



**Getting Started Guide v1**  
**by Marcus Dobeck and Rick Pollack**

## Important Note

At the time of writing, this is the current accepted standard and works well with MakerGear Prusa and Mosaic printers. There may be future upgrades that are not covered in this document. We encourage experimentation and also suggest you back up your existing version/settings before attempting to upgrade. This guide is meant to be followed sequentially.

## PreSetup

Download the following and place in a folder on your desktop labeled “MakerGear\_Source\_Files”.

### For Windows Users:

Download the latest Printron experimental from [here](#). This version does not require the following dependency files.

### Download Pronterface + Prerequisites (only required if not using the above file)

Latest Pronterface: <https://github.com/kliment/Printron>

PYSerial: <http://sourceforge.net/projects/pyserial/>

PYReadline: <https://launchpad.net/pyreadline/+download>

wxPython (Unicode): <http://www.wxpython.org/download.php>

### To use Skeinforge or SFACT, python is required

Python 2.x: <http://python.org/download>

### Download Latest SFACT

SFACT: <http://github.com/ahmetcemturan/SFACT>

**Slic3r is a new slicing program that is easier to use than Skeinforge but does not have as many features. If you don't need support material, try Slic3r first**

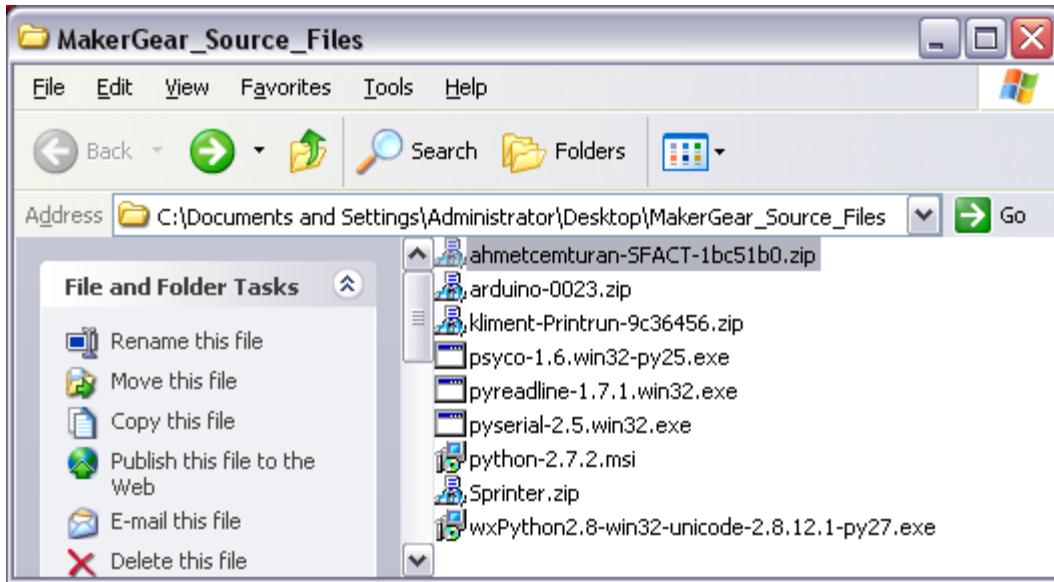
<http://slic3r.org/>

### Download latest Arduino IDE + Firmware

<http://www.arduino.cc/en/Main/software>

Firmware: (Optional) <http://groups.google.com/group/makergear> (select Sprinter for Prusa or Mosaic respectively)

Your MakerGear\_Source\_Files folder will look similar to this: (if you have downloaded all of the dependency files)



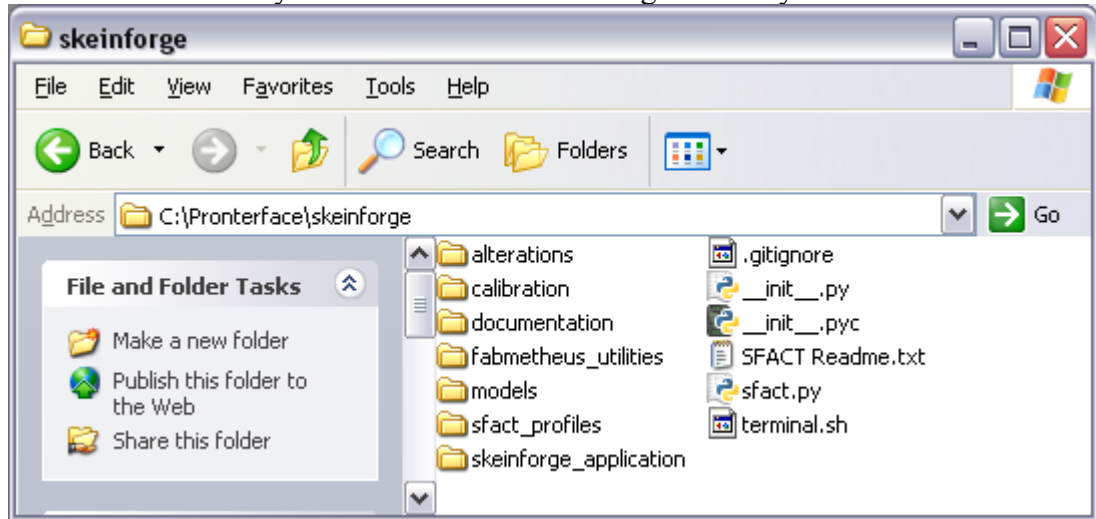
## Software Installs

### Pronterface

1. Install Python 2.7x from the MakerGear\_Source\_Files folder, select defaults.
2. Install PYSerial from the MakerGear\_Source\_Files folder, select defaults.
3. Install PYReadline from the MakerGear\_Source\_Files folder, select defaults.
4. Install wxPython from the MakerGear\_Source\_Files folder, select defaults.
5. Install Pronterface:
  - a. Unzip kliment-Printrun-\*\*\*.zip to a subfolder.
  - b. Rename the subfolder that contains the Pronterface files to “Pronterface”
  - c. Move the folder to the root of your “C” drive.
  - d. Create a shortcut on your desktop pointing to C:\Pronterface\Pronterface.py
  - e. Right click on the shortcut, select Properties, select “Change Icon”, browse to C:\Pronterface\p-face.ico hit OK to set the icon.
6. Briefly test Pronterface by double clicking on the desktop icon.
7. For additional help, please see the Pronterface Wiki:  
<http://reprap.org/wiki/Printrun>

## SFACT

1. Extract SFACT into your C:\Pronterface\skeinforge directory



2. For additional help, please see the SFACT wiki(s):

<http://www.reprapfordummies.net/>

<http://reprap.org/wiki/Sfact>

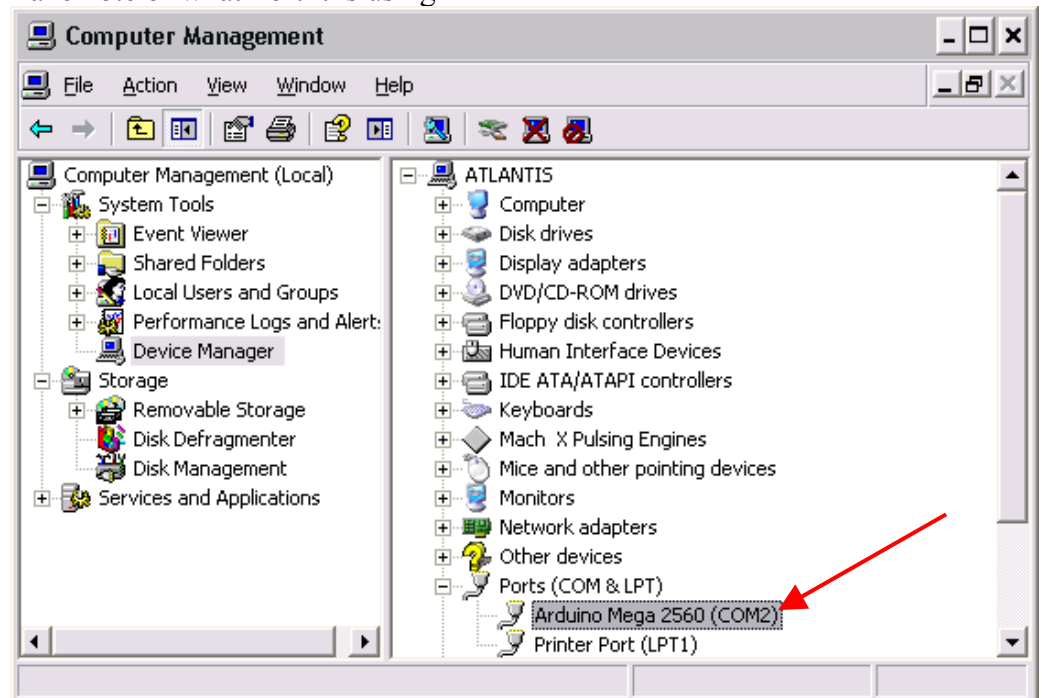
## Arduino IDE

1. Unzip arduino-00\*\*.zip to a subfolder and move it to C:\
2. Create a shortcut on your desktop pointing to C:\arduino-0023\arduino.exe

## Driver Install/Firmware Upload

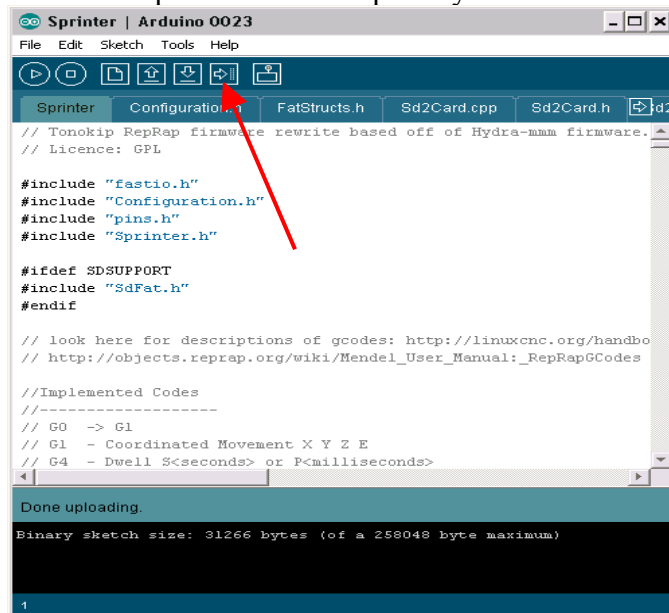
(Note\* This guide does not cover initial firmware configuration.)

1. Power up your machine and connect to your PC via usb.
2. When asked for a driver for your Arduino, choose C:\arduino-0023\drivers  
**NOTE\*\* Makegear ships Arduino's with Sprinter firmware pre-installed. You do not need to continue to upload firmware if you are not making changes or upgrading.**
3. Upload your firmware:
  - a. Start the Arduino IDE from your desktop.
  - b. Find the COM port your Arduino is using
    - i. Right click on "My Computer", click "Manage", and click Device Manager.
    - ii. In the right pane, under Ports (COM & LPT), locate the Arduino
    - iii. Take note of what Port it is using



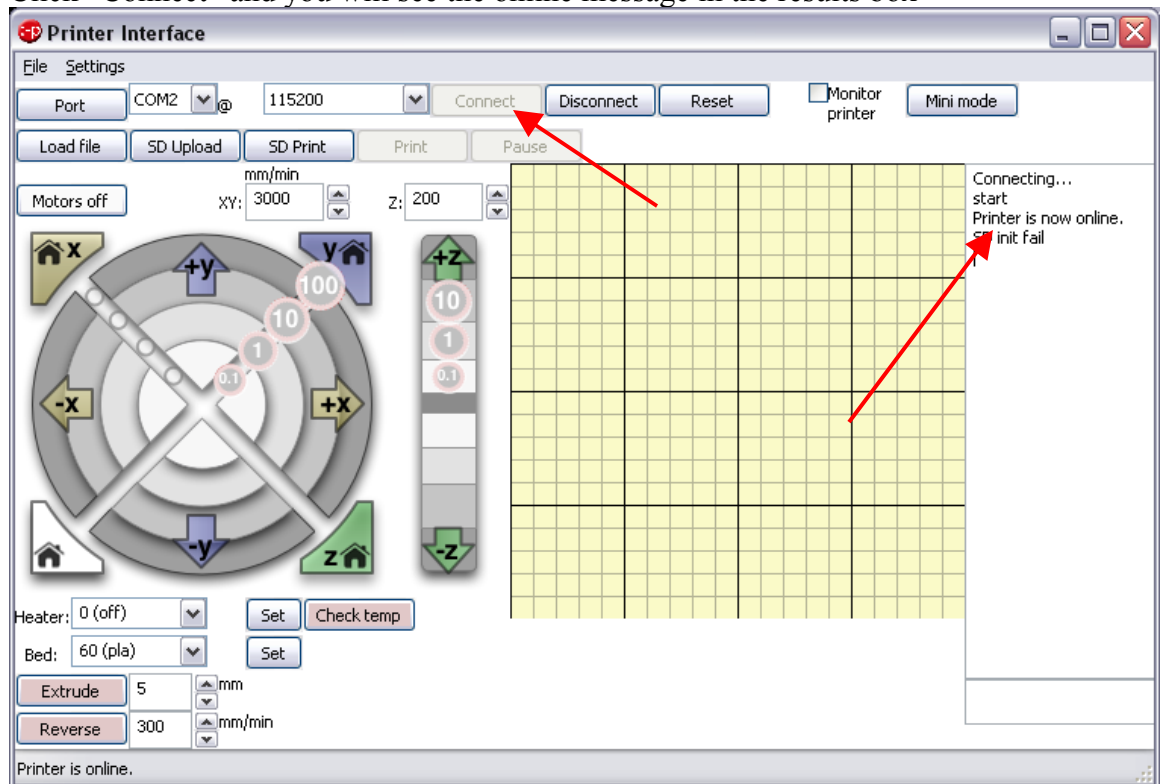
- c. In the Arduino IDE, select the Tools menu, Board, then choose Arduino 2560.
- d. Select the tools menu, Serial Port, and select the port # listed above.
- e. Extract the firmware zip file into a folder. In the Arduino IDE, click File/Open and select the "Sprinter.h" file in the firmware folder.
- f. **FOR RAMPS <1.2 USERS:** Edit configuration.h file and change "#define MOTHERBOARD 3" to "#define MOTHERBOARD 3" and save. Also, D10 should be connected to your hot end heater, D09 is reserved for an optional fan, and D08 should be connected to your heated bed. These values are different previous versions of the firmware.

- g. Click the Upload button to upload your new firmware

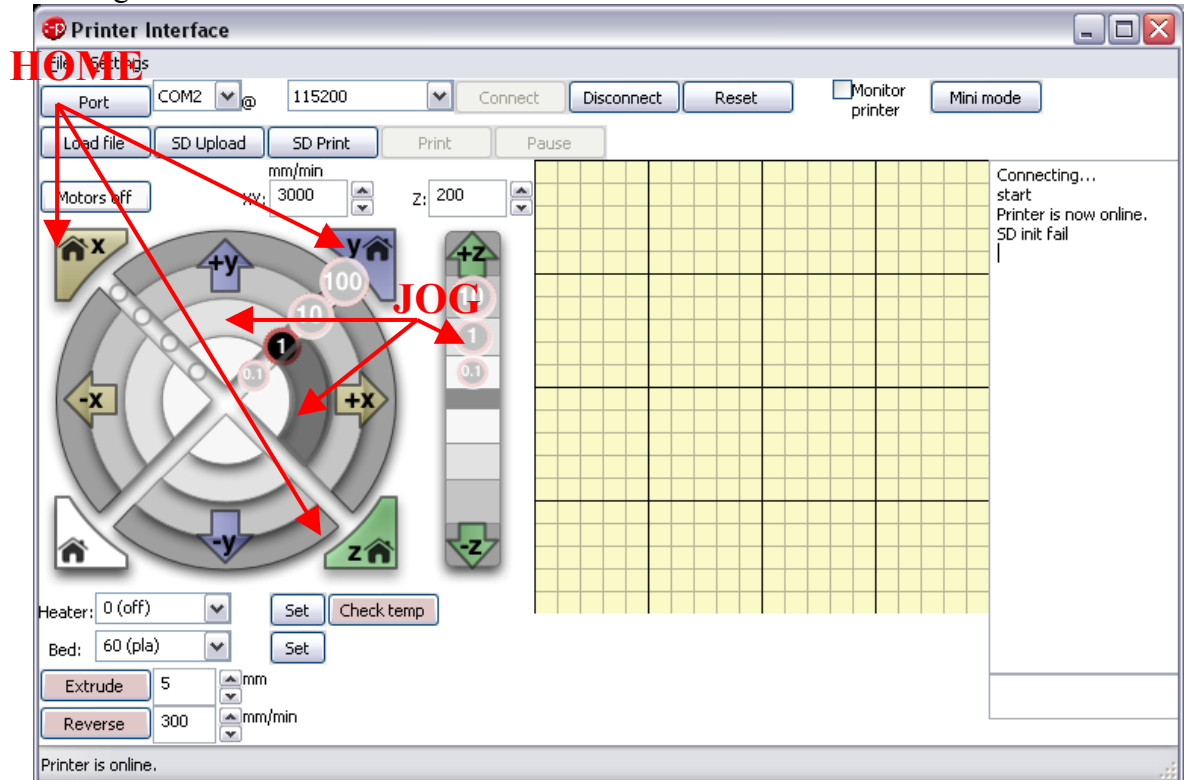


## Pronterface Configuration

1. Open Pronterface from your desktop.
2. Set the correct port you took note of earlier
3. Select 115200 for baud rate
4. Click "Connect" and you will see the online message in the results box



- The default settings in “Heater” and “Bed” have been tested and are appropriate for an unmodified Makergear Prusa build with PLA and ABS, respectively. Select the appropriate setting and click “set”. Wait for the temps to reach their set values (check by clicking “Check Temp”)
- For the movement testing, keep your hand on the power/usb connection so you can disconnect the printer quickly if something does not do what it is supposed to do. This is a good idea with ANY calibration steps.**
- Test movement in X, Y, and Z by clicking on the 1x, 1y, and 1z jog ring, as noted in image.



- Test the “Home buttons” as noted in the above image.  
**Mosaic only ships with the Z axis endstop so only Z homing is in place. You'll need to install the X and Y endstops to home the X and Y axes.**

If an axis moves in the wrong direction, power down the machine and flip the motor connector where it plugs into RAMPS.

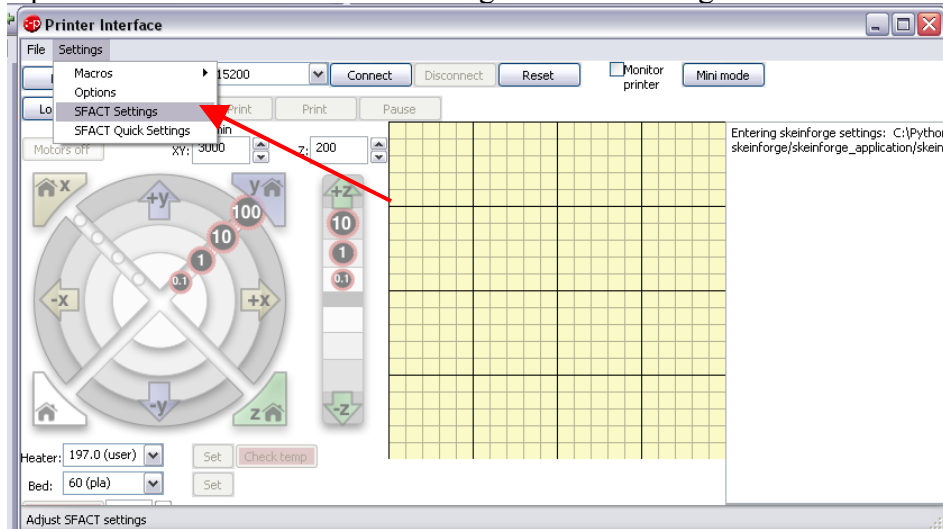
- This concludes the basic setup and testing of Pronterface.

## SFACT Configuration

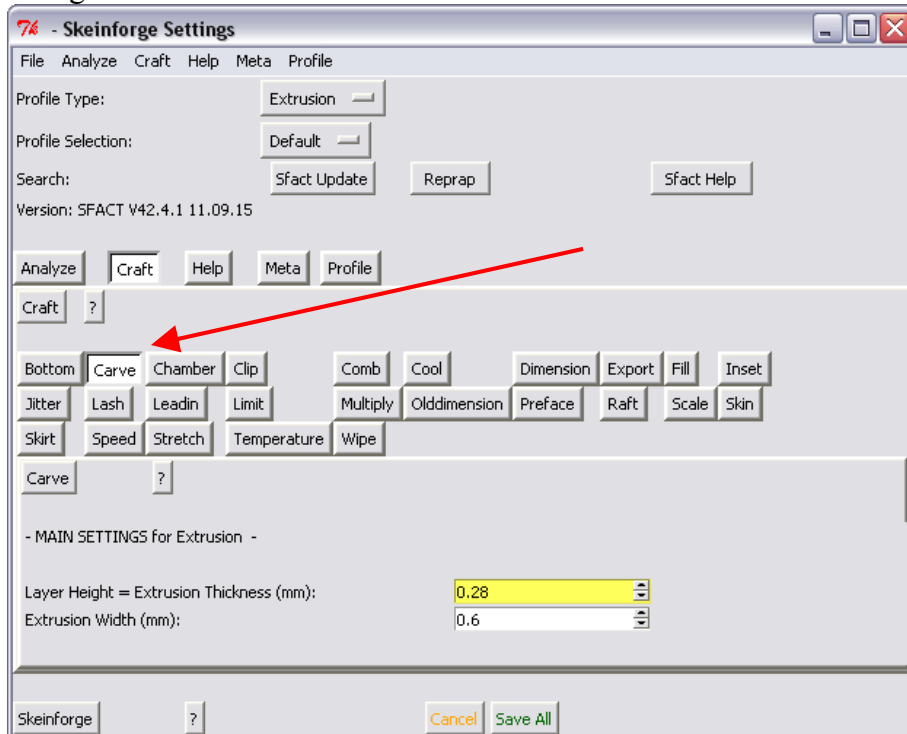
This section is in place to give you a “starting point” and to show you the basics of tuning SFACT. The settings provided will get you on your way to a good print, but further tuning is required on a trial and error basis.

**These settings are made on a few assumptions:** Your firmware is correct, you have a full MakerGear Prusa with heated bed, you are using 3mm filament, and you have a 0.35mm nozzle.

1. Open Pronterface and click on Settings/SFACT Settings



2. Click the “Carve” button and set the “Layer Height” to 0.28. The layer height setting increases or decreases the desired layer height. For example, if your nozzle is 0.35mm, 0.28 seems to give a perfect layer height, squishing the filament just enough.





3. Click the “Chamber” button. Check “Activate Chamber” and set the heated bed to 60 (110 for ABS). We also recommend checking the two boxes below (Turn print bed and extruder heater off at shut down.). When your print is finished, you don’t want hot ooze dripping all over your new bottle opener or kitten statue, do you?
4. Click on the “Dimension” button. The filament diameter is set to 2.8 by default (for 3mm filament) but can be set to 1.65 for 1.75mm filament.
5. Click the “Fill” button. Set the infill pattern to “Grid Hexagonal”. There are exceptions, but we’ve found that this type of infill gives the best all around results.
6. Click the “Inset” button and set your Nozzle Diameter to “0.35” (or whatever size your nozzle is)
7. (Optional) Click the “Raft” button. If you wish to add a “Raft” to your object, change the Base Layers to 1 and the Interface Layers to 1. This step is completely optional; we find that most prints do not require rafts. This is also the area where you add support material to overhangs too great to be printed alone. Change the “Where to add support” option to suit your needs.
8. Click the “Speed” button to set your feed rate. Set your feed rate (mm/s) to 40. This can be set higher once you are confident in your printers tuning. 40mm/s is a good starting point and has been proven to work well with you printer.
9. (Optional) Click the “Temperature” button. Check “Activate Temperature” to add temperature information to your output. This is for fine tuning and allows you to change your extruder’s temperature in various scenarios on the fly.
10. Click “Save All” at the bottom to save your settings and return to Pronterface.

## Z Axis Basic Setup

This is one of the most important settings for your printer and we believe it is worth mentioning in our quick start guide. 0.1mm can make a big difference in your print. We found simply using a piece of printer paper gives us the correct spacing. This guide assumes your print bed is level and even in all four corners.

1. Be sure there is no filament stuck to the bottom of your nozzle
2. Adjust your Z axis end stop so that you are just able to slide a piece of printer paper under it with little resistance. By “just slip” we mean barely just slip.
3. Be careful that you aren’t actually pushing down on your spring loaded bed when you are trying to move the paper, which will skew your results.
4. Now move your nozzle using the Pronterface 10mm jog button on X and Y. Your paper should not bind. If the paper binds at one point and not another, adjust your bed height to make it even.
5. Your end stop should now be setting your nozzle just about 0.1mm above the print bed and you are ready to print!

**For the latest updates check the [MakerGear Google Group](#). You can find firmware, skeinforge and slic3r settings for our Mosaic and Prusa 3D printers.**