

## Instructions: transferring .gcode files into .x3g files

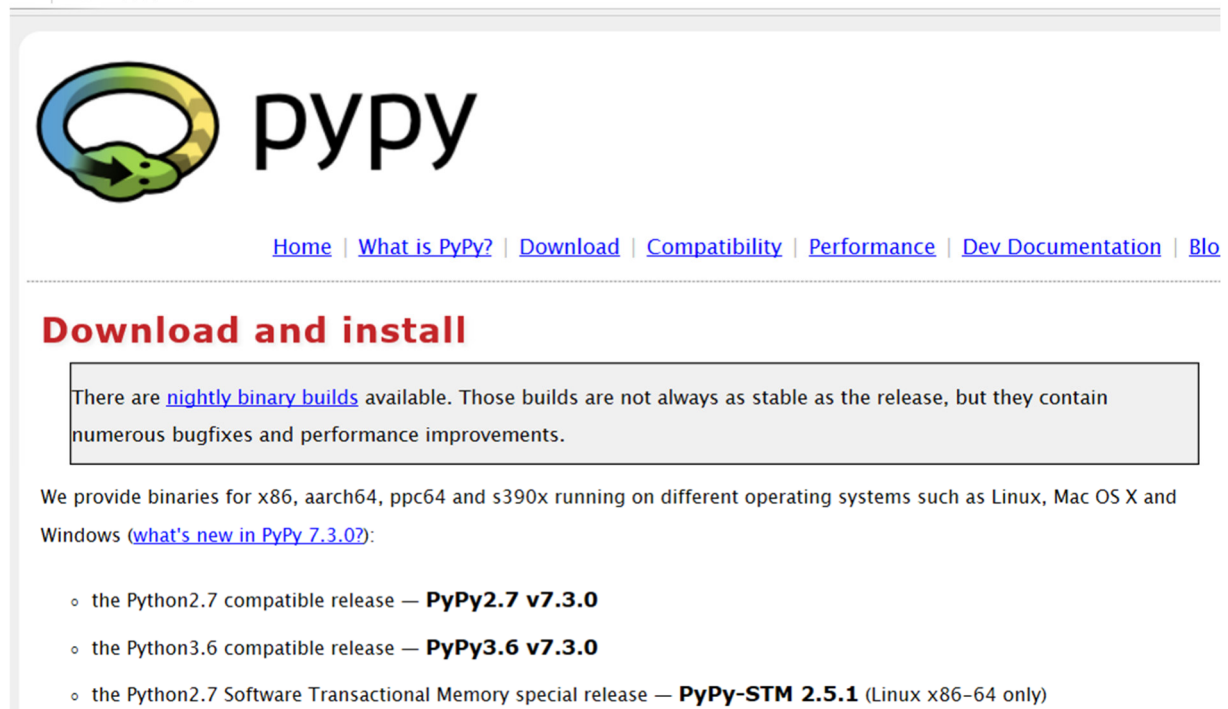
As the Flashforge creator pro printer is running on a makerbot based firmware (sailfish), you are not supposed to print .gcode files. Creating .x3g files that the printer is able to read can be achieved on two ways:

### 1. ReplicatorG

ReplicatorG is a open source 3D printing program you can use to modify your coded files. A free download is available on replicator.org or via google drive. Its recommended to use version replicatorG0040.

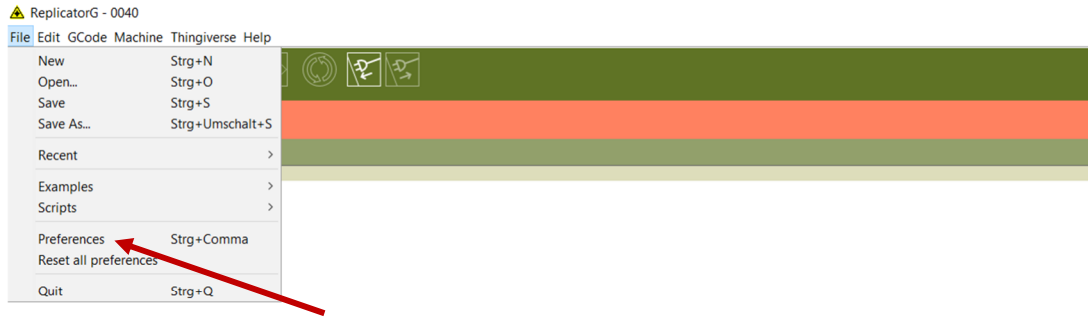
ReplicatorG0040 is based on Python which makes it necessary to download a Python interpreter, called PyPy. Find a free download of version PyPy2.7 on pypy.org.

 <https://pypy.org/download.html>

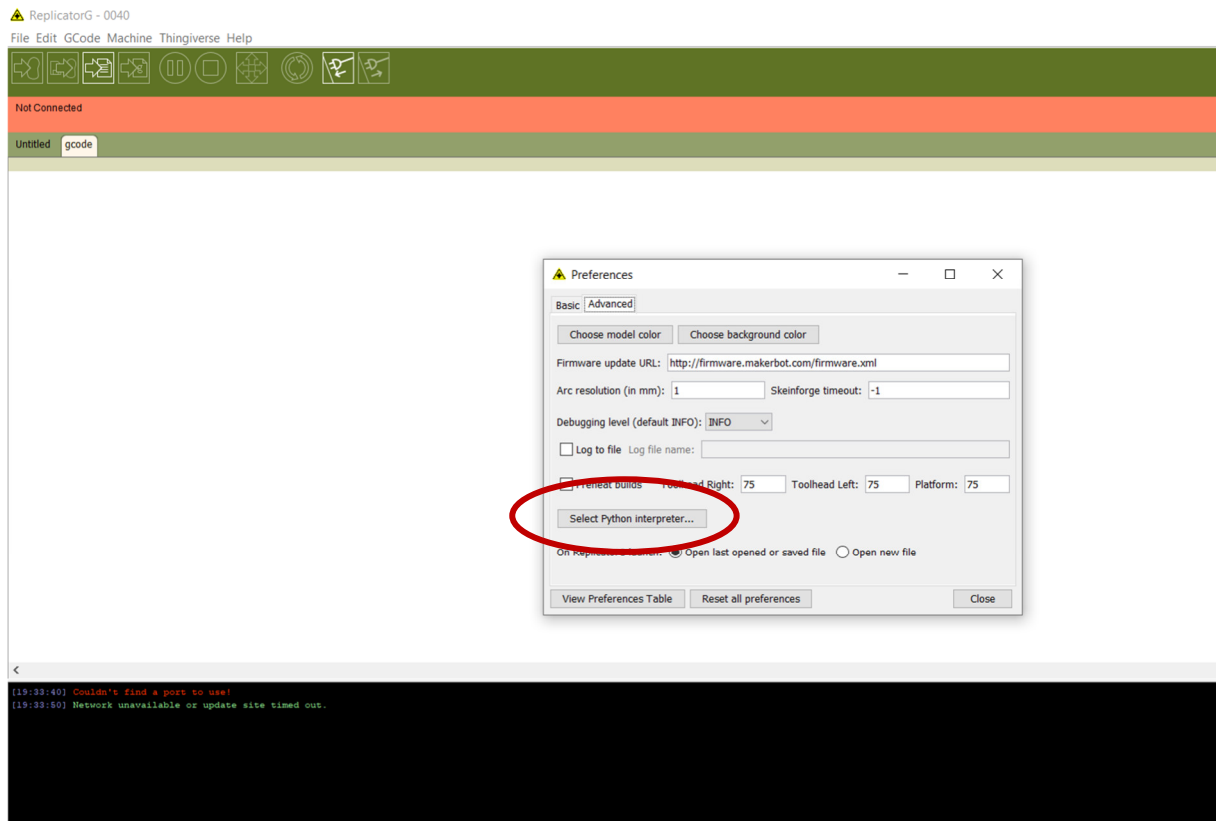


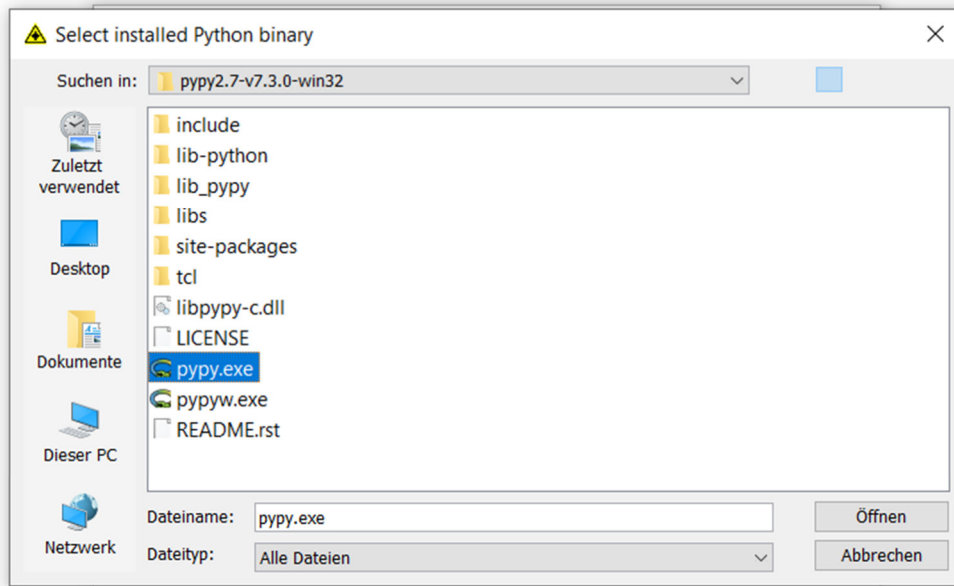
The screenshot shows the PyPy website's download page. At the top left is the PyPy logo, which consists of a green and blue circular arrow with a green arrow pointing right, followed by the text 'pypy' in a bold, lowercase font. Below the logo is a navigation menu with links: [Home](#), [What is PyPy?](#), [Download](#), [Compatibility](#), [Performance](#), [Dev Documentation](#), and [Blo](#). A horizontal dashed line separates the navigation from the main content. The main heading is 'Download and install' in a bold, red font. Below this is a grey-bordered box containing the text: 'There are [nightly binary builds](#) available. Those builds are not always as stable as the release, but they contain numerous bugfixes and performance improvements.' Below the box, the text reads: 'We provide binaries for x86, aarch64, ppc64 and s390x running on different operating systems such as Linux, Mac OS X and Windows ([what's new in PyPy 7.3.0?](#)):' followed by a bulleted list of three release options: 'the Python2.7 compatible release — **PyPy2.7 v7.3.0**', 'the Python3.6 compatible release — **PyPy3.6 v7.3.0**', and 'the Python2.7 Software Transactional Memory special release — **PyPy-STM 2.5.1** (Linux x86-64 only)'. The page has a light grey background and a white border.

Create a folder with the unzipped programs and start the installation of replicatorG. Once replicatorG is opened, you will need to find “Preferences” below the files column.



Switch into the advanced settings and click “select Python interpreter”. You’ll have to choose the place that you have saved **python.exe**. After entering the data path, close the pop up.





replicatorG is now ready to use: First step you are going to open the .gcode file whose format you want to change.

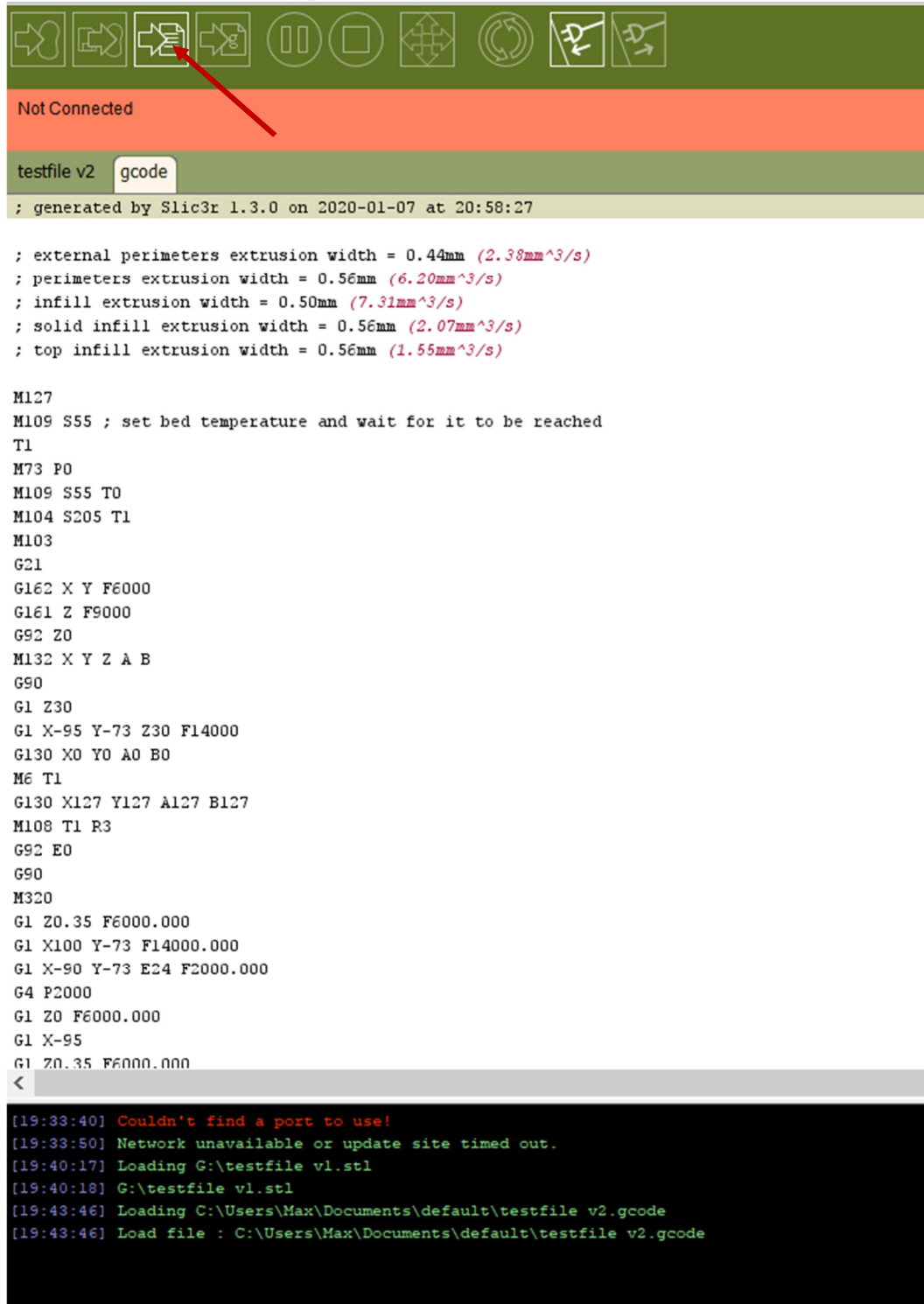


created: Winger Max

ReplicatorG is now showing the final gcode your slicer generated. You can run various operations as for example estimate the print time ("gcode" -> estimate). ReplicatorG is showing its operations in the bar at the very bottom. To create the .x3g file finally, you will have to select the symbol shown below. Choose .x3g as the output file and select your saving location.

⚠ ReplicatorG - 0040

File Edit GCode Machine Thingiverse Help



Not Connected

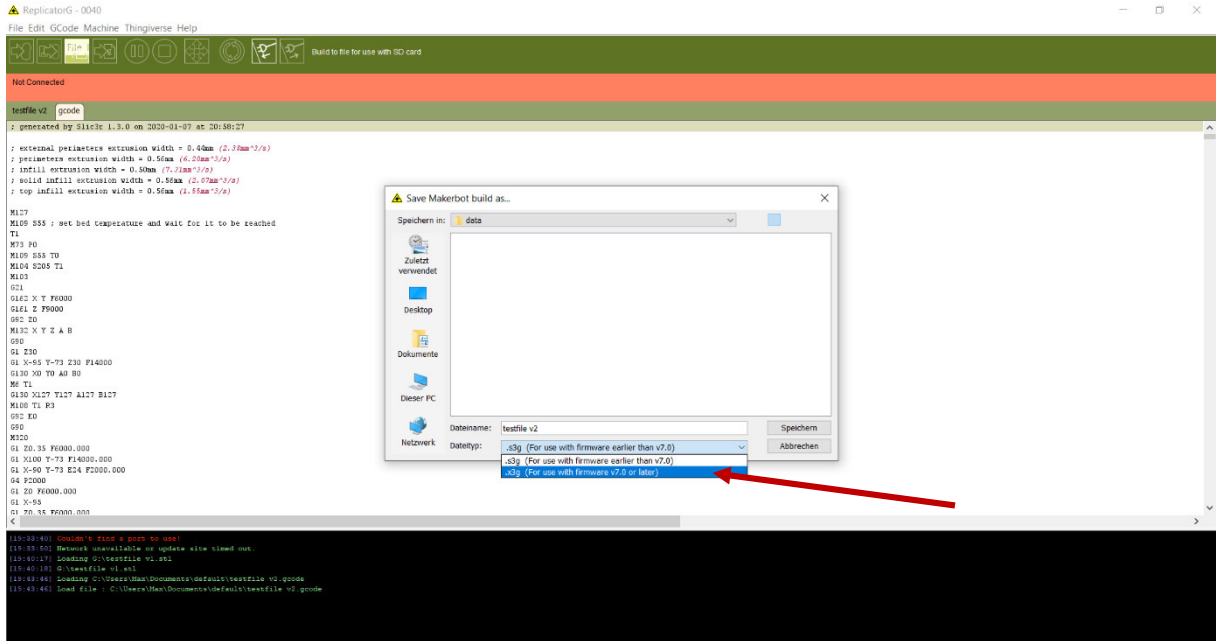
testfile v2 gcode

```
; generated by Slic3r 1.3.0 on 2020-01-07 at 20:58:27

; external perimeters extrusion width = 0.44mm (2.38mm^3/s)
; perimeters extrusion width = 0.56mm (6.20mm^3/s)
; infill extrusion width = 0.50mm (7.31mm^3/s)
; solid infill extrusion width = 0.56mm (2.07mm^3/s)
; top infill extrusion width = 0.56mm (1.55mm^3/s)

M127
M109 S55 ; set bed temperature and wait for it to be reached
T1
M73 P0
M109 S55 T0
M104 S205 T1
M103
G21
G162 X Y F6000
G161 Z F9000
G92 Z0
M132 X Y Z A B
G90
G1 Z30
G1 X-95 Y-73 Z30 F14000
G130 XO YO A0 B0
M6 T1
G130 X127 Y127 A127 B127
M108 T1 R3
G92 E0
G90
M320
G1 Z0.35 F6000.000
G1 X100 Y-73 F14000.000
G1 X-90 Y-73 E24 F2000.000
G4 P2000
G1 Z0 F6000.000
G1 X-95
G1 Z0.35 F6000.000
<
```

```
[19:33:40] Couldn't find a port to use!
[19:33:50] Network unavailable or update site timed out.
[19:40:17] Loading G:\testfile v1.stl
[19:40:18] G:\testfile v1.stl
[19:43:46] Loading C:\Users\Max\Documents\default\testfile v2.gcode
[19:43:46] Load file : C:\Users\Max\Documents\default\testfile v2.gcode
```



## 2. Direct format transfer with XPG

In case you are going to use an open source slicer like **sllic3r** you can directly export files to .x3g by using the program GPX. The following instructions are based on **sllic3r**. FlashForge is not providing custom gcodes to use with slic3r, cura or third-party programs. However, you are free to experiment with your own codes.

GPX.exe can be found on <https://www.thingiverse.com/> searching for “GPX gcode to .x3g converter”. Select the matching .zip folder in the drop down menu of “Thing files”, and install GPX on your computer.

**GPX gcode to x3g converter**  
by whgthomas Apr 29, 2013

Usage: gpx [-p] [-s <MACHINE>] [-c <CONFIG>] [-o] [-i] [-f] [-t]

Switches:

- p override build percentage
- s enable stdin and stdout support for command pipes

MACHINE is the predefined machine name

- c3 = Cupcake Gen3 XT2, M3/6 + Gen4 Extruder
- c4 = Cupcake Gen4 XT2, M3/6 + Gen4 Extruder
- cp4 = Cupcake Polaris 4, M3/6 + Gen4 Extruder
- cp6 = Cupcake Polaris 6, M3/6 + Gen4 Extruder
- t6 = T6 - single extruder
- t7 = T7 - single extruder
- t10 = T10 - single extruder
- r1 = Replicator 1 - single extruder
- r1d = Replicator 1d - single extruder
- r2 = Replicator 2
- r2x = Replicator 2x

CONFIG is the filename of a custom machine definition (.ini)

INPUT is the filename of the gcode file

OUTPUT is the name of the x3g output filename

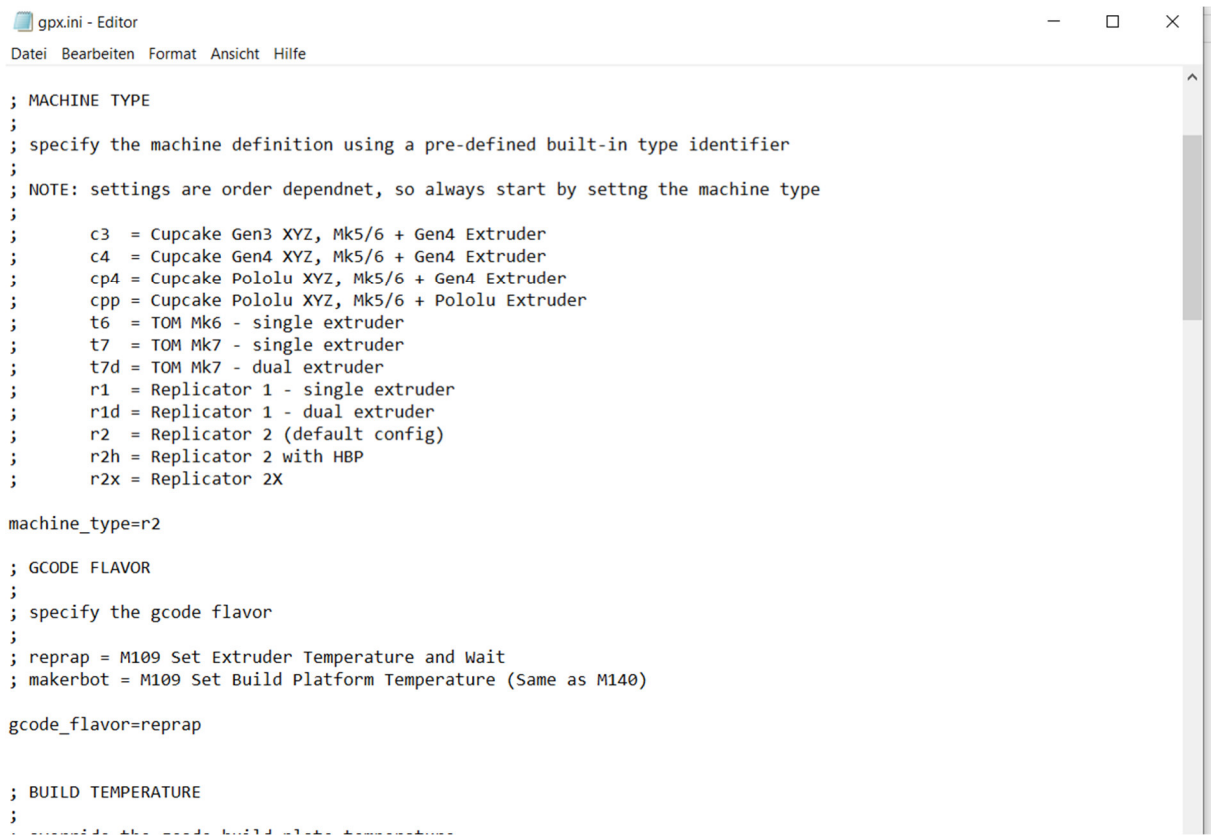
This program is free software; you can redistribute it and/or modify it under the terms of the GNU General Public License as published by the Free Software Foundation; either version 2 of the License, or (at your option) any later version.

Thing Details: 55 Comments, 0 Makes, 0 Collections, 0 Remixes

File Name	Downloads	Size
gpx-win32-1.3.zip	39999	45kb
gpx-osx-1.4.dmg	29195	66kb
gpx-osx-1.5.dmg	31961	66kb

AGREE & DOWNLOAD (162 kb zip file)

Next you are going to modify the configurators script that is listed in the extracted files as “gpx.ini”. Click on “gpx.ini” and select “edit”.



```
; MACHINE TYPE
;
; specify the machine definition using a pre-defined built-in type identifier
;
; NOTE: settings are order dependnet, so always start by setting the machine type
;
; c3 = Cupcake Gen3 XYZ, Mk5/6 + Gen4 Extruder
; c4 = Cupcake Gen4 XYZ, Mk5/6 + Gen4 Extruder
; cp4 = Cupcake Pololu XYZ, Mk5/6 + Gen4 Extruder
; cpp = Cupcake Pololu XYZ, Mk5/6 + Pololu Extruder
; t6 = TOM Mk6 - single extruder
; t7 = TOM Mk7 - single extruder
; t7d = TOM Mk7 - dual extruder
; r1 = Replicator 1 - single extruder
; r1d = Replicator 1 - dual extruder
; r2 = Replicator 2 (default config)
; r2h = Replicator 2 with HBP
; r2x = Replicator 2X

machine_type=r2

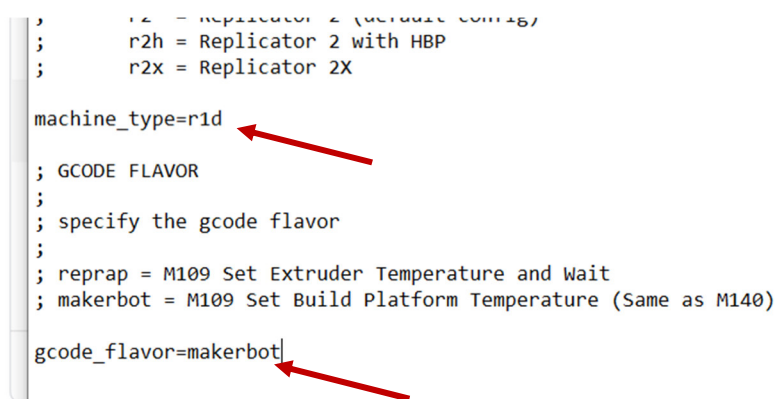
; GCODE FLAVOR
;
; specify the gcode flavor
;
; reprep = M109 Set Extruder Temperature and Wait
; makerbot = M109 Set Build Platform Temperature (Same as M140)

gcode_flavor=reprep

; BUILD TEMPERATURE
;
; .....
```

Hint: GPX is based on replicatorG, so need to make sure Replicator G is ready to run.

First you are going to change the machine type to “r1d”. This sets the program to the Replicator dual printer, which corresponds with the FlashForge Creator Pro. Second step is changing the gcode flavor to “makerbot”. After making those changes you can save the file and GPX is working. Your script should look like the following:

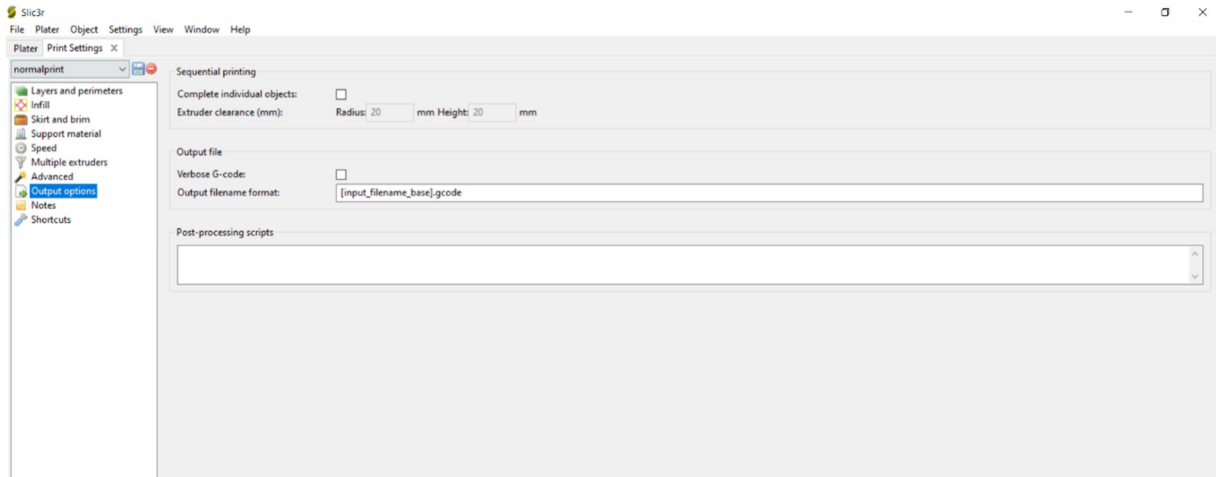


```
; r2 = Replicator 2 (default config)
; r2h = Replicator 2 with HBP
; r2x = Replicator 2X

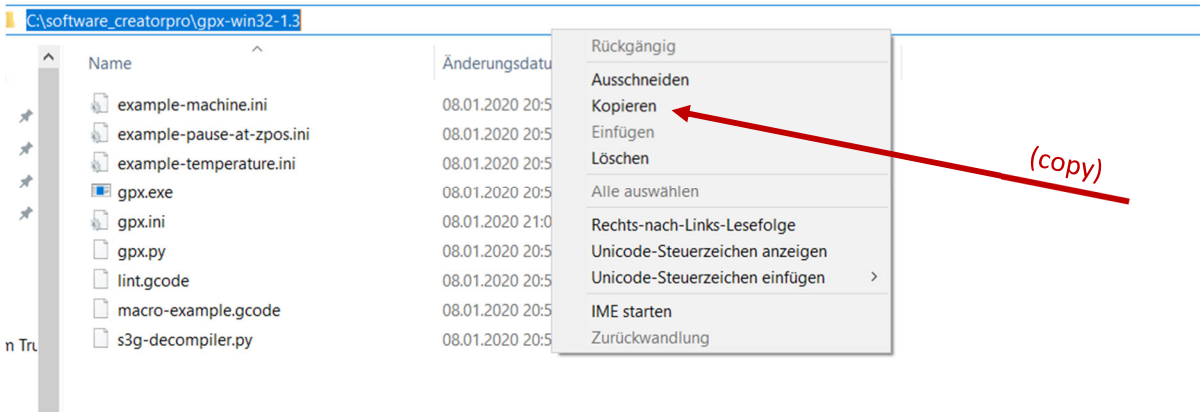
machine_type=r1d
; GCODE FLAVOR
;
; specify the gcode flavor
;
; reprep = M109 Set Extruder Temperature and Wait
; makerbot = M109 Set Build Platform Temperature (Same as M140)

gcode_flavor=makerbot
```

To make slic3r (or other slicers) directly export files to .x3g, you will need to open the print settings.



Below “output options” you will find the possibility to add a “Post-processing script”. This is the file path you have saved GPX.exe. Go to explorer (windows), open the folder that includes GPX.exe and copy the file path to insert the data into the Post-processing script.



Hint: Don't forget to insert the total file path by adding `\gpx.exe`.

Save the settings. Next time you are exporting a gcode file, the file is automatically changed to .x3g format.

