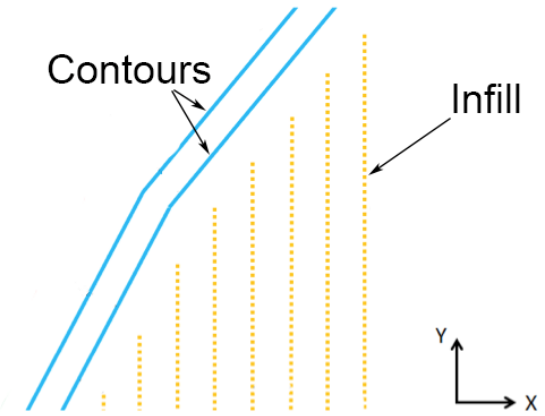

• Material: filament related parameters such as filament diameter, extrusion temperature and print speed, etc.

• Slicing strategy: parameters of slice thickness and accuracy.

• Build Strategy: parameters for printing contours (or perimeters), upskin and downskin layers and infill (fill density of a part).

• Support Strategy: parameters for support material generation. This section also includes settings of raft and skirt

**Print mode:**
The print mode is a specific option for printing with 2 nozzles to indicate for witch purpose you would like to use the nozzle.




• Part – Part: the printer will use both nozzles to print parts. You can assign a nozzle to a part by changing the color in the part list.

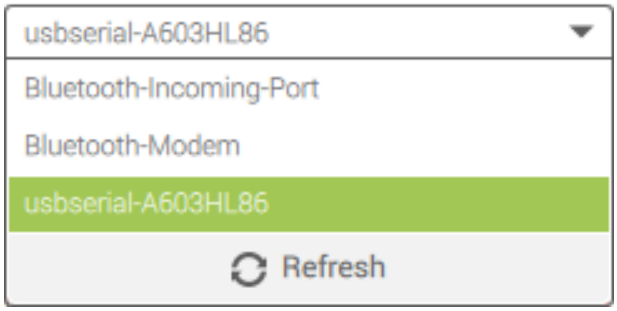
• Part – Support: the nozzle that is in part-mode will print the actual part, the other nozzle will be in support mode and will print the support material.

• Contour-infill: the nozzle that is in contour mode will print the contours of a layer and the other nozzle will print the infill.

### 3.6.2 Send to printer

The send to printer functions makes it possible to directly send your print job to the 3D printer via USB. In this functions we have 3 big sections:

- Connecting to a 3D printer
- Print settings: preparing a print job
- Printer control: perform manual printer movements, temperature settings,




First you will have to select the COM port where your 3D printer is connected to. To be sure that it is the right one you can insert the USB cable and press refresh. The port that appears should be connected to your 3D printer.

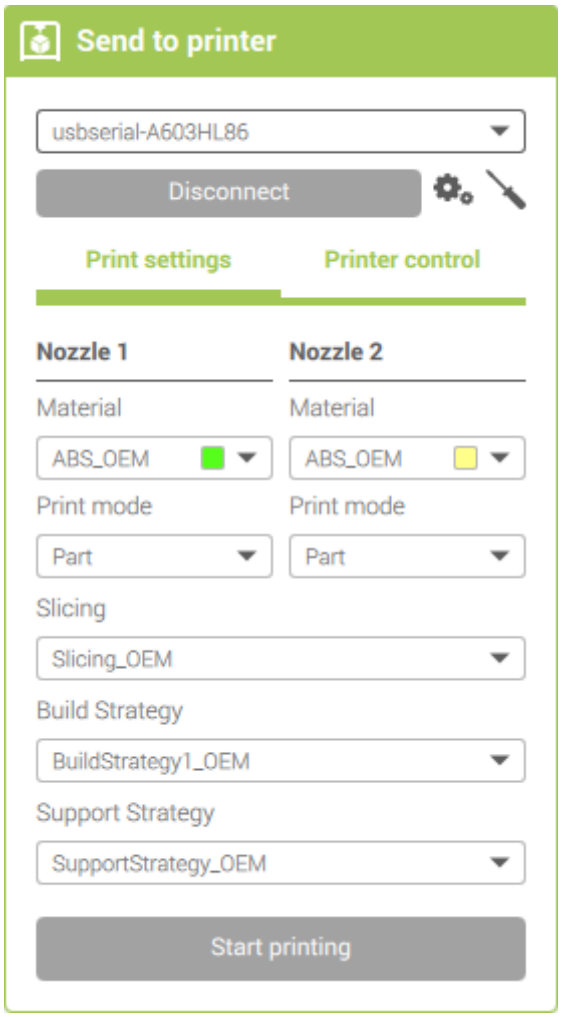
- Port- selection: select the com port where your 3D printer is connected to. Press refresh if the com port you like to connect with isn't in the list.

- Connect/disconnect button: By pressing this button you are going to setup a serial connection with your 3D printer. If you are connected the button will change to a disconnect button.

**Print settings tab menu**
To submit a print job that is appropriate for your 3D printer you will have to choose the right profiles and a print mode.

- Material: filament related parameters such as filament diameter, extrusion temperature and print speed , etc.

- Slicing strategy: parameters of slice thickness and accuracy.
- Build Strategy: parameters for printing




Note: If the port isn't appearing in the list please check if you have the right drivers installed.

contours (or perimeters), upskin and downskin layers and infill (fill density of a part).
- Support Strategy: parameters for support material generation. This section also includes settings of raft and skirt.

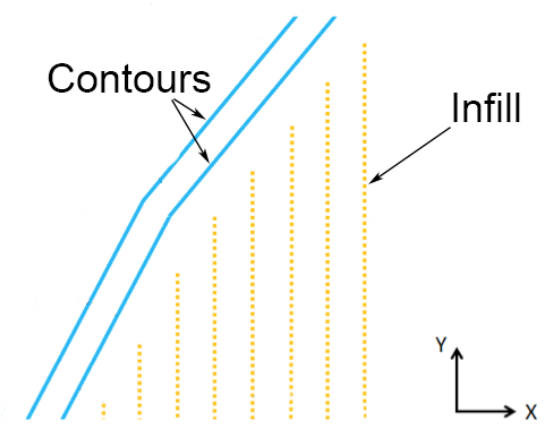
To change the profiles you need to press the edit profiles button. For specific information about the parameters in the profiles please refer to the profile editor section.

**Print mode**
The print mode is a specific option for printing with 2 nozzles to indicate for witch purpose you would like to use the nozzle.

• Part – Part: the printer will use both nozzles to print parts. You can assign a nozzle to a part by changing the color in the part list.

• Part – Support: the nozzle that is in part-mode will print the actual part, the other nozzle will be in support mode and will print the support material.

• Contour-Infill: the nozzle that is in contour mode will print the contours of a layer and the other nozzle will print the infill.

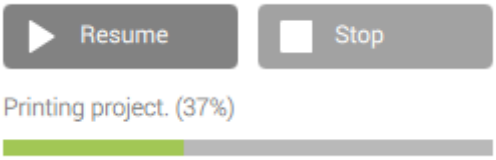


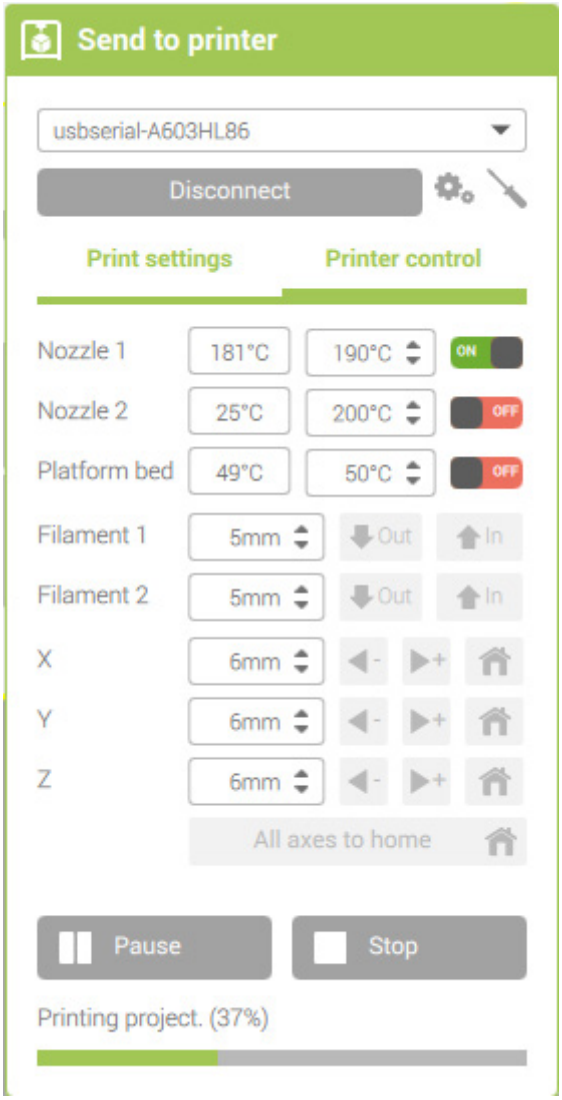

**Start printing**
When you are connected you can press "start printing", the button flips and a turns in a pause and stop button.

While printing a process bar appears to indicate the printing status. The first 37% of the bar is indicating the slicing progress, from that point on the bar moves by the progress of the print job.

The nozzle temperature will change to the ones that are set in the g-code. During a print you can monitor and manually change the temperature.

The pause button will pause a print job. On pause the printer will move up for 10 mm and to X/Y home.



### Printer control tab menu

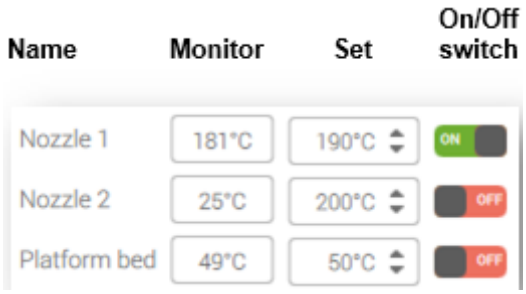
In the manual control tab menu you are able to control the hardware of your 3D printer directly.

At the top you see the printer temperature status

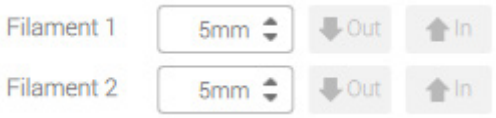
The extruder and print bed blocks allow it to change the temperature. The temperature can be set either by changing the value on the right text field or by clicking on the up and down arrows. If you change it in the text field, you need to press return or leave the field to set the value. By clicking the On/Off button the extruder or the bed is deactivated or reactivated. In the first column you can monitor the actual temperature nozzle and bed temperature.

The extrude filament controls are designed to let you manually extrude material out of the nozzle. In the textbox you can adjust the length of the filament you like to extrude out of the nozzle. The out arrow will let the material be extruded out of the nozzle on to the bed, the in arrow will retract the material back in to the tube.\

The next block controls the positioning of the extruder. With the arrow keys you move the extruder relative in any direction. The distance of one click on the arrows is specified in the text box before the arrows.

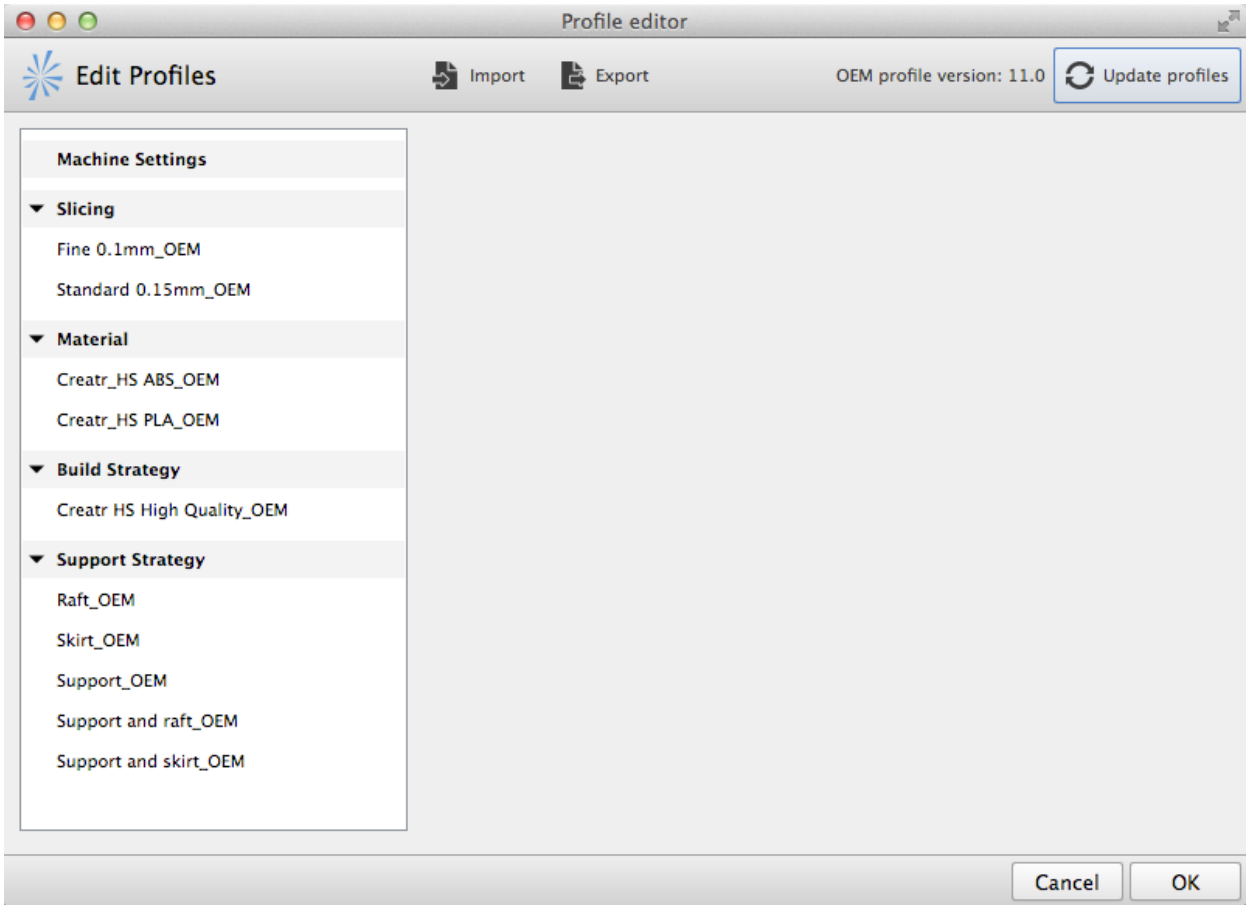



Note: You are also able to change the temperature while printing, but if a temperature is set in the machine control file this will overwrite you manual set temperature.



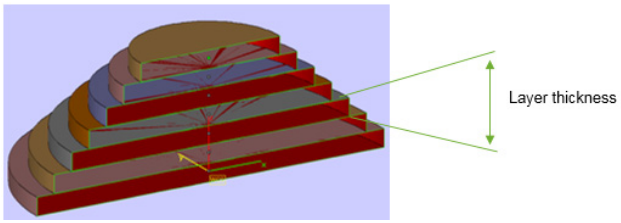

## 3.7 Profile Editor

The profile editor allows the user to setup and manage the print and printer parameters. These parameters are grouped into logical sections called profiles. Each profile has a parameter list attached to it that allow to make settings to the way the build processor behaves.



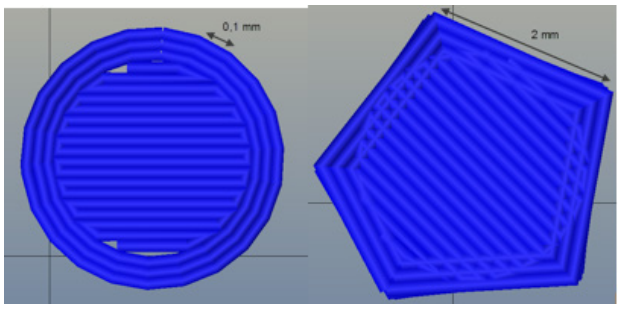

### Slicing

The slice profile section will allow the user to edit, manage and define the profiles that influence the slicing behavior of the build processor.




• Layer thickness: defines the thickness of the slicing process in mm

• Border accuracy: defines the maximum deviation in mm a toolpath is allowed to have towards the real toolpath.




TIP: For full description of all the available options in the profile editor, please refer to the Materialise Creatr Manual

**Material**
The material profile section will allow the user to edit, manage and define the profiles that define a material filament used for building a part.

• Extrusion temperature: here the user can set the temperature the nozzle needs to have to process the filament
• Bed temperature: here the user can set the temperature of the bed necessary to process the filament
• Diameter filament: here the user can set the diameter of the filament.
• Print speed: here the user can set the nominal speed at which the filament will print.
• Material density: here the user can set the density of the filament in order to calculate the consumed material during a print

In the section below 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 sugarcanes. After printing, the surface is instantly smooth. There are no toxic fumes coming 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 solved using D-Limonene |

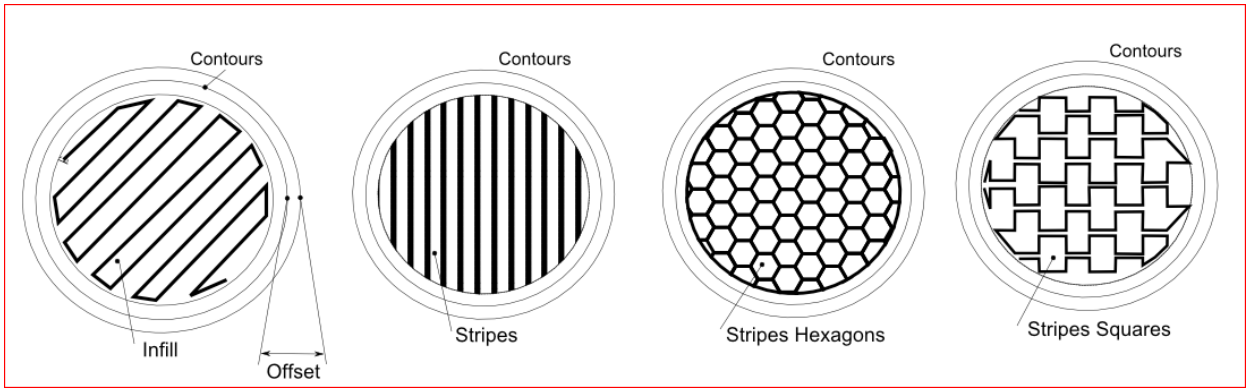
| | |
|---|---|
| Hybrid | 'Hybrid' is high in strength and toughness, extremely suitable for printing moving parts since it has an improved temperature resistance. This also means Hybrid has to be printed on a higher temperature to reach the right viscosity. It is a high strength engineering plastic 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. |

**Build strategy**
In the build strategy you can adjust the following parameters:

• Number of contours: here the user can set the number of contours that will be used to build up the model.

• Up skin/ Down skin

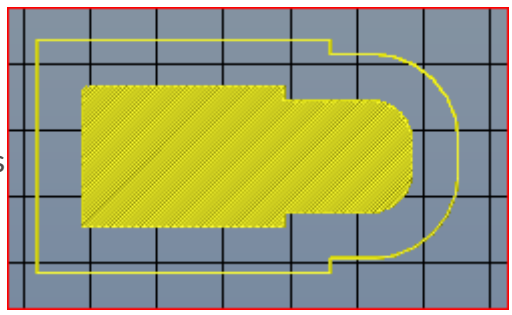
• Pattern: here the user can set the pattern that will be used for processing the infill area. There are 4 different patterns that can be used for infill:




• Raft: here the user can activate the generation of a raft structure.

• Skirt: here the user can activate the generation of a skirt for priming the nozzles.

• Base layer:here the user can set the layer thickness of the base layer.



**Support Strategy**

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

Structure with support overhang

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.

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 tab 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.

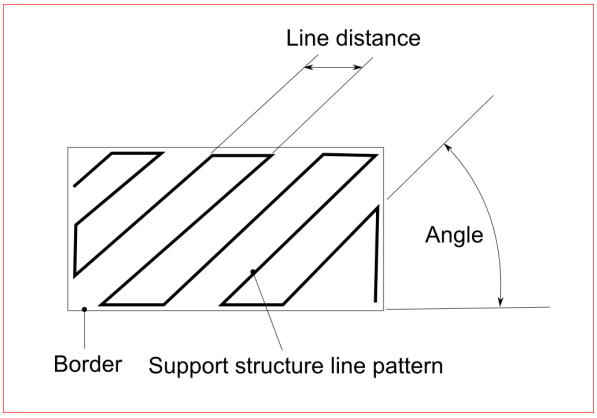
NOTE: we sometimes get feedback from clients that want to speed up the dissolving of PVA by using hotter water. Hot water might damage your print.

**Support options**

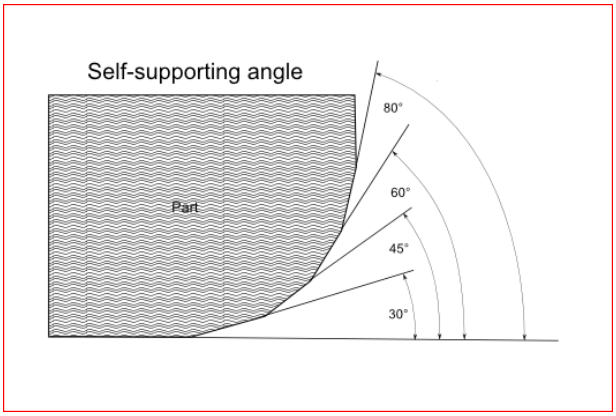
- Enabled: here the user can activate the support generation function
- Add border: here the user can activate the generation of a border around the support structure.
- Z-offset: here the user can set an additional offset on the layer height used to print the support structure.
- Line angle: here the user can set the angle that will be used to draw the support structure lines.
- Line distance: here the user can set the distance between 2 lines of the support structure pattern

- XY extension: here the user can set the off-set in X & Y that will be used to generate the support structure. The support structure will be extended in X&Y

- Distance to part: here the user can set the distance the support structure needs to keep from the part(s) during generation of the support structure.



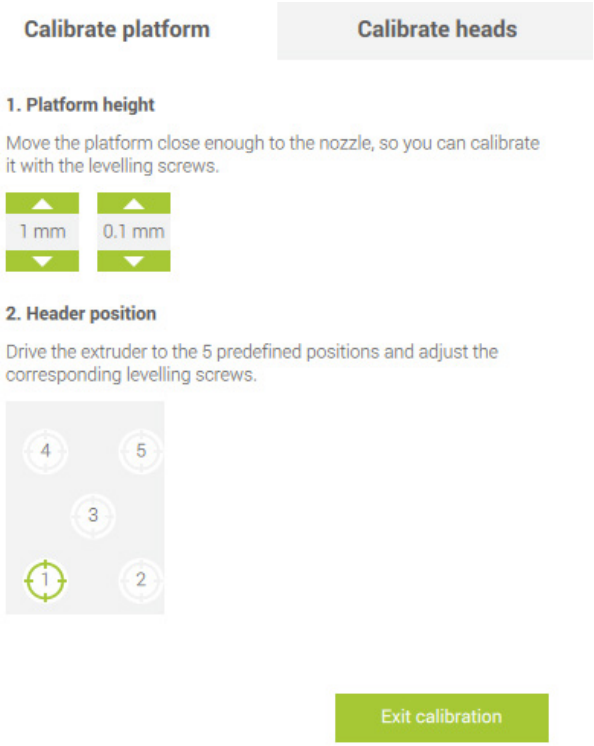

- Self-supporting angle: here the user can set the self-supporting angle that will be used to determine which parts of a model need support and which not during the support generation. All areas that have a support angle lower than the self-supporting angle will be supported.




TIP: For full description of all the available support options, please refer to the Materialise Creatr Manual.

### 3.8 Calibrate printer

This wizard consists out of two tabs:
- calibrate platform
- calibrate heads.




<u>Calibrate platform</u>
For a successful print it is important that the table is properly calibrated. The distance between the nozzle and table should be the same at every x,y position. The leveling mechanism of the table can be adjusted by turning the nuts underneath the heated bed. It is important to check it first, because the leveling can change during shipment.

Move the nozzle to all the points and check that the distance between the nozzle and the platform is equal across all the points. If it isn't adjust the nuts underneath the platform.

<u>Calibrate heads (extra calibration for dual nozzle printers)</u>
1. If both separate extruders have proven to print well, it is time to calibrate the distances between them. Print out the calibration parts by pressing the print calibration parts button. In this print 2 layers of squares are printed. Layer one is printed with extruder 1 and layer 2 is printed with extruder 2 and shifted.
2. After you have selected the squares that align on top of each other, you will send these settings to the eeprom of the printer by pressing the finish calibration button.
3. You finished the calibration and you should be able to print successfully with your dual head printer.

# 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?**

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. |

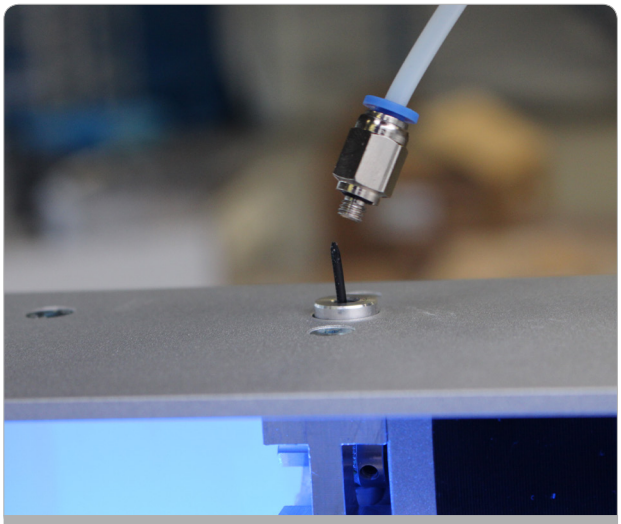
(For all recommended settings, please visit: creatrhs.lpfrg.com)

- 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 unsufficient 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.


Push-in coupling

**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.

1.5mm steel tool in nozzle

**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 Materialise Creatr 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.

# 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 HSunderstands and receives its instructions from. The Materialise creatr 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.

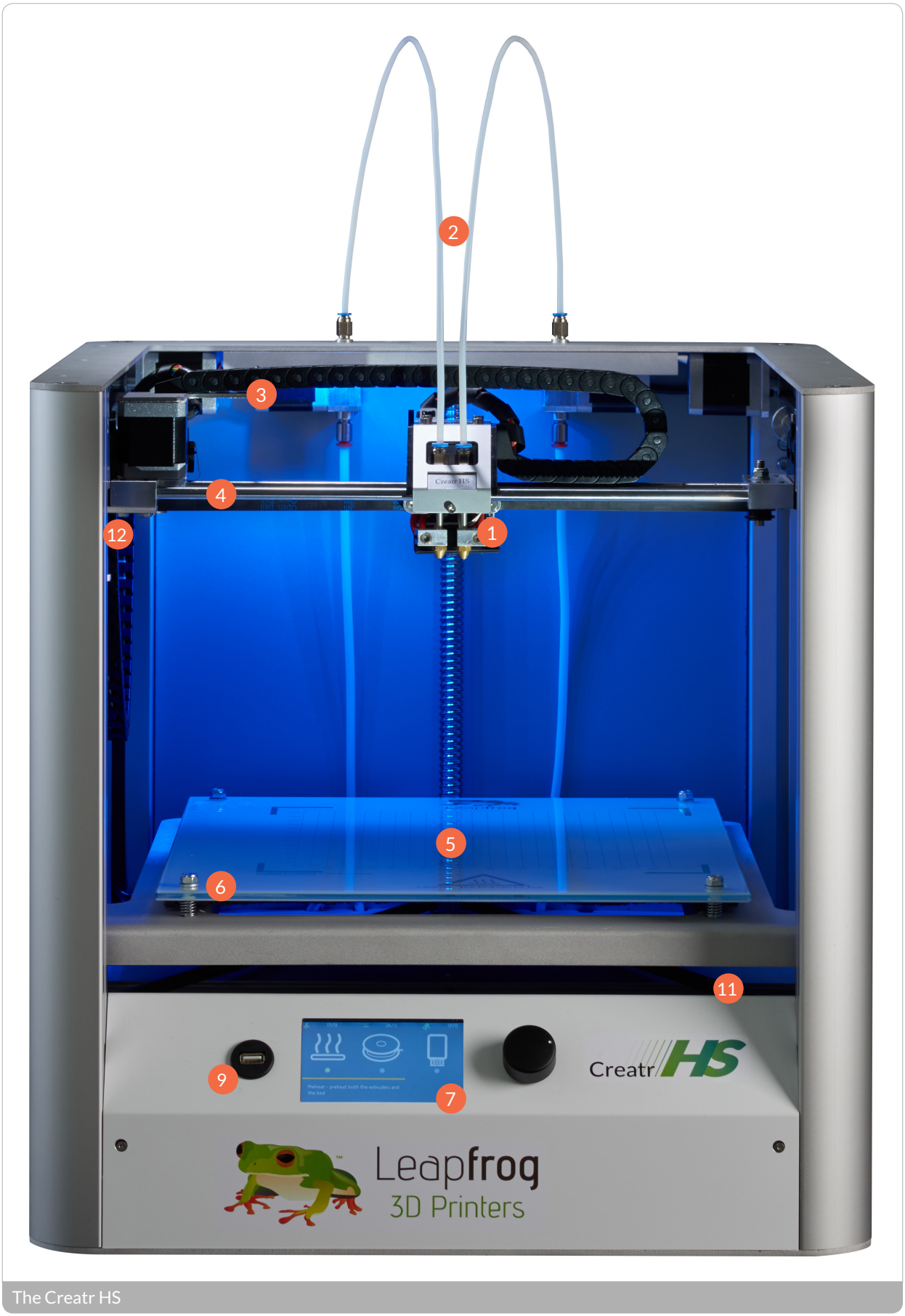
**Retracting**
Retracting means that the filament is pulled out of the extruder. It is the opposite of extruding.

**Slicing**
Slicing is the process through which the stl file (or the 3D model) is transferred into a gcode (printable file). Materialise Creatr 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.

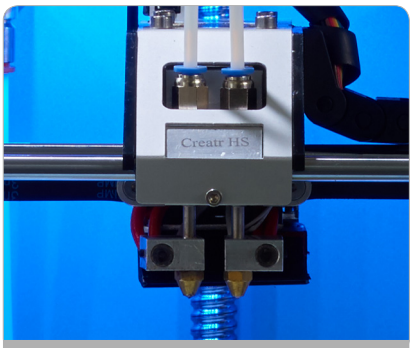
**Support material**
Due to the printing technique used, the Creatr HS has limitations in printing objects with more than 45 degrees overhang. To overcome this problem the printer can print a support structure which literally supports your actual print.

## 5.2 Getting to know your printer



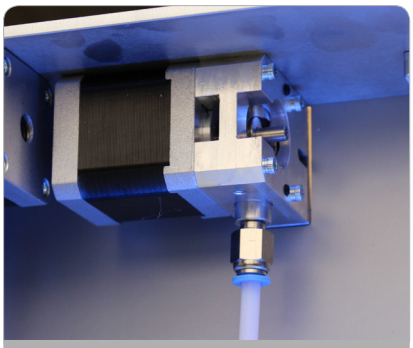

The Creatr HS

8 10 positioned at the back of the printer

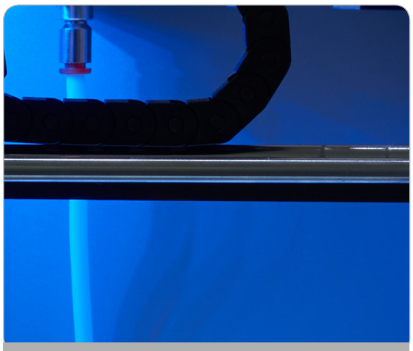



1. Printing head and extruders

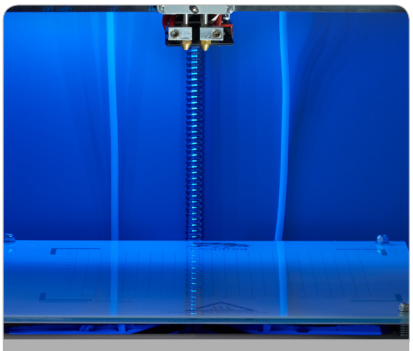
2. Filament guiding tubes



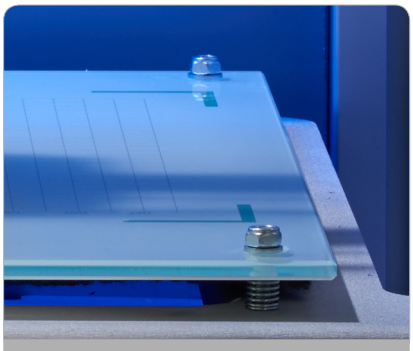
3. Filament drive unit



4. Axes



5. Print bed



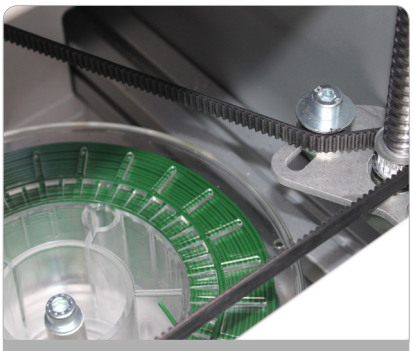
6. Print bed levelling knobs


7. Display

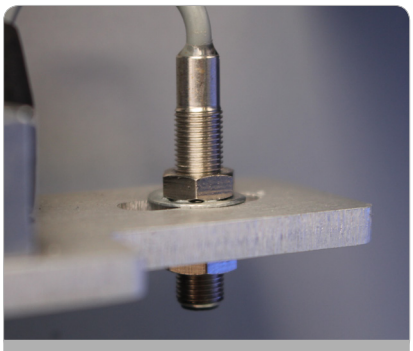
8. Quick release knob

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!)

**Leapfrog 3D Printers**
H. Kamerlingh Onnesweg 10
PO Box 252
2408 AW Alphen aan den Rijn
The Netherlands

T +31 (0)172 503 625
W www.lpfrg.com
E info@lpfrg.com

© Leapfrog 3D Printers