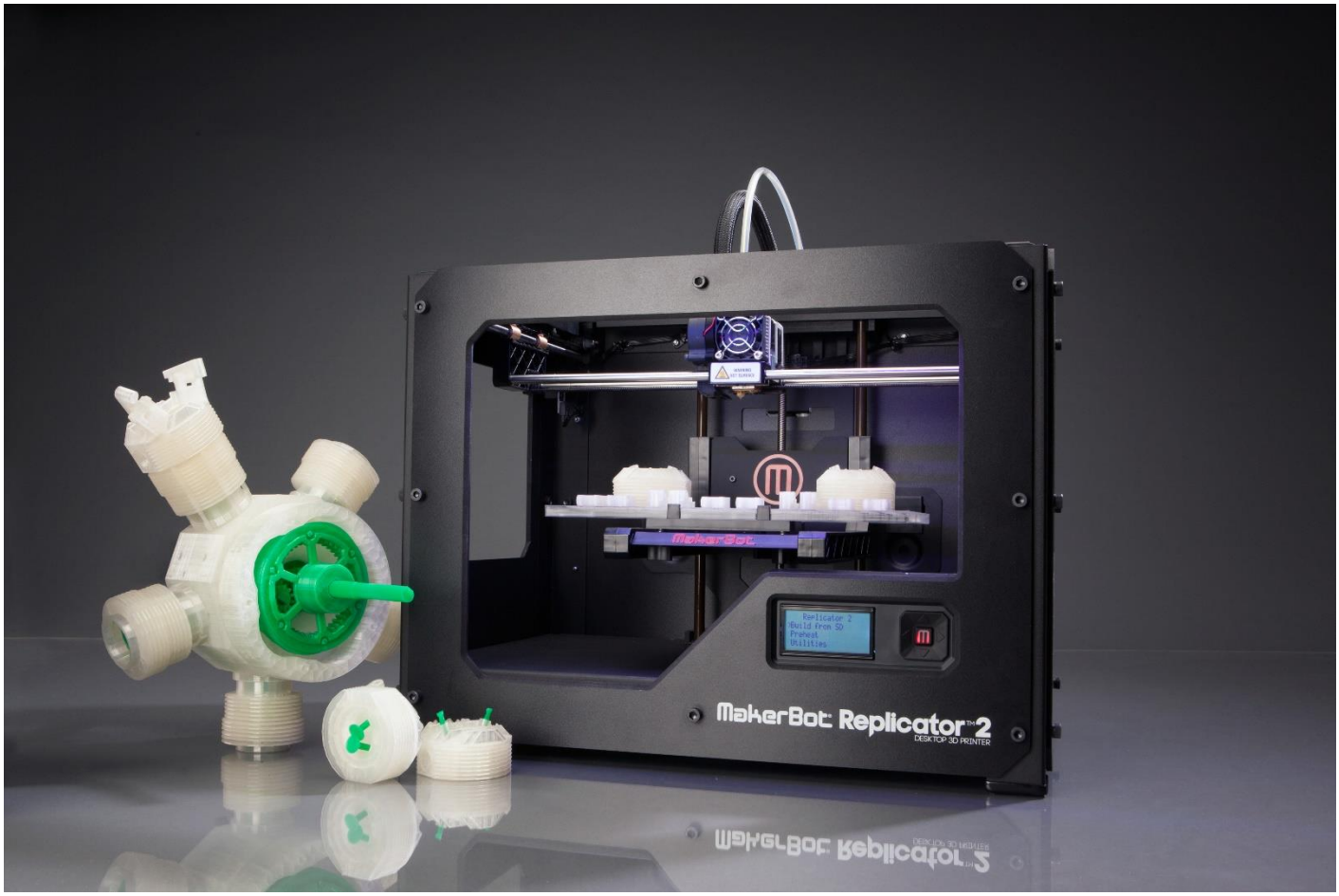
Makerbot Replicator 2 Guide

(This copy is written for all users of the MR2. Most of it can be used on other 3D printers)



Written by: Bryson Hicks

Includes basics, troubleshooting, and the use of Slic3r for reliable advanced prints.

First edition (May 2014)

Background

Basics of what a 3D printer is and what it does:

There are many different types of 3D printers. A Replicator 2 is a FDM (Fused Deposition Modeling) printer. A FDM printer melts plastic to form one layer at a time to slowly build up the object. Essentially, it works similar to a hot glue gun. The "extruder" will move around on the X and Y axis to place melted plastic for a layer. Then, the Z axis will move down to make room for the next layer. These objects are created three dimensionally in a computer. Then another program will "slice" the object to create each layer. Then the machine will print each one of those layers to create the object.

Benefits of a 3D printer:

- Direct from computer to product.
    - If you design a product in a computer, you do not need to manually create the part. You can create a part with minimal machining experience.
- Repeatability
    - You can create the same product over and over again with the same results.
- Complex fabrication.
    - You can fabricate complex parts and objects that would be impossible to create by hand.
- Rapid prototyping
    - An engineer can think of a concept, and print it to see if it works in real life. If it does, it can be used as the final part, or allow for more complex and expensive fabrication to be used.

Important terms to know:

- Filament: This is the material that is melted and used to create the part.
- Extruder: This is the device that moves around and extrudes the filament. It contains the heater and motor that pushes the filament into the heater.

- PLA: (Polylactic acid) This is the type of plastic that you will be using. It is biodegradable and easier to work with, but it is more brittle than ABS.
- ABS: (Acrylonitrile butadiene styrene) You cannot use this with the Replicator 2. It requires a different nozzle and a heated build place.

- STL: (STereoLithography) This is the type of 3D file that you will use to make the part. It is made of thousands of triangles and it has no information except the surface shape. It does not include the units, so it is important to export it with the same units. For example, if you export it in inches, and important in mm, the part will be too small.
- Gcode: This is a file that contains instructions that tells a 3D printer how to make a part. Simplified, it will say "move 1mm up at 80mm/s"

- Stepper Motors: These are extremely precise motors that move the extruder around and push the filament into the heating block. They move in tiny steps. When they are "activated" and not told to move, they will lock into place, so make sure that you do not attempt to move them while they are activated.

The Extruder

The parts of the extruder and their functions:

The extruder is actually much simpler than it looks. It is primarily made up of four components that can easily be removed by removing the two socket head bolts on the front of the extruder. These bolts screw into the stepper motor and hold everything together. Once they are removed, the majority of the extruder can be disassembled. (Make sure that you unplug the stepper motor first.) You should never have to disassemble the extruder more than removing these four components unless you are doing advanced repair.

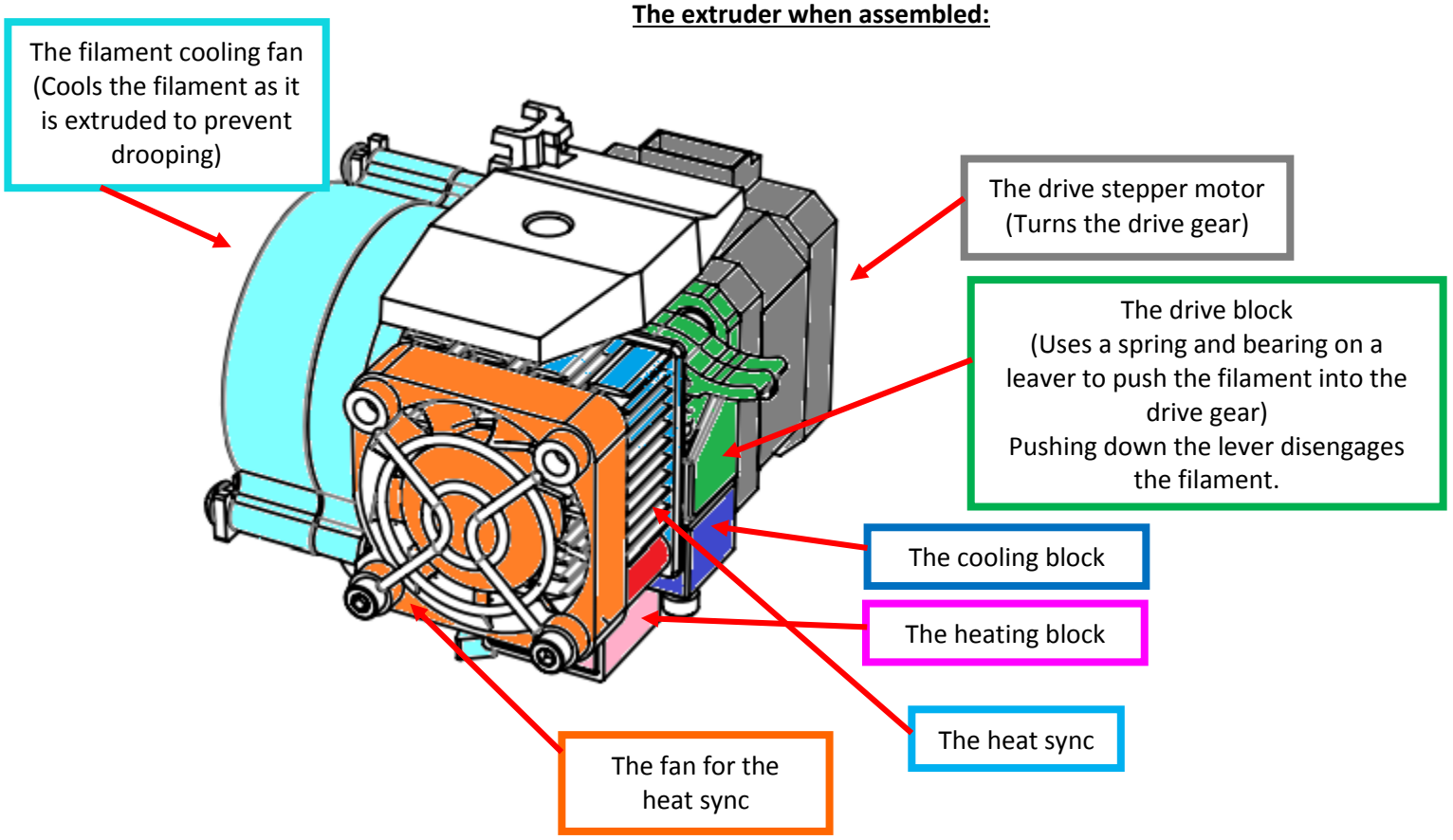
Below the four components that are connected via the bolts are two aluminum "blocks". These are removed via other methods, but you should never have to remove them. The top block is the cooling block, and the bottom block is the heating block. Knowing what they do is important for troubleshooting.

The purpose of the cooling block is to prevent the spread of heat from the heating block. It should always be in contact with the heat sync. This keeps the stepper motor and the filament that is above the heating block cool. If the filament above the heating block becomes too hot, the drive gear would not be able to grip it properly and it will jam.

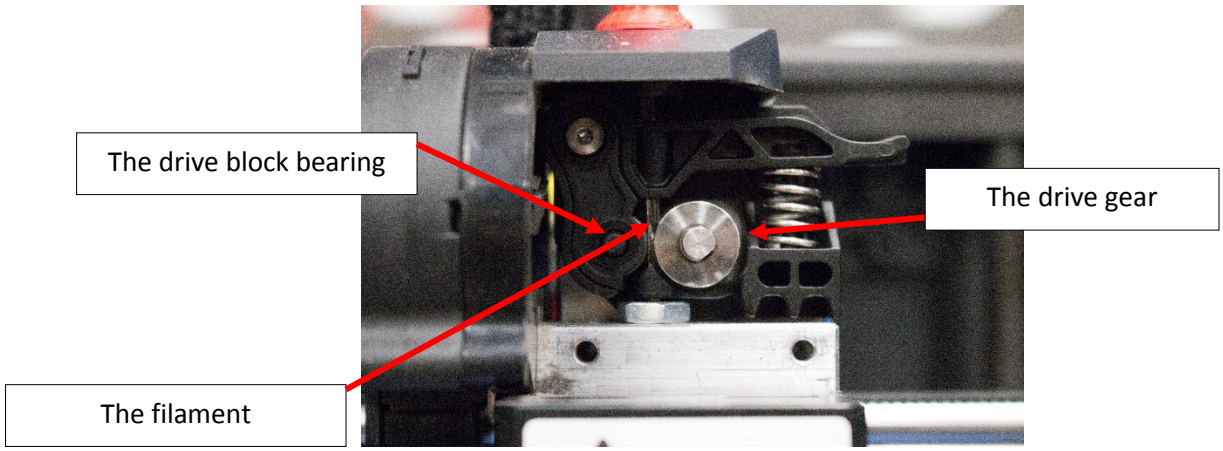
The purpose of the heating block is to melt the filament for extrusion. It is a small aluminum block with insulation wrapped around it. Inside of the block is a thermocouple (that measures temperature) and a resistor (that creates heat).

Below this is the nozzle. NEVER ATTEMPT TO REMOVE THE NOZZLE. With the Rep2, the nozzle is torque tightened in the factory. If you attempt to remove the nozzle, you will destroy it.

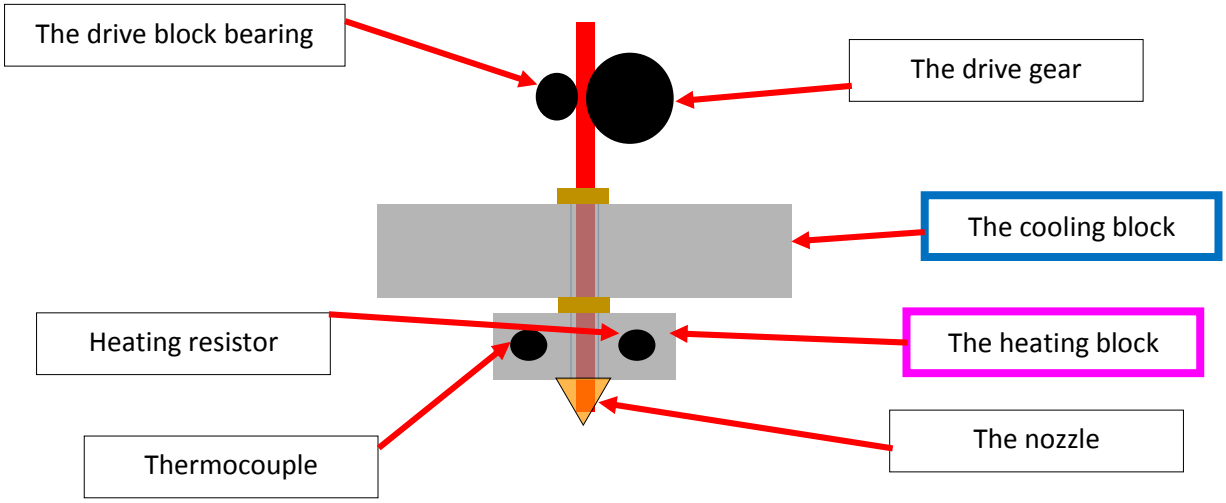
The extruder when assembled:



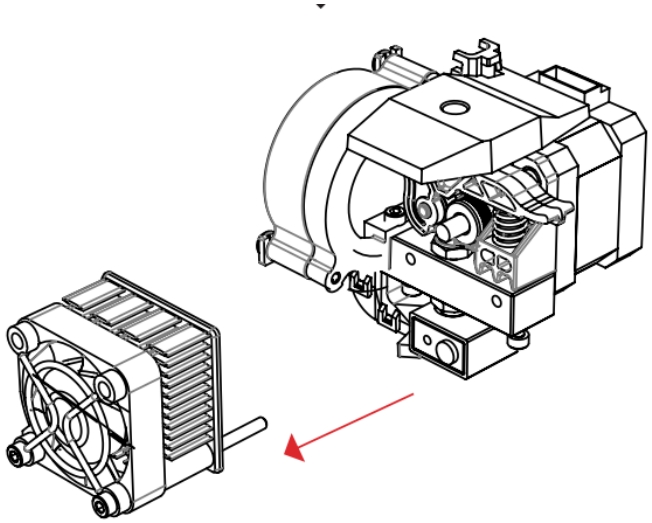
A view of the drive block:



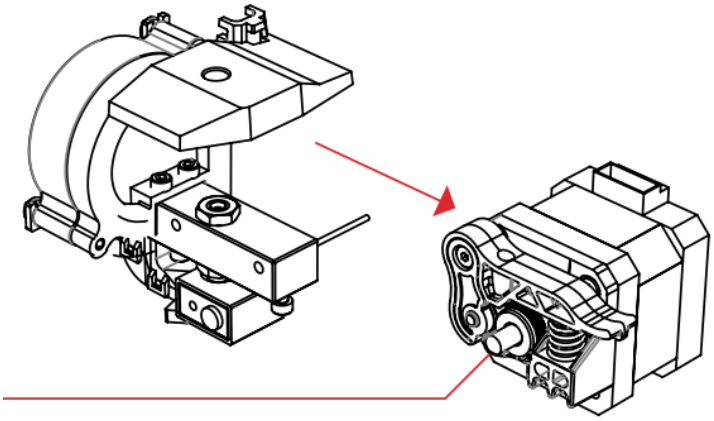
A cut away view of the inside:



With fan and heat sync removed:



With motor and drive block removed:



Maintaining the extruder:

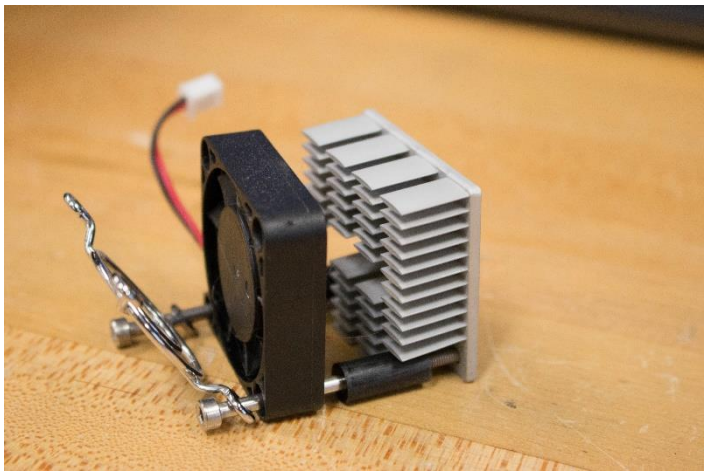
Every time the extruder jams and stops extruding during a print, you MUST clean out the extruder. If it jams, the drive gear will continue to turn and the teeth on it will fill up with scraps of filament.

Steps to take apart the extruder for basic cleaning:

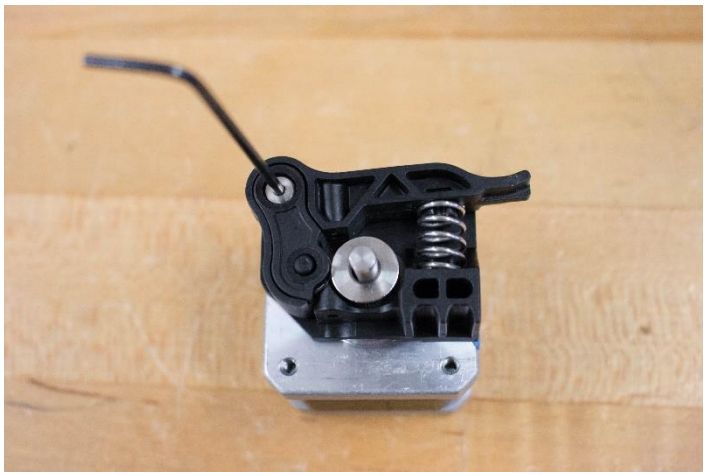
1. Preheat the machine via RepG or via the controller on the machine.
   I. To preheat via the machine, move down the menu until you select "Preheat".
   II. To preheat via RepG (assuming the Rep2 is connected), click on the control panel button (the four arrows) and enter the 220 into the temperature target, and then press enter.

2. Once it is above 220C, remove the filament. (Look below for more info on how to do that.)
3. Make sure that you unplug the motor first and don't forget to plug it back in.
4. Loosen the two Allen bolts in the front. Don't remove them.

5. Slide out the fan, spacers, and heat sync. You can let it hang carefully from the fan power cable. This is what it should look like:

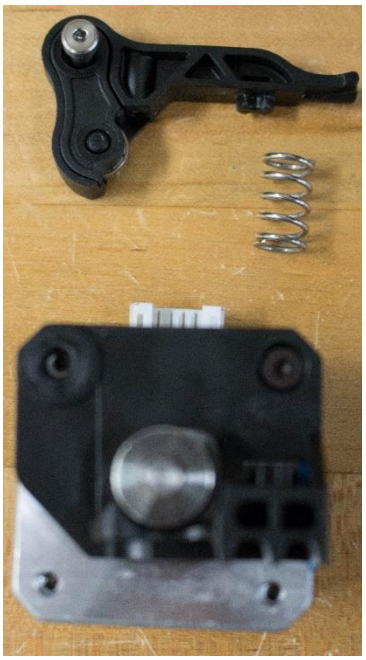


6. Slide out the motor and the drive block.



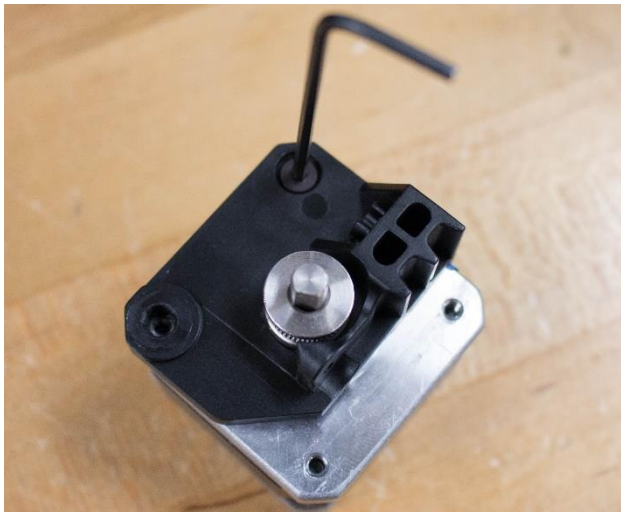
5. Remove the drive block.
   I. Use a smaller allen wrench to remove the screw on the top left. Remove the lever and spring.
   II. Blow off any partials on the bearing.
   III. Remove the screw on the top right and remove the rest of the drive block.
   IV. Use an air compressor or brush to remove any partials on the drive gear.
   V. If you are still having problems, use a degreaser or other cleaner to completely clean the drive gear teeth. Do not attempt to remove the drive gear.
6. Reassemble by going backwards.
   I. THE STICKER ON THE FAN MUST FACE INWARDS. Otherwise it will blow in the wrong direction. (The sticker should not be visible when assembled.)
   II. Don't forget to plug the motor back in.

I.

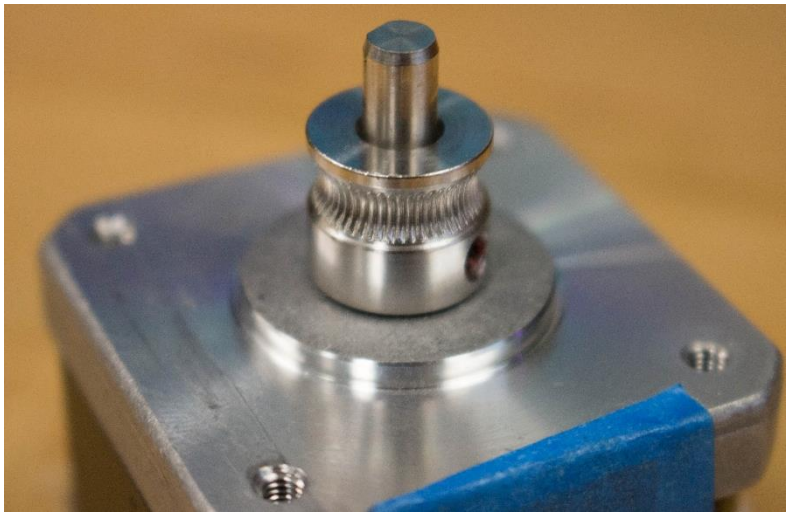


II.

III.



IV.



Advanced cleaning:

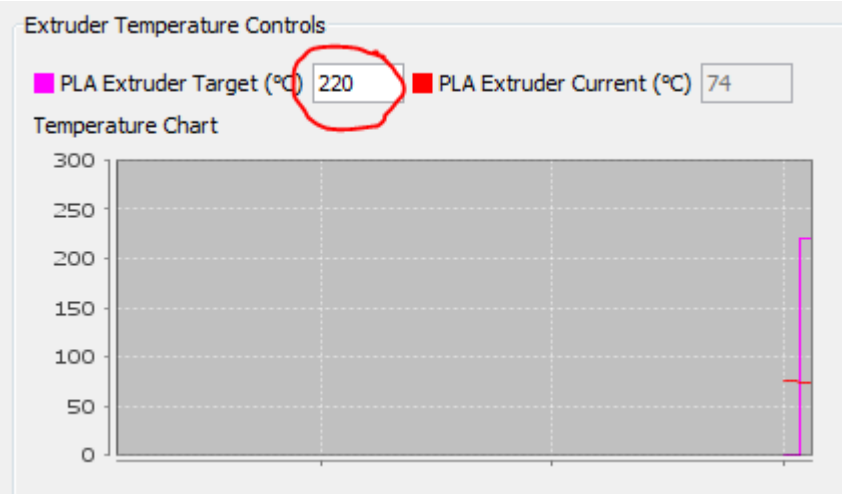
If you are still having problems, there is further cleaning you can do. This may seem extreme, but it is sanctioned by makerbot and if you ask them what to do, this is what they tell you to do.

1. Follow the instructions above 1 through 7.
2. With all of the four major components removed, use RepG to preheat to 260C. Do not leave it at this high temperature for long periods of time.
3. Quickly force filament down into the heating block.
4. Once you have used three inches or so, quickly pull the filament out to try and rip out whatever is causing the problem.
5. Do the quick pull method multiple times. It is generally good to use another color of filament and keep doing the quick pull method until you have removed all of the old color.
6. You can also use guitar string or other thin wire. Feed it up through the nozzle and push it out the top.
7. Reassemble.

Loading and unloading filament:

You can use the built in commands in the menu to do this. However, with the new type of drive block (with a lever) that is not necessary. You can just use the lever on the right side of the extruder to disengage the filament and pull it out.

1. Preheat the machine via RepG or via the controller on the machine.
   I. To preheat on the machine, move down the menu until you select "Preheat".
   II. To preheat via RepG (assuming the Rep2 is connected), click on the control panel button (the four arrows) and enter the 220 into the temperature, and then press enter.



2. Once it is above 200C. Push the lever on the right side down. Make sure to brace the bottom of the extruder by holding the cooling block. If you press down with no upwards support, you risk bending the gantry.
3. Remove the clear guide tube on the top and pull the filament out by hand. It should not be difficult, but sometimes it is. It may require pushing and pulling. Double check that you are at 220C if you are having problems.
4. Spray WD-40 on a rag and wipe it on the filament holder. This provides less resistance for the extruder when it pulls filament. If you are having widespread problems, resistance could be the source. I recommend building a rig where filament is held above the Makerbot on a low friction bar.

The steps to printing a part:

Order and uses of programs:

- CAD software
    - This is the computer program where the part is created. There are a huge variety of programs available for this.
    - Autodesk inventor is a very powerful CAD program that can be downloaded for free if you are a student.
    - Thingiverse.com is a huge database with thousands of objects created by people. Instead of using CAD software, you can download parts directly from here.
    - The CAD software provides a .stl file.
- Makerware (Optional)
    - You cannot three dimensionally rotate a .stl file in Slic3r. If your object is not oriented correctly. You can open the object in makerware, rotate it, and then resave it as an stl.
- Slic3r
    - This program will take the 3D model and "slice" it into individual layers.
    - After adding the .stl file, you will enter your settings such as the speeds and temperatures.
    - This creates the .Gcode file. These are the basic instructions of where the machine should move and print.
- RepG
    - This program will open the Gcode file and send it to the printer.
    - You can either send the data to the printer through a USB cable, or you can write a .x3g file that the printer can view on an SD card.
    - RepG will set how many steps the motors should take to move during a print. You can calibrate the steps/mm in RepG.

- Other programs available.

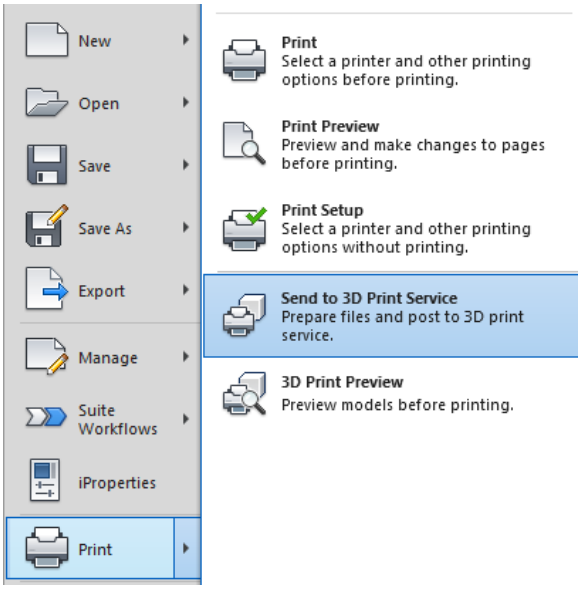
    - You can use RepG or Makerware as slicers. However, they are not nearly as effective as Slic3r and you will not have nearly as many options.

Exporting the CAD file:

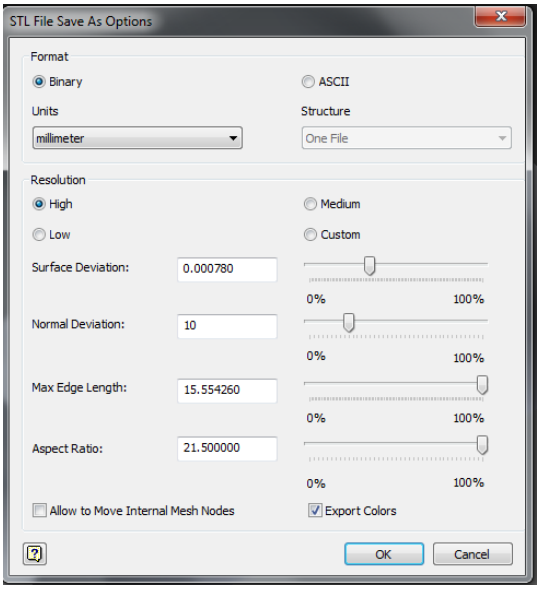
The CAD program needs to export a .stl file for you to use. It is made of thousands of triangles and it has no information except the surface shape. It does not include the units, so it is important to export it with the same units. For example, if you export it in inches, and import in mm, the part will be too small. Slic3r imports in mm, so export using your program of choice in mm.

If you are using Inventor:

1. With the part selected, go to print > Send to 3D printer service.



2. Click on options and make sure that units are set to mm and resolution is set to High.



3. Then click “OK” and save the .stl. If a page about 3D printing service pops up, close it.

Using Makerware (Optional):

Makerware can be downloaded from: makerbot.com/makerware/

If you open the .stl in Slic3r and see that it is not oriented correctly, you will need to rotate it first.

1. Click and drag your file into makerware or go to file > Open.
2. With the part selected, click on turn and use the buttons until it is placed correctly. It does not need to be centered.
3. Placing the part correctly can often be tricky. Unless it is over a short distance with support on both sides, you do not want to print over air. Rotate your object until it is oriented so that it has the least amount of area where it will print over air. If there is no way to prevent printing over air, you will use supports in Slic3r.
4. Now the file is ready for Slic3r. Save it as a new .stl .

Using Slic3r:

You can download slic3r from: http://slic3r.org/

This program will take the 3D model and "slice" it into individual layers. After adding the .stl file, you will enter your settings such as the speeds and temperatures. This creates the .Gcode file. These are the basic instructions of where the machine should move and print.

The first thing that you should do is make sure you are in "Expert" mode. You can change this under File > Preferences > Mode

Adding your object:

1. Press the "Add" button on the top left or drag your file onto the "build plate".
2. Click the "View" button to make sure that the object is oriented correctly.
3. If one file contains multiple parts, click the "Split" button. Then select all of the parts you do not want and click the "delete" button.
    a. It is preferable to not print multiple parts at the same time. You will lose print quality and add a significant amount of time.

Using pre-sets:

1. On the right side of the screen under the "Platter" tab are pre-sets that you can use. You will need to configure these yourself.
2. To use them, simply select one from a drop down menu.
3. To save one, click the "save" symbol while in a tab and give it a name.

Configuration:

You can load configurations via File > load config. I have uploaded a custom stable configuration here:

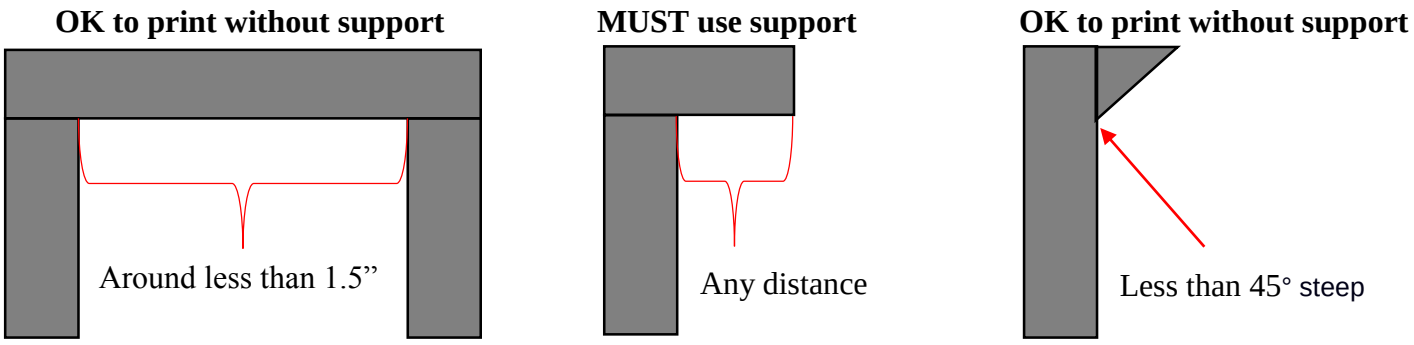
Mirror 1: http://bit.ly/megamirror

Mirror 2: http://bit.ly/boxmirror

Keep in mind that it is set to 40% infill with no support so you may need to change that.

Printing support:

It is generally best to not print with support if you can. Supports use up material, increase print time, and leave a bad finish. If you have something where the material is reaching out to so that it can create a "bridge", then in most situations you do not need support.



Advanced Slic3r Information

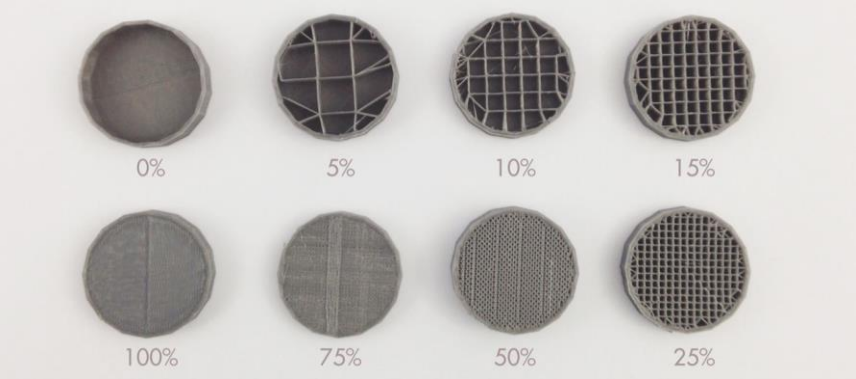
Print settings tab:

Layer and perimeters:

- Layer height
  - Layer height: This sets the height of each layer that is printed. Going above .2mm reduces print quality, but increases speed. Going below .2mm increases print quality, but significantly increases the chance of a jam.
  - First layer height: This overrides the above value to make the first layer thicker. This increases the chances of is sticking better. .4mm is the same diameter as the extruder nozzle and that is optimum. If you need a high quality first layer, you can drop this to .2mm, but you have a high chance that the print will not stick to the build plate.
- Vertical shells
  - Shells are the number of solid loops around the perimeter of the object. If you have three shells on an object, there will be three solid loops before the infill begins. The more layers there are, the stronger it will be, but the longer it will take. 2 is generally a good value.
  - Vertical shells are the number of solid layers on the top and bottom faces of a part.
  - Spiral vase: Only works for single width objects. Do not activate this.
- Horizontal shells
  - This is the same as the vertical shells, but this controls the layers on the top and bottom of the object.
  - Three top horizontal shells will result in three solid layers on the top.
- Quality
  - Extra perimeters: leave this on
  - Avoid crossing perimeters: leave this off unless you are trying to force the printer to stay within one area before moving on to the next. This significantly increases processing time.
  - Start perimeters at: I have tried this before, but it did not make a difference.
  - Detect thin walls: Leave this on. Normally, the printer can only print even layered walls, (walls are NOT shells). This is because each shell will go on each side of the wall at least once to count as a shell. If you have an area that is only one extrusion thick, then this will detect that and override the shells.
  - Detect bridging perimeters: Leave this on
- Advanced:
  - Leave these both off unless you want to do experimental work.

Infill:

- Infill
  - Fill density:
    - This changes the density of the infill. Keep in mind a little goes a LONG way.
    - 10% is good for objects under no stress. 25% is good for objects under medium stress. 50% is good for objects under high stress. 100% is good for small objects under high stress.



  - Fill pattern: Leave this on rectangular
  - Top/bottom fill pattern: This changes the pattern for the fill of the top and bottom visible solid layers. Leave this on rectangular unless you want a different look.
- Reducing print time
  - Combine infill every: Leave this at 1. I have tried higher values and I did not notice a change.
  - Only infill when needed: This will make it only infill when it is necessary for supporting the print ceiling. Leave this off.
- Advanced:
  - Solid infill every __ layers: This can add extra strength to an object.
  - Fill angle: This changes the angle of the infill pattern.
  - Solid infill threshold: Do not change this. I have not seen it have an effect on the print.
  - Only retract when crossing perimeters: Leave this on. If this extruder is not crossing perimeters, then it is inside of the object. There is no need for retraction inside of the object.
  - Infill before perimeters: This is self-explanatory. Leave it deactivated.

Speed:

(I have included my recommended speeds in the format of (Fast print speed, high quality print speed)

- Speed for print moves
  - Perimeters: This is all of your shells. (120, 80)
  - Small perimeters: This overrides the above speeds for perimeters that are small. (Have a radius > or = 6.5mm.) (80,60)
  - External perimeters: This is the outer shell that will be visible. It should be slower for a better surface finish. (80,60)
  - Infill: This is self-explanatory. (120, 100)
  - Solid Infill: These are the layers that are solid. (100, 80)
  - Top solid infill: This is the top visible layer. (80,60)
  - Support material: Also self-explanatory. (120, 100)
  - Bridges: This recognizes where there is no material below an area of a layer, but there is material to form a bridge. See "printing support" for more info. (100, 80)
  - Gap fill: This fills in areas that are very small. If this is set high, it will induce dangerous vibrations. (20,20)
- Speed for non-print moves
  - Travel: This is when the extruder moves from one spot to another without extruding. (120,120)
- Modifiers
  - First layer speed: This is the speed of the firs layer. It will override everything on the first layer. (30,30)
- Acceleration control
  - Acceleration is VERY important. However, it is controlled in the machines firmware, so LEAVE THIS OFF

Skirt and brim:

- This primes the extruder and tests boundaries by extruding one line around the object before the print starts.
- This is self-explanatory and it is best left on defaults.

Support material

- First read over "Printing support"
- Support material
  - Generate… : This is how you completely disable or enable support material.
  - Overhang threshold: This sets the angle where it will auto detect that support is needed. 45 is a good value, but you can push it to 30 with good results.
  - Enforce support for the first ___ layers: Leave this disabled unless you have an object that only touches the build plate at small areas (like an upside down pyramid).
- Raft:
  - This uses a lot of filament to create a sticky raft below the material that is difficult to remove. With the slower and wider first layer that we have set, that is not necessary. A raft is only needed in inferior slicers like makerware.
- Options for support material and raft:
  - Pattern: Leave this at rectangular.
  - Pattern spacing: This sets the distance between the lines in the support. It is similar to setting the density. For supporting small things, 2.5mm is good. For supporting large areas, you can push this to 6mm or higher.
  - Pattern angle: This is self-explanatory. Leave it at default.
  - Interface layers: This creates a layer between the support and the object to make it easier to remove. However, I have found it to create more of a mess than a benefit.

Notes

- This just adds custom text to the beginning of the Gcode. It does not affect the print.

Output options and multiple extruders

- You don't need to mess with these.

Advanced

- You can leave most of this at default.
- Change "First layer" to 200%. This makes the width of the first layer twice as wide as normal, and it keeps it from peeling off of the build plate. You can set this back to default if you need a high quality first layer, but you have a much higher chance of peeling off.

Filament settings tab:

Filament

- Filament
    - Diameter: Normally leave this at 1.75mm. If you are having problems with too much filament or too little filament being used, you should update this by measuring the filament with calipers. PLA will very slowly expand in humid environments, and some rolls are slightly thinner or thicker than others. However, you should avoid changing this from 1.75 unless you are having problems.
    - Extrusion multiplier: Don't change this. It is better to update the diameter instead.
- Temperature:
    - I have had great results with setting the temperature to 220 on the first later and 210 on the rest. Some filaments have slightly different optimum temperatures, so if you are having problems you can change this.
    - Too high of a temperature will result in drooping and ooze.
    - Too low of a temperature will result in layers not sticking and a rough finish.
    - Black filament is notoriously tricky.

Cooling

- This allows you to toggle the fan on the left side of the extruder that blows on the part. This helps to keep the part from drooping by cooling it quickly.
- I have printed with "keep fan always on" enabled and had great results. I recommended just leaving it on.

Printer settings tab:

General

- Everything should be left at default unless otherwise stated.
- Bed size: x 280 y 150
- G-code flavor: Makerware

Custom Gcode:

- This is the code that tells the printer the beginning and ending instructions for printing. It is different for every printer. This code works with the Makerbot Replicator 2:

Start code:

```
(**** Start.gcode for Replicator 2 ****)
M103 ;disable RPM
M73 P0 ;enable build progress
G21 ;set units to mm
G90 ;set positioning to absolute
(**** begin homing ****)
G162 X Y F2500 ;home XY axes maximum
G161 Z F1100 ;home Z axis minimum
G92 Z-5 ;set Z to -5
G1 Z0.0 ;move Z to "0"
G161 Z F100 ;home Z axis minimum
M132 X Y Z A B ;Recall stored home offsets for XYZAB axis
(**** end homing ****)
G1 X-141 Y-74 Z75 F3300.0 ;move to waiting position
G130 X20 Y20 Z20 A20 B20 ;Lower stepper Vrefs while heating
M6 T0 ;wait for toolhead, and HBP to reach temperature
M70 P5 (YOU CAN PLACE ANY TEXT HERE THAT YOU WANT)
G130 X127 Y127 Z40 A127 B127 ;Set Stepper motor Vref to defaults
M108 R3.0 T0
G0 X-141 Y-74 ;Position Nozzle
G0 Z1 ;Move above plate
G1 Z0.3 ;Position Height
M108 R5.0 ;Set Extruder Speed
G4 P3000 ;Create Anchor
G1 X-137 Y-70 Z0.1 F1000.0 ;Slow wipe
G0 Z0.3 ;and lift
(**** end of Start.gcode ****)
```

End code:

```
(**** End.gcode ****)
M103 ;Turn extruder off
M104 S0 ;Zero temperature
M73 P100 ;End  build progress
G0 Z155 ;Send Z axis to bottom of machine
M18 ;Disable stepper
M104 S0 T0 ;Cool down the Right Extruder
G162 X Y F2500 ;Home XY endstops
M18 ;Disable stepper motors
M70 P5 (YOU CAN PLACE ANY TEXT HERE THAT YOU WANT)
M72 P1 ;Play Ta-Da song
(**** end of End.gcode ****)
```

New layer code:

```
(**** New Layer ****)
```

Extruder 1:

- Everything should be left as default unless otherwise noted
- You can disable retraction entirely by setting length to 0. This should only be done for troubleshooting.
- Nozzle diameter: .4mm
- Retraction length: .5mm
- Retraction speed: 15mm/s

Move to next page for information on Replicator G

Using Replicator G:

This opens the GCode "instructions" and turns it into a file that the printer can read. This will convert the Gcode instructions of "move 1mm up" to "move 80 steps up." Replicator G also allows for control over the printer so you can preheat the extruder to a higher temperature and control the stepper motors.

Always check that you have the right 3D printer type selected before you do anything. Go to Machine > Machine type > Replicator 2. If you don't do this, you could cause collisions.

Adding your Gcode:

1. Go to File > Open and select your file.
2. The Gcode will open and you should see the code itself.
3. If you click on the "model" tab, you will see the .stl model that has the same name and is in the same folder as the code. RepG automatically finds this file. It does NOT necessarily show what the print will look like.

Printing via USB:

1. This will actively send data packets to the printer. Do not close RepG during printing or use other processor heavy programs as it could stop the packets and mess up the print.
2. With your Gcode open, press the button on the top left showing the arrow and "loop". This will start the print. The printer will automatically preheat.

Printing via SD card:

1. Make sure you are using a 2GB or less SD card that is formatted to FAT16 (also known as just FAT and NOT FAT 32)
2. With your GCode open, click the third button to the left that shows an arrow and a document. This will write the Gocde to the SD card in your computer.
3. Make sure to choose a short name. A long name may not show on the printer.
4. MAKE SURE to save it as a .X3G. NOT a .s3g.
5. After it has written, inset the SD card into the printer. Select "build from SD card" and then select the file. It will automatically begin printing.

Calibrating RepG:

- If you are not using the drivers I have already modified on the designated computer, or you are using an account on that computer other than the designated one supplied, you will have to update the driver with the correct steps/mm.
- This is not necessary for simple models. It is only necessary for parts that need to be accurate for engineering use.

1. First, print a model of known size that you can measure. Thing number 28109 on thingiverse.com is good.
2. After printing this with your current settings, measure the X and Y axis length and compare that to what it should be. For thing # 281109, use this formula: (110mm) / (Measured length of each axis) = _____%
3. Now you need to update the driver's step/mm.
4. Go to (file location of RepG program) > replicatorg-0040 > machines > replicator (file not folder)
5. Start notepad as an administrator. Do this by finding notepad (just search it) and then right click on it > Run as administrator.
6. Open the file found in step four.
7. For each axis, find "axis id = (the one you want to update)"
8. Do the following to update the "stepspermm=" number that corresponds with the correct axis id:
   (the currently set number) x (the percentage that was calculated before) = (the new value.)
9. After updating each axis, save the file and restart RepG.
10. Go to machine > machine information and check that the new steps per mm is there.

Preparing the printer:

Adding tape:

Painters tape is necessary to keep the object attached to the build plate. If you are starting to have problems with the object peeling, replace the tape. Simply remove the old tape and add a new layer. Make sure to minimize gaps.

Leveling:

In order to make sure the first layer is applied evenly, you must level the plate periodically. This is relatively simple to do and instructions are built in. Simply find the "level build plate" utility in the printer menu. When performing this procedure, make sure that you use a piece of paper or a card that is .1mm thick. If you are having problems, this video is very helpful. http://youtu.be/Hedb_g2kcXg

You know that the plate is correctly leveled if the curtain is printed correctly. When the extruder first moves around to print the curtain, the curtain should have an even thickness throughout. If one side of the curtain is thinner, then the plate is too low on that side. If one side of the curtain is not printing at all, then it is too high and the plate is preventing the filament from being extruded. Adjust accordingly. The curtain should be about 1.1mm wide.

Troubleshooting:

It stopped extruding mid print.

Cancel the print and clean the extruder. If you still have problems, do the advanced cleaning. Make sure the sticker on the fan is facing inwards.

The size of the object is slightly off.

Double check that you calibrated the driver code in replicator G.

The size of the object is VERY off.

Make sure that you exported your file in the correct units of mm and that you have selected the correct machine in RepG.

There are strings of filament all over the object where there should be nothing.

Make sure that retraction is enabled.

Layers are not sticking, or there is a rough surface finish.

Raise the temperature 5 degrees.

RepG crashed or I got an SD card error part of the way through the print.

Restart the print. This is random and cannot be avoided. If it happens continuously, reslice the object. Also try a new SD card.

My object won't show up under "print from SD card"

Make sure that the name is not too long and that it is saved as a .X3G file type.

The SD card wont show up at all.

Make sure that the SD card is under 2GB and that it is saved in Fat 16 format. (Also known as just FAT.)

I'm having problems connecting Rep G to the printer.

Make sure that the printer is connected via USB. Close RepG and shut off the printer. Turn back on the printer, then wait five seconds. Open RepG and it should connect.