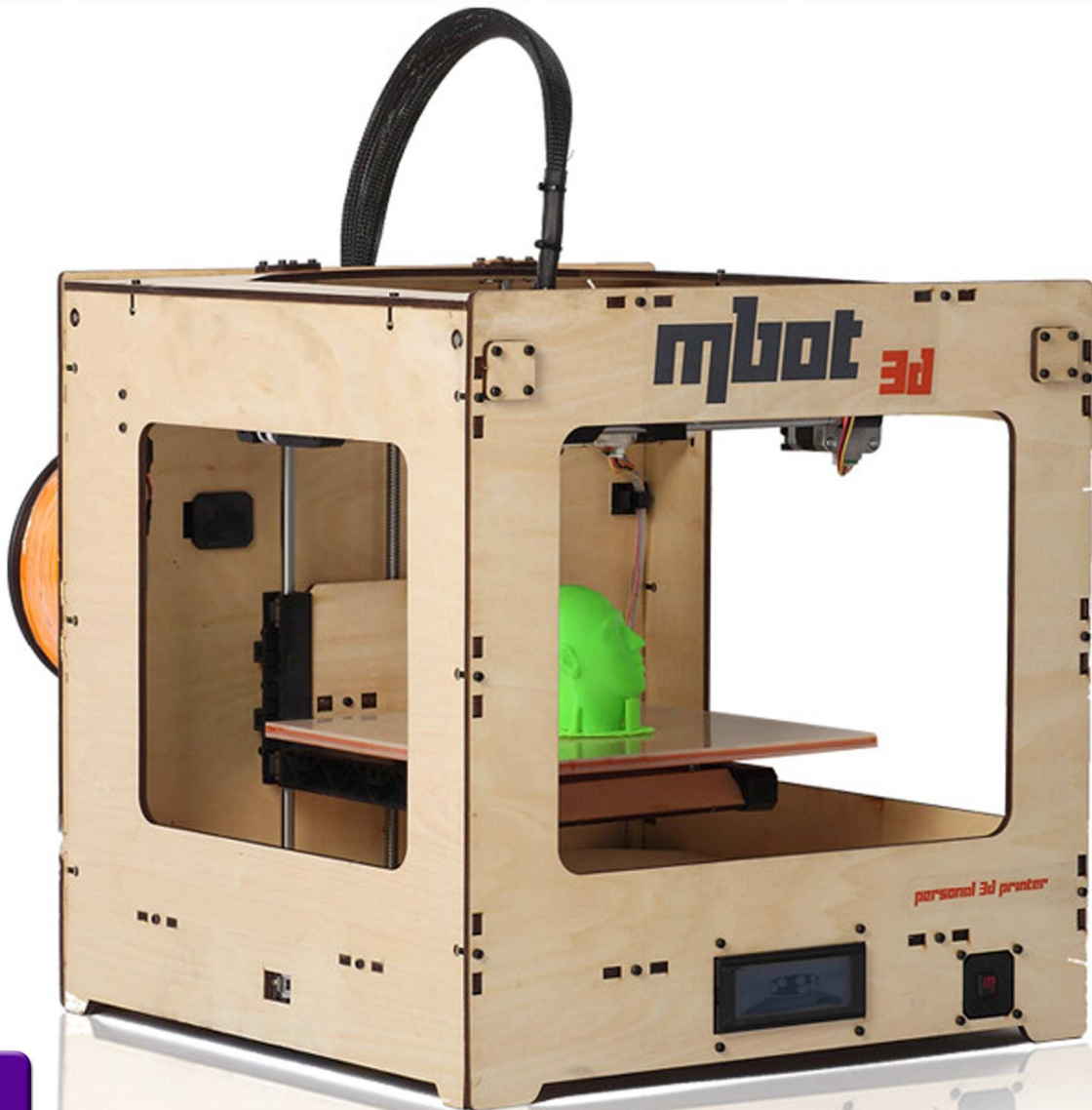




CUBE : Desktop 3D Printer PLY or PVC Models User Manual



Distributed in Australia by :

BilbyCNC

www.BilbyCNC.com.au

Australian freecall **1800 245 297**

International : **+61 2 4403 0352**

Contents

1 Installing software

1.1 Installing Python.....	3
1.2 Installing ReplicatorG.....	4

2 Pre-print checks

2.1 Power up.....	8
2.2 Connect to computer.....	9
2.3 Setting up software.....	10

3 Setting up model data

3.1 Importing model file into ReplicatorG.....	13
3.2 Move, scale, rotate and estimated build time.....	13
3.3 Generating GCode (Slicing)	15

4 Test printing *(If plastic stop coming out the nozzle during printing, please read this chapter)*

4.1 Using the ReplicatorG Control Panel	18
---	----

5 Printing

5.1 Printing from computer.....	20
5.2 Printing from SD card.....	22
5.3 Finished Prints.....	24
5.4 Other Important Printing Information and tips <i>READ BEFORE PRINTING ANYTHING!!!!</i>	25

6 Two-color (Dual) printing..... 28

This manual was last updated 24 July 2013

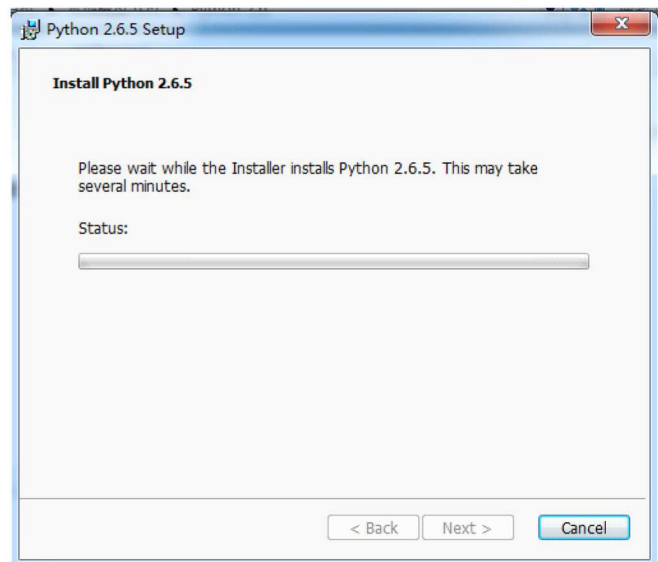
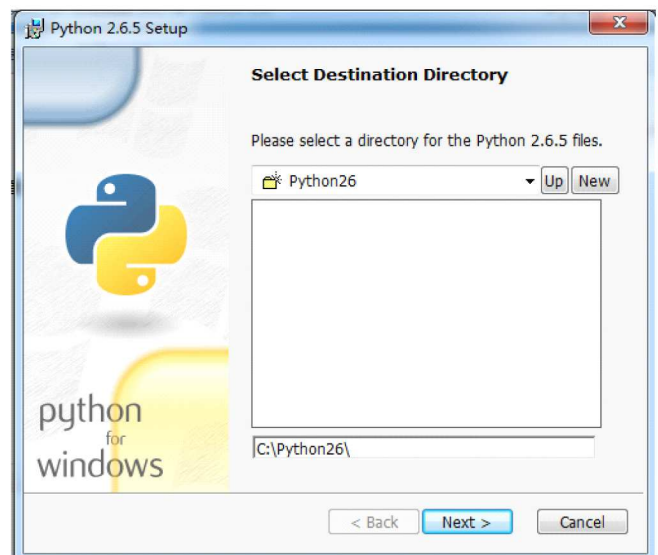
To get a better printing experience, we suggest you to take time reading this manual. It is designed to illustrate the procedures from the instalment of software to operation , and get you up and running as quickly as possible.

1 Installing software

1.1 Installing Python

- 1) Go to <http://www.mbot3d.com/pages/software>, and download ReplicatorG and python2.6
IMPORTANT : You MUST use MBots modified ReplicatorG and not the standard one available at replicatorG.org
- 2) For Windows XP, double-click python2.6.5 to install. For Windows 7, run the installer with administrator rights. (How? Right click on the installer to get the 'Run as Administrator' option.)
- 3) Keep clicking the “Next” button to finish installing.

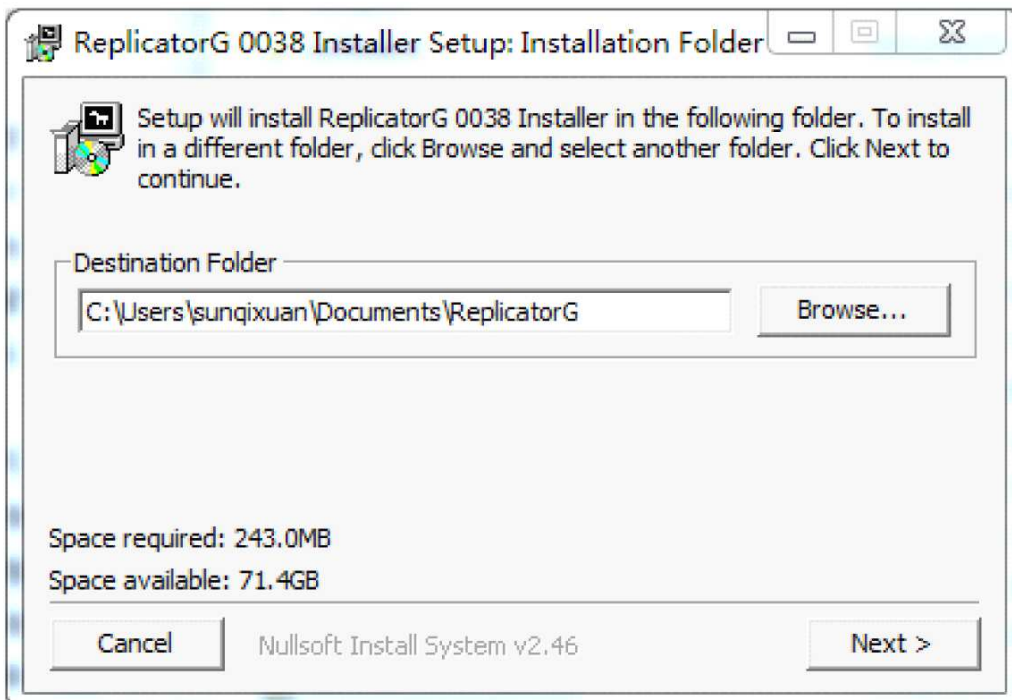
NOTE: Use the default installation directory.





1.2 Installing ReplicatorG

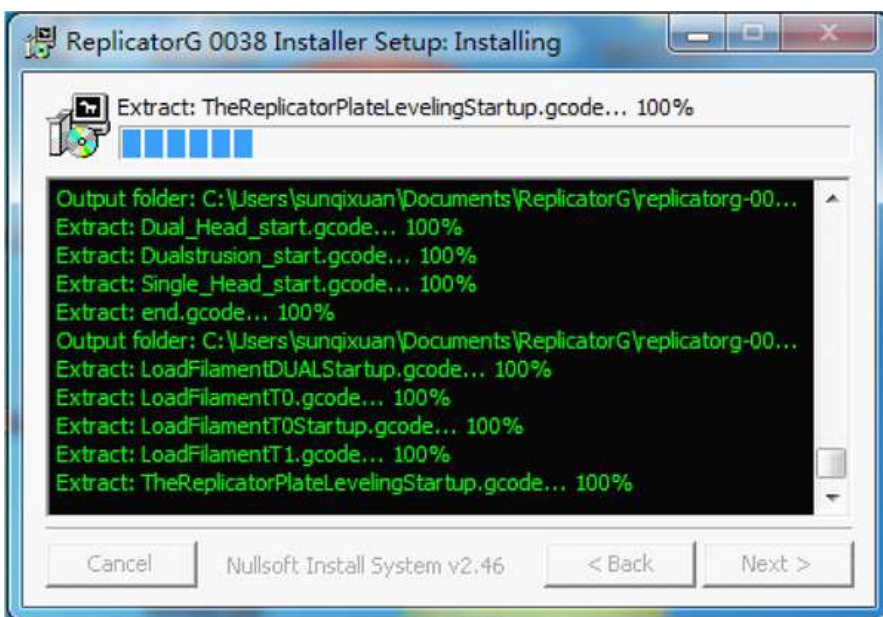
- 1) Double-click ReplicatorG, and you'll see the following window.



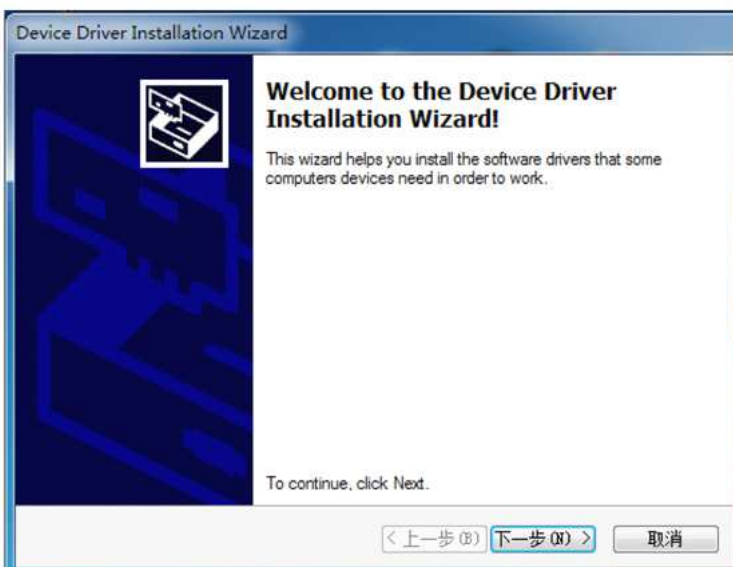
- 2) Click the "Next" button. Then on the following window, click the "Install" button. Suggest using the default destination folder.



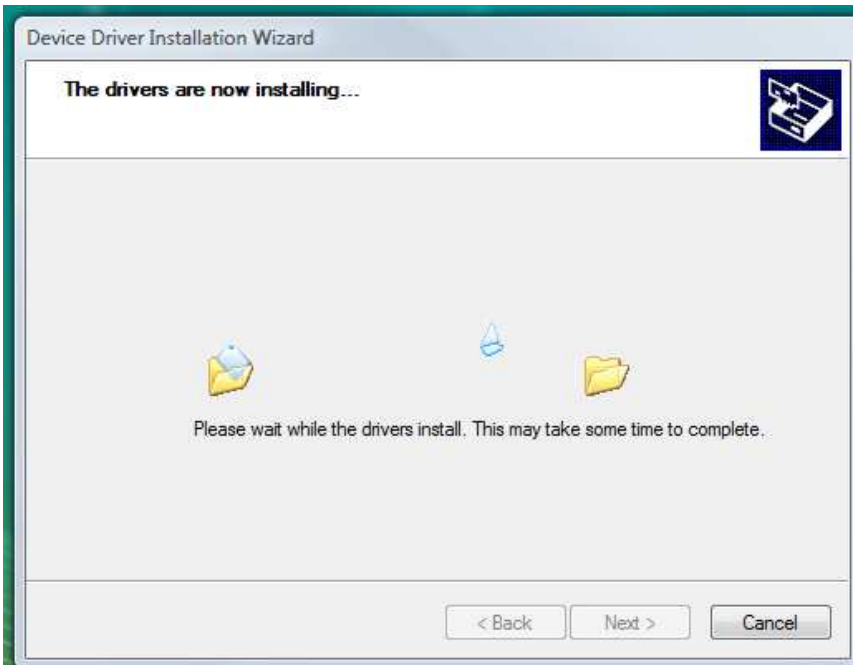
3) On the following window click the “next” button



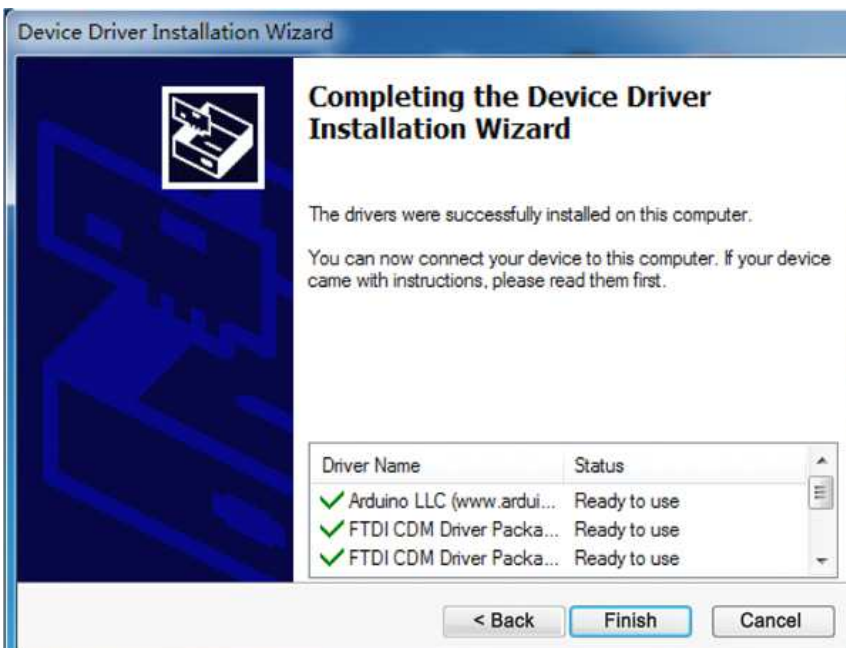
The driver installation wizard will pop up. Click the next button



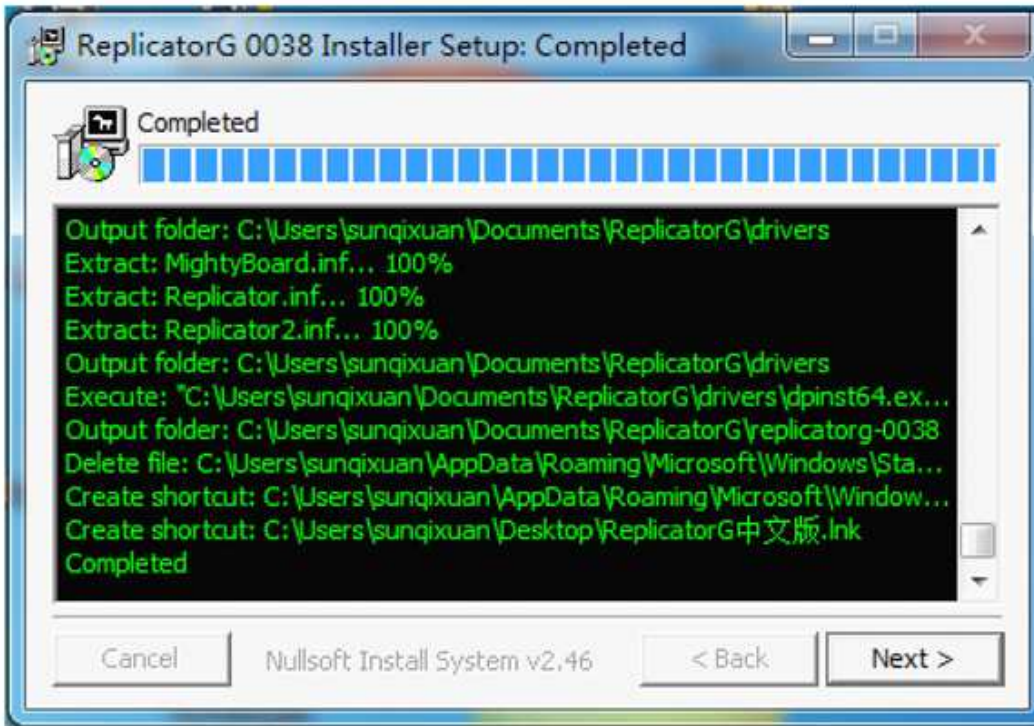
4) Click the “Install” button on the “Window Security” pop-up to install the printer drivers. When another “Windows security pop-up appears, click “always install the drive(i)”.



5) Click the "Finish" button to complete driver installation



While on the ReplicatorG installer window, click the "Next" button to finish the installation of ReplicatorG



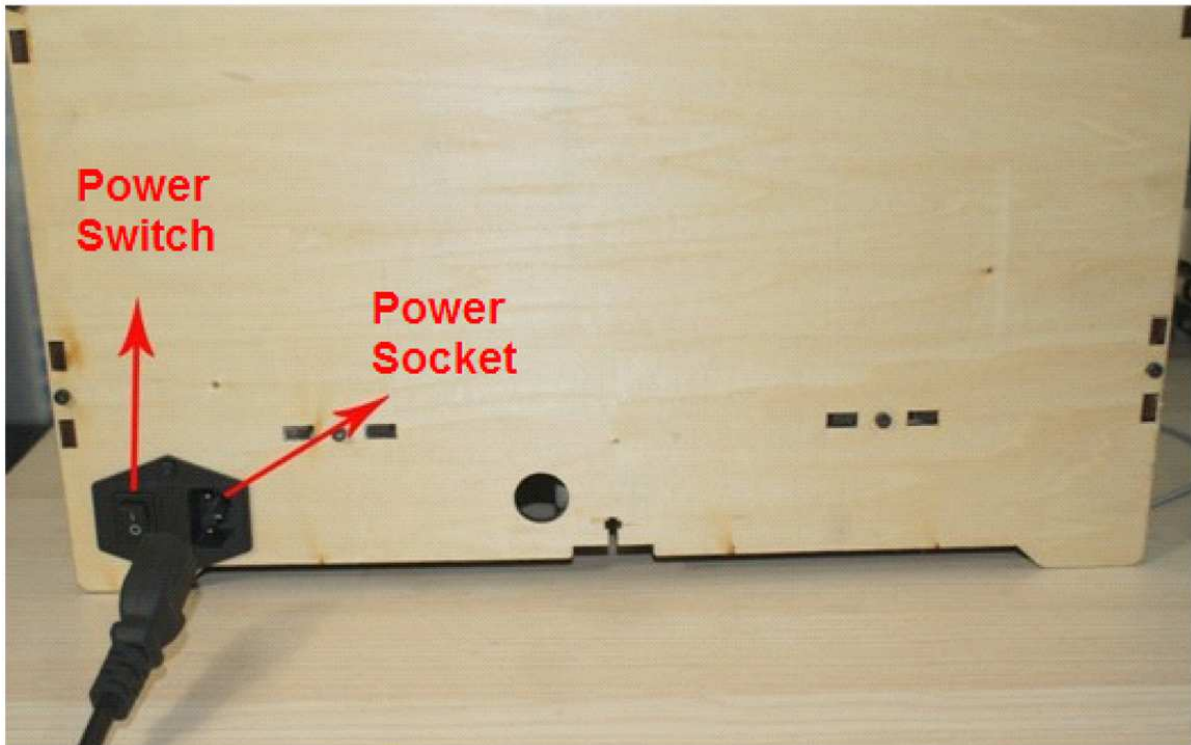
6) A short cut icon for ReplicatorG is created on your desktop



2 Pre-Print Checks

2.1 Power up

- 1) Ensure the power cable is attached (see the picture below), and then switch on.



- 2) The LCD screen reads "Heater Error!", which can be fixed by heating up through software. Press any button at the right hand and you'll see the message changes on the LCD, which means the machine is ok.

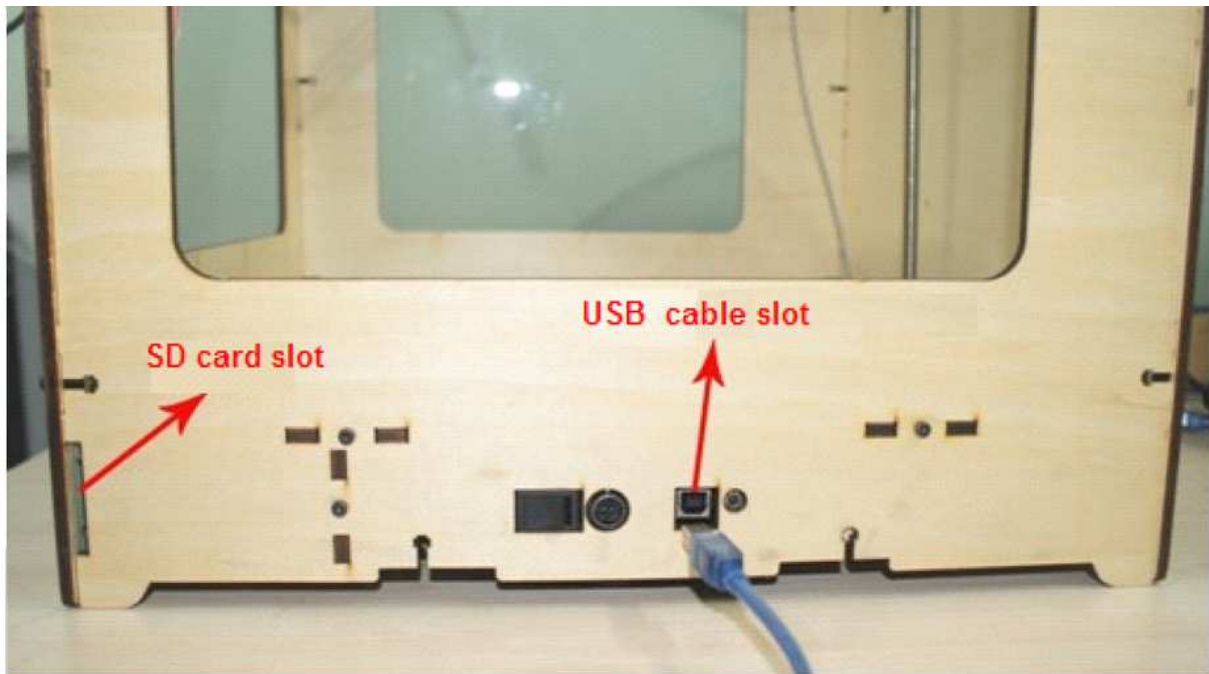
By default the firmware assumes there is a Heated Bed. As there is not you get the error. Simply use the LCD screen menu and navigate as follows:
Info and Settings -> General Settings -> Change "Heat Plate" to NO





2.2 Connect to computer

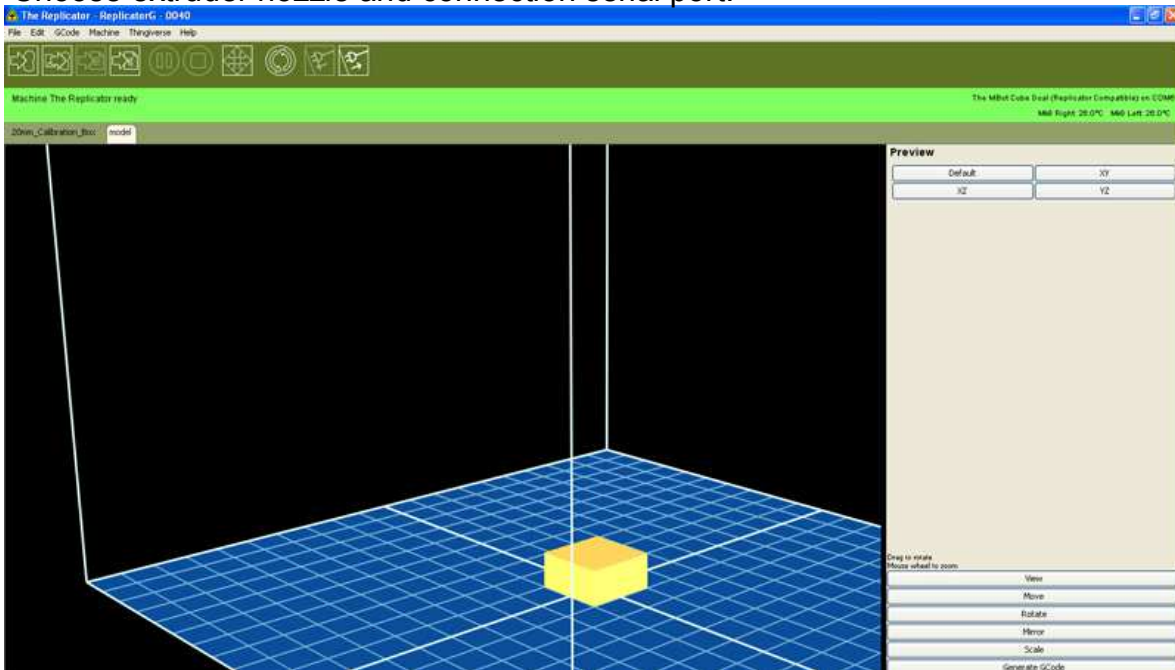
Make sure your computer is connected to your MBot with the supplied USB cable.



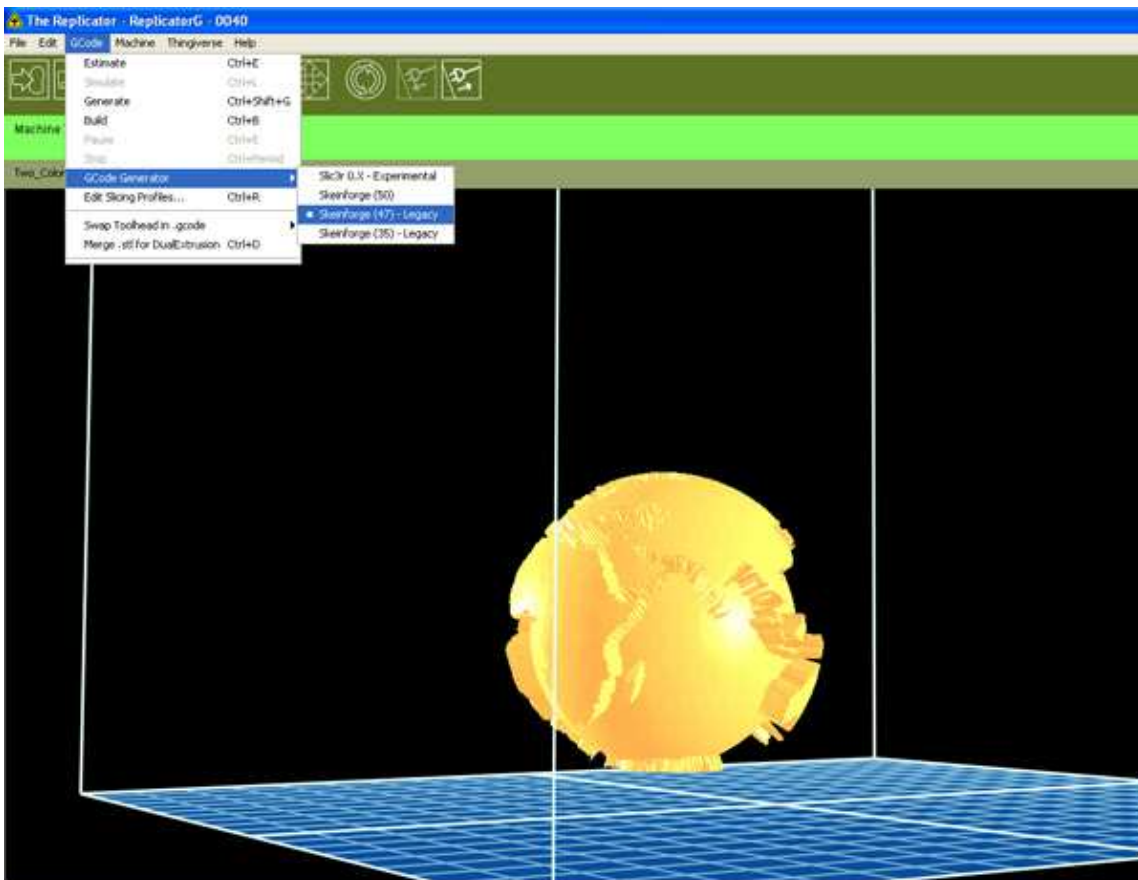
2.3 Setting up software



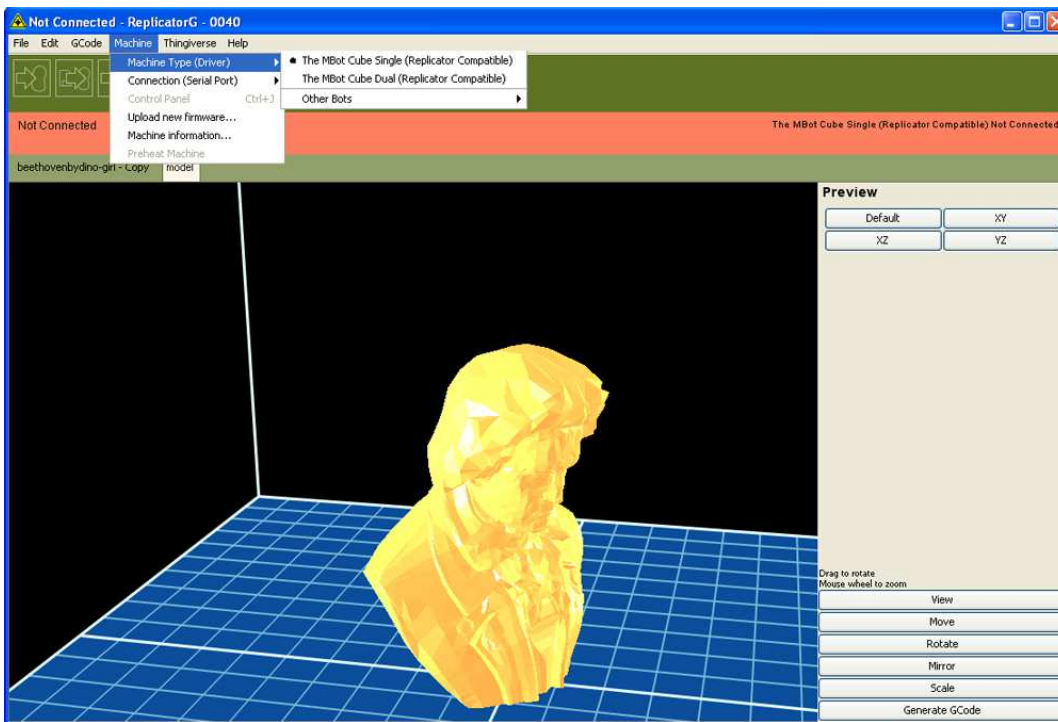
1. Double-click the desktop icon to run ReplicatorG.
2. Choose extruder nozzle and connection serial port.



A) Click GCode -> GCode Generator, and select "skeinforge(47)-Legacy".

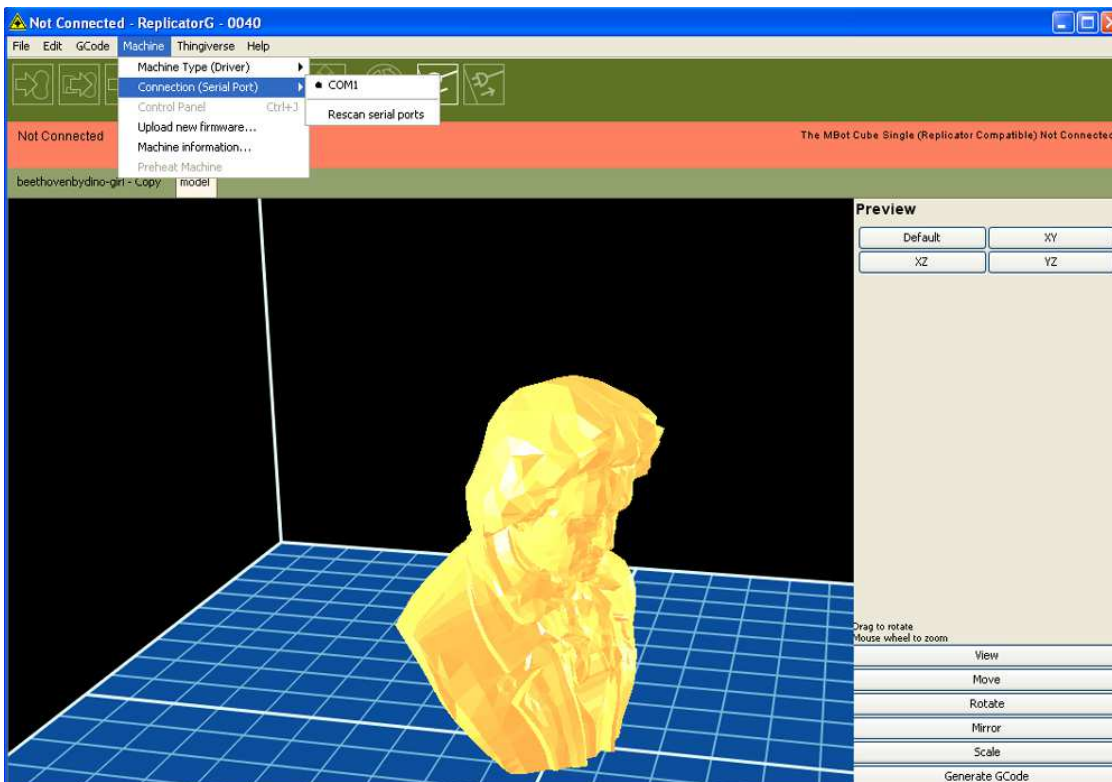


B) Click Machine -> Machine Type (Driver). If your printer is dual-extruder, select "The MBot Cube Dual". If it is a single select "The MBot Cube Single"



- 3) Click Machine->Connection (Serial Port) ->COM(Number). If you have only one printer, select the latest port. If you have more than one printer, please select the port according to your printer.

If there's no serial port on the menu, please click Machine->Connection (Serial Port)->Rescan Serial Port. If it doesn't work, please close ReplicatorG, disconnect the USB cable from your computer, and wait for 5 seconds. Then connect the USB cable to the computer, run ReplicatorG, and click Machine->Connection (Serial Port)->Rescan Serial Port. Wait for 10 seconds, and you'll be able to select serial port.



- 4) Make sure you select the corresponding serial port. Then click the "Connect to Machine" button (as shown below). Once connected, GCode generator and connection serial port can't be changed. If you want to change, please click "Disconnect from Machine" button (at the right side of the following icon).

- 5) When the machine is connected successfully, the message area below toolbar turns green. Otherwise, red. When the machine is running safely, it turns yellow.



Helpful Hint: When the machine can't be connected, please check :

- Power is being supplied to the printer.
- Your computer is connected to your MBot with the supplied USB cable.
- Driver is installed successfully.

3 Setting up model data

3.1 Importing model file into ReplicatorG

- 1) The machine supports STL, OBJ, Gcode and collada model files. You can use 3D design software (such as Rhinoceros, Solidworks, Sketchup, UG, etc.) to convert file format. Click "File->Open" to open the model file.

3.2 Move, scale, rotate and estimated build time

- 1) After opening the model file, you can **Position/Move/Scale** your model it as follows.

NOTE: When adjusting, please make sure the model is inside the printable area (i.e.the blue box on the model view window).The bottom of the box (i.e.the blue platform) stands for the build bed (with blue tape) in your printer.

The model can be zoomed by scrolling the mouse wheel. The view can be rotated by holding the mouse wheel and moving. There're five buttons at the lower right corner of the window.

A) View

Default: The build area is facing us.(i.e. The position of the model in view is exactly the same as in build bed when Finishing printing .)

XY : Top view

XZ : Front view

YZ : Right view



B) Move

Center : Place the model in the middle of the platform.

If the model can't be seen after opening the model file, which means it may be far from the printable area, please click the "Center" button.

Put on Platform: The model is resting on the surface of the platform, not suspended in air or below the platform. **Check it before printing.**

X-, X+ : Move left or right.

Lock Height: When it's checked, the height won't change while holding mouse to move the model.

C) Rotate

Z+, Z— : Rotate your model by 90 degrees in z.

Flat on Platform: It keeps the model in balance, which is very helpful for inclined models.

Strongly recommend using this function before printing.

Rotate around z-axis: When it's checked, the model rotates around z-axis by holding the mouse and moving.

D) Mirror Image

Reverse X: Right mirror image.

Reverse Y: Back mirror image.

Reverse Z: Bottom mirror image.

E) Scale

Input numbers to scale. When the number is less than 1, the model is zoomed in.

When it's greater than 1, the model is zoomed out.

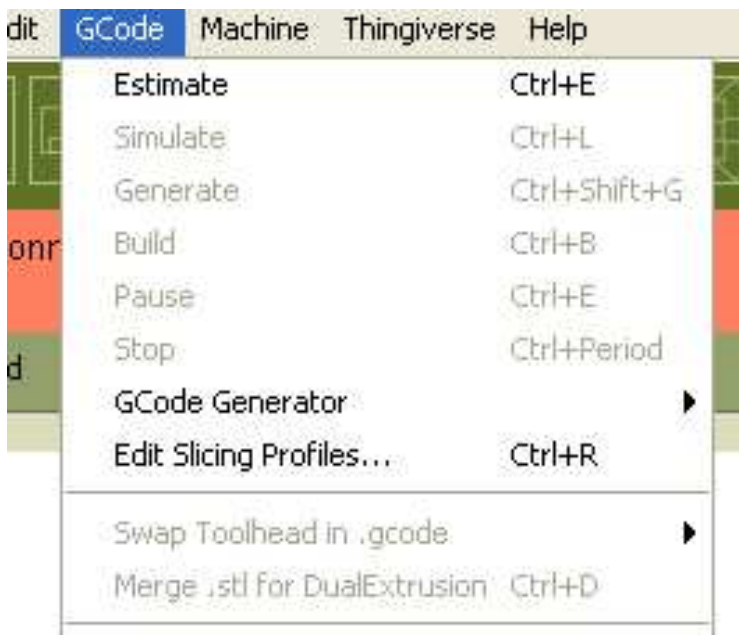
The options below is to convert model dimension between inch and millimeter.

Click the "Fill the Build Area" button to maximize the model, which is the maximum print size.

2) Estimate time for printing

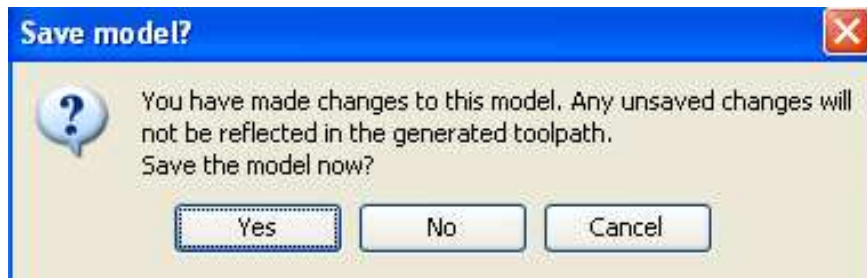
Click GCode-> Time Estimate (as shown below).

After about 30 seconds, the estimation is displayed at the bottom of the window (as shown below). Usually, it takes longer than estimated.



3.3 Generating GCode (Slicing)

- 1) Click the “Generate GCode” button at the lower right corner of the window. Then the following prompt pops up, if the model is adjusted but not saved. Click “Yes” to save changes. Otherwise, click “No”.



But please click “Cancel” and reopen the sample file, if you’re following our example. We have to make sure the model size is proper for holding ABS material.

- 2) After clicking “Generate GCode”, the following window pops up. Red arrows point to our helpful hints.

Set up as shown above. Then click “Default ” tab to set up as shown below.

Select MBot Cube

When checked Rafts and support will automatically be created.

Click both these

Fill :0% = hollow, 100% = solid. Percentages in between are filled with hexagonal Structure

Temperatures have to change for different filaments.

You can influence where support material is / isn't used.

If the model has overhanging features inside and you are printing it hollow, select 'full support'. If the overhangs are outside the model select 'outside'

If there are no overhangs you can select 'None'

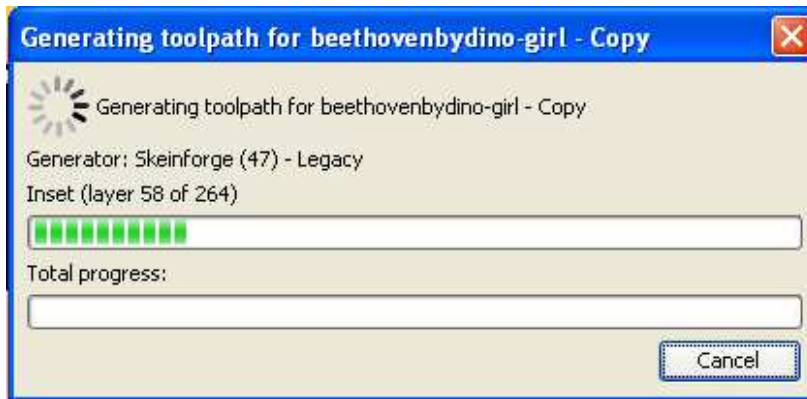
Layer height = resolution. 0.1 = 100 microns. It is best to start at 0.27

Shells = how thick the outer wall is

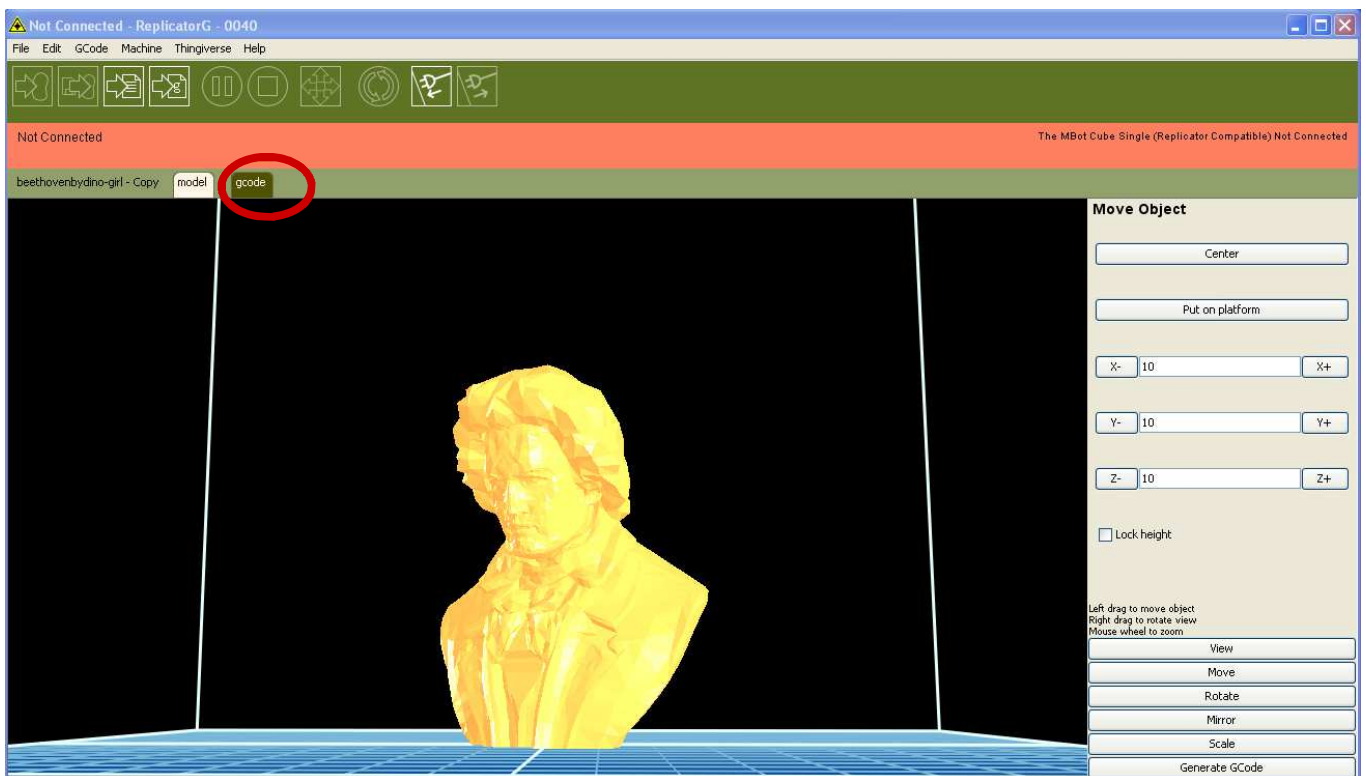
Feed rates and travel speeds see Chapter 5 :Printing

Generate GCode **Cancel**

- 3) When finishing setting up, click the “Generate Gcode” button and you’ll see the following window showing progress bars.



- 4) After GCode is generated, you’ll see the “gcode” tab next to the “model” tab.



- 5) Click the “gcode” tab and you’ll see the window as shown on the next page.

yoda model gcode

```
(** This GCode was generated by ReplicatorG 0034 **)
(* using Skeinforge (35) *)
(* for a Dual headed The Replicator *)
(* on 2012/08/12 14:21:16 (+1000) *)
(**** start.gcode for The Replicator, dual head ****)
M73 P0 (enable build progress)
G21 (set units to mm)
G90 (set positioning to absolute)
G10 P1 X-16.5 Y0 Z0 (Designate T0 Offset)
G10 P2 X16.5 Y0 Z0 (Designate T1 Offset)
G54 (Recall offset coordinate system)
(**** begin homing ****)
G16Z X Y F2500 (home XY axes maximum)
G16L Z F1100 (home Z axis minimum)
G9Z Z-5 (set Z to -5)
G1 Z0.0 (move Z to "0")
G16L Z F100 (home Z axis minimum)
M13Z X Y Z A B (Recall stored home offsets for XYZAB axis)
(**** end homing ****)
G1 X11Z Y-73 Z155 F3300.0 (move to waiting position)
G130 X0 Y0 A0 B0 (Lower stepper Vrefs while heating)
M109 S100 T0 (set HBP temperature)
M6 T0 (wait for toolhead, and HBP to reach temperature)
M104 S220 T0 (set extruder temperature)
M6 T0 (wait for toolhead, and HBP to reach temperature)
G130 X127 Y127 A127 B127 (Set Stepper motor Vref to defaults)
M108 R3.0 T0
G0 X11Z Y-73 (Position Nozzle)
G0 Z0.2 (Position Height)
M108 R4.0 (Set Extruder Speed)
M101 (Start Extruder)
```

```
[15:50:58] Connecting to machine using serial port: COM6
[15:50:58] Motherboard firmware v6.2 (□)
[15:51:21] Loading Z:\CNC\Models\Fun\FacesFigures\yoda.gcode
[15:51:21] Load file : Z:\CNC\Models\Fun\FacesFigures\yoda.gcode
[15:51:25] Z:\CNC\Models\Fun\FacesFigures\yoda.stl
[15:51:25] Got name Default
[15:51:34] Parsed object name[Default] facets 198530
```

4 Test Printing

Please test nozzles before your first printing, which is required for first printing only.

4.1 Using the ReplicatorG Control Panel

- 1) Click the button as shown right to open the control panel.



The control panel pops up.

Use arrows to "Jog" the machine around.
X = Left-Right
Y = Front-Back
Z = Up-Down

Press these buttons only when temperature reaches 220°C
REVERSE : unloads filament
FORWARD : loads filament

Tells you the current Temperature of your left and right extruders. Single extruder cubes will appear as "right" extruders

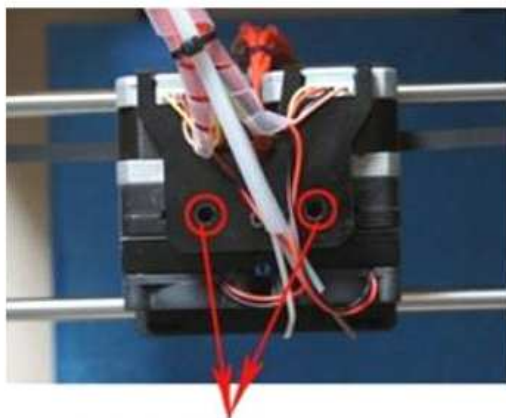
Type in target temperature
Usually :230°C for PLA
210°C for ABS

When testing make sure the nozzle is well above the build plate. You can "disable" stepper motors in an emergency to stop the head crashing into the bed

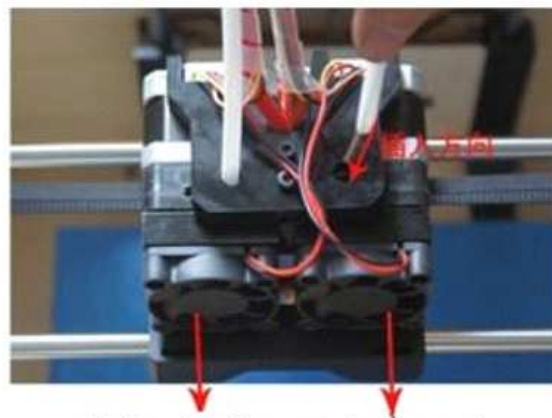
- 2) Once the nozzle is at temperature (220° C), load filament as shown below.

Click "Forward" button, and then take the filament and push it. You'll have to use some force. After a few moments you should feel the motor pulling it in. Then let go. You'll see some plastic start to come out the nozzle.

When that happens you may hear the noise like "tah-tah-tah". Please maintain pressure on the filament for 25 seconds. Then the noise will disappear. It also works when changing material.



Push the filament into these two holes.



Left extruder Right extruder
Once the nozzle is above 50° C, the fan starts working.

3) If the noise doesn't disappear:

Click the "Backward" button on the control panel, to pull the filament out. Cut off the used segment. Input "230" as target temperature and press "Enter" on keyboard. When current temperature rises to 230° C, click the "Forward" button. Then take the filament and push it with some force until some plastic comes out the nozzle.

If it still will not load please remove front fan and check that there is nothing blocking the filament path and try again.

NOTE fans go on with the voltage info at the back of the fan not facing forward.

Close the control panel if the machine works well.

If plastic stop coming out the nozzle or the noise reappear during printing, please stop printing and rotate z-axis manually to keep the nozzle from the build bed. Then take the measures described above .

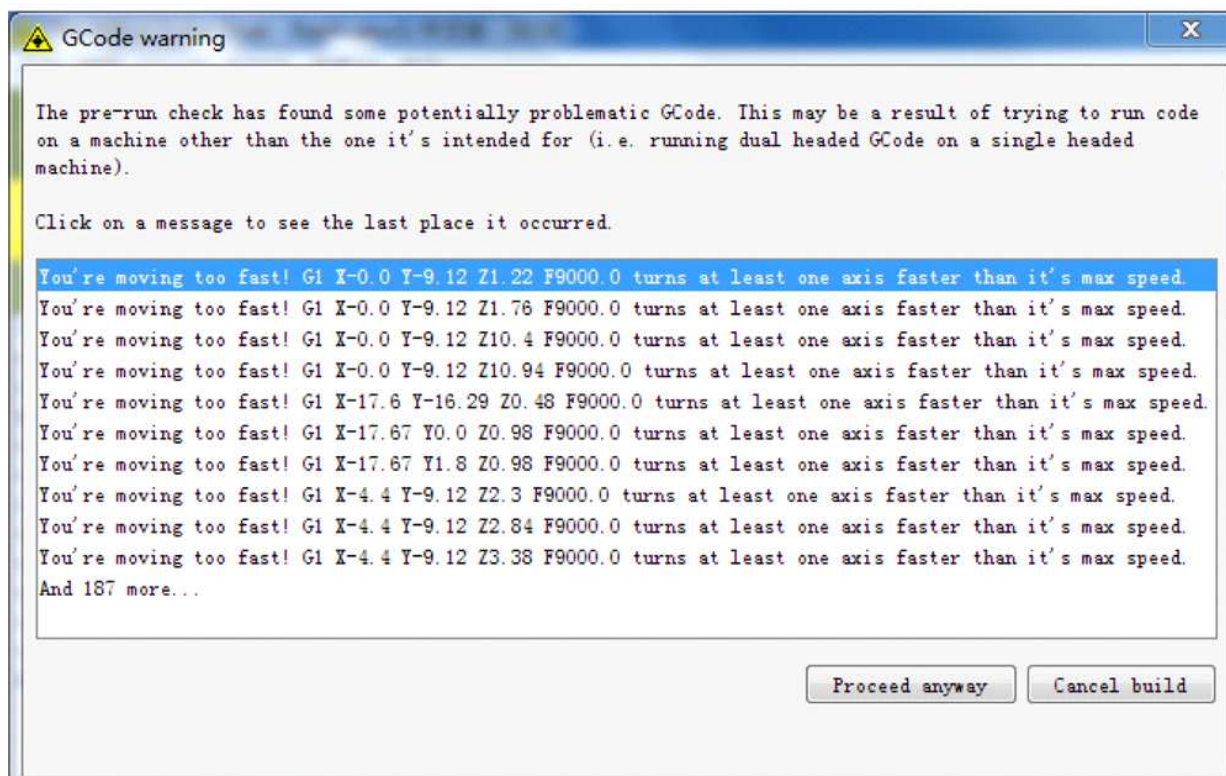
5 Printing

5.1 Printing from computer



- 1) Clear the blue build bed. Then click the “Print” button (as shown right) at the upper left corner of the window.

If a prompt pops up warning “You’re moving too fast!”, never mind. Click the “Proceed anyway” button to keep printing. The machine can handle that.

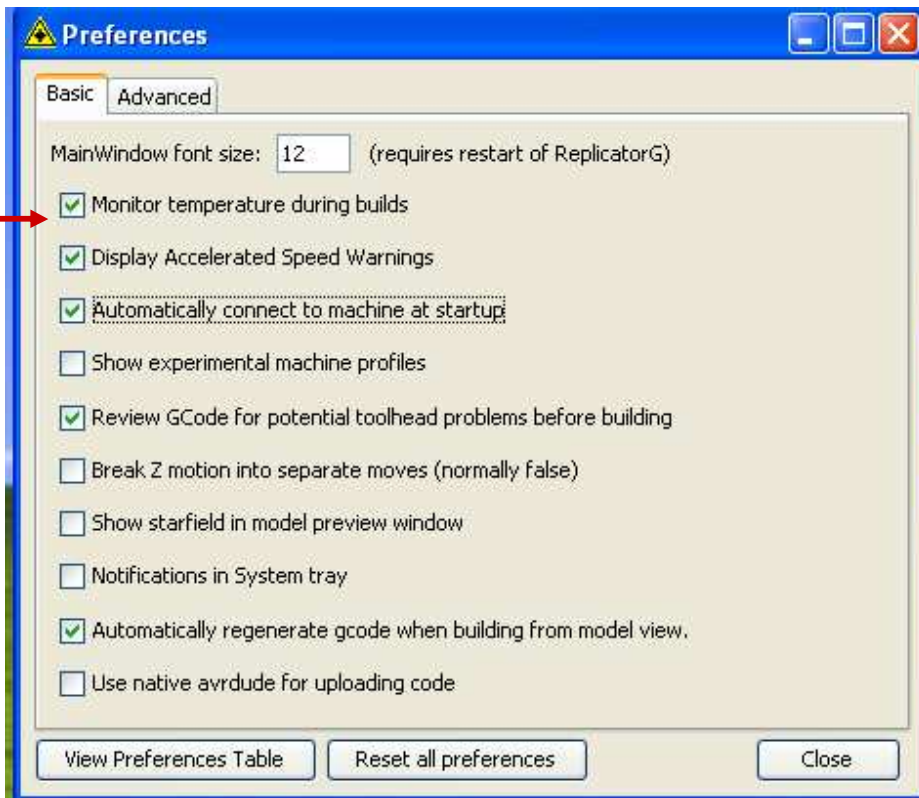


- 2) The print head will move to its warming position before starting. This may take 5 or so minutes if starting from cold. When the nozzle approaches your set temperature plastic may start to come out the extruder (more with PLA than ABS). This is normal.

- 3) If a prompt appears “Monitor print temperature: Closed ” at the top right corner of the window, click File->Preferences.

On the “Preferences” window, check the “Monitor print temperature” option and then close the pop-up window.





- 4) If you want/need to **pause or stop printing**, click the buttons circled below. NOTE: In the initial phase of printing (when the message area below toolbar is yellow), the Pause and Stop buttons are disabled. You can close ReplicatorG to pause printing.

While printing, it may take 30 seconds to pause after clicking the Pause button, which depends on computer configurations. DON'T hit the Pause button again.



- 5) **When finishing printing**, you'll see the icon circled below light up. You can now remove your model by hand, or using a spatula or paint scraper. If printing PLA take care not to dig into the blue tape. If the blue tape is damaged you may need to replace it before your next print

NOTE: Before powering off the printer, click the "Disconnect" button (as shown below).



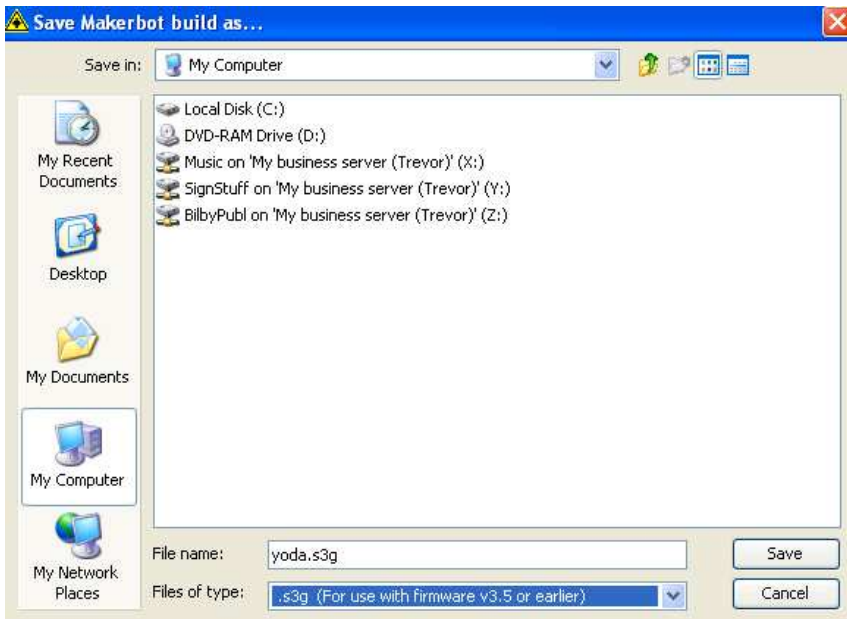
5.2 Printing from SD card

NOTE: The printer can recognize 1G or 2G SD card only.

- 1) Make sure your SD card is in the SD card slot of your computer. After GCode is generated, click the button as shown below.



A window will pop-up, find and open SD card.



File Name : Input file name
Note : Names over 10 characters will not show on the screen.

BilbyCNC recommends the following naming convention:

L = layer height

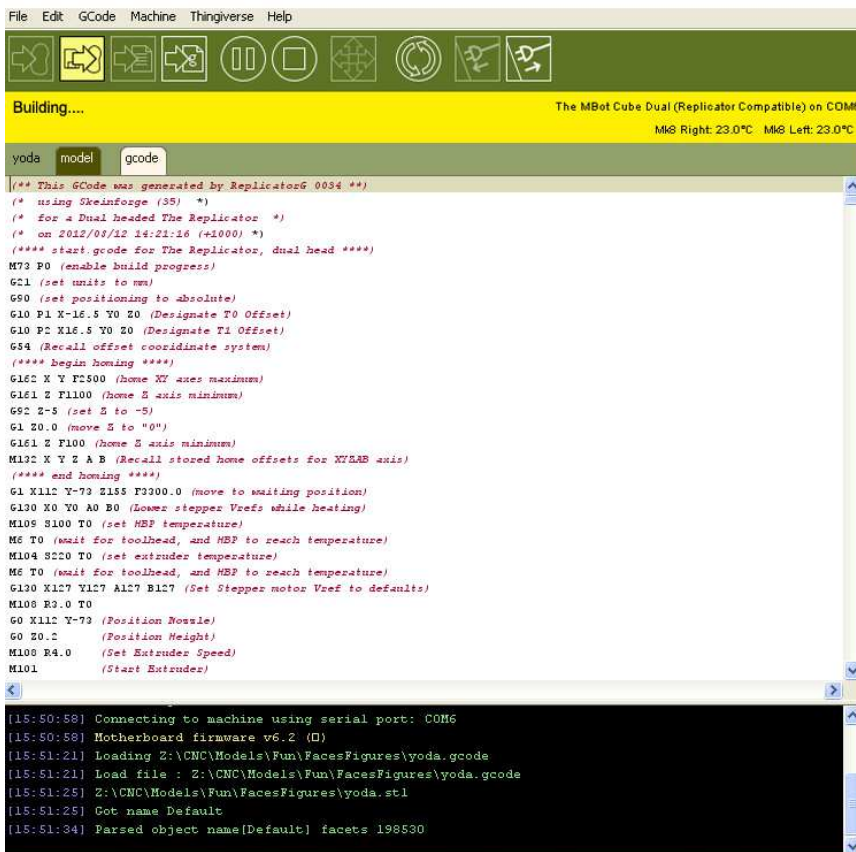
S = shells

F = Fill

EG *yoda1L2SH25F.x3g*

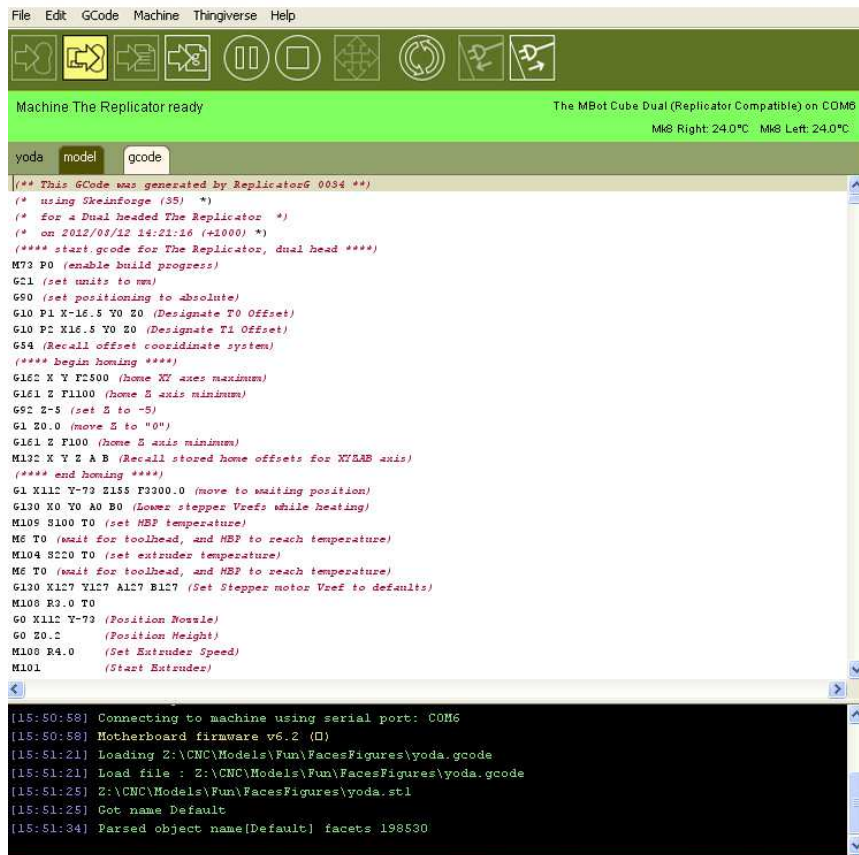
Tells us that the model is Yoda with a 0.1 layer height, 2 Shells and 25% fill

File Type : x3g format.



Then click the “Save” button, and you’ll see the window as shown left with the message “Building”, but actually the file is saving to the SD card.

2) After the Gcode file is saved, the window shows as below.

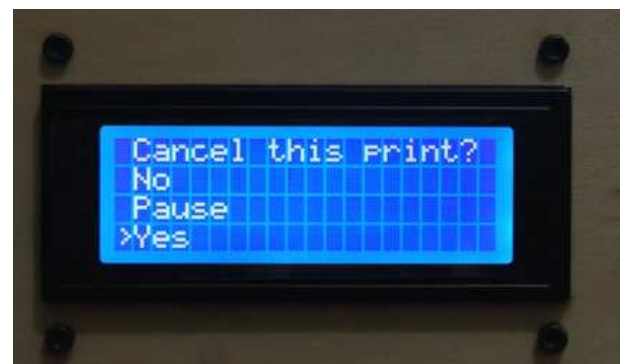


3) Take out the SD card from your computer and insert it into the printer. Then Select “Print from SD” and choose the model file, by up and down arrow keys beside the LCD screen. Press the center key to enter after selecting.



After selecting file and pressing the center key, the printer starts to heat up. The LCD screen shows as above right.

4) If you want to **cancel printing**, press the left arrow key (i.e. the back button). A message appears as right. Select “yes” and press the centre button.



5.3 Finished Prints



Above is a spool holder that was printed with a raft and support. After removing the print from the bed most support material will peel off with your fingers; however sometimes you will need a sharp cutting tool like a packers blade. We have also found wood carving tools helpful for fine areas.

The raft is often very very difficult to get off a flat surface and will need cutting off.

Other finishing techniques

You can also pain, guild and treat your finished model in a large variety of ways. You will find articles on these techniques at:

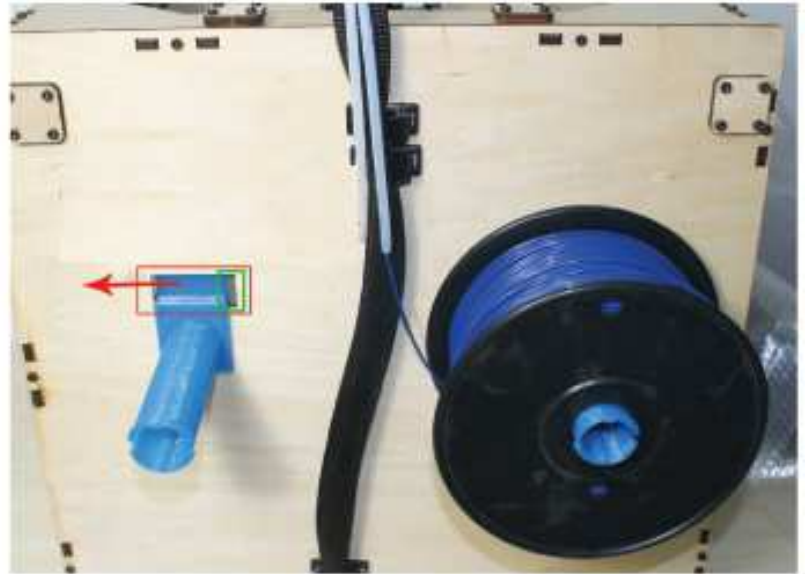
www.BilbyCNC.com.au under SUPPORT—> ARTICLES



5.4 Other Important Printing Information

1. Spool placement

When assembling the spool holder bracket (i.e. our printing example since chapter 3) to hold material, please push in the direction as the red arrow points, in order to get the maximum space.



The blue filament pictured right has been loaded so that the filament loads clockwise and consequently can drop back off the spool and across the spool holder. It will then get stuck between the black spool and the back of the machine, or even get jammed between the spool holder and spool!



The red filament shows the correct placement of the spool, with the filament being draw up from the bottom of the spool straight up to the filament feeder tube. The blue spool should be turned around to mirror the red.

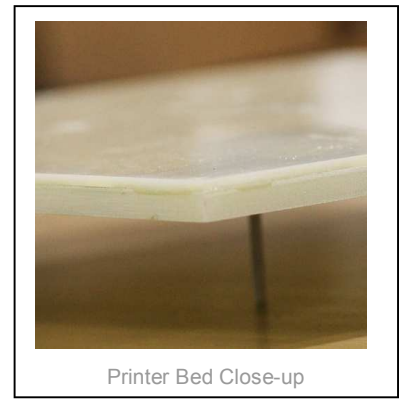
2. The Printer Bed

The composite printer bed supplied as standard with the Cube is a thick layer of fibreglass with a top layer of special material that makes your print stick whether in ABS, PLA or other material.

Other 3D printers use a heated bed so that ABS prints sticks when it is hot, but releases when cool.

BilbyCNC have found the printout sticks to the bed better than using a heated bed—but the downside is that releasing the printout can be difficult.

BilbyCNC consider the printer bed to be a wearing part and damage to it's surface is not covered under warranty. We suggest the following tips to help you get the maximum life out of your bed.



PRINTING WITH PLA—BLUE TAPE

If you are printing with PLA, blue painters masking tape still seems best at having your print stick during printing, but easily removed when complete.

Apply strips of the tape over your printing area evenly and smoothly. You may find you need to adjust the bed level after applying to allow for the thickness of the tape.

When the tape gets “worn” or damaged. Simply remove the tape and apply new tape.



PRINTING WITH ABS—ABS SLURRY

Acetone is flammable and not good for you—please read the safety precautions on the bottle.

ABS can stick so well to the bed that the bed is damaged when removing your print—this is especially true for prints with a large “footprint” - where the area of plastic directly in contact with the bed is large. **We strongly suggest making an ABS “slurry” to apply to your bed.**

To make the slurry, put some scrap ABS filament in Acetone in a safe container such as a glass jar (acetone will dissolve many plastics). It is freely available, try your local hardware store.

The filament will dissolve over a few hours to form a liquid. Acetone is normally very runny—like water. The resulting liquid can be very thick, but you want to keep it runny. It will dry almost instantly, so you want to err on the runny side, any excess will evaporate. Just add more acetone if required.

Use some folded paper towel to dip in the slurry and wipe across the bed, getting it as smooth and even as possible. It will dry almost instantly.

The layer you've just created will stick very well to the bed, and your print will stick well to it. In some ways it creates a sacrificial layer between the bed and printout. It will be the weakest point if you need to use a spatula to remove the print, so neither the bed or your printout are damaged. Simply reapply the slurry when it gets damaged or scraped off. You can remove the layer with paper towel and acetone if you require.

3. Temperatures and Feed Rates

INTRODUCTION

The MBot Cube and Cube 2 are based on the open source Makerbot designs and software for the Replicator 1. It works well with a special version of the ReplicatorG software, which takes into account the different size, offsets and setup of the Cube. Please read our accompanying sheet on preparing the printer bed before printing.

PRINTING SPEEDS

The Cube is not the fastest of printers, but it has the advantage that you can experiment with the setting in the ReplicatorG software. We have found the following settings advantageous when used with the standard, BilbyCNC or nylon type filaments

	Default	Faster	Experimental
Feedrate	41	60	97
Travel	56	70	95
Temperature (C)	210	210	215

Up-to-date Information

We have set up a MBot section on our *solutions* and *support forum* online and will continue to post articles and information there for our customers.

Both are accessible by going to :
www.BilbyCNC.com.au and clicking **SUPPORT**

6 Two-Colour (Dual) Printing

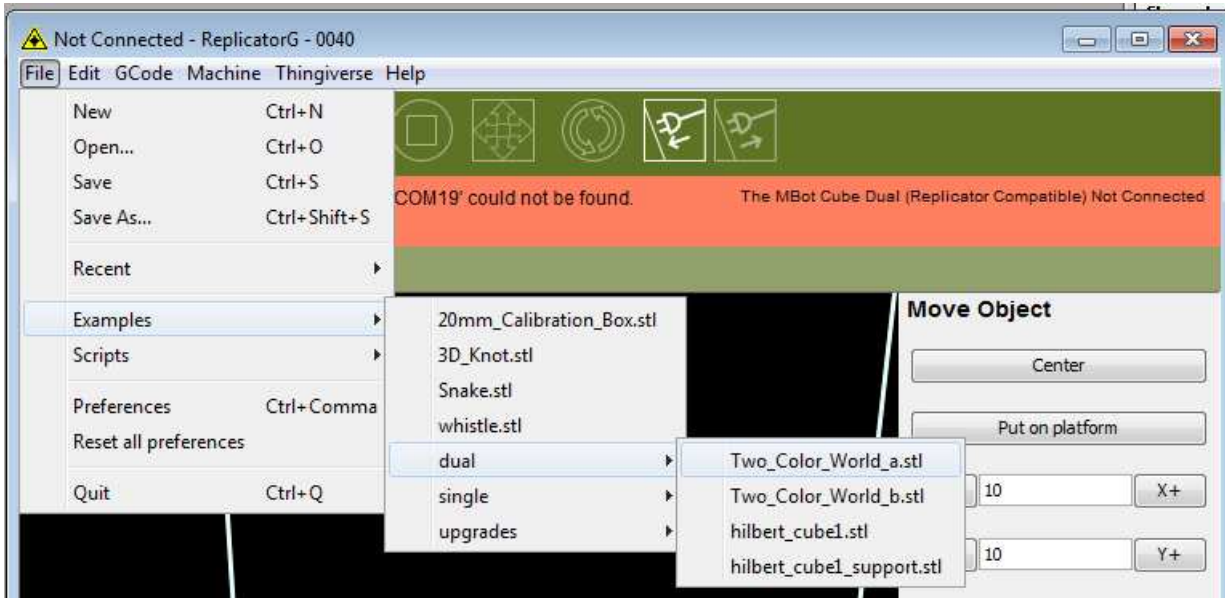
To print a model with two colors, you'll first have to have both parts saved as separate .stl files.

Then generate GCode by ReplicatorG for each model to generate one x3g print file.

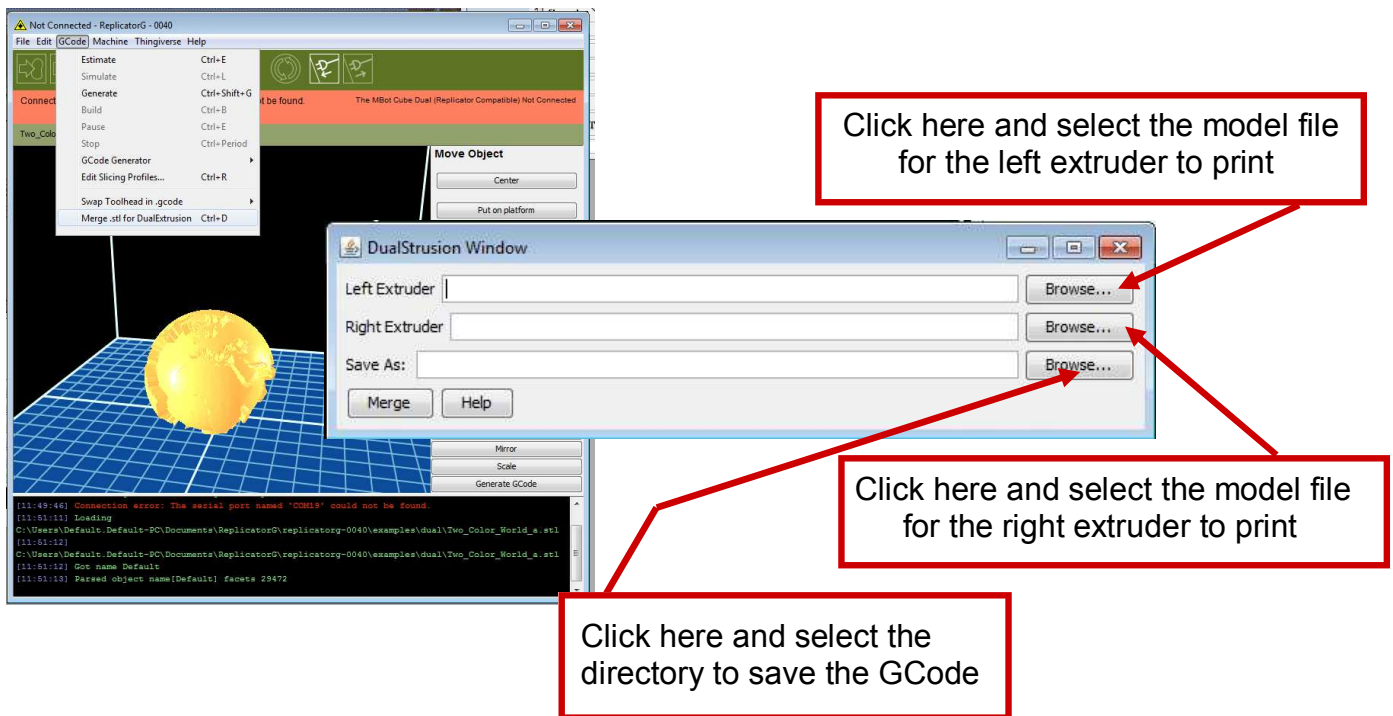
Step—By—Step example : Earth

The picture below shows the model after slicing.

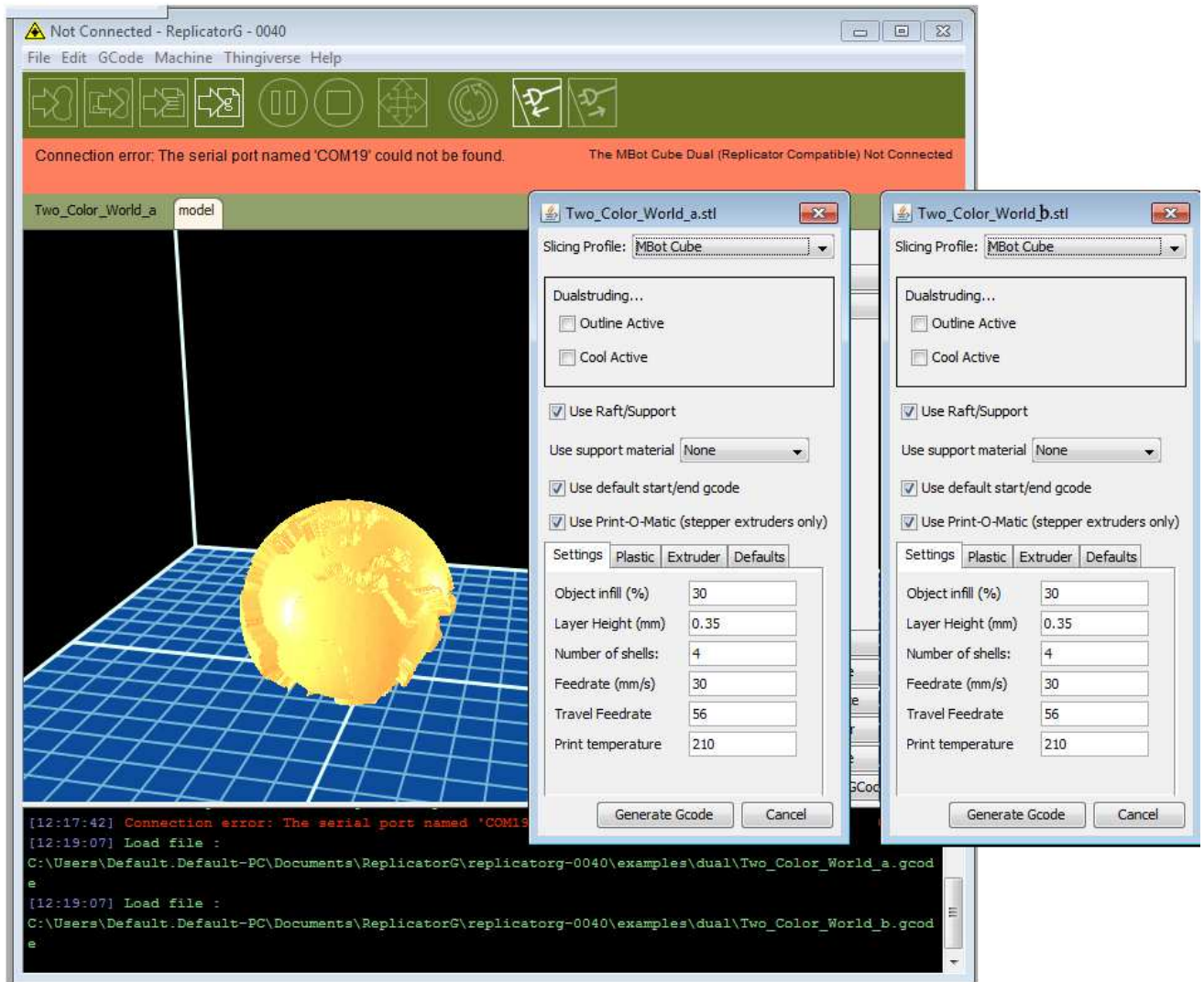
- 1) Click the "File" menu and choose the file for Part 1.



- 2) Then go to "GCode" -> "Merge .stl fir DualExtrusion)
The pop-up window shows as below.



3) Click the “Merge” button and you’ll see three pop-up windows as shown below.



In the EARTH example : on the two setting windows, uncheck the “use raft/support” option and select “None” in the “Support Material” list; because two models can support each other.

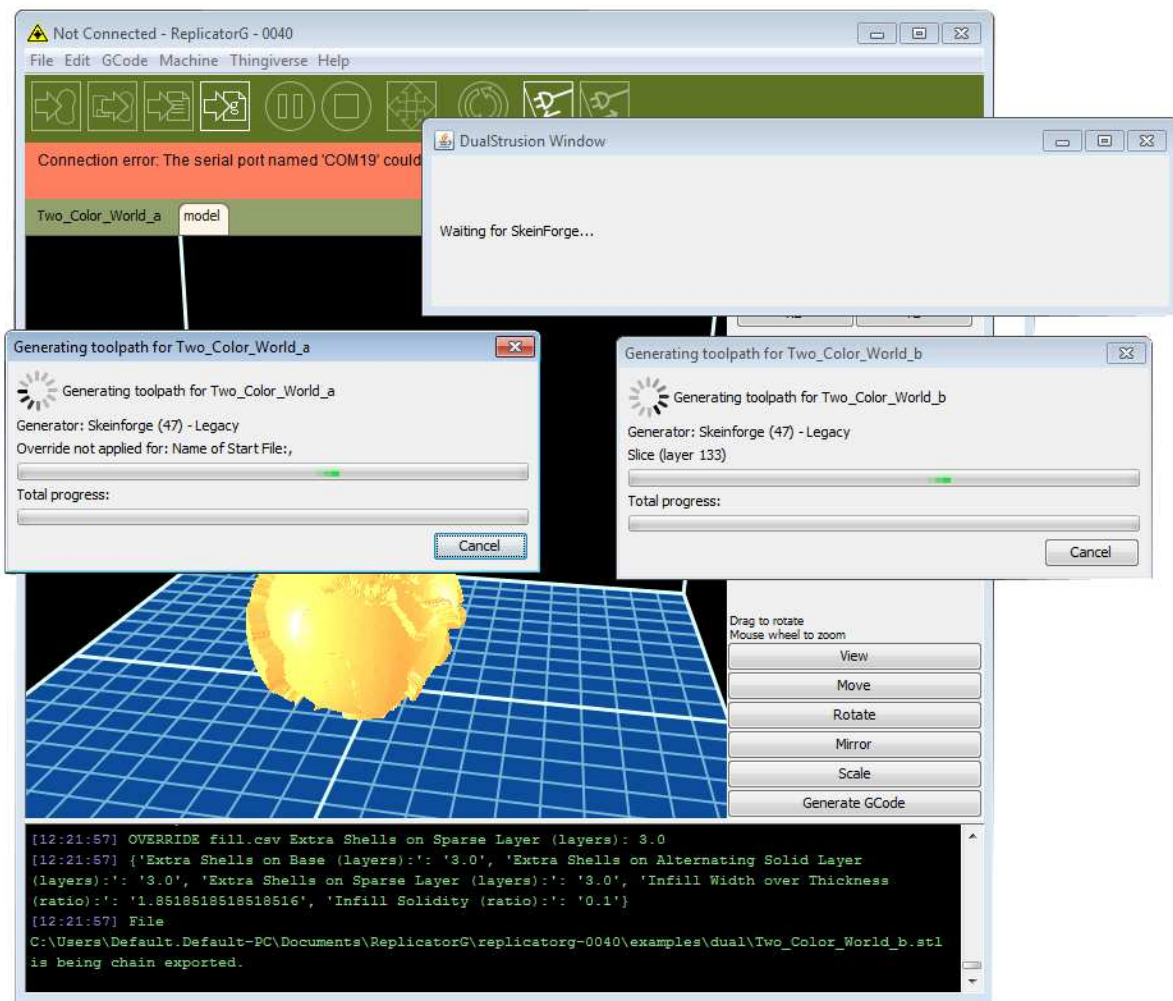
You can now enter all your desired settings for Layer height, Shells etc.

We suggest :

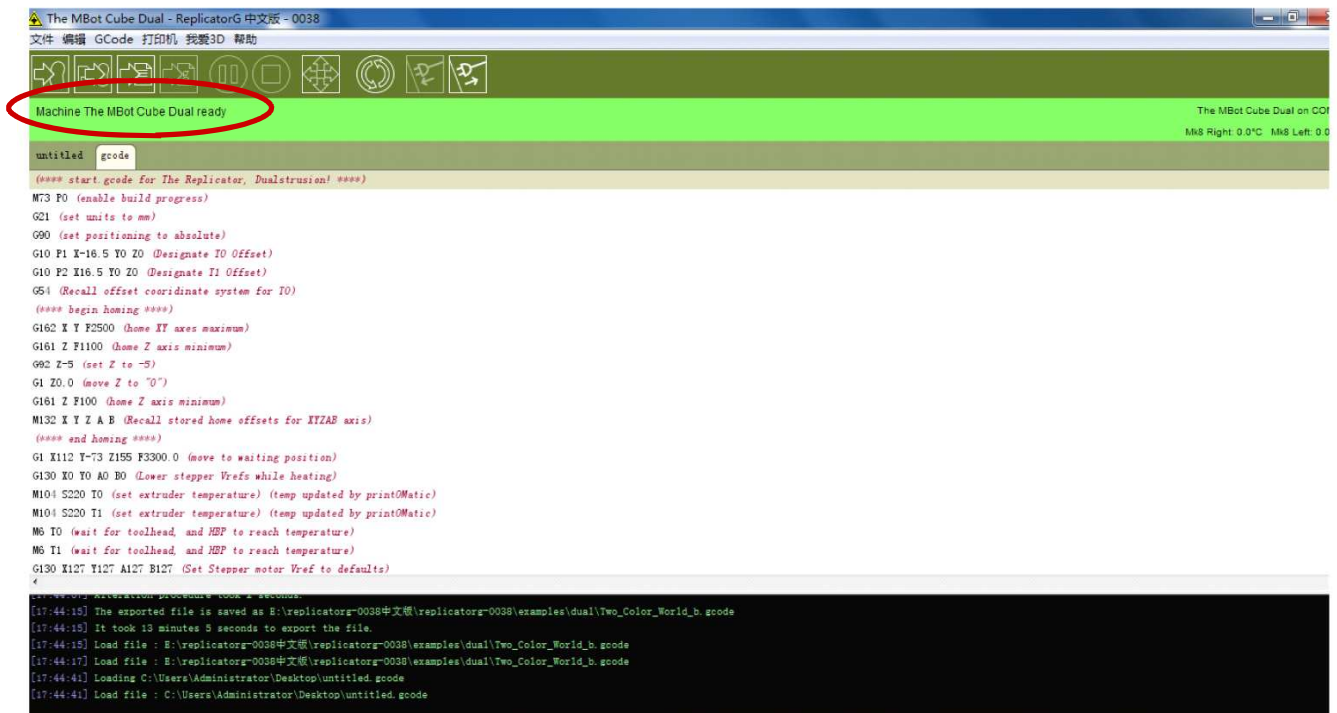
- Layer Height : 0.27
- Shells: 3
- Fill : 10%
- Temperature : 210
- Feederate : 41
- Travel Speed : 60

Then click “Generate GCode” on both windows

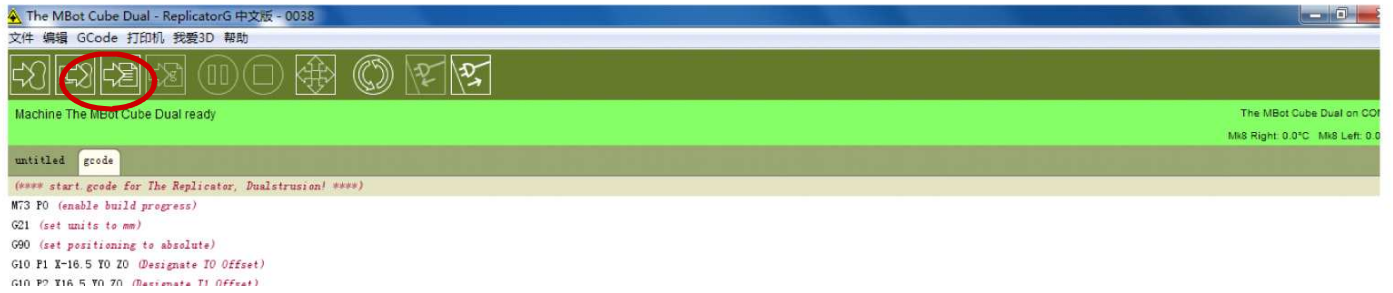
You will then get 2 pop up progress windows as ReplicatorG slices both files



It can take up to 30 minutes to slice the Earth example file. When finished it will look like this :



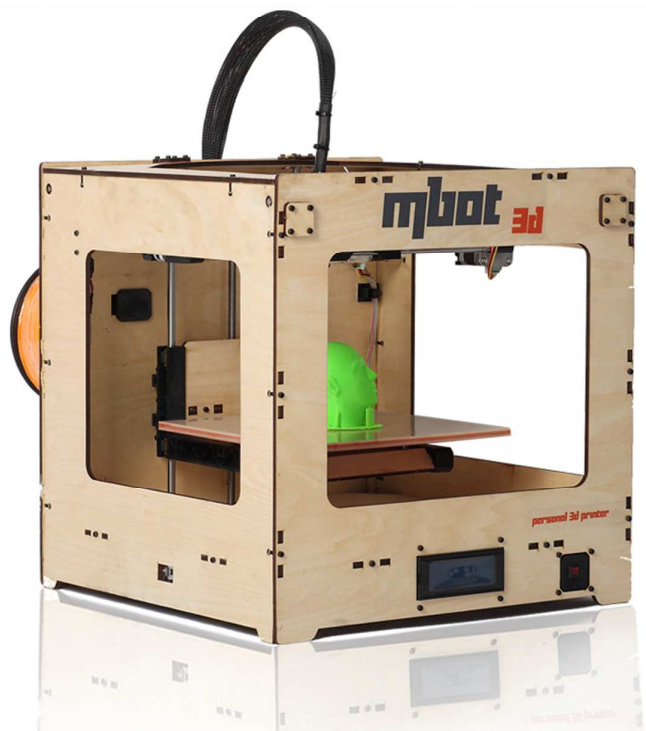
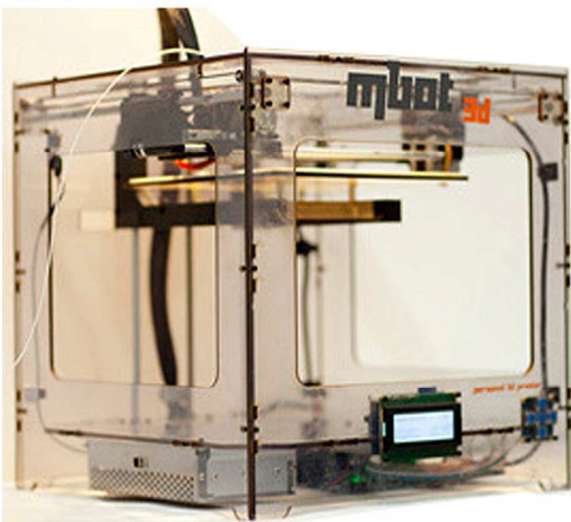
4) Click either the *Print to Machine*, Or *Print to SD Card* buttons and print your file



Thanks for choosing an

mbot 3d

as you desktop 3D printer



mbot 3d
OFFICIAL RESELLER

BilbyCNC
www.BilbyCNC.com.au

Australian freecall **1800 245 297**
International : **+61 2 4403 0352**