



Manual Leapfrog Creatr



Manual Leapfrog Creatr

Dear customer,

Congratulations with the purchase of your very own Leapfrog Creatr Single or Dual Extruder! We would like to help you get started so you can make your first 3D print on the Creatr and many more.

This is why we have developed this document; it will guide you through unpacking, installation and the print settings. We also provide you with guidelines to get you back on track if something should go wrong.

It is possible that after reading this document or while operating your printer you have a question that is not listed in this document. Therefore, we also explain you how to contact our support department.

We wish you happy printing!

With kind regards,

The **Leapfrog** team



Manual Leapfrog Creatr



Table of content

1. Installation

1.1 Unpack	Page 7
1.2 230 V → 115 V	Page 9
1.3 Software	Page 10
1.4 Connect	Page 16

2. Printing

2.1 Print sticker	Page 18
2.2 Connection	Page 20
2.3 Preheat	Page 20
2.4 Filament	Page 23
2.5 Print settings	Page 25
2.6 Start your print	Page 29

3 Troubleshooting - Knowledgebase

3.1 Oh, oh, my filament is not feeding through	Page 36
3.2 My object won't stick to the bed	Page 37
3.3 Extruder stops mid-print	Page 39
3.4 Min-/max temp triggered error messages	Page 47
3.5 Nozzle seemed clogged	Page 50
3.6 My extruders are not the same height	Page 54
3.7 Uploading firmware	Page 55
3.8 Need support?	Page 59

1. Installation

The following steps are steps you only have to follow when your brand new Creatr arrives at your doorstep for the first time. Please follow each step thoroughly to make sure the installation will be a success and you can find yourself as a member of the Leapfrog eco-system.

This section contains the following:

1.1 Unpack

- How to unpack your Creatr and where to plug in your cords

1.2 230 → 115 V

- If necessary, how to switch your Creatr to 115 V

1.3 Software [Windows]

- How and where to download the necessary software

1.4 Install

- How and where to install the drivers and the software

1.5 Connect

- How to let the computer recognize your brand new Creatr

1.1 Unpack

In this step we will show you how to unpack your Creatr when it arrives at your doorstep. Before you start, make sure that you have cleared a spot where you want to place the printer. We advise you to not place the Creatr next to a window or at a place where it is draughty.

- Unscrew all the top screws of top plate of the shipping crate and remove the top plate.



- Remove the filament you ordered and open the cardboard box. Remove the plastic that is placed on top of the printer and take out the printer while holding it as shown in the picture below. It is best to do this with two people.



- After positioning the printer on the desired spot, remove all plastic that is used to wrap the outside of the printer.

Manual Leapfrog Creatr

- Cut the tie wraps on the axes [3 in total].



- Insert the power cord and USB-cable in the backside of the Creatr.



- Plug the power cord in the outlet, the USB-cable in your computer and turn on your Creatr.



Manual Leapfrog Creatr

1.2 230 V → 115 V

!!! This step is only necessary if your electrical grid provides 115V !!!

WARNING: NEVER PERFORM THIS OPERATION WHEN THE POWER PLUG OR USB CABLE IS PLUGGED INTO YOUR CREATR!

- If you already connected your Creatr to your computer and power grid, unplug your power cord and USB cable!
- Tilt your Creatr so you can access the bottom. Look for the square hole.
- In this hole you can see a yellow sticker and under it a red switch.
- Use an Allan key or screw driver to carefully move the switch to the left. Your Creatr now operates on 115 V.

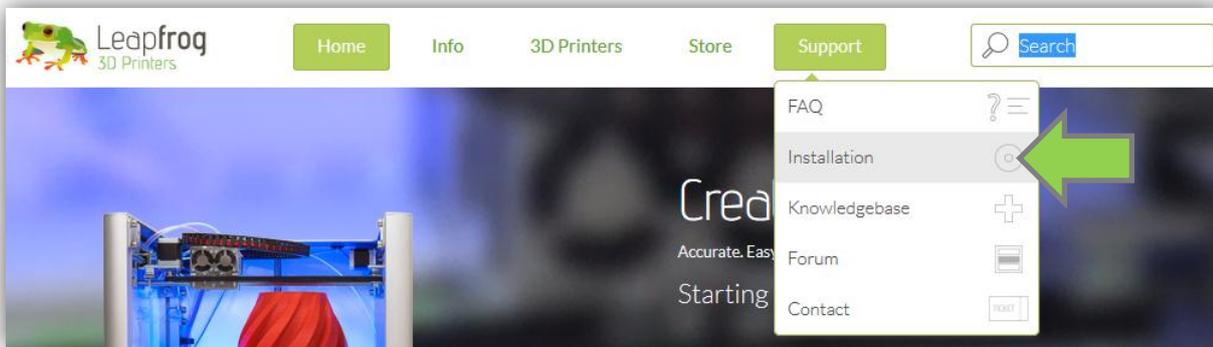
WARNING: NEVER PERFORM THIS OPERATION WHEN THE POWER PLUG OR USB CABLE IS PLUGGED INTO YOUR CREATR!

1.3 Software (Windows)

Here, we will explain you where to download and install the necessary software that communicates to your printer and turns your design into a printable file.

Download

- Open the Internet browser and go to www.lpfrg.com > “Support” > “Installation”.



- At “Part 2 of 3: Software and configuration” select your operating system [Windows].

Part 2 of 3: Software and configuration

In this movie, we will show you how to install the necessary software. You will need to download and install the Arduino software to make sure your 3D printer and your computer can communicate with each other. The next step is to install the Repetier software that controls your 3D printer. We have tried to make this as simple as possible and also provided direct downloading links per operating system.

Please select you operating system and follow the descriptions in the movie.

Operating system

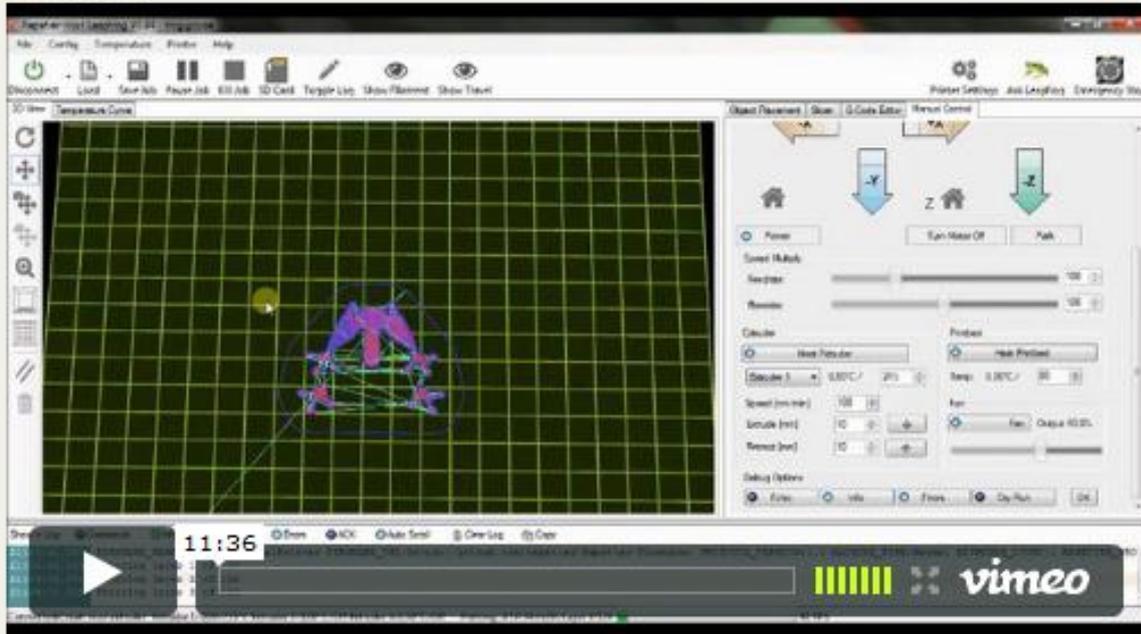
Windows
Mac



- On the website, click on the text “Arduino software”.

Operating system

Windows



The first step is to download and install **Arduino software** to make it possible for your computer to communicate with your printer.



- Download “Arduino 1.0.5” (ZIP file) and open the file with an archive utility like Winrar or WinZip.

Download

Arduino 1.0.5 ([release notes](#)), hosted by [Google Code](#):

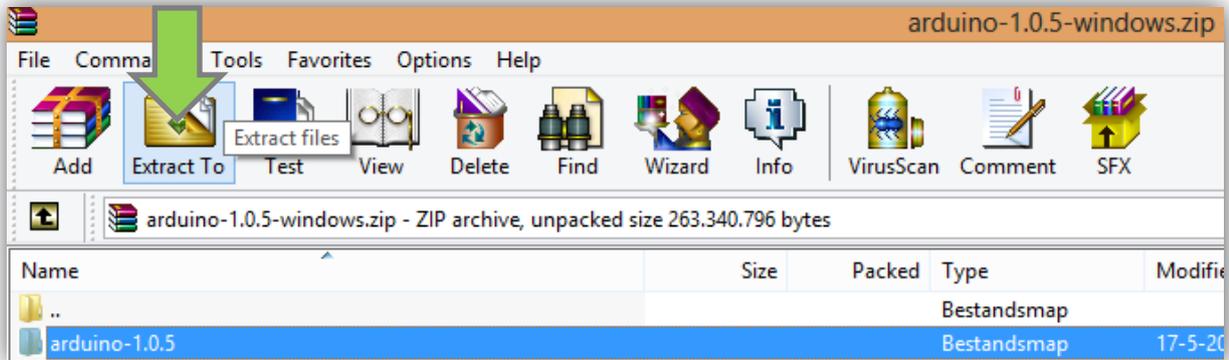
- + [Windows Installer](#), [Windows \(ZIP file\)](#)
- + [Mac OS X](#)
- + [Linux: 32 bit, 64 bit](#)
- + [source](#)



Next steps

- [Getting Started](#)
- [Reference](#)
- [Environment](#)
- [Examples](#)
- [Foundations](#)
- [FAQ](#)

- Unpack the content to any location.



- Go back to www.lpfrog.com/support/installation and on the website click on the text “Leapfrog Repetier software” to download “Repetier-Host Leapfrog setup” from the website.



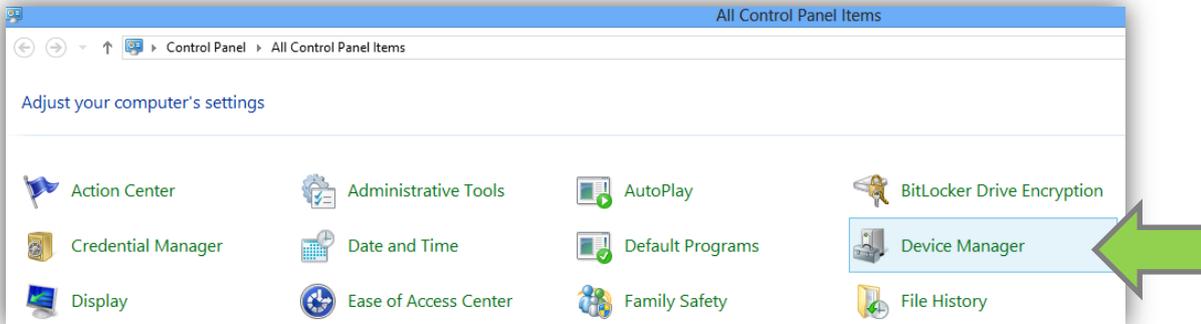
The first step is to download and install **Arduino software** to make it possible for your computer to communicate with your printer.



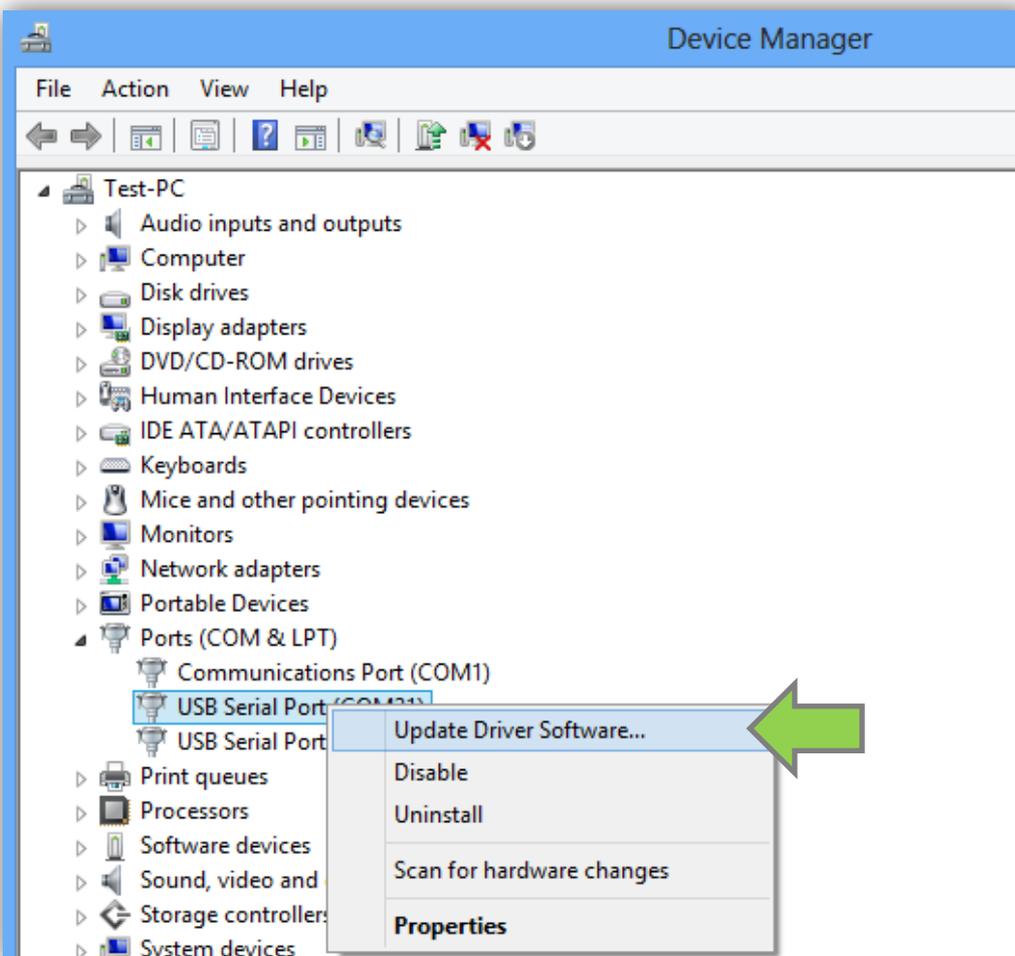
The next step is to download the **Leapfrog Repetier Software** as shown in the video. This software allows you to convert your 3D software file [STL for example] to a printable file [G-Code] and allows you to control your printer. The Leapfrog Repetier software contains standard Creatr Print Profiles. Just select the extruder, the

Drivers

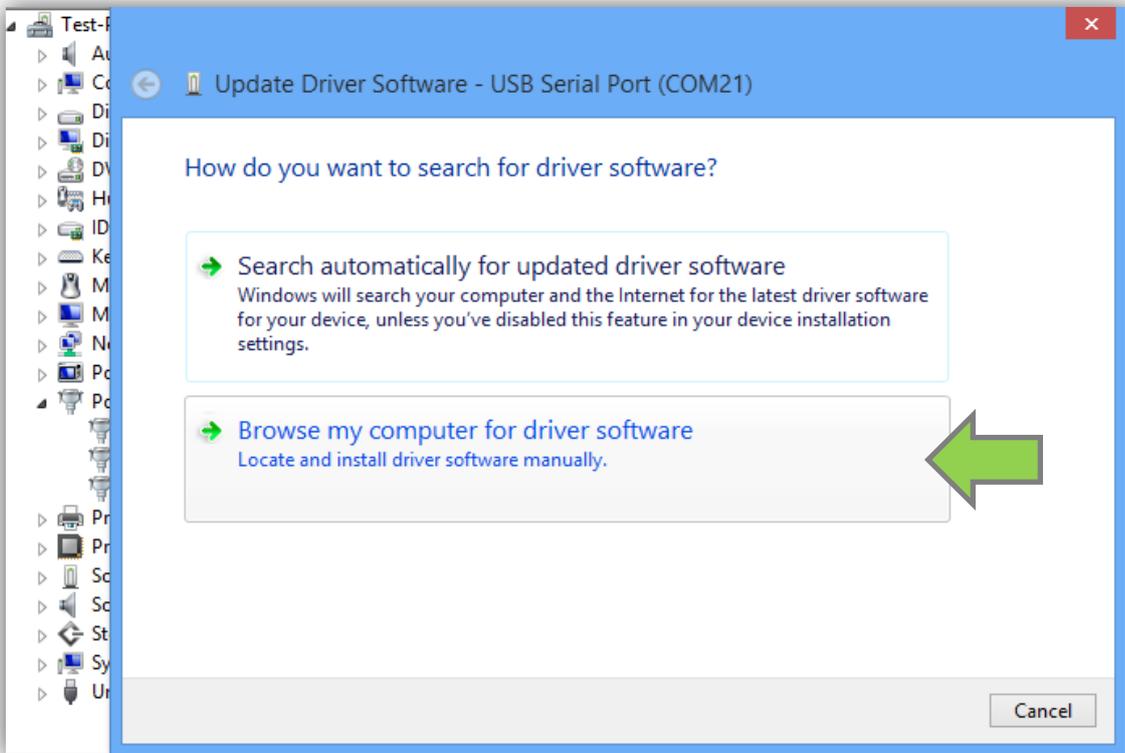
- On your computer, go to “Control Panel” and select “Device Manager” .



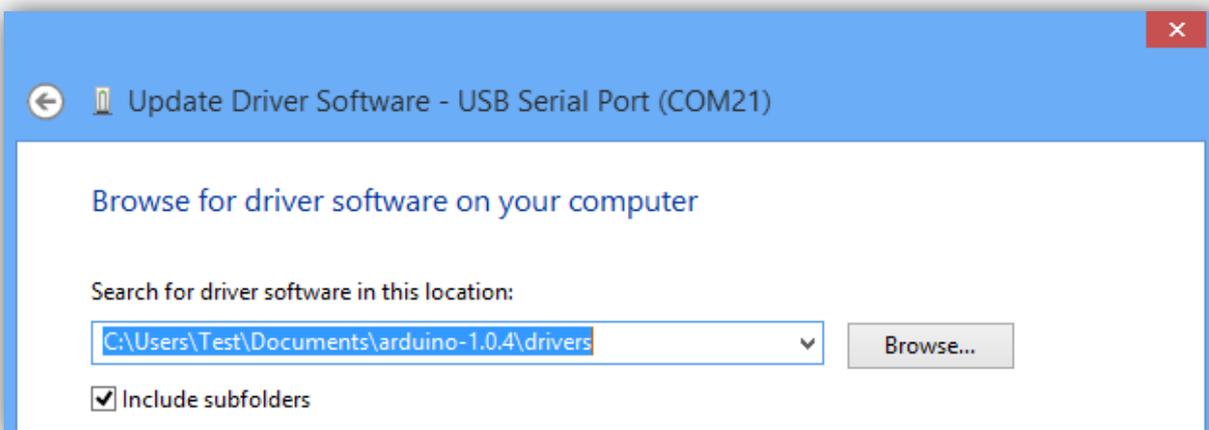
- Go to Ports, right-click on “Unknown Device” and select “Update Driver Software”.



- Click on “Browse my computer for driver software”.

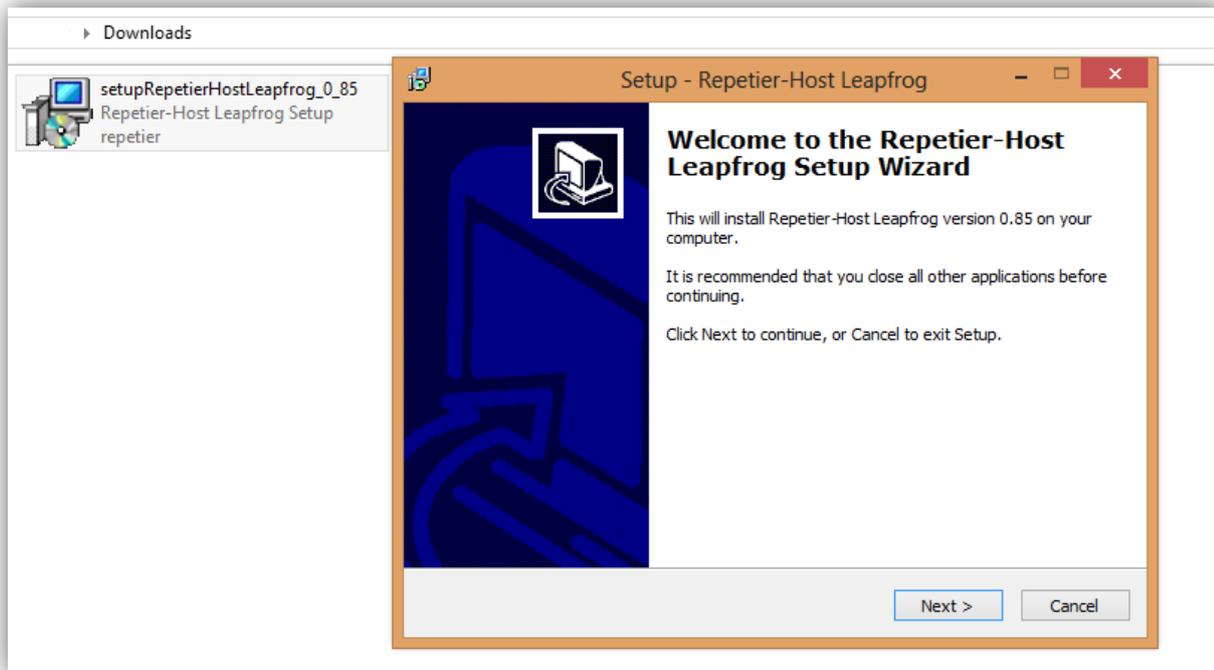


- Select the location of your Arduino files. Select the “driver map” and click “Next”. The printer will now be recognized by your computer.



Leapfrog Repetier-Host Software

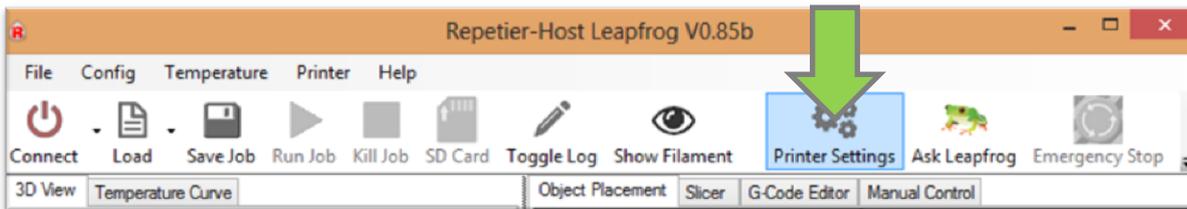
- Go to the location where you downloaded your “*Repetier setup file*” and run the setup.



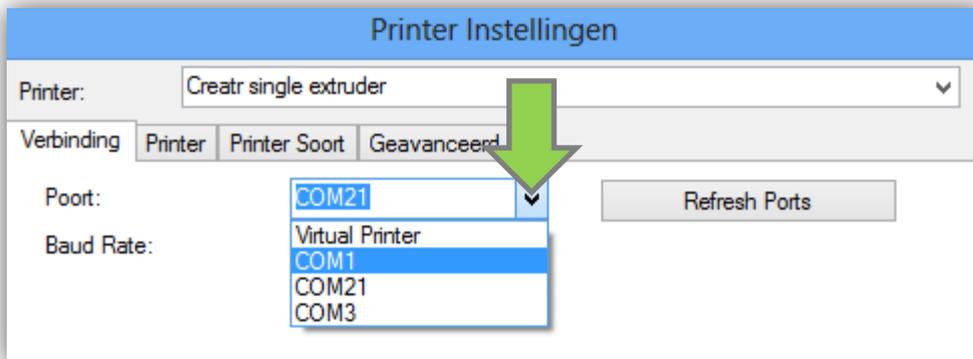
1.4 Connect

In this step we will show you how to connect the printer to your computer.

- Open Repetier-Host Leapfrog and go to “Printer Settings”.



- In the *Device manager* you can see which *COM port* you need to select [the number is displayed after the driver installation].





Printing

The following steps are general steps you need to follow when making a 3D print with your Creatr. Please follow each step to make sure you will be able to print a beautiful print.

This section contains the following:

2.1 Print sticker

- How to check the sticker and how to replace it if necessary

2.2 Connection

- How to connect the printer to the computer using the software

2.3 Preheat

- How to preheat your printer so you are able to print

2.4 Filament

- How to insert or change your filament and extrude a test piece

2.5 Print settings

- Which printing settings are there and when to select which one

2.6 Start your print

- How to start a print using a G-code or a STL file

2.7 Dual color printing

- How to print a dual color print

2.1 Print sticker

In this section, we will show you when you will need to replace your sticker and how to do this.

- Remove your [test] print from the print bed. You can use a putty knife to make it easier to remove your printed object. If the print still sticks on the printbed, turn on the print bed to 40 degrees Celsius [in this case please first follow step 6 & 7]. Heating the printbed can make it easier to remove prints. It is easiest to remove prints immediately after finishing printing.
- The print sticker is used to make sure the print material will stick to the print bed. Without a print sticker, adhesion is not possible. First check if the print sticker can be re-used. You can re-use the sticker until there is no space left that is undamaged. We advise you to position your object on an undamaged part of the printing sticker.



Example of a sticker that needs replacing

- 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.

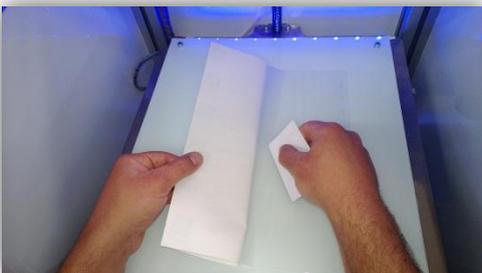
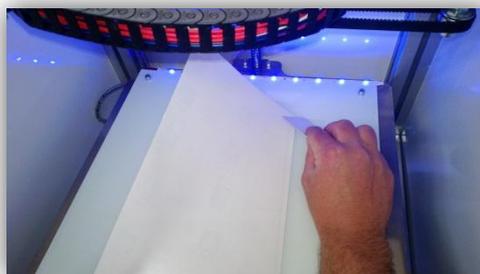


Manual Leapfrog Creatr

- Remove the sticker glue by applying sticker remover or glass cleaner on the printbed. 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.



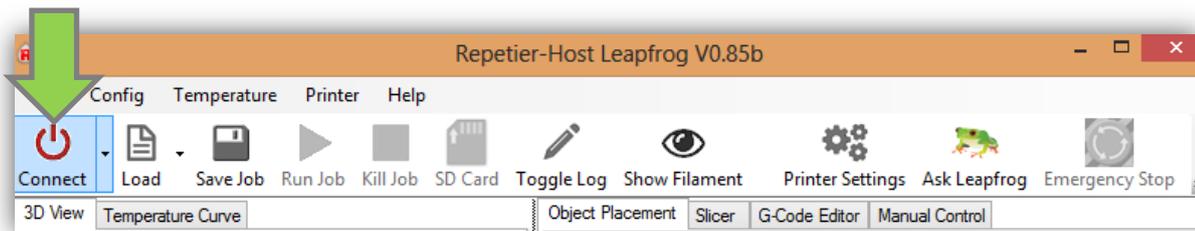
- Apply the new sticker by placing the sticker on your print bed. If you are satisfied with the position, take of the small part of the sticker. Use a plastic card to apply the sticker, work from the inside to the outside and avoid air bubbles. Take off the big part slowly and use a plastic card to apply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



2.2 Connection

In order for your computer to tell your printer what to do, you will need to connect your computer to your printer. In this step we will show you how to do this.

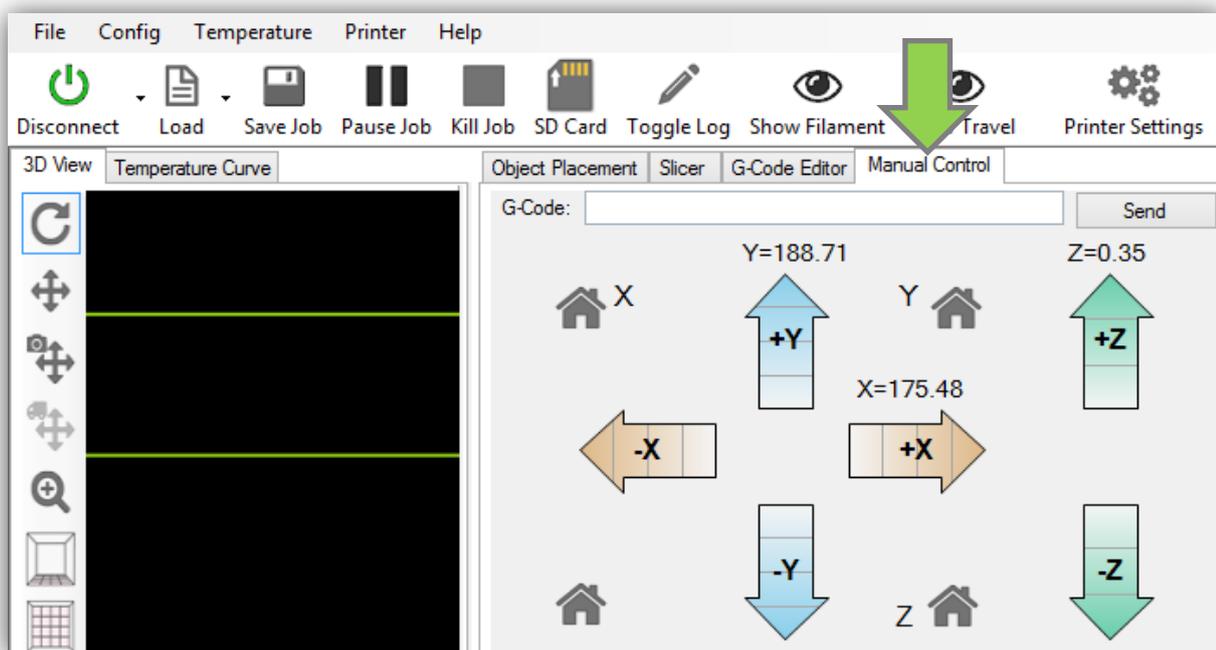
- Open Repetier Host Leapfrog and click the “Connect” button. The button will change from red to green. Your printer is now connected to the computer.



2.3 Preheat

Before you start printing, it is necessary to preheat your printer. The print material needs to be melted and your printbed needs to be warm to make sure the material will stick to your bed. Here, we will explain you how to do this.

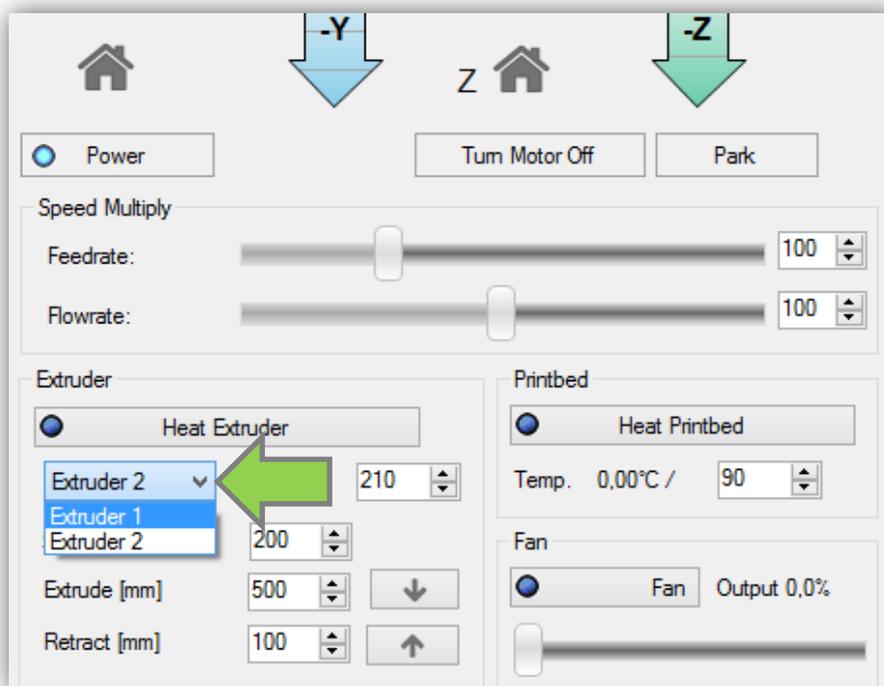
- Go to the “Manual Control” tab.



- Select the extruder you want to print with. If you have a Creatr Single Extruder it is only possible to select one extruder.

If you have a Creatr Dual Extruder, it is possible to select two extruders. Follow the below mentioned steps for each individual extruder if you are making a dual colour print [a print with two colors] or if you are printing with a combination of ABS, PLA or PVA.

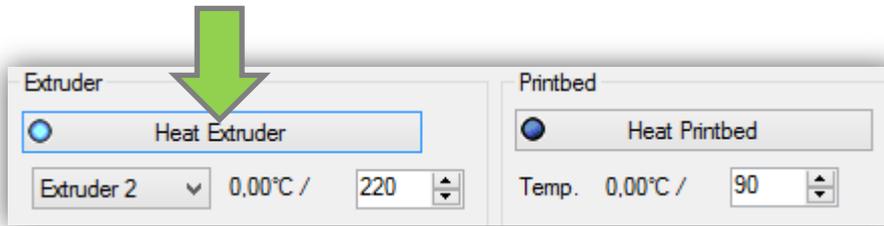
“Extruder 1” is the right extruder, “Extruder 2” is the left extruder. If you are printing with two extruders, make sure you heat them both.



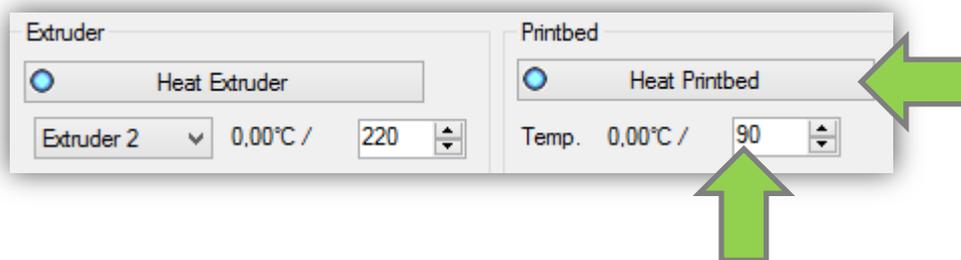
- Enter the temperature in degrees Celsius for each extruder. For ABS enter 230 degrees, for PLA enter 220 degrees. When you are printing your object, the printer will automatically use the predetermined temperature that matches with the filament you have selected in the software.



- Click on *"Heat Extruder"*. The extruders need to be heated in order for the filament to melt to enable you to print your object. Depending on your print, select to heat one or two extruders.



- Set the temperature for the printbed [80 degrees Celsius for ABS, 60 degrees Celsius for PLA] and click *"Heat Printbed"*. The printbed needs to be heated in order for the filament to stick on the printbed.



2.4 Filament

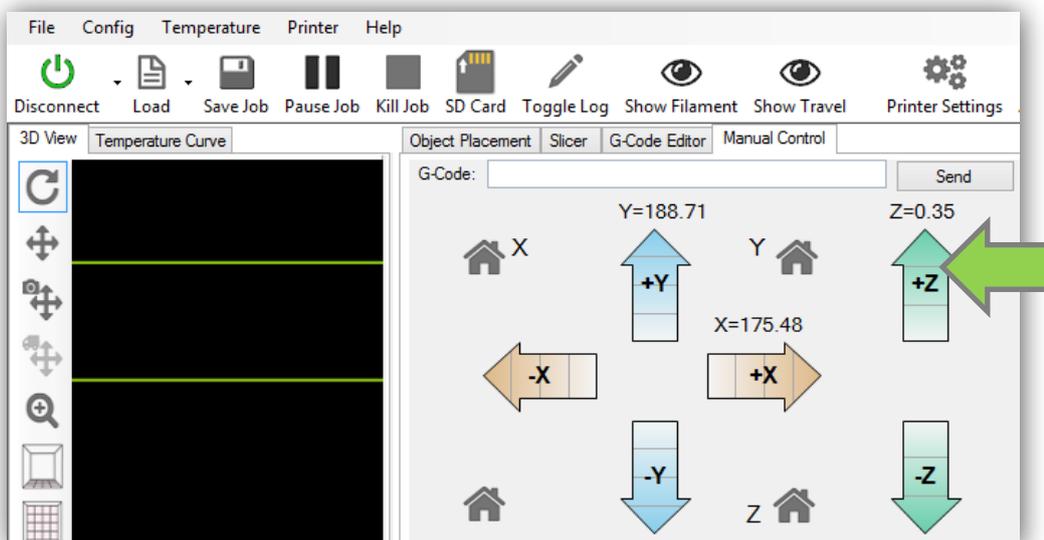
In this section of the manual we will explain you how you can load new filament and how to switch between filaments that you already have loaded.

Insert new filament

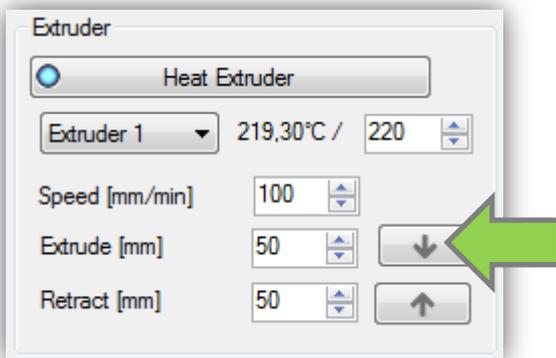
- Place your filament on the turntable in your Creatr.
- Feed the filament through the holes until it reaches the drive gear of the extruder.



- Lower the printbed by clicking on the “+Z” button. This is necessary because you will be extruding some filament and this needs space.



- Go to the “Manual Control” tab, at “Extrude” enter “50” mm and click on the “Extrude button” [arrow] to manually extrude filament. Your Creatr will start extruding when the extruder is hot enough to extrude [above 220 degrees]. We recommend to always extrude some filament before you start a new print.

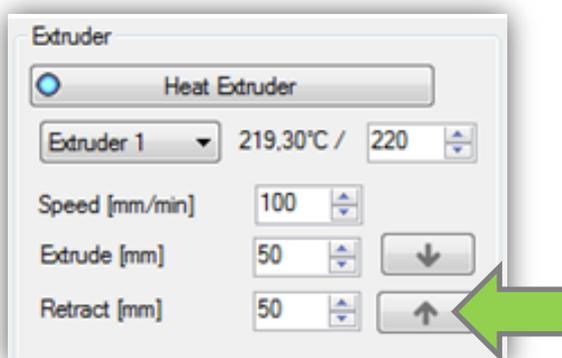


- Extrude until the material comes out the nozzle and then remove the printed filament.
!!! Be careful, the nozzle of the printer is very hot !!!

Remove old filament

[Only necessary if there is already filament in your printer and you want to switch to a different filament]

- Heat the extruder, when it is heated to 220 degrees click the “Retract button” until the filament comes loose at the top of the extruder.



- Pull the filament by holding on to the cold filament out the filament hole upwards and remove the spool.

2.5 Print settings

To fully understand the print profiles, it is best to first explain certain settings involved in the printing process.

Support material

- General explanation

Support material is added during the printing process in order to support the overhanging parts of your design. Not all shapes need support material. For example, the left cup you see below can be printed without support material, because all layers can be easily printed on the layers below.

However the cup on the right is not printable without support, because the printer cannot print the “ear” part into thin air. If you would print this design without support material, the print material will just drop down on the place where the ear begins. In this case you will need to print your object using support material.

When printing with support material, there is no need to design the support structure yourself, the software will automatically calculate this for you when selecting one of the support profiles.



- Breakaway support

There are two types of support; 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. Here, the support material is printed with one and the same extruder.

Therefore, this type is great for those owning a Creatr Single extruder. When printing the support material, the support that is generated results in an easy to break away structure when the print is done. If the prints are very small this may become a lot harder. The downside of breakaway support is that you will see the marks of the support on the surface of your print, as you have broken it away. Depending on your demands, your object needs a finishing treatment.



Manual Leapfrog Creatr

- Soluble support

This support type is only accessible for those owning a Creatr Dual extruder, as you will need to print with two different materials and extruders. The advantage of soluble support is that it dissolves and does not leave any marks on the surface of your print.

The Creatr supports PLA and PVA as soluble support material. PLA is soluble in sodium hydroxide [drain cleaner] mixed with water and PVA is solvable in ordinary tap water. The soluble support material needs to be inserted in the second extruder. **Warning, you should follow the guidelines as listed on the package while using sodium hydroxide!**

After your Creatr 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 will dissolve.

- No support

When your print does not need any support material because it does not have any overhanging parts that need support, just like the left cup in the example, you can select the option “No support”.

High quality or fast printing

- Now that you have become familiar with the support types available, we will tell you the difference between high quality printing and fast printing.

With 3D printing you will always ask yourself: “Do I need a quick prototype or the final product?” The answer to this question affects your choice for the print profile.

- High quality printing

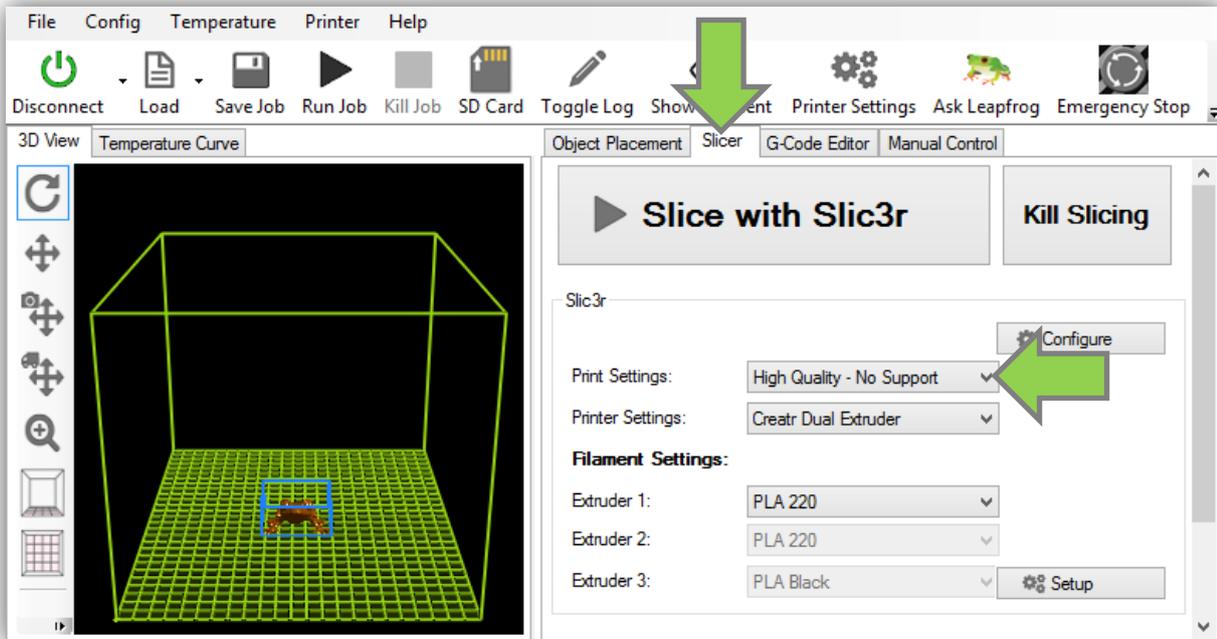
Choosing for high quality will give you the best results. This will mean that the layer height of each layer is very small [0.15 mm, experimental to 0.1 mm], leaving an almost flat finish. When selecting a high print quality, your layers become more than twice as small compared to the fast printing setting. This will result in twice as much movements needed by the print head and therefore it also takes twice as much time to print your object.

- Fast printing

When you need a very quick prototype, just to check dimensions of your product or to quickly show someone the idea, we would recommend the fast printing profile. The fast print profile prints with a layer thickness 0.35mm. Here, you will notice a less smooth finish surface, but choosing the fast printing profile will also speed up the print time at least twice.

Each print setting explained

Below, you can see an overview of all the pre-determined print settings. Please read through all of them to understand the difference between the settings. Print settings can be selected in the Slicer tab. Print settings need to be selected before slicing your design as they cannot be altered afterwards.



- **High quality – No support**

We advise to select this setting when:

- Your print does not need support &
- You want a high quality print

Printing without support will affect the print time positively. If your print does not need support and you want a high quality print, select this option. This printing setting can be used by the Creatr Single and Dual extruder.

- **High quality – Breakaway support**

We advise to select this setting when:

- Your print needs support &
- You want a high quality print &
- You do not have a Creatr Dual extruder or soluble support material [PLA or PVA]

Here, you will use ABS for both your object as for your support material. The software will take into account that you will need to breakaway the support material. Depending on your requirements you will need to perform finishing techniques like sanding etc. This printing setting can be used by the Creatr Single and Dual extruder.

- **High quality – Soluble support**

We advise to select this setting when:

- Your print needs support &
- You want a high quality print &
- You have a Creatr Dual extruder & soluble support material [PLA or PVA]

Here, you will need to print your object in ABS and your support material in PLA or PVA [use the second extruder for support material]. This means that after printing, you can place your object together with the support material in the correct solvent for your support material [see sub-chapter Support] and your support will dissolve. This leaves a beautiful printed object without any traces of any support material. This printing setting can only be used by the Creatr Dual extruder.

- **Fast printing – No support**

We advise to select this setting when:

- Your print does not need support &
- You want your print as fast as possible &
- You do not need a high quality print

Printing without support will affect the print time positively. When selecting the fast printing option, your layers will be thicker and your print will be printed faster. This printing setting can be used by the Creatr Single and Dual extruder.

- **Fast printing – Breakaway support**

We advise to select this setting when:

- Your print needs support &
- You want your print as fast as possible &
- You do not need a high quality print &
- You do not have a Creatr Dual extruder or soluble support material [PLA or PVA]

When selecting the fast printing option, your layer will be thicker and your print will be printed faster. By choosing Breakaway support, you will be able to breakaway the support material after printing. Depending on your requirements, you will need to perform finishing techniques. This printing setting can be used by the Creatr Single and Dual extruder.

- **Fast printing – Soluble support**

We advise to select this setting when:

- Your print needs support &
- You want your print as fast as possible &
- You do not need a high quality print &
- You have a Creatr Dual extruder & soluble support material [PLA or PVA]

Here, you will need to print your object in ABS and your support material in PLA or PVA [use the second extruder for support material]. This means that after printing, you can place your object together with the support material in the correct solvent for your support material [see sub-chapter Support] and your support will dissolve. This leaves a printed object without any traces of any support material. This printing setting can only be used by the Creatr Dual extruder.

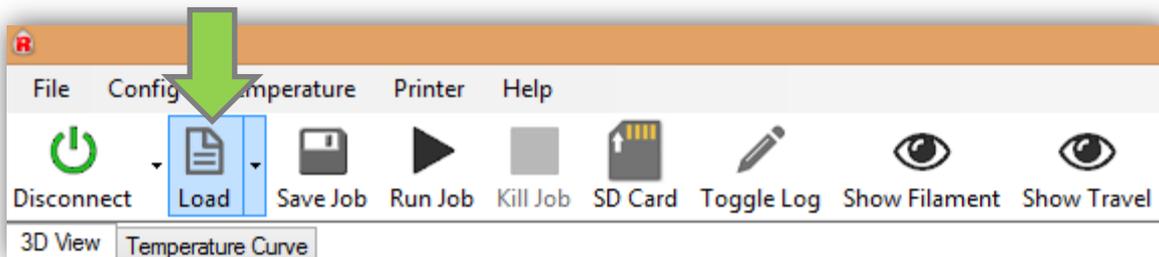
2.6 Start your print

This section of the manual will explain you how to print a file if you already have a printable G-code or if you start with a STL file. STL files are files that contain your design, but do not contain the movements the printer needs to make while it is printing. This is why a STL file needs to be converted into a G-code, a file format that tells the printer how to move in order to create your design.

I have a G-code

A G-code is a file format the printer uses to print. It is comparable with a PDF file in a sense that it cannot be altered after you have created it. All your settings like print material, speed, layer thickness etc. are in the G-code.

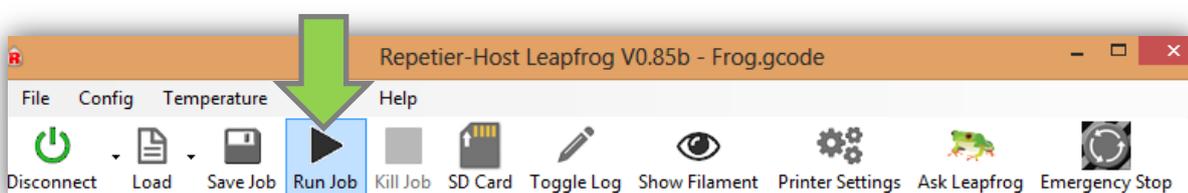
- If you want to print a pre-existing G-code, click “Load”.



- Open a G-code file.

- Make sure the print bed is heated to 80 degrees for ABS or 60 degrees for PLA and click on the “Run Job” button to start your print. Your print will not start until the printbed and the extruder[s] are heated to the temperature you have selected.

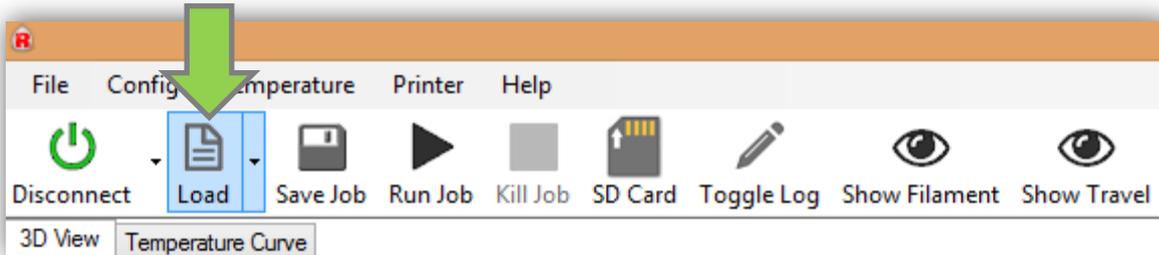
Also note that the printbed will automatically move to the “Home” position when starting a new print. The home position is the position where the value for the X-, Y- and Z-direction is zero. You can see that the bed is placed at the highest point and the extruders are positioned at the front left side of the printbed.



I have a STL file

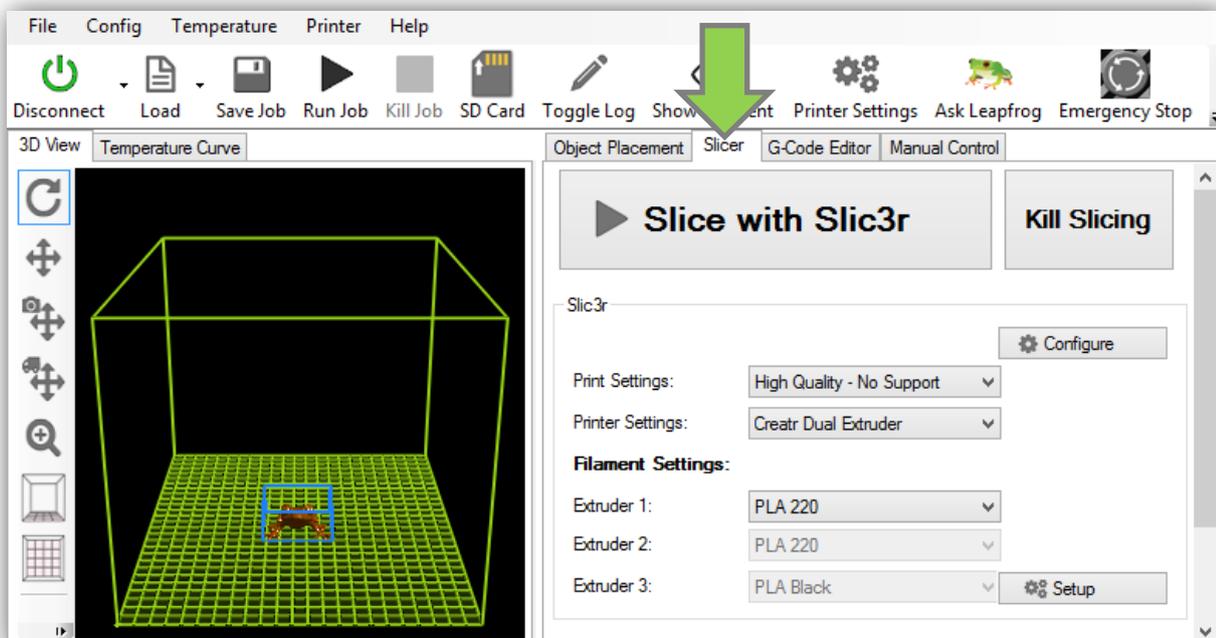
A STL file is a file format that contains the 3D design of your file. If you have a STL file, you will need to convert it to a printable file format called G-code. Below, the steps how to do this are explained.

- Click on the “Load” button.

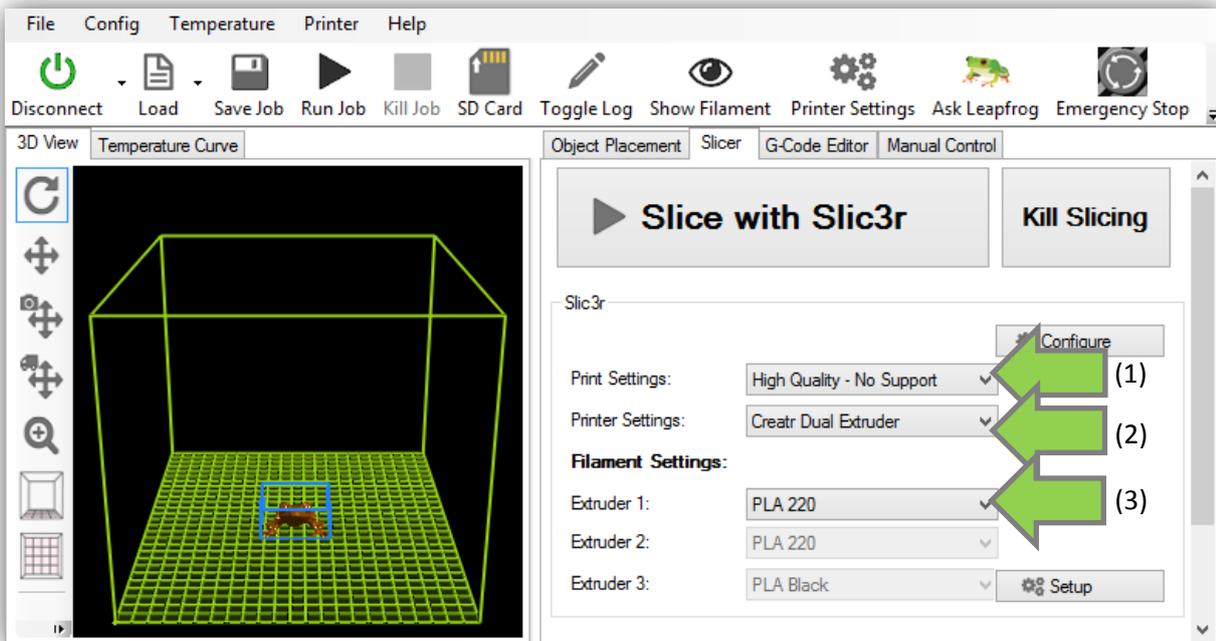


- Open a STL file.

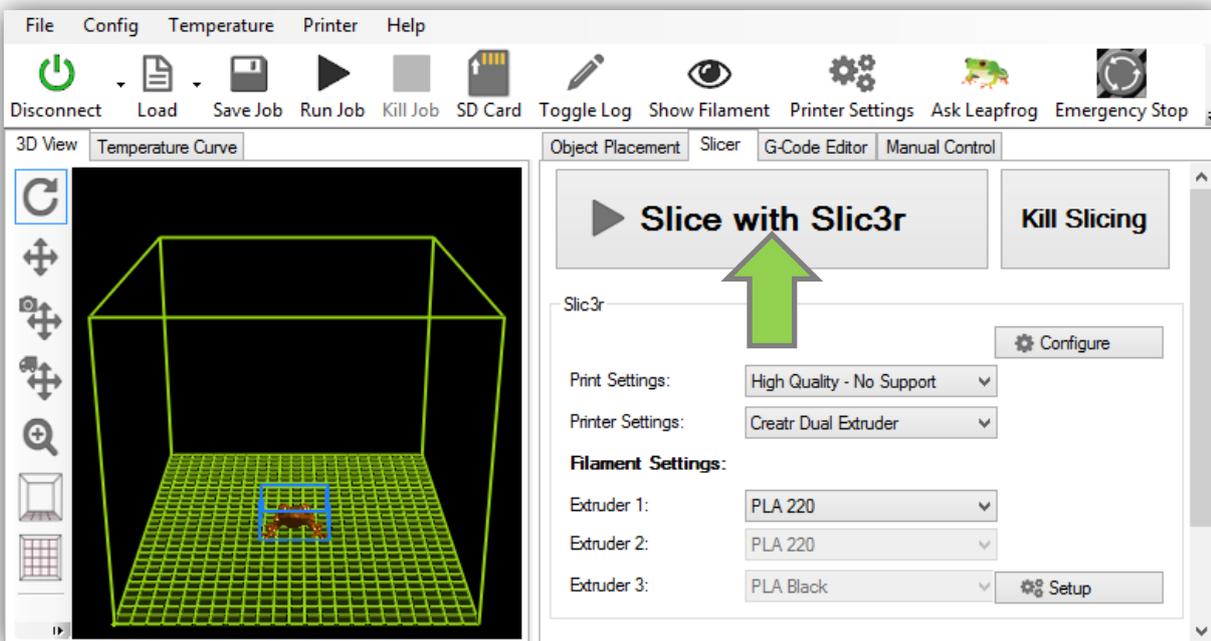
- Go to the “Slicer” tab.



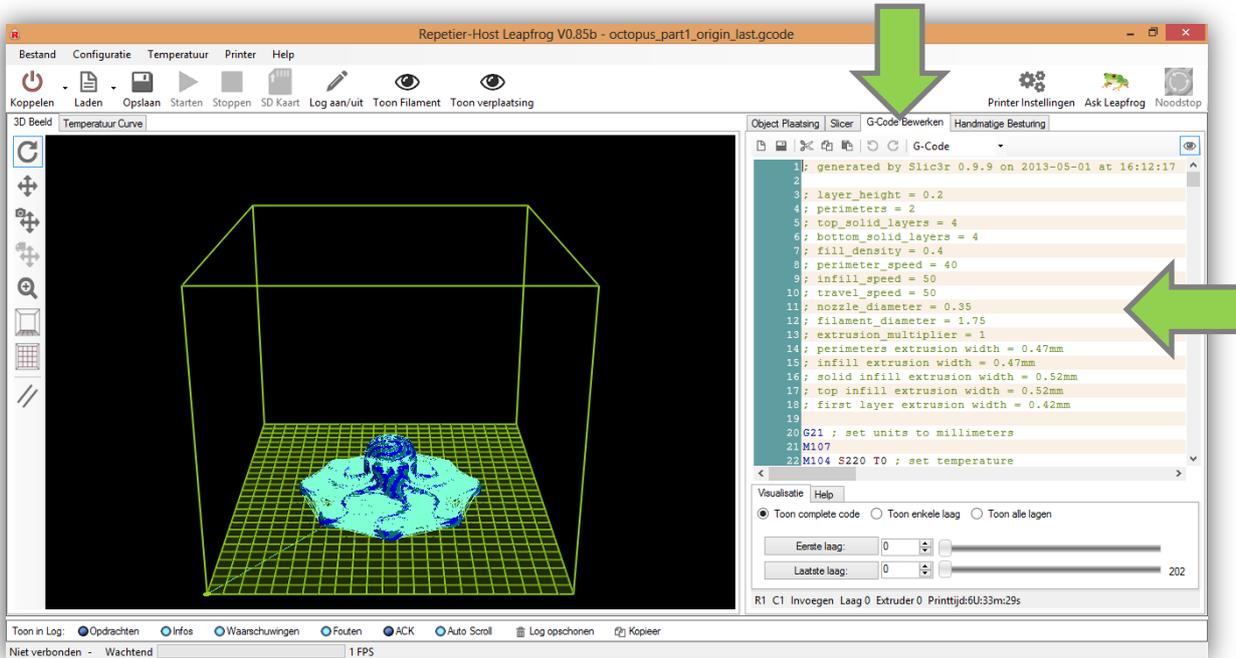
- Select your desired print settings (1).
For an explanation of the printing settings, look at the next chapter Print settings.
- Select single or dual extruder under printer settings (2).
- Choose your type of filament (3) It is possible to print your design in ABS, PLA or PVA.
If you are printing your design in ABS and your support in soluble PLA or PVA, always position your support material in the right extruder!



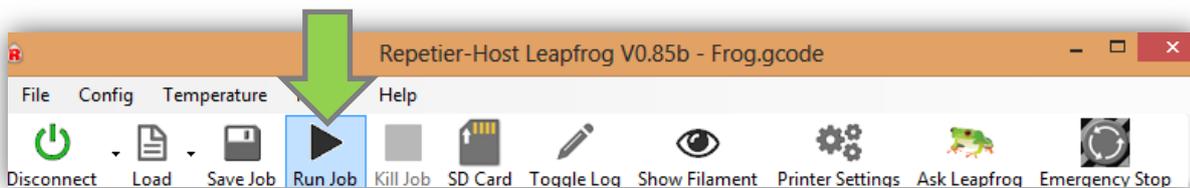
- Slice your STL file by clicking the "Slice with Slic3r" button.



- Wait until the slicing process has finished. Your STL file will be turned into a printable G-code. This could take a few minutes. The slicing is finished when a code appears at the “G-code” tab.

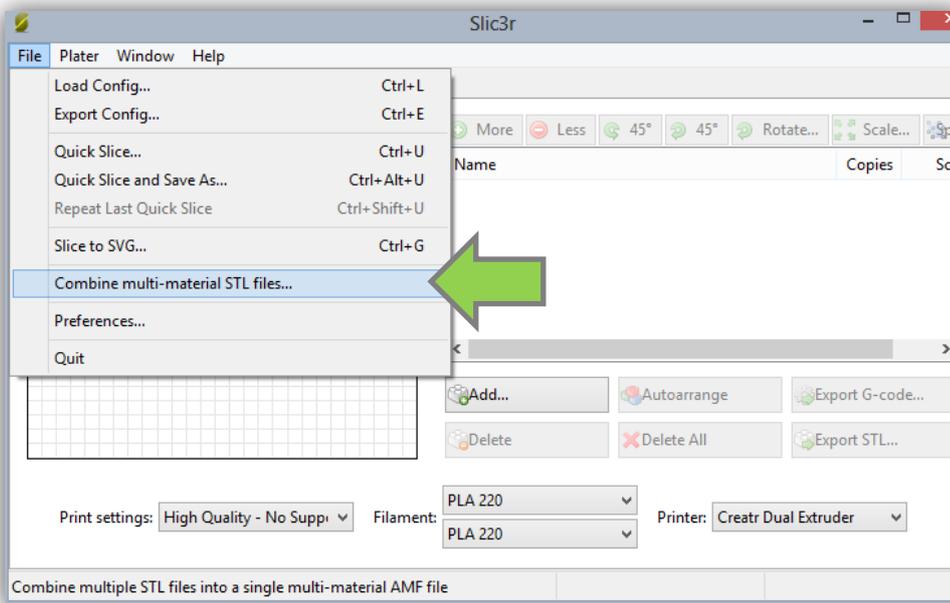


- When the slicing has finished, press the button “Run Job”, the printer will automatically home the X-, Y- and Z-axis. Your printer will start printing! On the bottom of the Repetier-Host Leapfrog screen you can see the estimated print time and which layer is being printed.



2.7 Dual color printing

- To be able to print in multiple colors you will first of all need the latest version of Slic3r, which is available at www.slic3r.org Please download and install the latest version.
- The trick to printing in two different colors is that you need two different STL files; one for each color. These two files have to be combined into one file by Slic3r. Follow the instructions below:
 - Open Slic3r
 - Go to submenu “File”
 - Select “Combine multi-material STL files...” as shown in the screenshot below



- A window will then pop up for you to browse for your two STL files. Browse to, select and open STL File 1 for the first color.

When you have done so, a new similar window will pop up in which you have to browse to, select and open STL file 2.

After selecting and opening the second STL file a similar window will pop up again, allowing you to add a third file and this continue for as many files desired, but since there are only two different extruders in the printer, two files is the maximum amount you wish for so therefore press “Cancel”.



Manual Leapfrog Creatr

- Now that the files which are to be combined have been selected, a new window will pop up to save the new combined file. Browse to the location you wish to save this file. Give it name and save it. It will be saved as an *“.amf file”*.
- Now load the .amf file in Repetier Host Leapfrog software just like you would do with a regular STL file. You are now good to go to start multi-colour printing by treating this file just like you would treat any STL file.



Troubleshooting - Knowledgebase

In this section, we provide you with guidelines to get you back on track if something should go wrong.

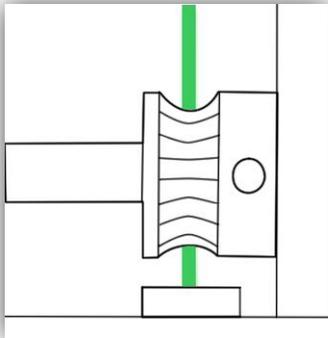
Of course it is possible that you encounter something that is not listed here. In section 3.8 we will explain what you can do in such a case.

This section contains the following:

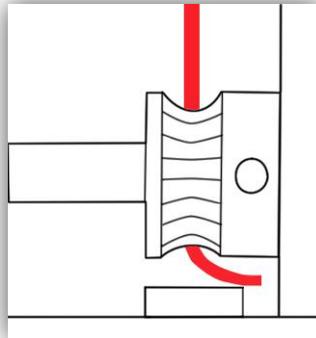
- 3.1 Oh, oh, my filament is not feeding through
- 3.2 My object won't stick to the bed
- 3.3 Extruder stops mid-print
- 3.4 Min-/max temp triggered error messages
- 3.5 Nozzle seemed clogged
- 3.6 My extruders are not the same height
- 3.7 Uploading firmware
- 3.8 Need support?

3.1 Oh, oh, my filament is not feeding through

- This can happen, and it is not a big problem. The filament is kept on spools, which means that the filament is curved into one direction. When your filament is not feeding through, make sure to feed the filament completely straight and from the top.
- Please note that the hole where the filament should feed into is very accurately aligned underneath the bearing and drive gear. Make sure that you align your filament with this hole.



Correct alignment



Wrong alignment

3.2 My object won't stick to the bed

- Print sticker

It is possible to make more than one print on one print sticker. However, if you print several prints on the same sticker, it is possible that your object won't stick on the bed any more. In that case, please replace your sticker.

After printing several prints on one sticker, you will notice the print sticker will tear/rip on some places. For smaller parts this is not a problem, because you can position them on a place that is not damaged. However, if bigger parts do not stick on your bed, always place a new print sticker first.

- Warping

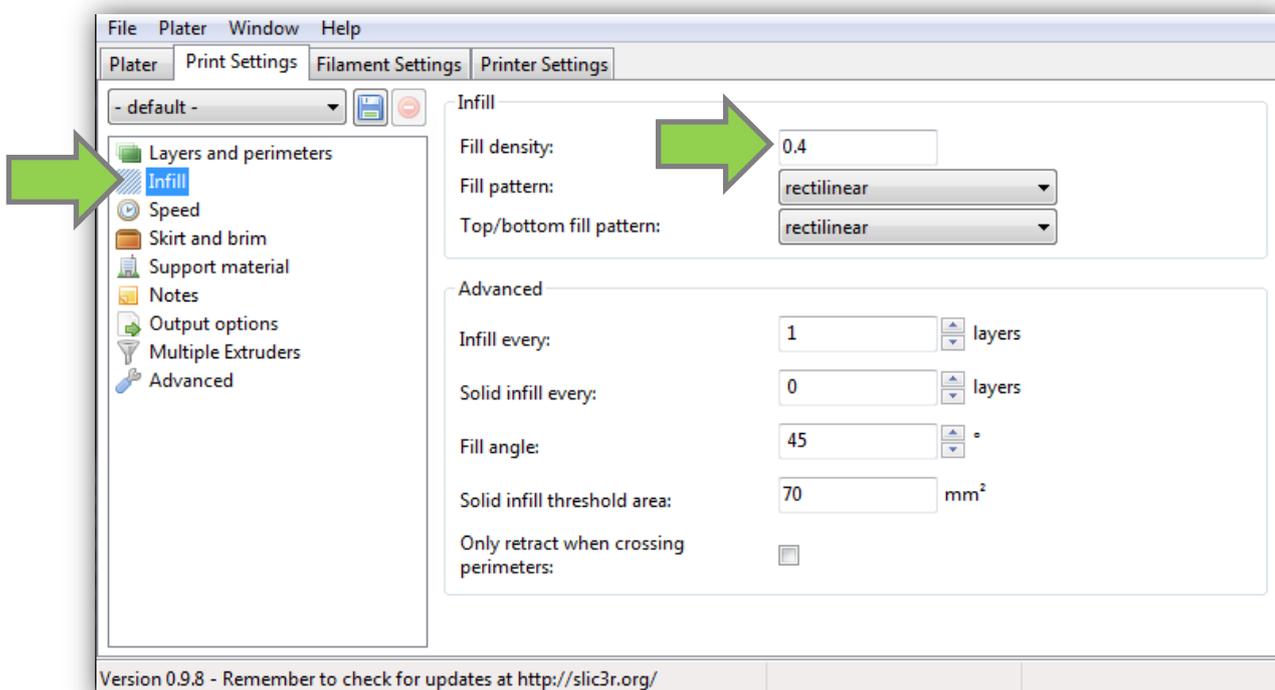
When ABS cools down after extrusion, it has a certain percentage of shrinkage. This means that the object tries to get a bit smaller. Resulting in the infamous warping. You can reduce warping by changing the Infill or adding Brim.

- Infill

If you are experiencing warping, we would advise to decrease the infill if possible.

Reducing the infill will reduce the shrinkage as you decrease the amount of material that can shrink. An infill of 0.1 will be strong enough in most cases and also solves the warping issue.

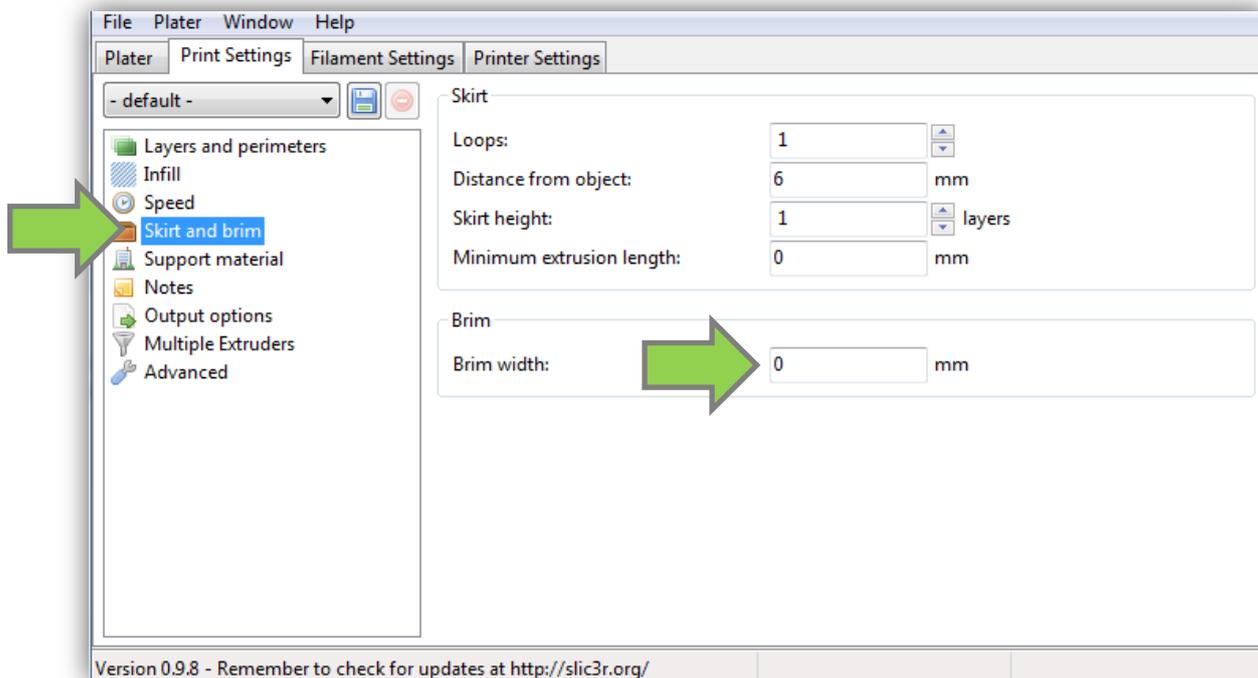
In Repetier Host Leapfrog, go to the "Slicer" tab, click on the button "Configure". Now Slic3r will open, go to the tab "Print Settings", select "Infill" and change the "Fill density" value. A fill density of 1 is 100% infill [solid], a fill density of 0.1 is 10% infill. Do not forget to click on the "Save" button.



- **Brim**

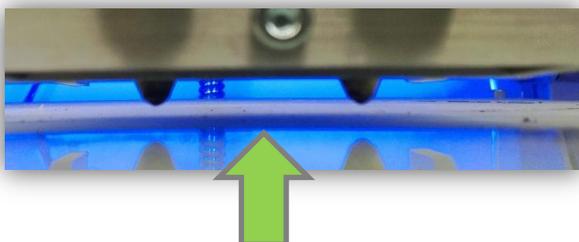
In Slic3r you can also add a brim; this feature increases the first layer size by implementing a border around your object when printing the first layer. This increases the surface tension and reduces the chance on warpage significantly.

In Repetier Host Leapfrog, go to the “Slicer” tab, click on the button “Configure”. Now Slic3r will open, go to the tab “Print Settings”, select “Skirt and brim” and change the “Brim width” value. The value you fill in will be the size of the border that you add around your object on the first layer only.



- **Decrease the Z-offset**

If your object won't stick to the bed, it is possible that the X-offset is too big and you need to decrease the Z-offset. Placing the bed a little bit closer to the nozzle will result in better adhesion between the first layer and the heated printbed. Since the print sticker is made of plastic, the first layer and print sticker can melt together. We would recommend placing the bed 0.1mm closer if your object does not stick to the bed correctly. Putting in a negative value will place the bed closer.



3.3 Extruder stops in mid-print

This manual will describe step by step what to check and how to solve any issues you encounter if your filament stops extruding in the middle of your print

Basic checks

To start there are a few basic things to check that can make a big difference:

- Laptop screensaver
If you connect your laptop to your printer, it is important to disable your screensaver. If your screensaver is enabled, the printer stops printing when the screensaver turns on. If you connect your PC to your printer, it is not necessary to disable your screensaver.
- Filament spool
Check if the filament on the spool is entangled. This can make it hard/impossible for the driver gear to pull through. If this is the case, retract the filament as mentioned before. Make sure the filament of the spool is free to roll of and reinsert the filament.
- Filament
Check if you find scrapings on the filament near the inlet in the electronics cover plate, located inside your printer. If the inlet is too sharp or the filament is not let in perpendicular, pieces of the filament can be cut off. Deformation of the filament can result in the driver gear not being able to get grip on the filament or the hot end not heating the filament properly. Readjust your filament if necessary.



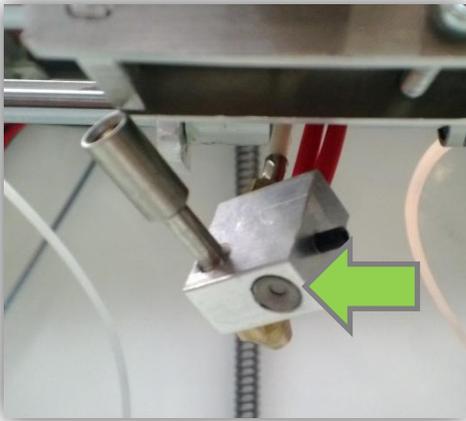
- Extruders
Check if both the extruder fans are working properly. If the fans do not work the extruder will not be cooled properly causing large fluctuations in the nozzle temperature, which can cause clogging. If you see that the fans are not spinning, try to give them a push with for example a pen to get them going. If they still don't start spinning please check if the wiring is connected properly.

- Heat cartridge

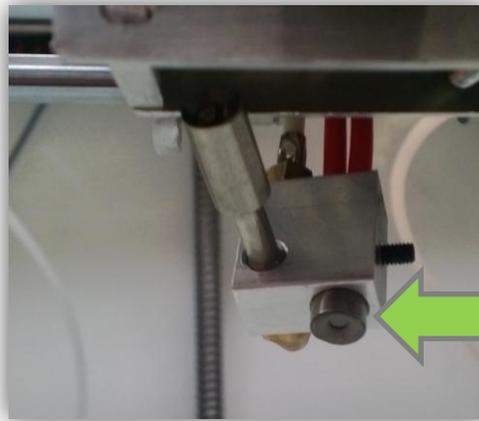
Check whether the heat cartridge is sufficiently centred in the heat block. In the pictures below you can see a heat cartridge that is centred and one that is not.

It is important that the heat cartridge is centred properly so that the heat is distributed equally over the filament and that the thermistor reads the correct temperatures, preventing too cold extrusion which can cause clogging.

If you see that the heat cartridge is too far outside the block, please loosen the index bolt that holds it and adjust the position.



Correct: Centred



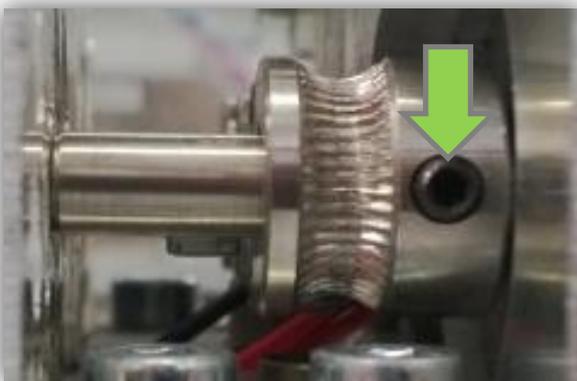
Wrong: Not centred

- Nozzles

Check if the nozzles are still hot after the extruder stops and if they have the correct temperature. **Do not check this with your bare hands, the nozzles can be very hot!** Please look at the software for the current temperature. If the nozzle heating is somehow turned off or a different temperature was set, this will result in clogging due to cold extrusion. It is important to check if the software is set at the correct temperature.

- Driver gear

Check if the driver gear is tightened to its motor correctly. If the driver gear is not tightened correctly, the motor will deliver power but the driver gear won't spin. If this is the case, please tighten the screw that is indicated on the picture below.



The next steps

If all the checks mentioned earlier turn out to be okay, we advise you to follow the next steps

- Clean your nozzle with your filament

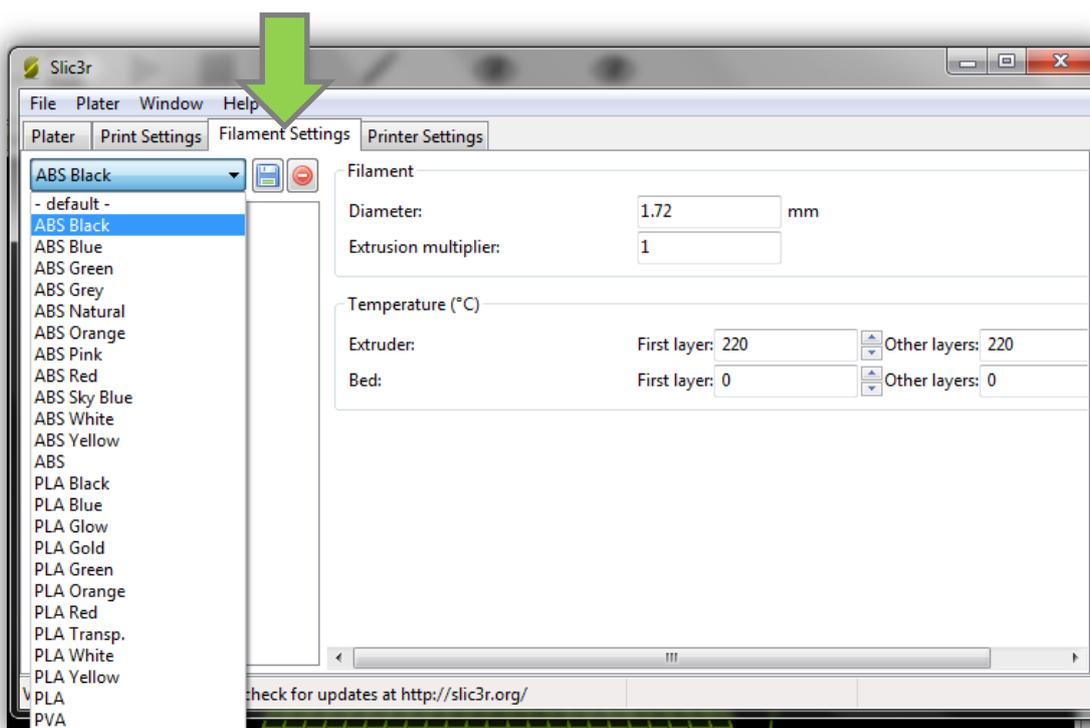
It is important that the nozzle has no leftover filament inside it that are stuck. Please heat the nozzle to 220/230 degrees and retract the filament until it is completely out of the extruder. Cut off the last part from the just retracted filament and make sure your cut is perpendicular.

Heat the nozzle to 240 degrees Celsius, feed and extrude the filament. Hereby, the filament should push any clogging parts through the hot end. This is a great way to clean your nozzle without the hassle of removing the nozzle and placing it in bowl of dissolvent. We would recommend doing this on a regular basis when you start your prints as it is a fast and a good regular check.

- Printing at higher temperatures

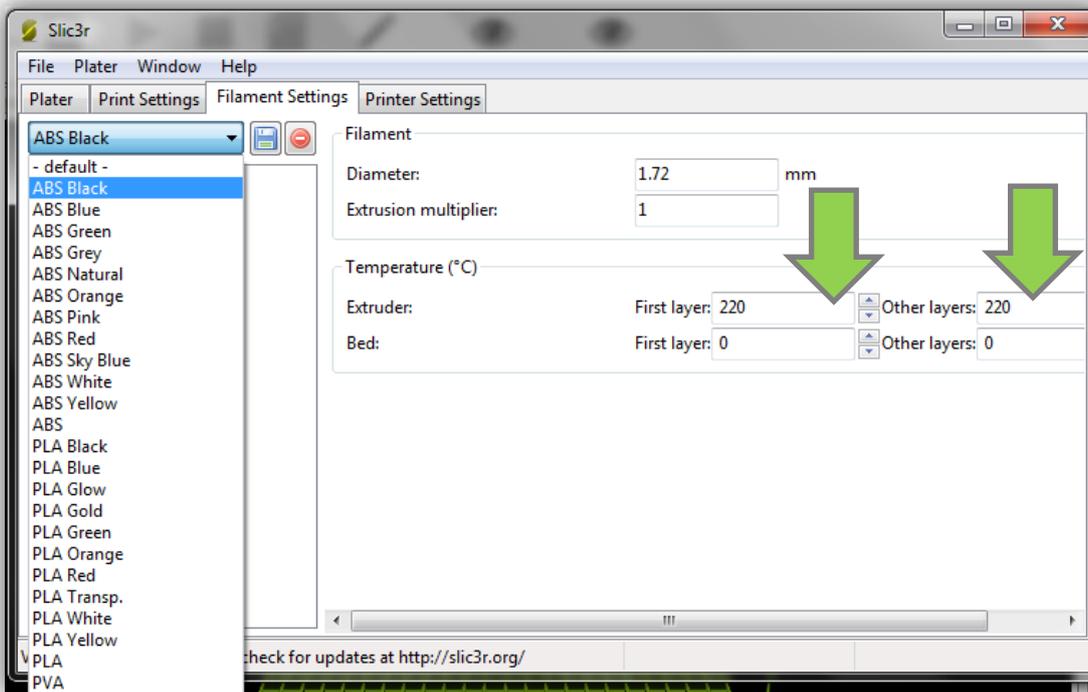
Printing with a nozzle that is too cold can cause clogging. This can occur mid-print, because the nozzle temperature fluctuates with a few degrees. If this happens, please increase the temperature of your nozzle.

You can increase the print temperature by going to Repetier-Host and selecting the “*Slicer*” tab. In the Slicer tab select “*Configure*” to enter the Slic3r settings menu. In the menu go to the “*Filament settings*” tab and select the filament type of which you wish to change the setting as indicated in the screenshot below

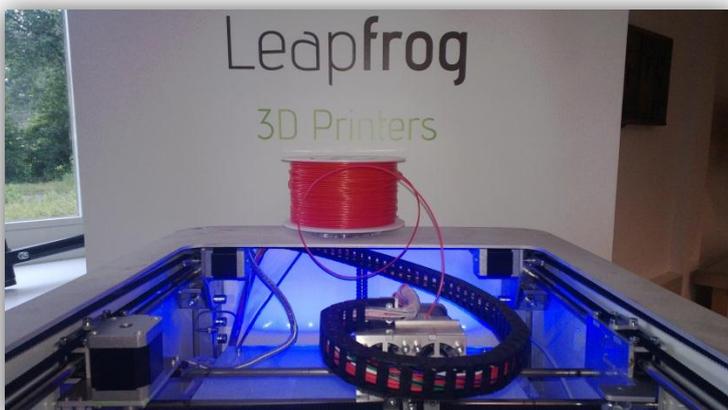


Once you have selected the correct filament type, adjust the temperatures by increasing the temperature given for the first layer and the other layers set for the extruder. In the screenshot shown you can see that these are set to 220 degrees Celsius. When you have adjusted the temperature values press the “Save”-icon underneath the “Filament Settings” header to save the changes in the settings.

Try to increase the temperatures in steps of 5-10 degrees with a limit of 245 and see if this improves the printing result.



- Print without using the filament guiding tubes
The filament guiding tubes through which the filament is fed into the extruder can cause drag that makes it hard for the driver gear to pull the filament through. Printing without the use of the tubes will reduce this drag. So remove the filament spool holders from the bottom of your Creatr and place this holder on the top plate of your Creatr. Feed the filament directly from the spool into the extruder and try to see if this improves the print result



- Changing the distance between the bearing and the driver gear

If none of the above has proven to be a solution, you can check if the distance between the driver gear and the bearing of the extruder is set correctly.

In order to determine this, you will have to extrude some filament and then retract the filament. If the bearing is too far away from the driver gear you will hardly see any marks on your retracted filament. If this is the case, you will need to change the distance between the driver gear and the bearing.

If the gear is a little bit too close to the driver, your filament will be deformed too much (see the figure below). This causes the filament to be unable to pass through the hot end properly and it will jam. The following description explains how to change the distance between the driver gear and the bearing.

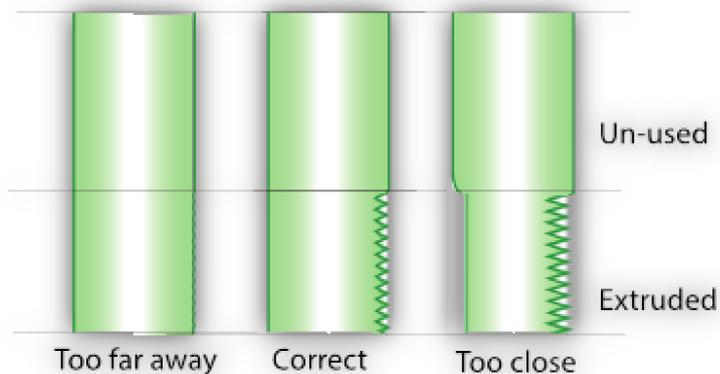
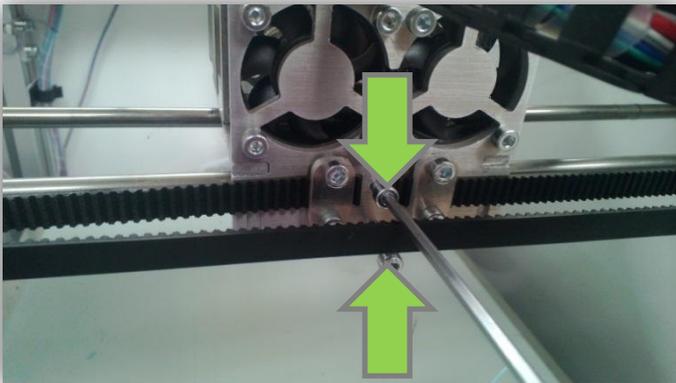


Figure: Sketch of the filament for three types of distances between the driver gear and the bearing. Left shows when they are too far apart. The middle shows the correct distance and the right sketch displays the filament when the distance is too small.

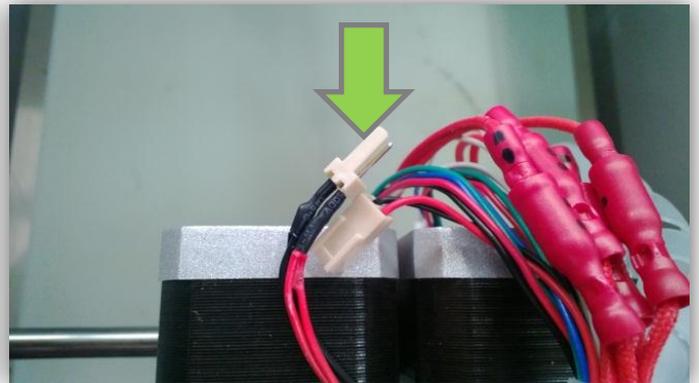
- **Step 1: Gain access to the bearing plate**

In order to gain access to the bearing plate you will have to remove the 2 middle bolts at the front of the extruder carriage first as shown in the next figure.

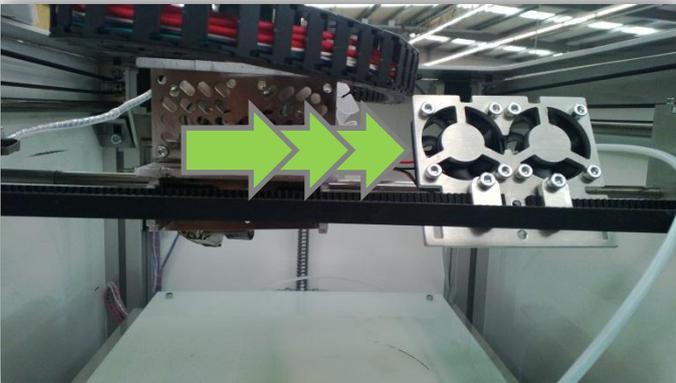
Once these bolts are loosened you have to disconnect the connectors of the fan cables in the cable chain so you can slide the front of the carriage sideways. Now you gain access to the plate in front of the driver gear. This plate can easily be removed by loosening the bolts in the bottom corners of the plate.



Loosen the middle bolts

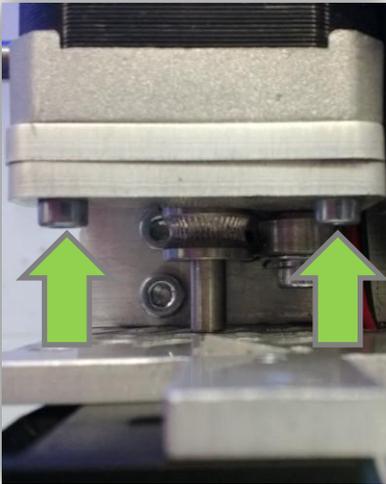


Disconnect the fan cables



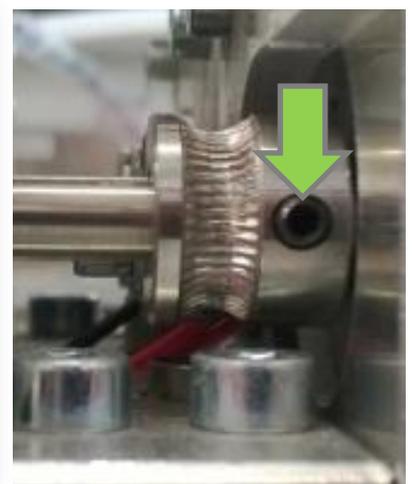
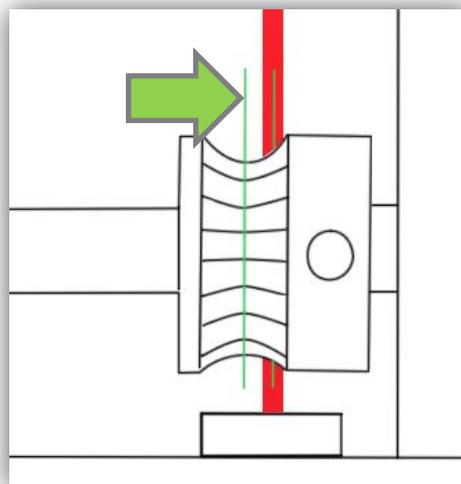
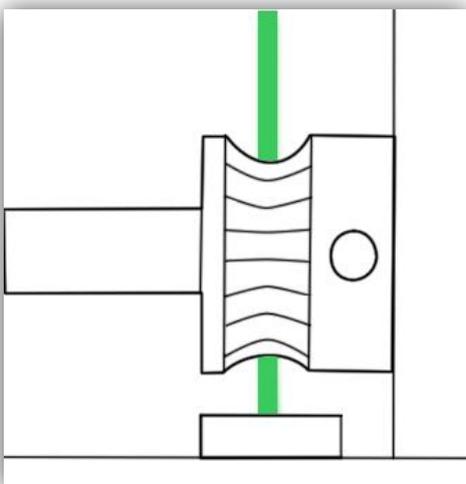
Slide away the front of the carriage

- **Step 2: Loosening the bearing plate**
The plate where the bearing is attached to, is fastened with two bolts as shown in the pictures below. Loosen these bolts a little to be able to change the distance between the driver gear and the bearing. Do not take them out.

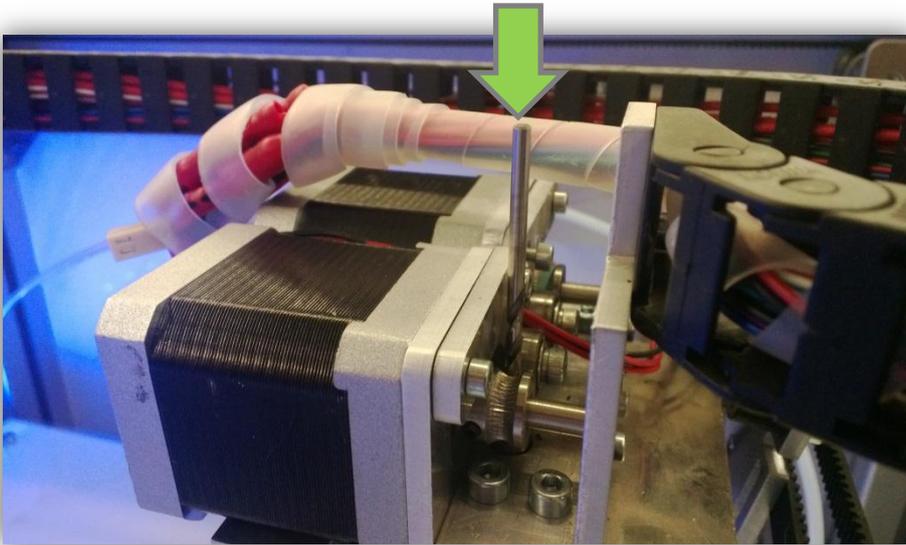


- **Step 3: Checking the driver gear**
Now is a good time to check if the groove of the driver gear is neatly in line with the entry of the hot end tube. The left picture displays a good alignment; the middle picture displays a misaligned driver gear. Here, the green lines should be on top of each other.

If your driver is misaligned, please loosen the small index bolt [see right picture] and slide the driver gear on the axis of the motor to the correct position. Don't forget to tighten it again.



- **Step 4: Determine the proper distance of the bearing**
Grab a drill with a diameter of 1.5mm (or anything else with this diameter) and stick it on the hole of the hot end tube. It should also be positioned against the groove of the driver gear. Next, place the bearing plate in such a way that the bearing is tight against the drill bit. Then tighten the two bolts holding the bearing plate in order to fixate the bearing.



- **Step 5: Testing**
Now that you have adjusted the distance, rebuild the extruder carriage as it was and try to extrude 10 mm of filament and afterwards retract your filament again to check if the filament is severely deformed or has no marks at all. If it does not display any marks, insert the filament again and try to extrude a couple of centimetres to make sure the extruder is working properly

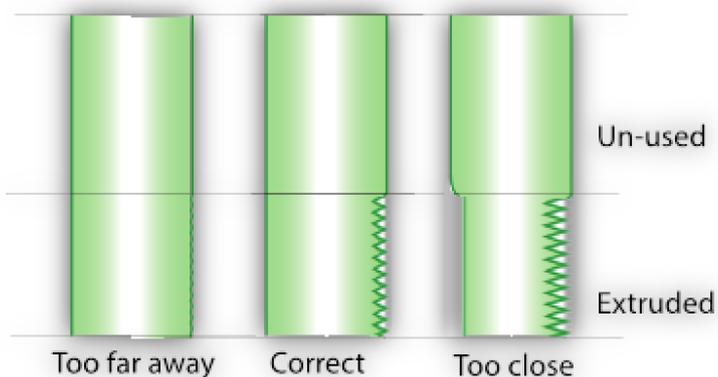
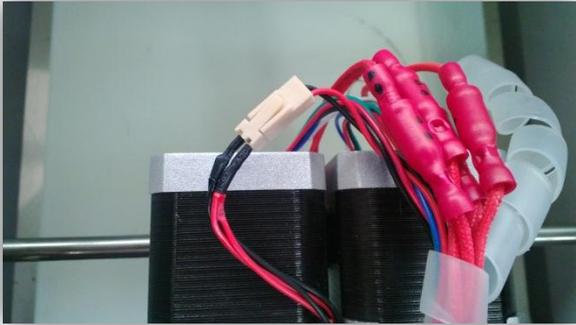


Figure: Sketch of the filament for three types of distances between the driver gear and the bearing. Left shows when they are too far apart. The middle shows the correct distance and the right sketch displays the filament when the distance is too small.

3.4 Min-/Max temp triggered error messages

- Always first check all the wiring. Most of the times it comes down to a loose connector, always add some pressure to the connectors to make sure that they are thoroughly connected. You will need to unwind the plastic cover that covers the wires to check this. Don't forget to cover it again.

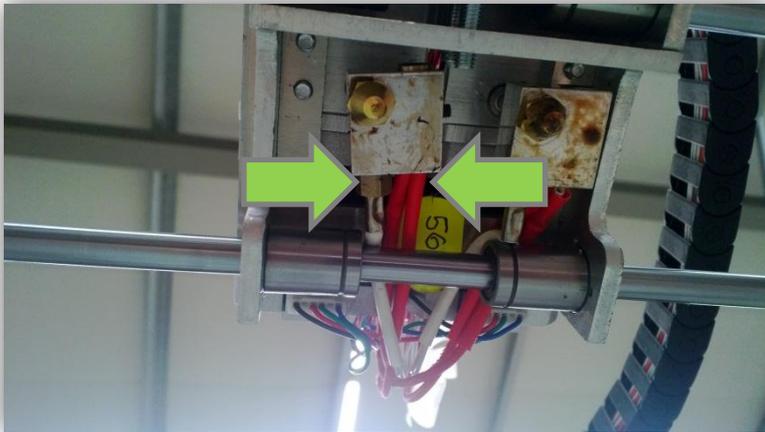


Correct: Connected



Wrong: Disconnected

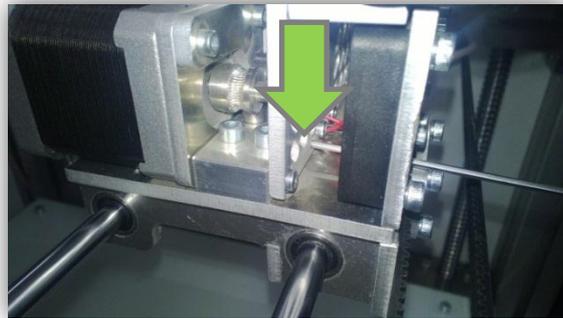
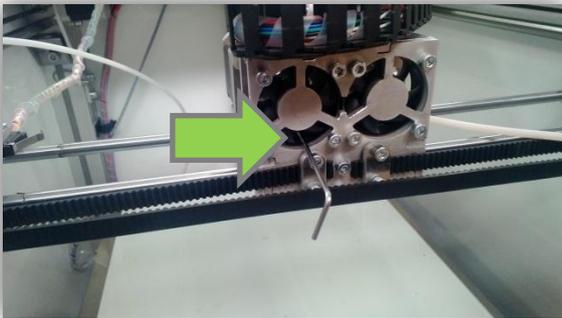
- Lower your bed and take a look underneath the extruder carriage. This way you can make sure both the heat cartridge and thermistor are properly connected.



- If both of the previous steps are checked properly, we will have to conclude that the thermistor has failed. In this case we should place a new thermistor. Please contact our support department via support.lpfrg.com by entering a support ticket.

Manual Leapfrog Creatr

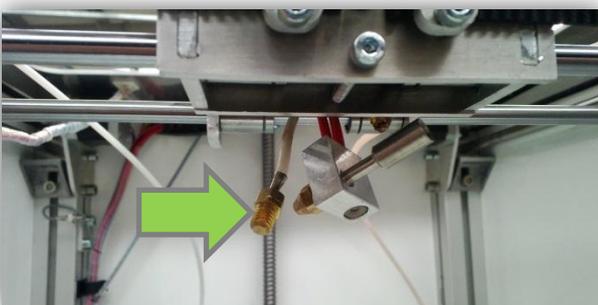
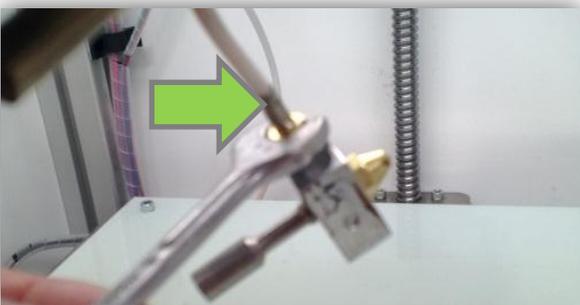
- After receiving your new thermistor, the following steps will guide you through replacing the thermistor.
 - Make sure that the filament is not loaded in the extruder. If it is loaded, please retract the filament.
 - Loosen the Allen screw indicated on the picture with an Allen wrench.



- Loosen the Allen screw holding the heat cartridge in place, and slide the heat cartridge out of the heating block.

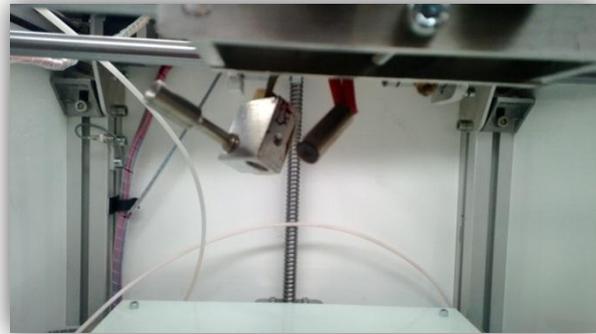


- Loosen en remove the broken thermistor.



- Place and tighten the new thermistor in its place and connect the connector.

- Slide the heat cartridge into place, and tighten the Allen screw again.

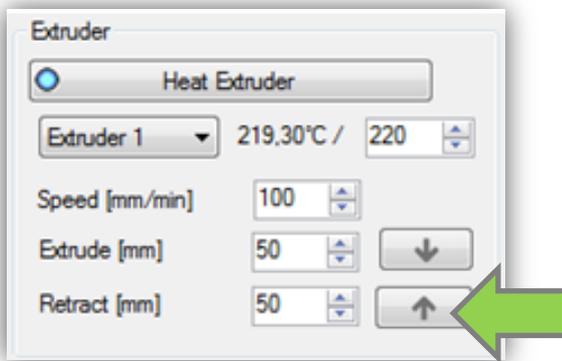


- Slide the hot end into the extruder carriage and tighten the other Allen screw again (go through the height difference extruder steps to calibrate the height, see 3.6).

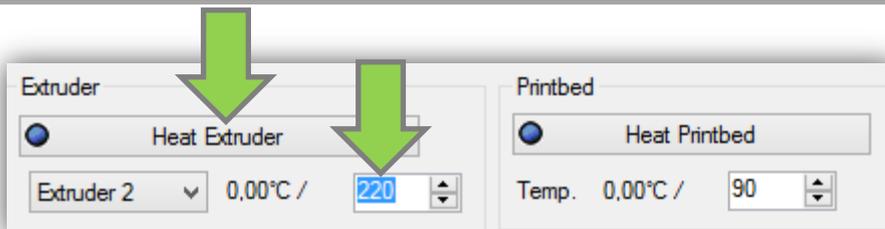
3.5 Nozzle seems clogged

In most cases the problem can be resolved by increasing the temperature by 10 degrees and extruding. Please follow the steps as described below.

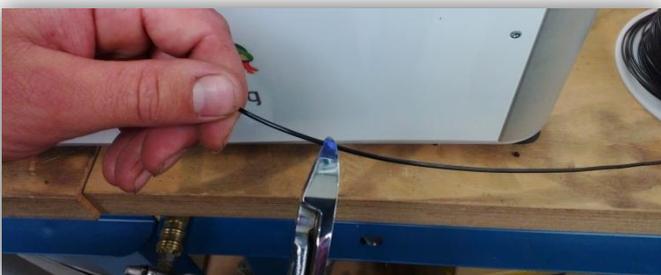
- Retract the filament
First, in the “Manual Control” tab select the extruder of which the nozzle is clogged. Make sure that it is heated, set “Retract” to 50 mm and click on the “Retract” button [the arrow pointing upwards]. Retract until the filament is retracted from the nozzle and remove the filament in an upward direction from the extruder.



- Increase the extruder temperature by 10 degrees
Increase the temperature of the extruder that is clogged [Extruder 1 or Extruder 2] as shown in the field below and click on “Heat extruder”.



- Snip of the used filament, and make sure the cut is perpendicular to the direction of the filament.



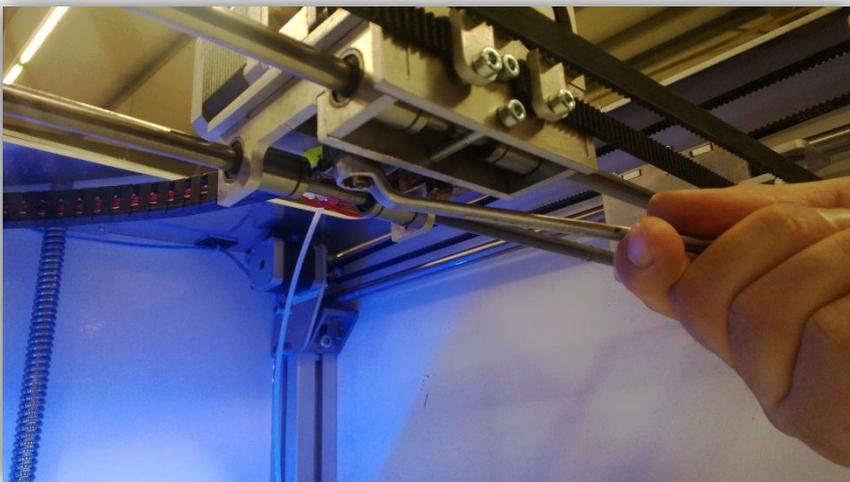
- Insert the filament in the extruder.

- Extrude the new strand of filament. Make sure the extruder is heated, set “*Extrude*” to 50 mm and extrude until filament is extruded through the nozzle.

If the previous step was not the solution, you will need to replace the nozzle with a new nozzle [delivered with the printer]. Please follow the below mentioned steps carefully.

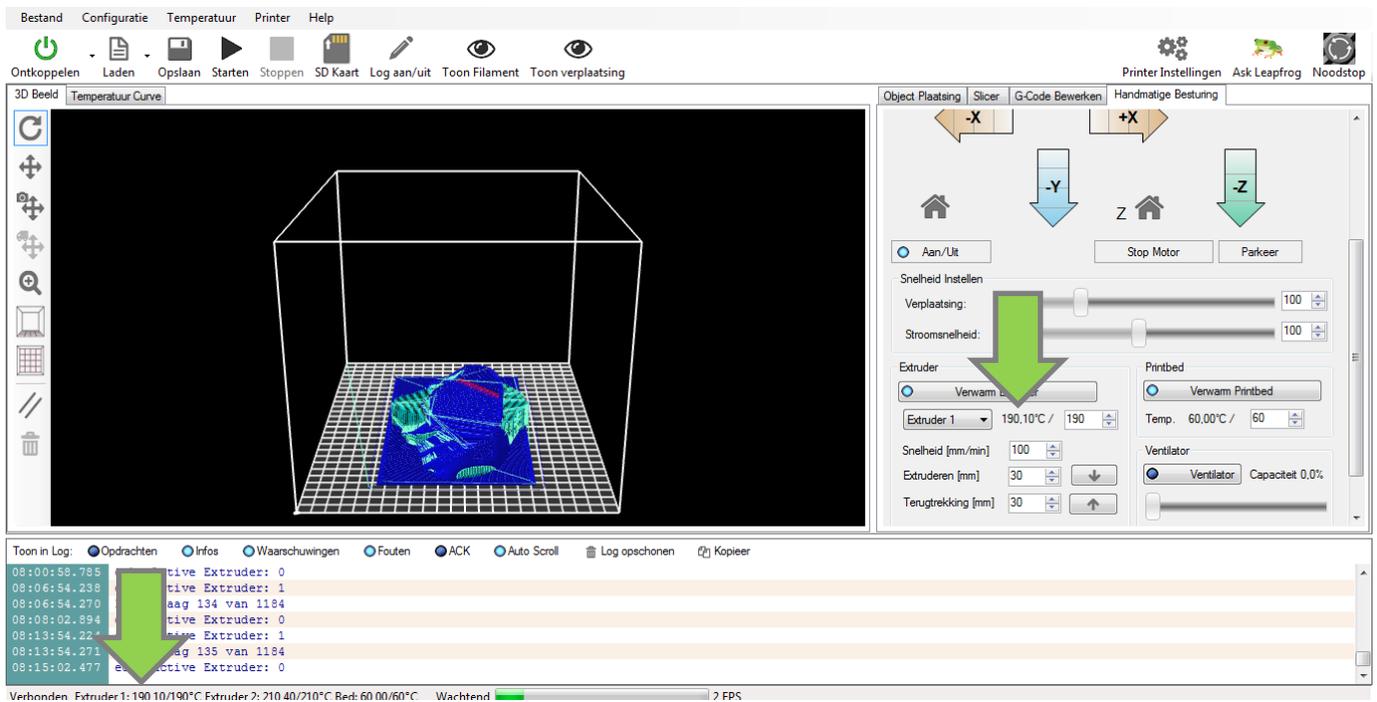
- Heat the extruder that is clogged to 230 degrees.

- Take your wrench and loosen the nozzle, do not take it off completely.
Be careful, the extruder is very hot !!! Do not touch it with your bare hands !!!

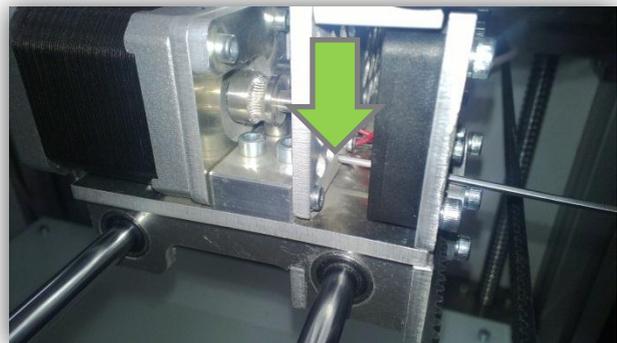
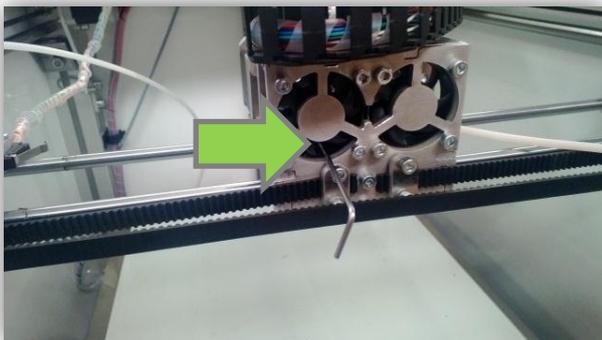


- Let the extruder cool off, you can monitor the temperature of the extruder in Repetier as shown in the picture.

Do not check the temperature of the nozzle with your bare hands !!!

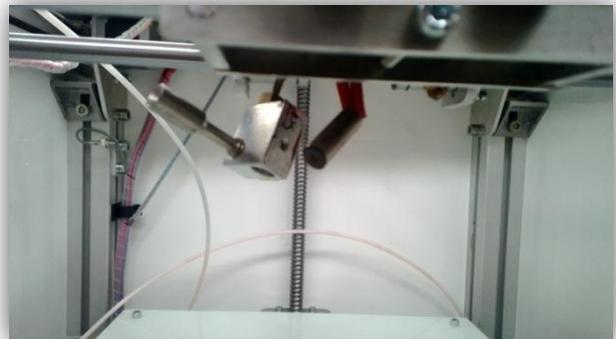
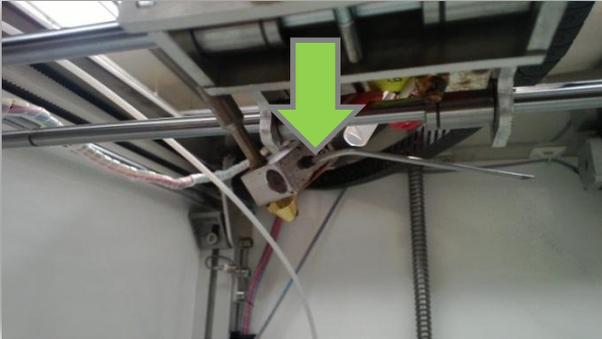


- Remove the hot end from the extruder carriage by loosening the Allen screw that is located at the front side of the extruder behind the fan as indicated on the picture.



Manual Leapfrog Creatr

- Remove the heat cartridge by loosening the Allen screw in the heating block.



- Turn the hot end, this will allow you to remove the thermistor.



- Screw off the old nozzle, screw on the new nozzle, and tighten its position by mounting the heating block against it.

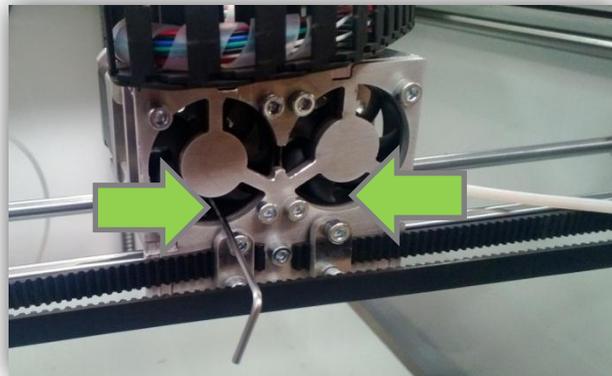
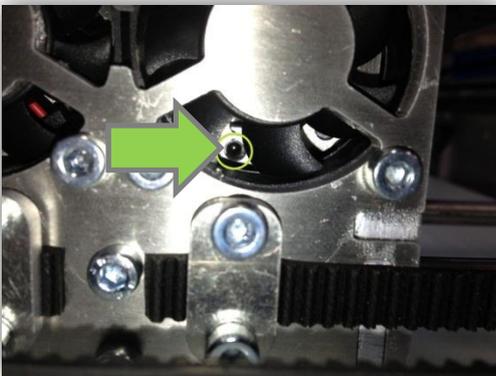


- Re-attach the hot end in the opposite order as you disassembled it, and follow the extruder height calibration as displayed in section 3.6.

3.6 My extruders are not the same height

When there is a height difference between the extruders, support material will not properly attach to the printed part or will melt into the printed part. This guide will explain how to level both extruders to the same height.

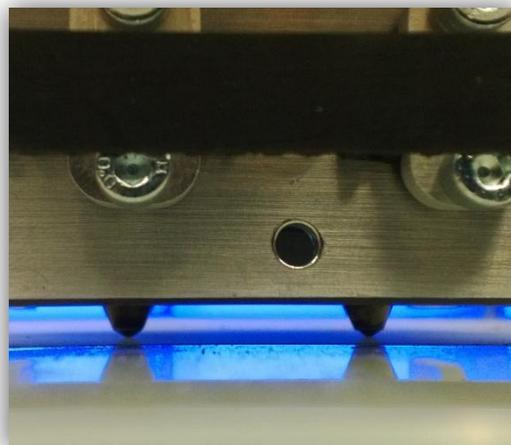
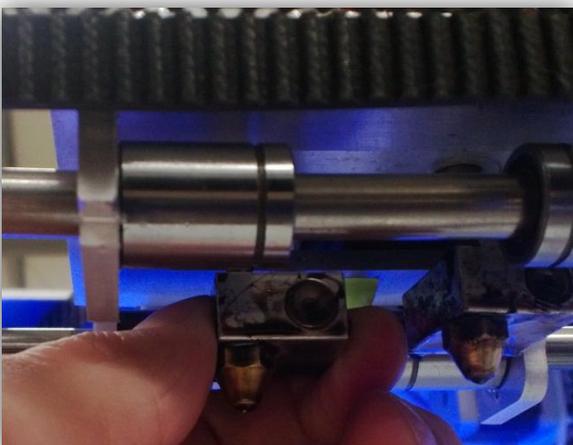
- **Loosening the index bolt**
The index bolt can be found when you look through the fans on the extruder carriage as indicated on the picture. Use an Allen wrench to loosen the index bolts of both extruders. It is not necessary to take them out, only loosen them



- **Sliding the hot end down**
After loosening the index bolts, the hot ends will be able to slide up and down. Use pliers to move the hot end which is a little bit too high downwards.

The next step is to home the Z-axis by pressing Z-home at the “Manual control” tab. The other hot end will start to move upwards. At this moment in time, both your hot ends are positioned on the same height.

Make sure you tighten the index bolts of both extruders to secure their position.



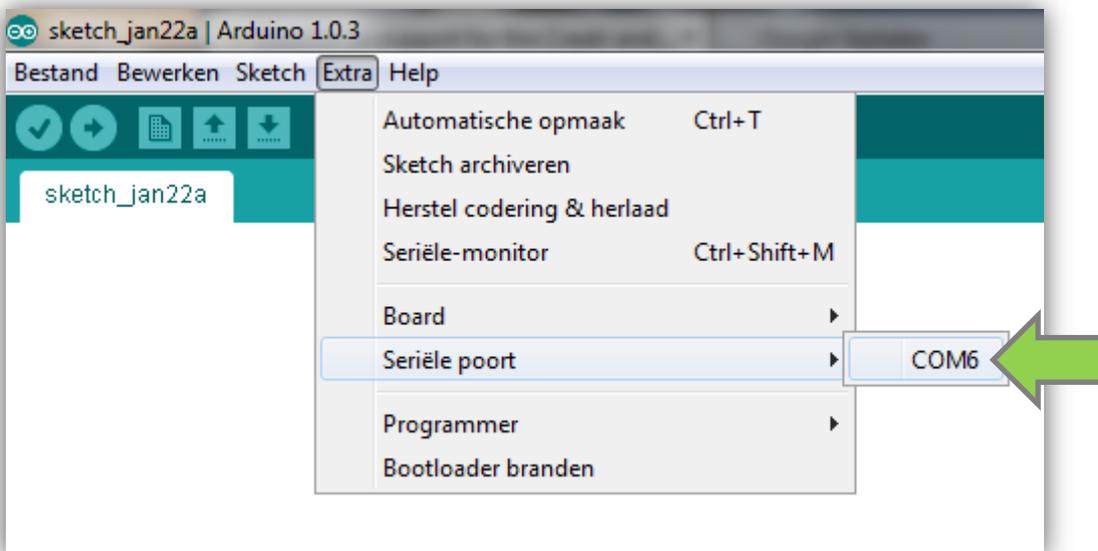
3.7 Uploading firmware

It is a fairly easy process to upload the firmware onto the Arduino board. Please follow the following steps one by one

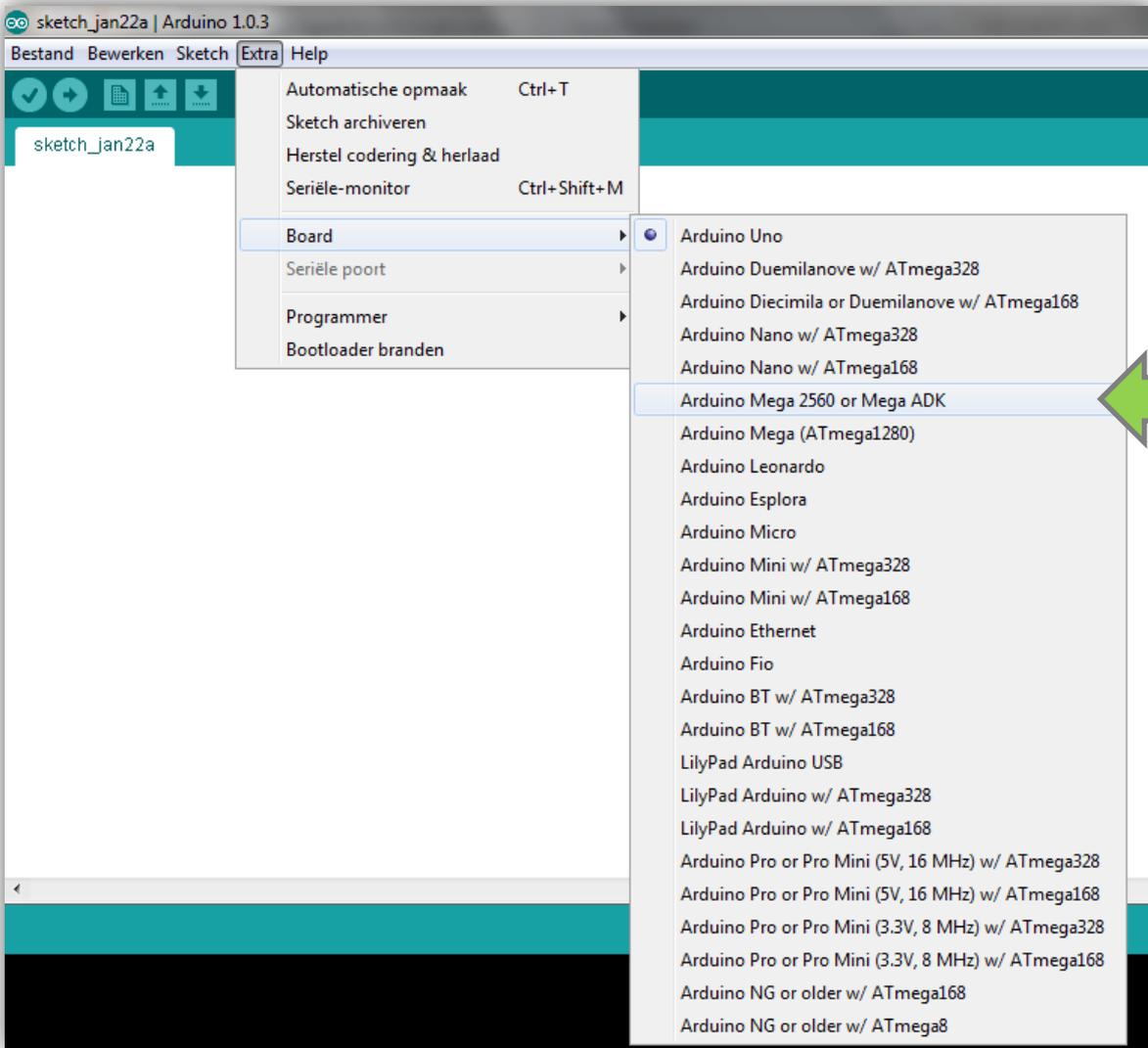
- Make sure your USB cable is connected to your Arduino board.



- The next step is to start the Arduino software, go to the tab “Extra” in the main menu, select “Serial Port” and select the “right COM port” [here indicated as COM6].

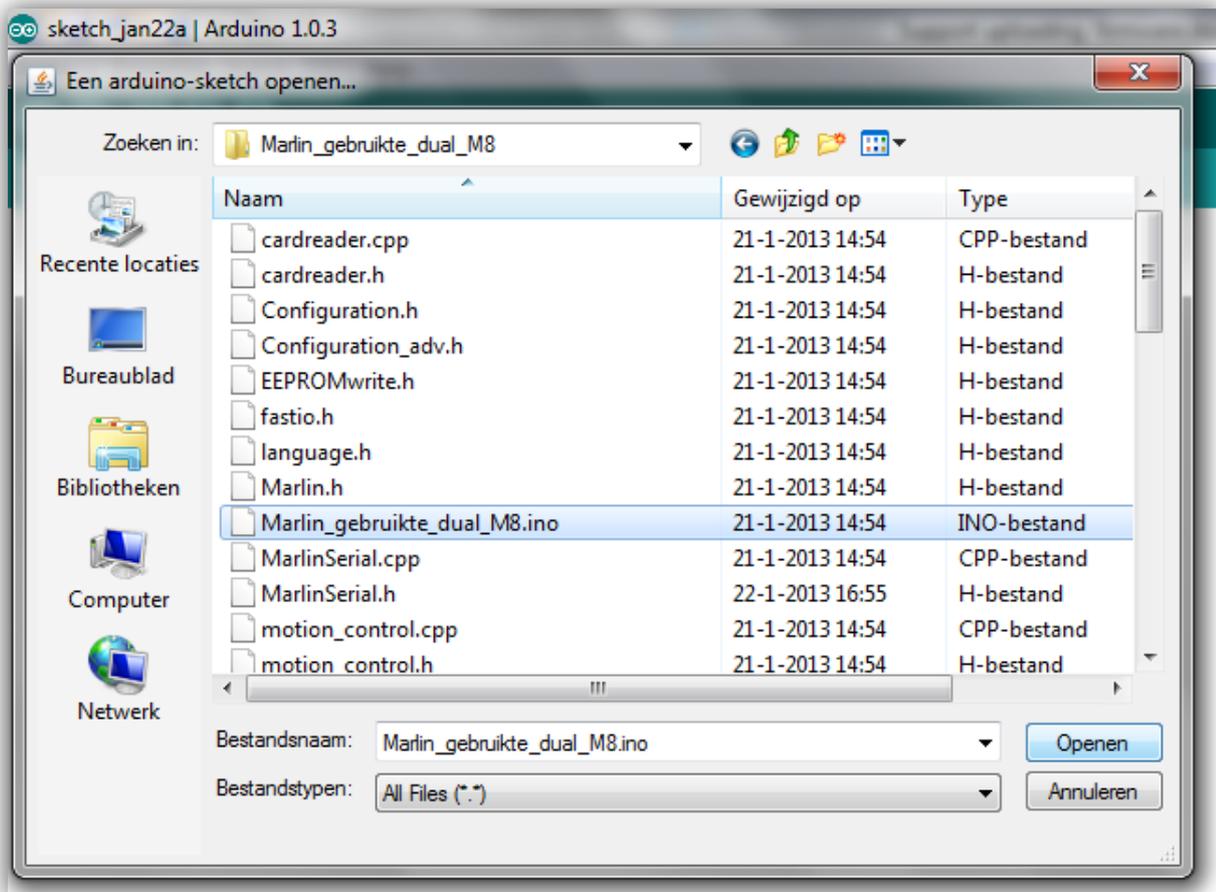


- Select the Arduino Mega 2560 board by going to the tab “Extra” in the main menu, select “Board” and select “Arduino Mega 2560 or Mega ADK”.

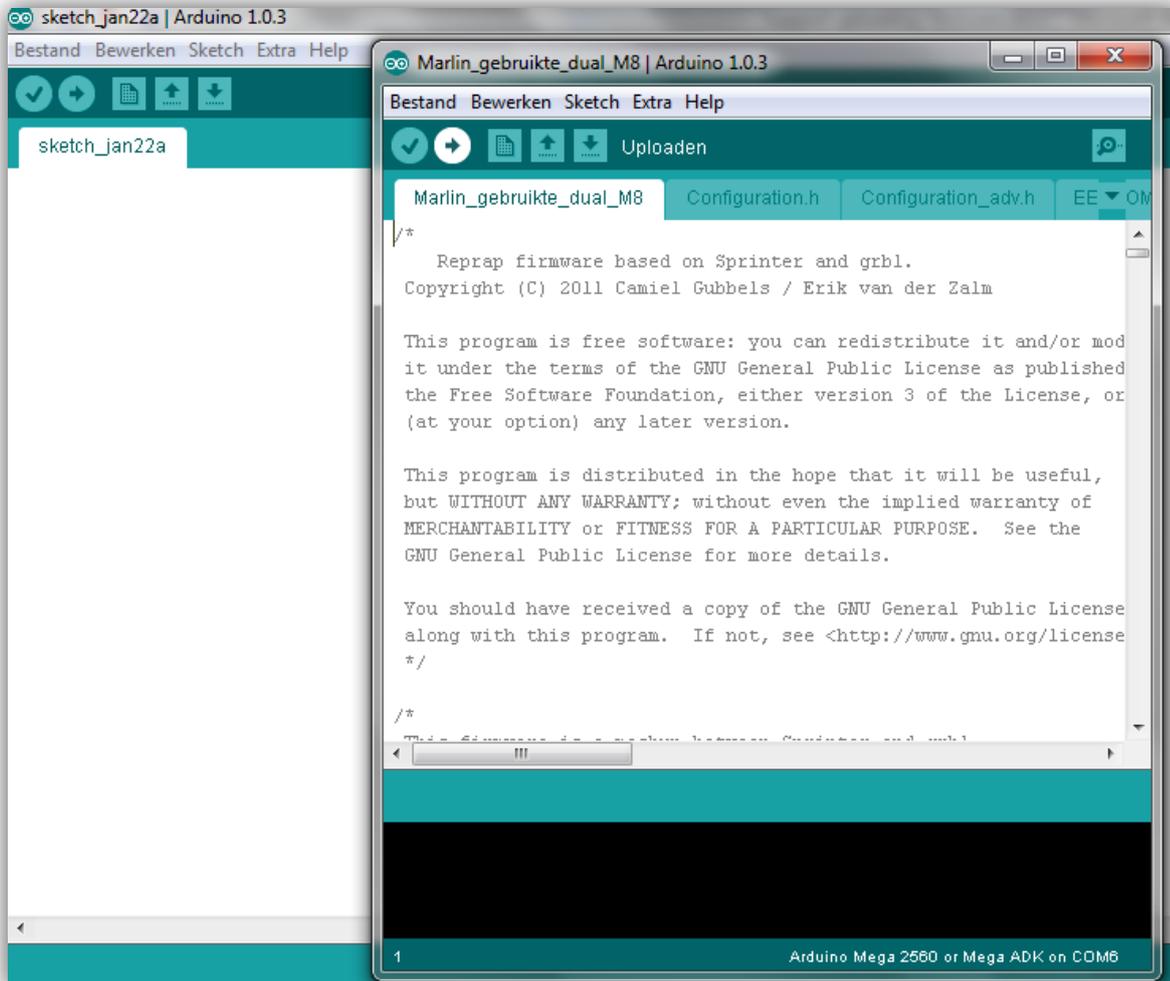


- Next go to the submenu “File” > “Open”
A window will pop up to select a file. In this window select the folder you received from support. In this folder you will find a file called either: “Marlin_gebruikte_*.ino” or: “Marlin_gebruikte_*.pde”. Open this file

In the screenshot example below the file has the name: “Marlin_gebruikte_dual_M8.ino”



- The final step is to upload the firmware. This is done by pressing the upload button which is indicated by an arrow pointing to the right as shown lighted up in the screenshot below.

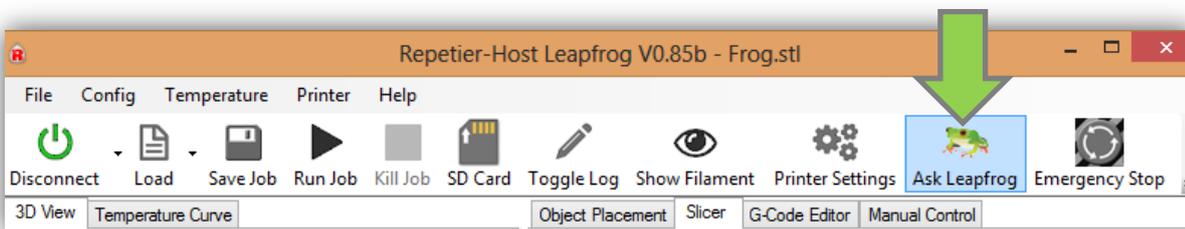


- Once the firmware has been successfully uploaded you can close the menu and all is set for you to start printing!

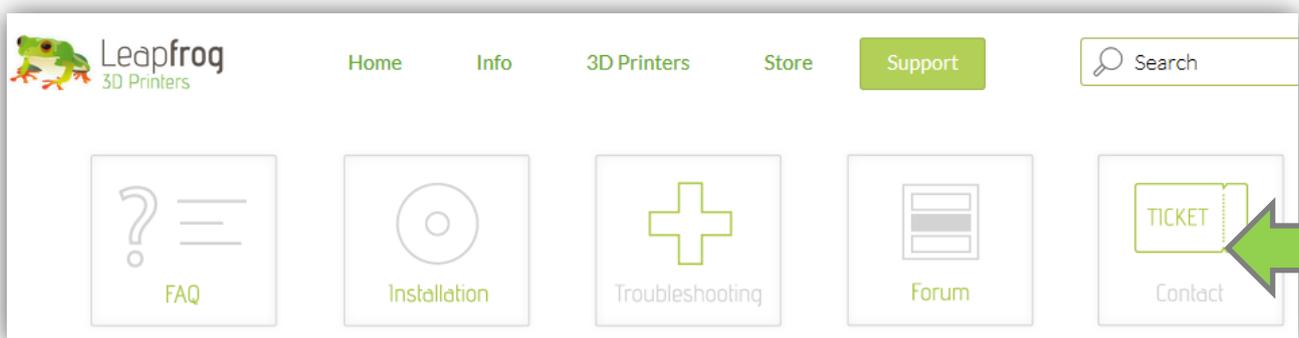
3.8 Need support?

Of course it is possible be that you run into something that was not explained in this document. Here we explain you what to do in such a case.

- If you need support, click on the “Ask Leapfrog” button in Repetier-Host Leapfrog or go to <https://www.lpfrg.com/support/>.



- By clicking on the “Ask Leapfrog” button in Repetier-Host Leapfrog, you will be directed to the Leapfrog support page.
- If you cannot find the solution to your problem at “FAQ”, “Knowledgebase” or on our “Forum”, you can always ask our support team by creating a ticket.



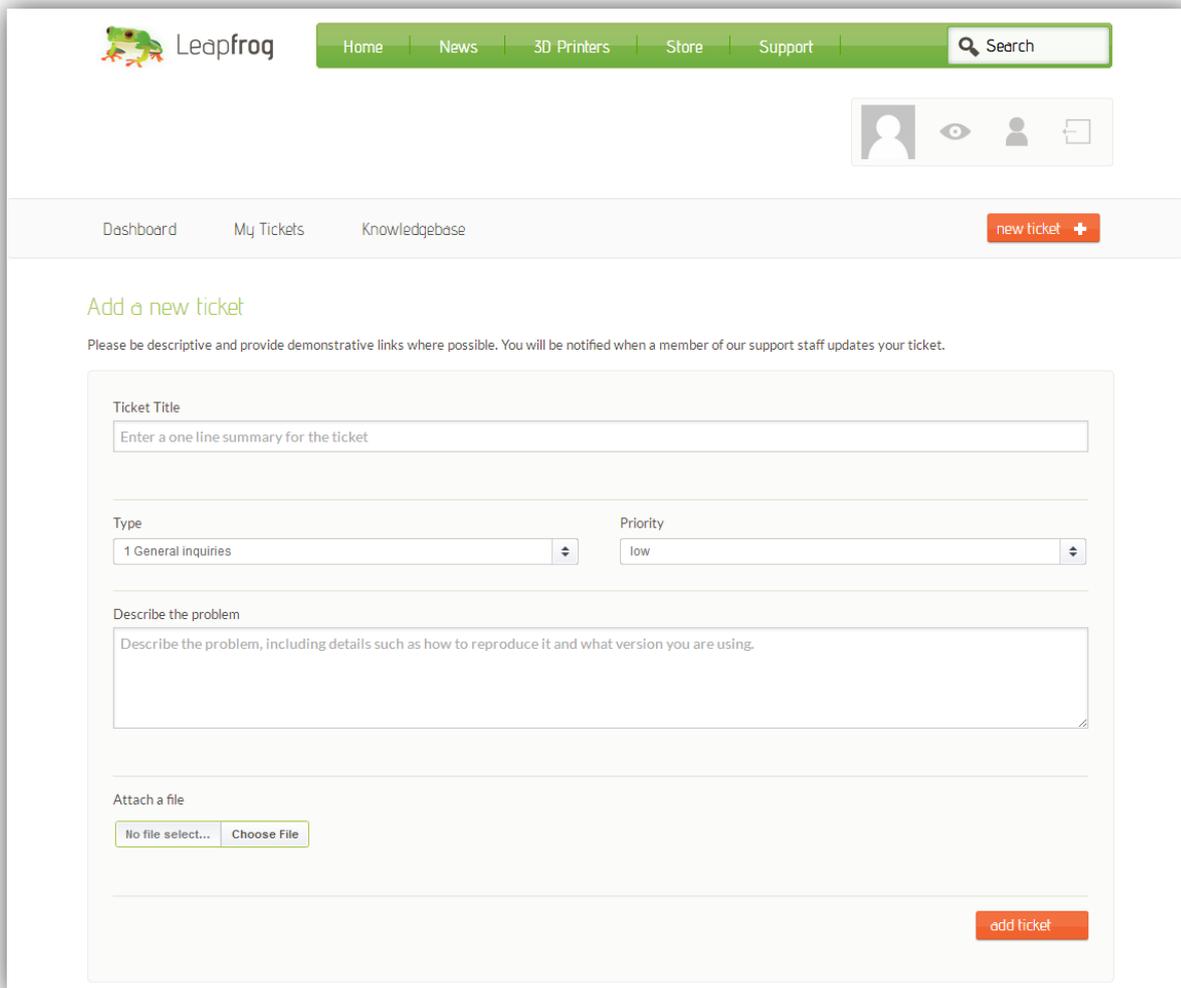
- Click on the button with the Ticket icon, select “Existing customer” and log in with your account. If you do not have an account or if you have forgotten your account, please create one.

- In our support ticket system you can again see the “*Knowledgebase*”. If you have not yet checked if your question is listed here, please do so.

If your question is not to be found in the Knowledgebase, create a new ticket by clicking on “*New ticket*”

- Summarize your question in one sentence and enter this in the “*Ticket title*” field.
- Select the category which best suits your question.
- Select the priority of your question.
- Next, please describe your question as detailed as possible so our support team can answer you as best as possible.
- Clarifying your question with pic’s and movies will significantly increase the speed of providing a clear answer, you can add files below “*Attach a file*”.

You will receive an automatic email when your ticket has been updated. This way, you do not have to check if you have received an answer but we will let you know.



The screenshot shows the Leapfrog website's support ticket creation interface. At the top, there is a navigation bar with links for Home, News, 3D Printers, Store, and Support, along with a search bar. Below the navigation bar, there are icons for user profile, eye, and document. The main content area has a breadcrumb trail: Dashboard > My Tickets > Knowledgebase, and a red button labeled "new ticket +". The "Add a new ticket" section includes a sub-header and a note: "Please be descriptive and provide demonstrative links where possible. You will be notified when a member of our support staff updates your ticket." The form itself has several fields: "Ticket Title" with a placeholder "Enter a one line summary for the ticket"; "Type" dropdown menu set to "1 General inquiries"; "Priority" dropdown menu set to "low"; "Describe the problem" with a text area and placeholder "Describe the problem, including details such as how to reproduce it and what version you are using."; and "Attach a file" with a "Choose File" button. A red "add ticket" button is located at the bottom right of the form.