
How to Use ReplicatorG

ECE 480 Application
Note

Annalin Davis

April 3, 2015

Abstract

This application note discusses ReplicatorG and its role in the 3D modeling process. It goes through the entire process using ReplicatorG from loading an STL file to finally outputting code for a 3D printer. It covers editing the model, generating GCode and understanding the options associated with generation, and further editing GCode after generation. It ends with a discussion about this team's work with ReplicatorG and the improvements this team is making to the program.

Keywords

GCode – The term GCode generally refers to the code produced by the program. The code produced is actually composed of GCodes and MCode but includes more GCode than MCode. Specific GCodes are referenced later in Table 1 and describe the movement of the 3D printer's extrusion heads.

MCode – MCodes are similar to GCodes, but they control other aspects of 3D printing such as temperatures and extrusion speeds.

Slicing – Slicing is the method by which the program breaks apart a 3D model. By slicing it into thin layers, it is possible to describe it as many stacks of 2D shapes.

STL – STL stands for STereoLithography, it is a type of file produced by 3D modeling programs. This file type allows the slicer to work efficiently because it describes the model with many small triangles.

Toolpath – Toolpath is the path followed by the extruder head on the 3D printer. The extruder heads are referred to as tools sometimes in the program.

Introduction

ReplicatorG is open source software that turns 3D models into directions for a 3D printer to print that model. It works with machines that run MakerBot, Thing-O-Matic, CupCake and RepRap. It also works with most generic CNC machines. ReplicatorG is capable of slicing, creating GCode and generating code for the 3D printer.

ReplicatorG is needed to slice and generate code as described above. It is what connects the 3D model to the process of actually creating that model in the real-world with a 3D printer. It changes the STL file, comprised of many triangles, into instructions for the 3D printer to follow. ReplicatorG is used because it is simple and fairly intuitive. For example, after generating GCode, ReplicatorG will let you view the GCode in another tab. There you can edit the GCode and then save the edited GCode to your SD card.

ReplicatorG slices the 3D model into small layers. Each layer will be a layer that is printed by the 3D printer. After slicing each layer, it will generate the toolpath that the extruder will need to follow to actually print the entire layer. GCodes are generally responsible for controlling the position and movement of the print bed and extruder heads of the 3D printer. Another thing that the program has to consider when generating code is when the extruder should be extruding (on) and when the extruder should not be extruding (off). These kinds of operations are dealt with through MCodes.

The main objective of this Application Note is to demonstrate how to import a model, generate GCode, edit key settings and export to the printer.

Importing Files

Before using ReplicatorG, an STL file must be produced. The STL file will communicate the model's dimensions to the slicer program in a way that is simple for the slicer program to interpret. STL files can be produced by most 3D modeling software. There are also many STL files available for download on the web. One site that has a large collection of them is Thingiverse. Once an STL file has been produced, it can be opened with ReplicatorG. Click File->Open to bring up the file selection dialog box, shown in Figure 1. From there, navigate to where the STL file is saved, select it, and click the Open button.

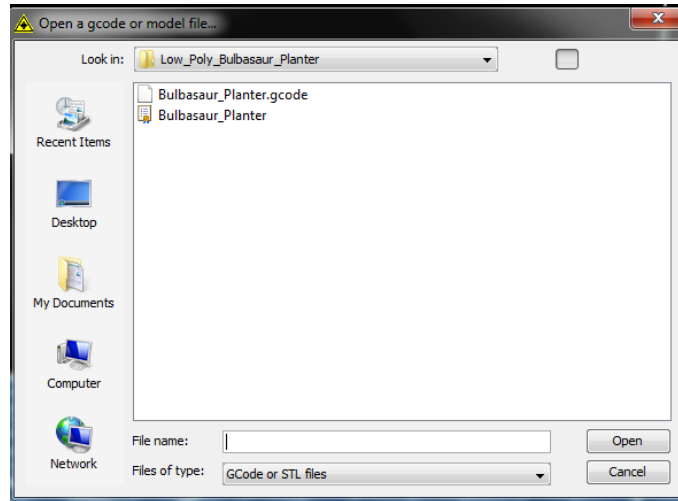


Figure 1: A Dialog Box Prompting a User to Open an STL File.

After loading the STL file, ReplicatorG will display the model centered on the printing platform. At this stage, the model can be checked for errors and a few manipulations can be performed. The model is in View mode by default. This means that clicking and dragging will rotate the model and scrolling the mouse wheel will zoom in or out on the model. In this mode, there are also buttons near the top right of the screen to snap the camera to the XY, YZ and XZ planes (Figure 2).

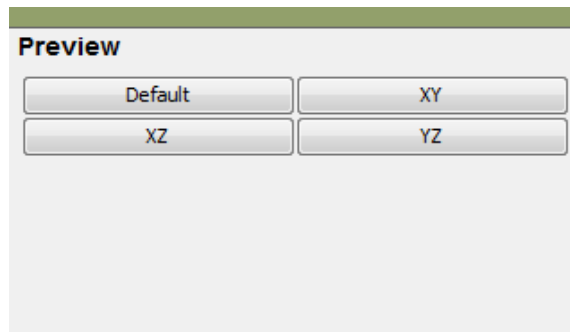


Figure 2: Camera Angle Default Buttons.

Editing the Model

There are a few ways that the model can be edited from ReplicatorG. They all have to do with how the model eventually prints on the 3D printer's print bed. The first button under View is Move. This allows the model to be printed anywhere on the print bed as long as it will fit. Rotate allows the model to be rotated to be printed from any orientation. Mirror will mirror the model around an axis, and Scale allows the user to print a bigger or smaller version of the model. These operations can be performed by manipulating the model with the mouse, or they can be performed more precisely by entering values in the input boxes that appear when the respective mode is selected. Figure 3 shows the input options for

the Move mode. There is a button to center the model, and there are inputs to show precisely where the model is located on the X, Y, or Z axis. Generally, the model should not be moved in the Z direction. By default the model is loaded to be printed directly on the printer bed. Moving the model above or into the printing bed will cause inaccurate and undesirable prints.

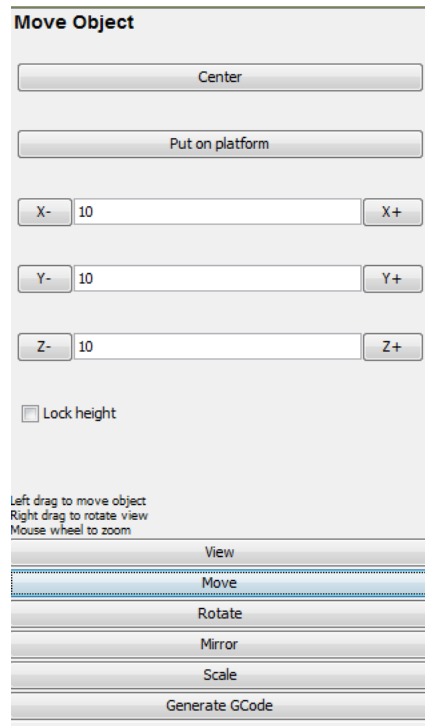


Figure 3: The Input Options for Move Mode.

Generation of GCode

Underneath the Scale button is the Generate GCode button. Clicking this button will bring up a dialog box with editing options to further influence how the object is printed (Figure 4).

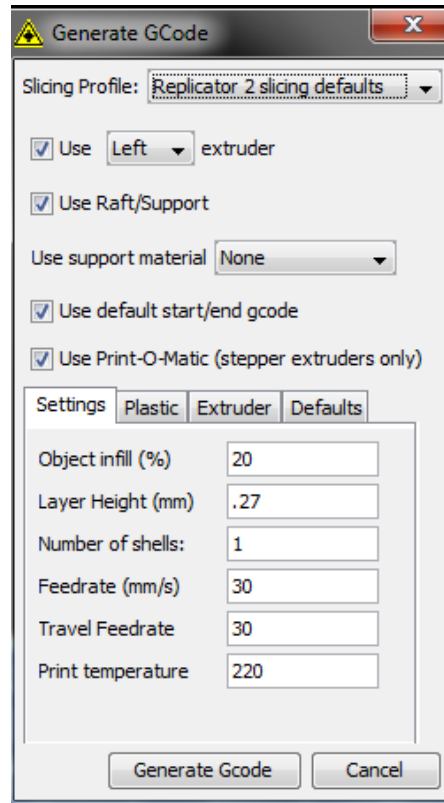


Figure 4: The Option Dialog Box for Generating GCode.

Figure 4 displays many of the settings the user can set before generating the GCode that will influence the generation of the GCode. The most important ones will be further discussed. There is an option to select the left or right extruder for systems that have been set up to acknowledge dual-head extruders. The 'Use Raft/Support' option will add a raft to the instructions for printing. The raft is a thin grid of plastic that is printed prior to printing the 3D part. It helps the printed part stick to the print bed, which helps the part print smoothly. It can be easily snapped off after printing is completed and the part has cooled down. The 'Object infill (%)' tells the program how much of the interior of the 3D model to fill in. The default is 20%, this will cause the outside of the 3D model to be printed solidly, but the inside will look like a hexagonal grid, instead of being fully filled in. To create sturdier, smoother models, the object infill percentage can be increased. 'Layer Height' is related to the thickness of the plastic that is being printed, 0.27 mm is the default for 1.75 mm plastic filament. 'Feedrate' and 'Travel Feedrate' determine how much plastic, in millimeters, is extruded per second. The 'Print Temperature' is the temperature setting for the extruder heads. This number is in Celsius and must be high enough to melt the plastic.

Further Editing of the GCode

There are a few things that can't be edited with the menu, such as the temperature of the heated bed. If there is a feature that is not handled by the menu, ReplicatorG gives the user access to the GCode so it can be edited further after generation. The GCode will be in the GCode tab, as can be seen below.

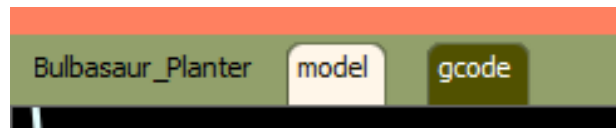


Figure 5: The Tab System of ReplicatorG.

The GCode that ReplicatorG generates is comprised mostly of G Codes and M Codes. Figures 6 and 7 are common G Codes and M Codes and their uses. Using the example of changing the temperature of the heated bed for the printer, the code that needs to be changed is M109. After M109 there is an S110, which means that the heated bed is currently set to heat to 110 degrees Celsius. This is the default the ReplicatorG assumes when working with PLA plastic. If a 60 degree Celsius temperature was desired, then S110 would need to be changed to S60. The third part of an M109 command is the tool it is associated with. This should stay the same so as not to affect the rest of the code.

- G0 - Rapid Motion
- G1 - Coordinated Motion
- G2 - Arc - Clockwise
- G3 - Arc - Counter Clockwise
- G4 - Dwell
- G10 - Create Coordinate System Offset from the Absolute one
- G17 - Select XY plane (default)
- G18 - Select XZ plane (not implemented)
- G19 - Select YX plane (not implemented)
- G20 - Inches as units
- G21 - Millimeters as units
- G28 - Home given Axes to maximum
- G30 - Go Home via Intermediate Point (not implemented)
- G31 - Single probe (not implemented)
- G32 - Probe area (not implemented)
- G53 - Set absolute coordinate system
- G54-G59 - Use coordinate system from G10 P0-5
- G90 - Absolute Positioning
- G91 - Relative Positioning
- G92 - Define current position on axes
- G94 - Feed rate mode (not implemented)
- G97 - Spindle speed rate
- G161 - Home negative
- G162 - Home positive

Figure 6: Useful GCodes from Replicat's Website.

- M101 Extruder on, fwd
- M102 Extruder on, reverse
- M103 Extruder off
- M104 Snn set temperature in degrees Celsius
- M105 get extruder temperature
- M106 turn fan on
- M107 turn fan off
- M108 Set Extruder's Max Speed (Rnnn = RPM, Pnnn = PWM)
- M109 Snnn set build platform temperature in degrees Celsius
- M110 Snnn set chamber temperature in degrees Celsius
- M120, M121, M122 Snnn set the PID gain for the temperature regulator (not currently supported by ReplicatorG)
- M123, M124 Snnn set iMax and iMin windup guard for the PID controller (not currently supported by ReplicatorG)
- M128 get position
- M129 get range (not currently supported by ReplicatorG)
- M130 set range (not currently supported by ReplicatorG)
- M200 reset driver
- M202 clear buffer (not currently supported by ReplicatorG)
- M300 Snnn set servo 1 position
- M301 Snnn set servo 2 position

Figure 7: Useful MCodes from Replicat's Website.

Exportation Methods

There are two main methods to send the finished code to the 3D printer. The code can either be saved to an SD card to be plugged into the printer, or the code can be directly sent to the printer if the printer is plugged into the computer running ReplicatorG. The two buttons to do this are located at the top of the window. In Figure 8, the button to save to an SD card is on the left and the button to send it straight to the printer is on the right.

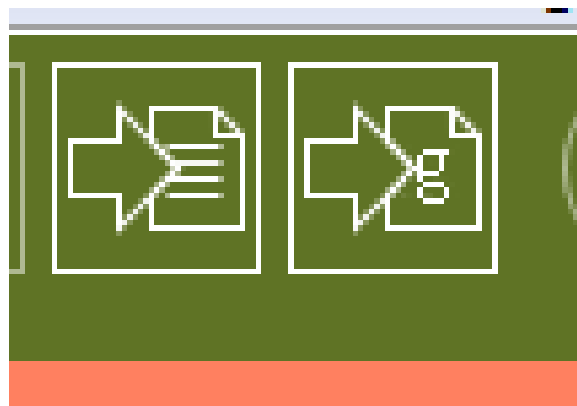


Figure 8: Buttons for exporting the GCode.

Known Issues

When using ReplicatorG, there is no way to set the bed temperature of the 3D printer. It loads defaults and automatically uses those in the GCode. While this can be changed directly in the GCode tab, it is not convenient or easy for the user. ReplicatorG is not fully compatible with all 3D printers.

Conclusion

ReplicatorG is useful for turning STL files into code for a 3D printer to run. It is simple and efficient to use. There are other programs that do similar things to what ReplicatorG does. Some of them are very hardware specific, the MakerBot software runs best with MakerBot Hardware. Other programs include Repetier and Slic3r.

Currently, the team is creating a program that will run concurrently with ReplicatorG and allow the user to have more control over the generated GCode without having to manually edit the GCode themselves. The features that are being added are: temperature controls for both extruders, full temperature control for the print bed and a multipass feature. Allowing the user to turn off the temperature of an extruder is useful for printing materials that may not need a temperature setting. It increases the flexibility of the program to work with different types of printers. It also allows the user to set the temperature of the bed without having to go into the GCode and edit the code itself. The multipass feature allows the user to have the left or right extruder print a certain number of layers for every one layer that the other head prints. This will be useful for a printer that can print two different materials, if one material prints much thinner layers than the other, this will help compensate for that and ensure an even print.

Sources:

Getting Started with the FlashForge Creator X: Replicator G on Windows (3D Universe). (n.d.). Retrieved March 31, 2015, from https://www.youtube.com/watch?v=IPbMDegy4_4

Know Your Slicing and Control Software for 3D Printers - Make:. (n.d.). Retrieved April 3, 2015, from <http://makezine.com/magazine/guide-to-3d-printing-2014/know-your-slicing-and-control-software-for-3d-printers/>

MakerBot Thingiverse. (n.d.). Retrieved March 30, 2015, from <http://www.thingiverse.com/>

Makerbot/ReplicatorG. (n.d.). Retrieved April 2, 2015, from <https://github.com/makerbot/ReplicatorG>

ReplicatorG User Manual. (n.d.). Retrieved April 2, 2015, from http://wiki.ultimaker.com/ReplicatorG_User_Manual

ReplicatorG. (n.d.). Retrieved April 2, 2015, from <http://replicat.org/>