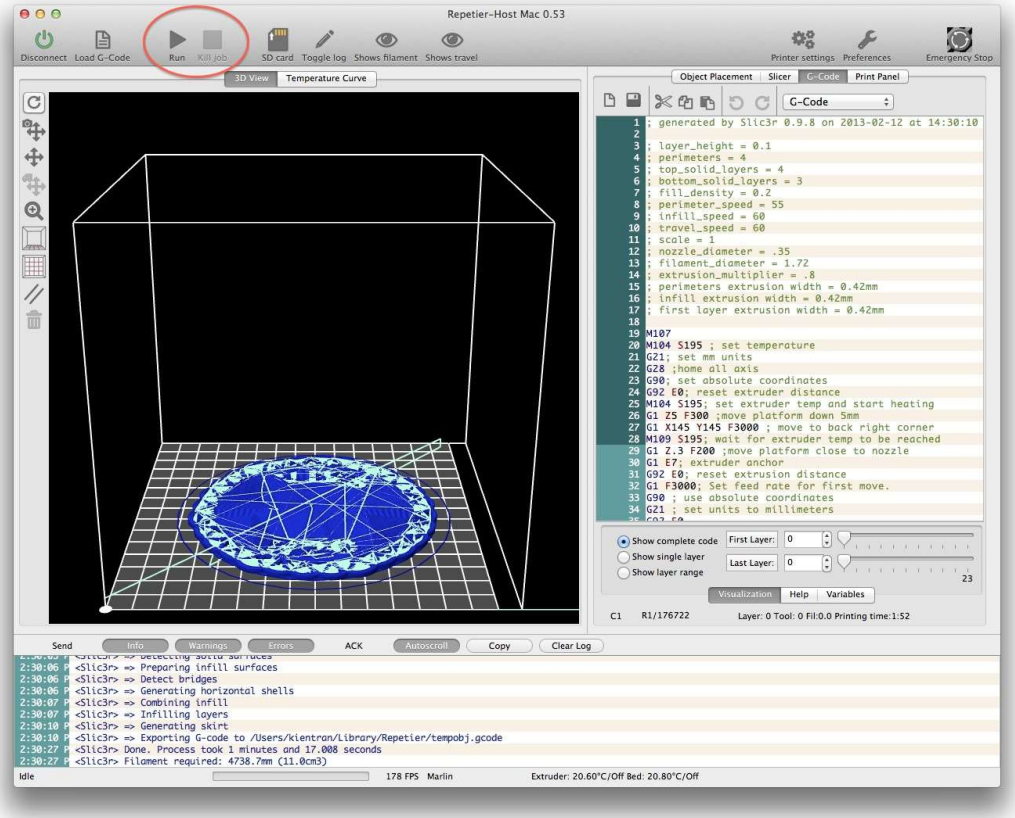

Solidoodle Setup and Tutorial for Mac

Recent Posts

Best CBD Oil Brands to Buy

Go Mic Mobile Wireless System for Mobile Filmmakers

A RAY OF LIGHT

Keep Devices Charged Anywhere in the World with the Twist Plus World Charging Station

Final Leaked iPhone 7 Details Before The Big Event

Recent Comments

Ricardo Garza on Streambox Announces 360 Live Video Streaming to YouTube

Sunbya on Review: Blue Lola Headphones bring unparalleled quality

Users

February 13, 2013 by Kien Tran

In this post, we'll show you step by step how to download, load, and print an object on the Solidoodle 3D printer using your Mac. A future post will deal with operation for Windows based systems. If you haven't read our review of the Solidoodle 2, be sure to check that out.

The Solidoodle is a great 3D printer available at a very affordable price point. Though, like the regular printers of the 1970s and 1980s, setting it up and sending it print files is not for the faint of heart. Following these directions though, you can be underway and quickly printing all kinds of great knickknacks.

First Time Software Installation and Configuration

Download and install the Repetier-Host software.

- Choose the `"Printer Settings"` button on the upper toolbar.
- In Printer settings, the connection tab defines how the program talks to the printer. Use the following settings.
    - Port: usbserial-A######## (The printer will be on some usbserial

Jay Anderson on CES2016 – Autel Robotics X-Star Quadcopters

Scott Ellis on CES2016 – The Parrot DISCO – The Worlds First Fixed Wing Consumer Drone

Ricardo Garza on CES2016 – The Parrot DISCO – The Worlds First Fixed Wing Consumer Drone

Tags

3D Amazon Apple apps audio bluetooth business CE Line Shows 2013 CES computers fitness gadgets games gaming geek geek beat GeekBeat geek house Google health how-to ios john p. kickstarter Microsoft mobile Mobility Music nabshow new building Other photography Reviews robots Samsung social media speakers storage tech

port)

- Baud Rate: 250000
- Stop Bits: 1
- Parity: None
- Transfer Protocol: Auto
- Receive Cache Size: 63
- Ping Pong Communication : Unchecked

- The `Dimension Tab` defines the workable area of the printer. Follow the table below for the appropriate settings for your device

table below for the appropriate settings for your device.

	Soliddoodle 2	Solidoodle 3
X-Min	0	0
Y-Min	0	0
X-Max	159	205
Y-Max	150	200

Print Area Width	159	205
Print Area Depth	150	200
Print Area Height	150	200
Printer Type	Classic Printer with Dump Area	Classic Printer with Dump Area
Dump Area Left	150	200
Dump Area Front	0	0
Dump Area Width	9	9
Dump Area Depth	150	200
Bed Left	0	0
Bed Front	0	0
Home X	Max	Max
Home Y	Max	Max
Home Z	Min	Min

- The `Behavior Tab` controls the printer behavior while printing. These settings can change based on user preferences.

Travel Feed Time	4800
Z-Axis Feed Rate	100
Default Extruder Temperature	195 Celsius
Default Heated Bed Temperature	95 Celsius
Check Extruder and Bed Temperature	1 second
Number of Extruders	1

Don’t Log temperature Requests (M105)	Unchecked
Dump Area Position	X=145, Y=145, Z=15
Go to dispose after job/kill	Checked
Disable Extruder after job/kill	Checked
Disable Heated Bed after job	Unchecked
Disable motors after job/kill	Checked
Add to comp. printer time	8

Some Notes about the Behavior Tab

- You must set “Check Extruder and Bed Temperature” or the printer will not print.
- “Disable Extruder after job/kill” will keep the nozzle from overheating and burning out once a job is done.
- “Disable Heatbed after job/kill” will turn off the bed after a job, but if you are printing several jobs, leaving the bed heated will save time on reheating the bed for the next job. Be sure to check this if you anticipate not being around when the job is complete to prevent the bed from burning out.

Download the Slicer profile files and extract them.

- Back in the main application, select the Slicer tab, and click the configure button. This will load the Slicer engine.
- Go to the **File Menu -> Load Config** and select the “Solidoodle2 .1mm.ini” profile file you extracted.
- Go to each tab and make sure “Solidoodle2 .1mm.ini” is selected in the drop down and click the blue save icon. This will save the profile back to Repiter so it will slice your files without asking for a profile every time.
- You can now close the Slicer window and are ready to print

Downloading, Loading, and Printing Model Files

Repetier uses the industry standard **STL 3D format model file** to print out objects. The fastest way to get started is to find some interesting things to print via Thingiverse and download the STL format files. You can also make them yourself using Google Sketch up or Autodesk 123D.

On the main application, go to the "`Object Placement`" tab, click on "`Add STL File`", and select the file you just downloaded or created.

Next you need to "slice" the STL into a G-code format that Solidoodle uses to print a file. Click "`Slice with Slic3r`", and allow the application to run though it's sequence. When it is complete, the `G-code tab` will appear with the printing

instructions that Repetier will send to the printer.

Click on “Connect” to have Repetier connect to the printer.

If there were no warnings or errors, you are now ready to send the file to the printer. Simply click “Run” in the main toolbar and allow the program to run. The printer will take some time to warm up ahead of time, especially the printing bed.

If there is an error in printing or other issue during the print, be sure to

select “`Kill Job`” to cancel the job, or if there is a serious issue, select “`Emergency Stop`” which will immediately shut off all motors and heated elements.

Extra Tips

- Before you load and slice the STL file, go to the Print Panel tab and manually start the heated bed. It takes a significant amount of time to warm up to the requisite 95 degrees. As some files take a long time to slice, it is a good opportunity to warm the bed. A hot bed is critical in allowing the filament to stick and create a base for your print job.

Otherwise, the print object will slide around as the nozzle moves.

- You can rescale the object larger or smaller in the Object Placement tab before you slice the file.
- On the G-code tab, you can see an estimate of how long the print will take on the bottom of the section.
- If you need to load/unload the filament, go to the Print Panel and turn on the Extruder to the normal 195 degrees. You can manually extrude or retract the filament from the nozzle via the buttons on the Extruder section.

That's about it! If you have any other questions just drop them in the comments and we'll do our best to help you through. The Solidoodle is a great low cost tool for home use, so don't be scared to get one and give 3D printing a whirl!

Filed Under: News, Tutorial
Tagged With: 3D, Tutorial

About Kien Tran

Based out of of the Dallas-Fort Worth Metroplex, Kien is a long time technology and internet enthusiast. With a background in corporate IT and software development, he spends much of his time freelancing as an IT consultant and

is currently the studio engineer for Livid Lobster. You can reach him via @kientran or via +KienTran