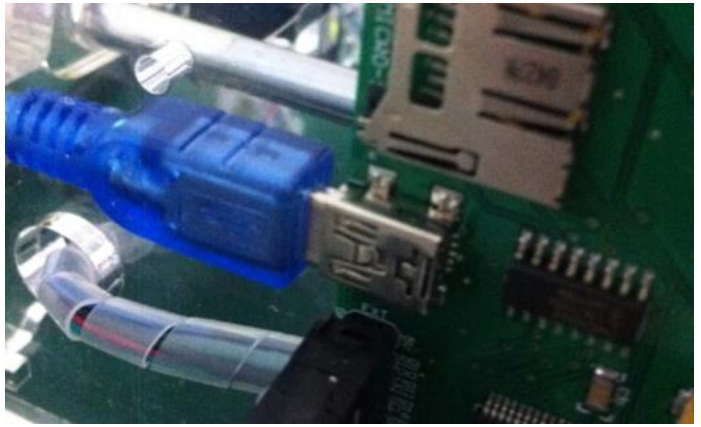
1. Connect the machine to the electricity supply.

2.Plug one end of USB cable into USB port of your computer

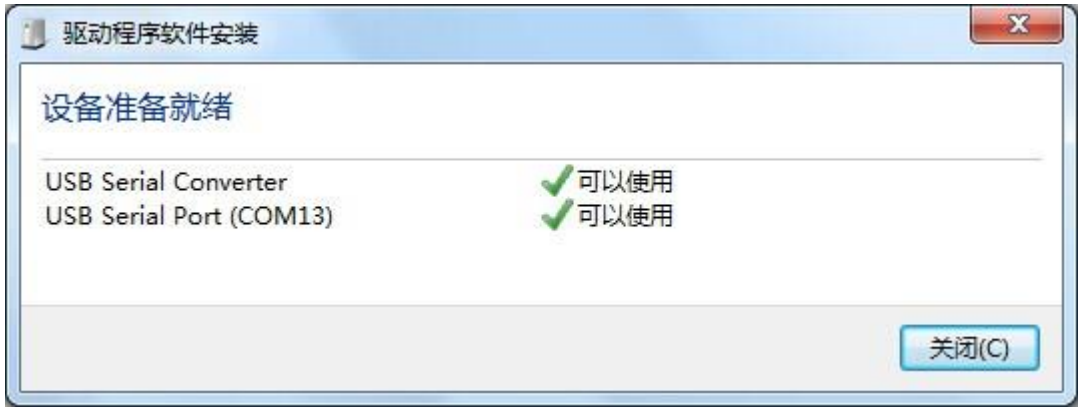


3. Plug the other end into PBC board of the machine

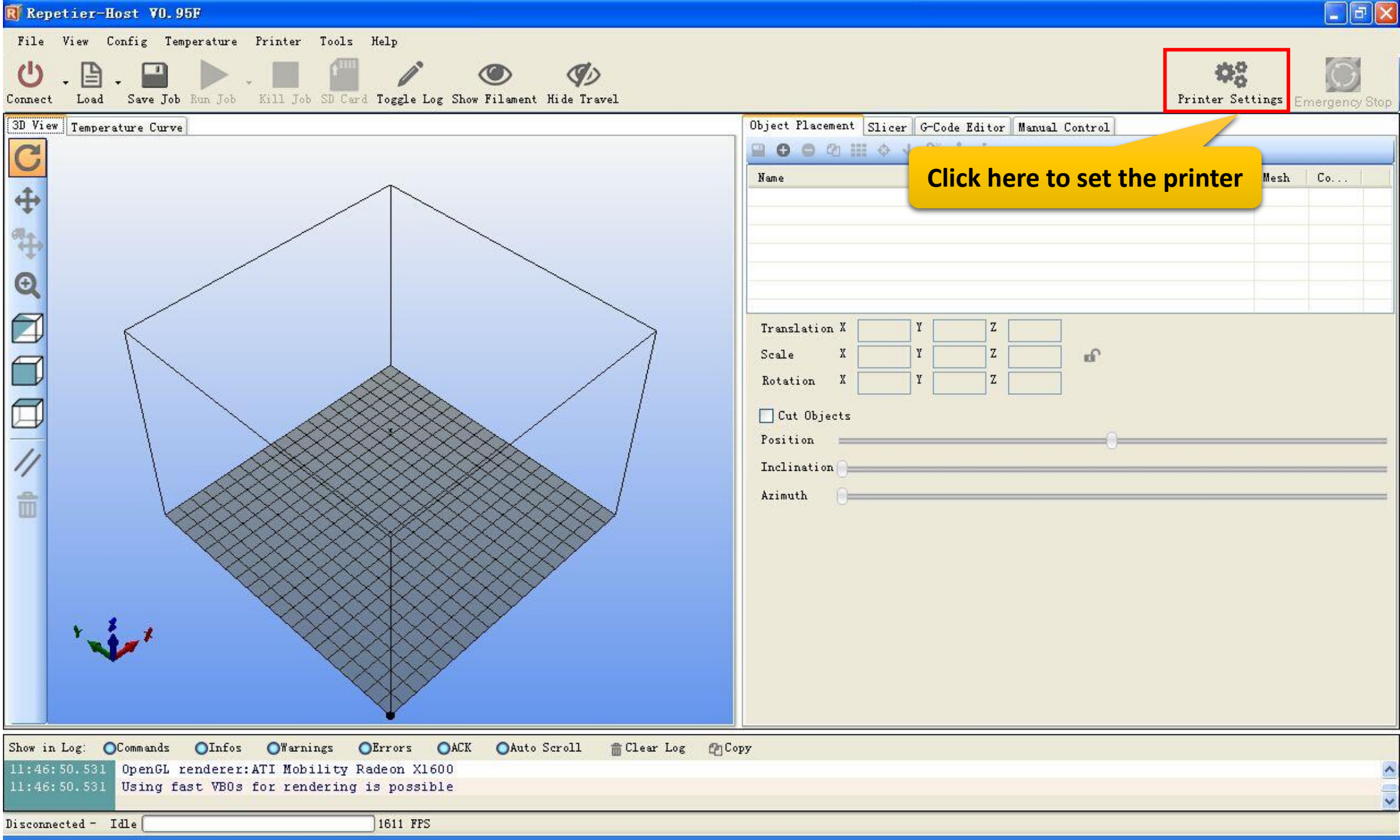
The firmware is optimized by my company ,and then loaded on mainboard, please do not upgrade The firmware by yourself.

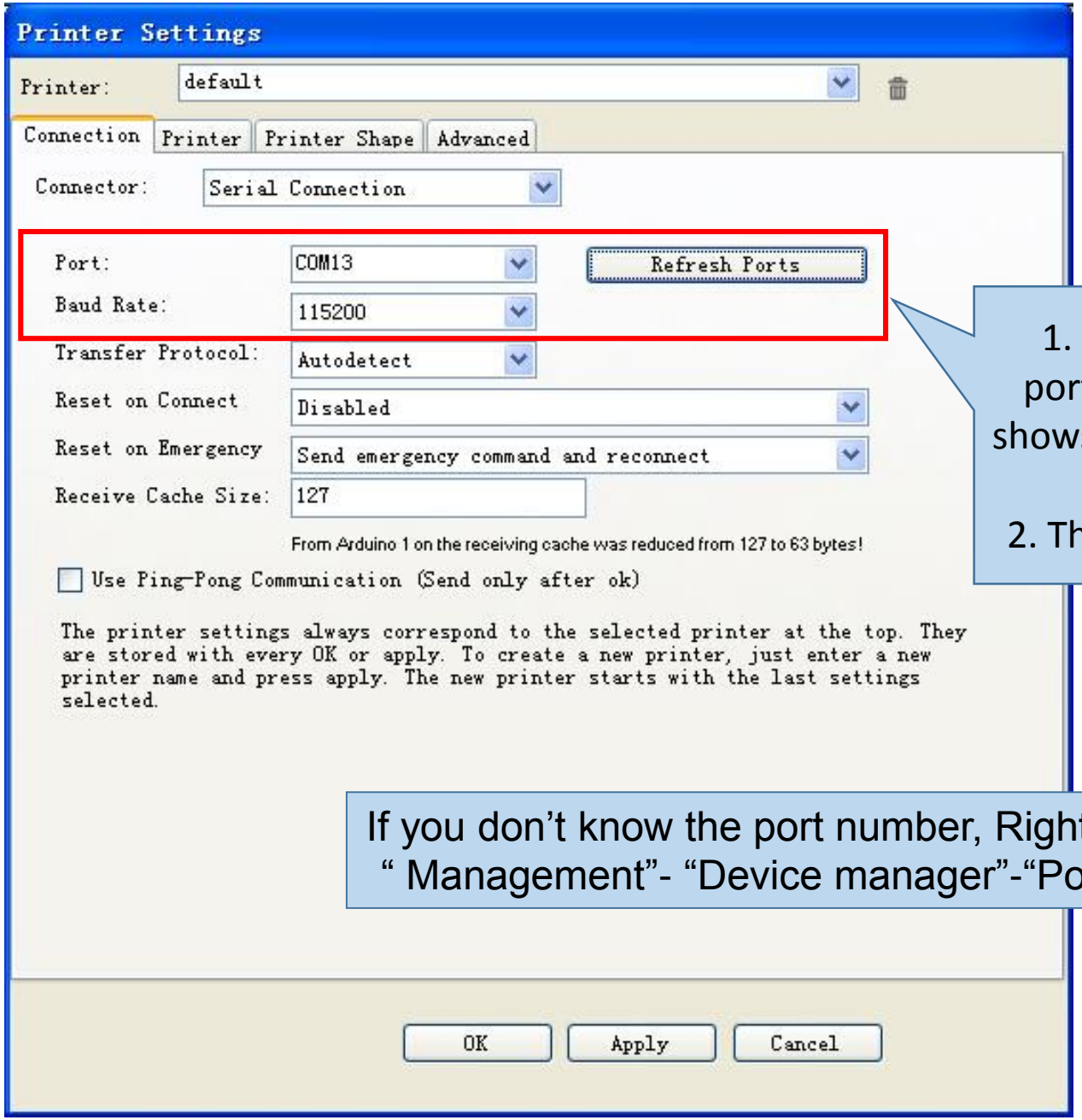
After the USB cable plugged, the USB driver will auto-install on Win7 system, as picture below

1. System prompted the USB driver is installing



2. Setup complete, different 3d printer has different USB Serial port, here shows **COM13**

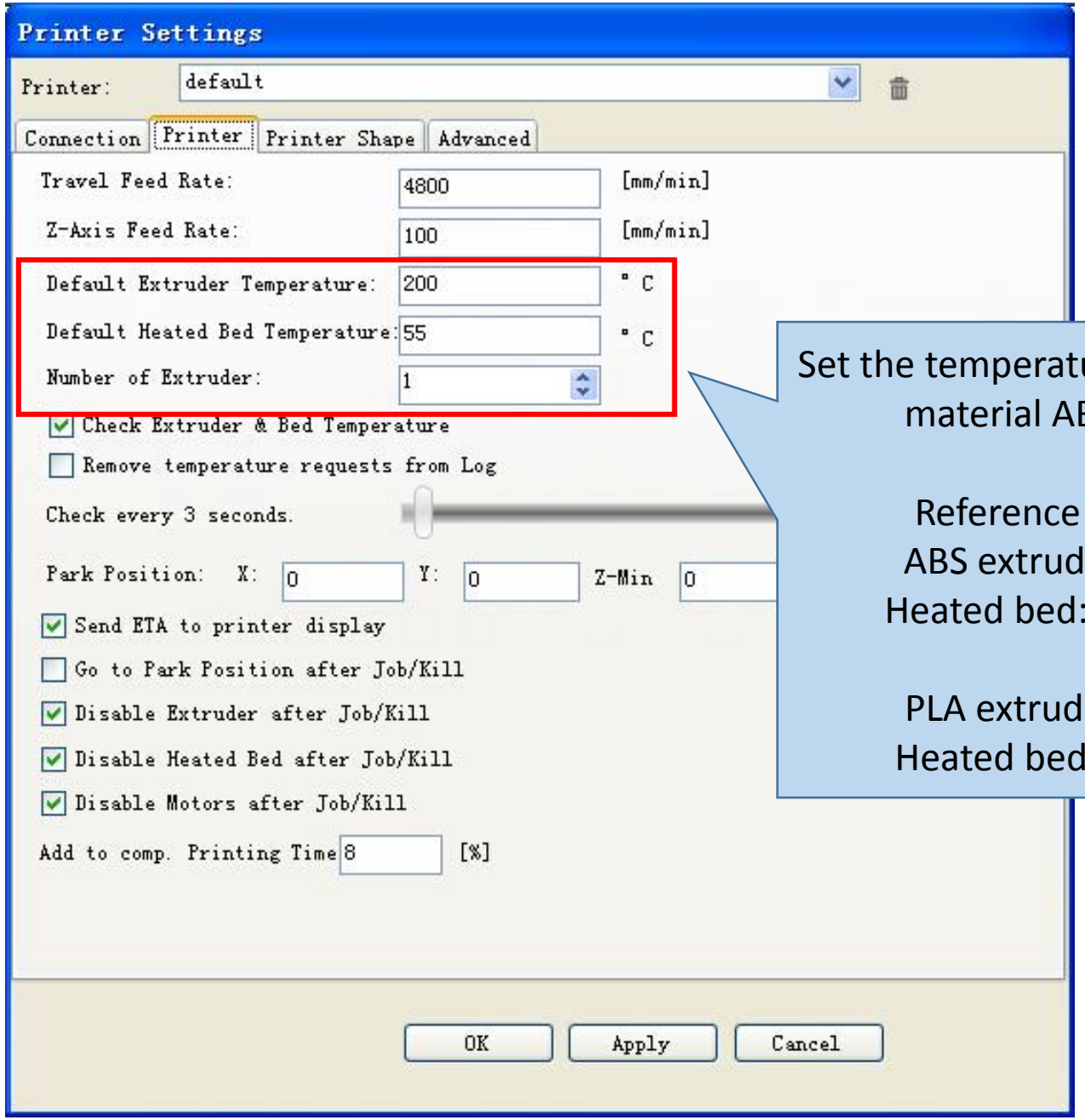

Repetier-Host V0.95F
File View Config Temperature Printer Tools Help
Connect Load Save Job Run Job Kill Job SD Card Toggle Log Show Filament Hide Travel
Printer Settings
Emergency Stop
Click here to set the printer
3D View
Temperature Curve
Object Placement
Slicer
G-Code Editor
Manual Control
Name
Mesh
Co...
Translation X Y Z
Scale X Y Z
Rotation X Y Z
Cut Objects
Position
Inclination
Azimuth
Show in Log: Commands Infos Warnings Errors ACK Auto Scroll Clear Log Copy
11:46:50.531 OpenGL renderer:ATI Mobility Radeon X1600
11:46:50.531 Using fast VBOs for rendering is possible
Disconnected - Idle
1611 FPS



1. Select the USB serial port ,Different 3D printer shows different port number

2. The Baud Rate is **115200**

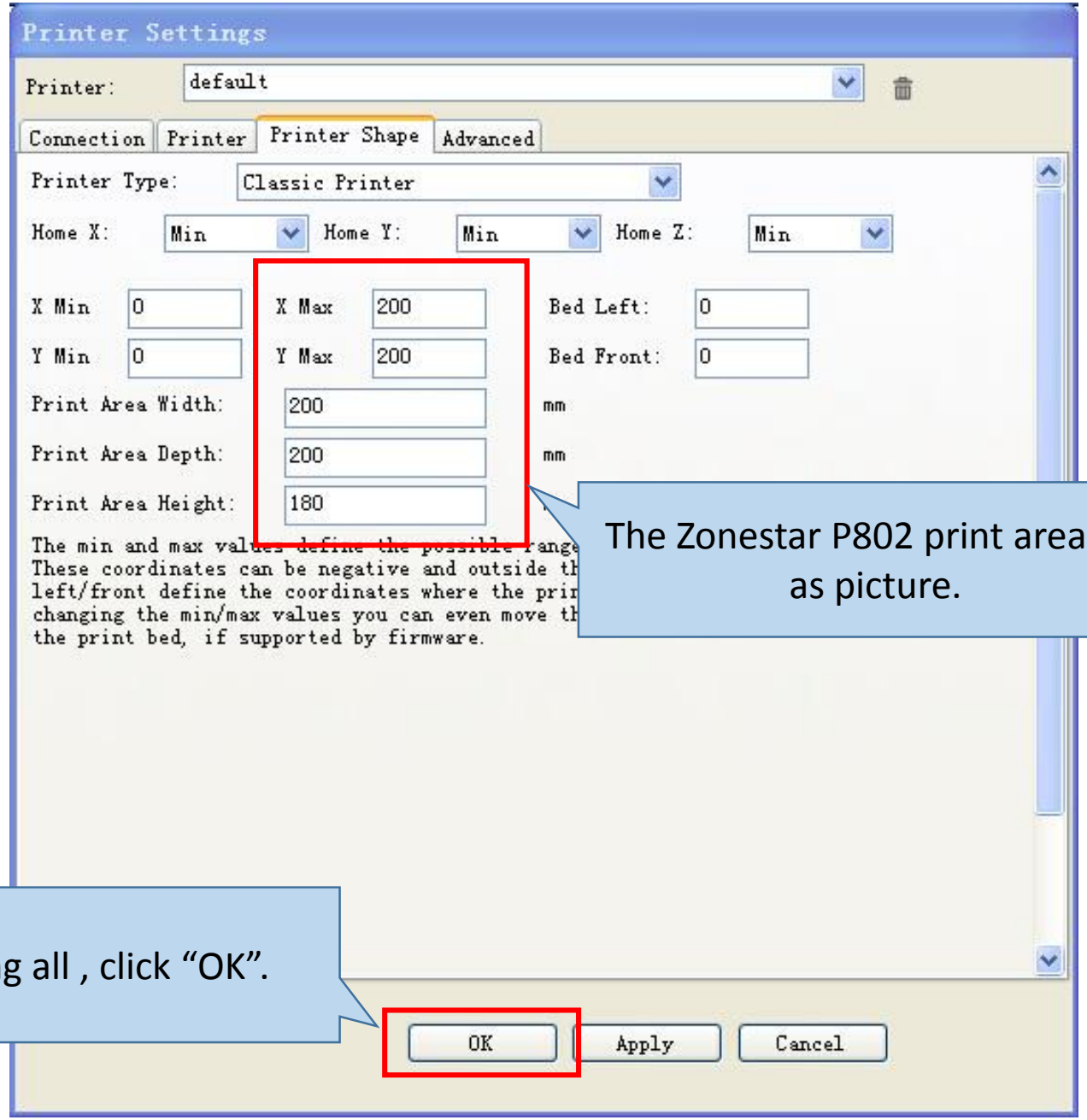
If you don’t know the port number, Right click “My computer”- “ Management”- “Device manager”-“Port” ,find the number.)



Set the temperature according to material ABS or PLA.

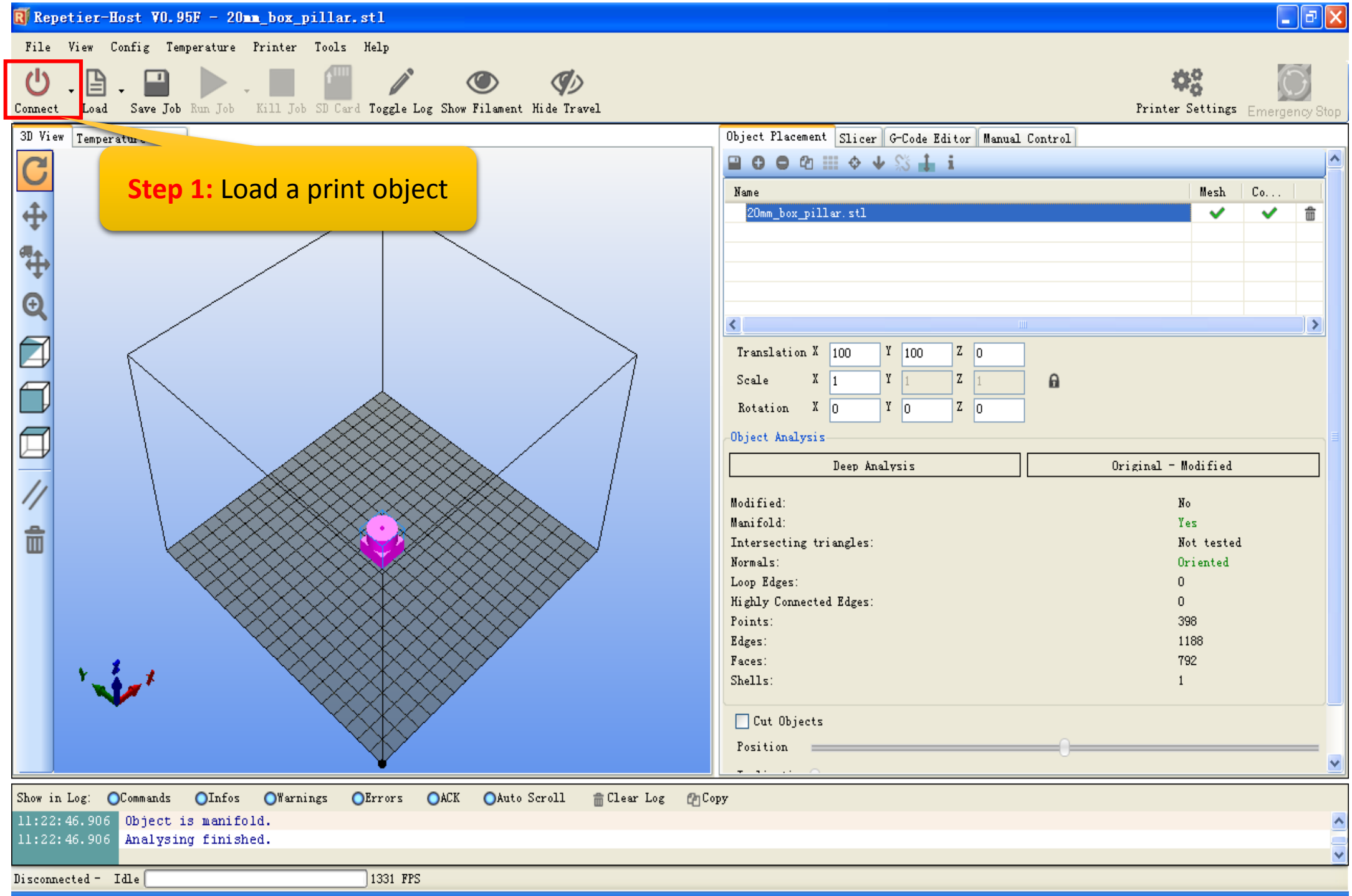
Reference Temp.:
ABS extruder: **230℃**
Heated bed: **80~100℃**

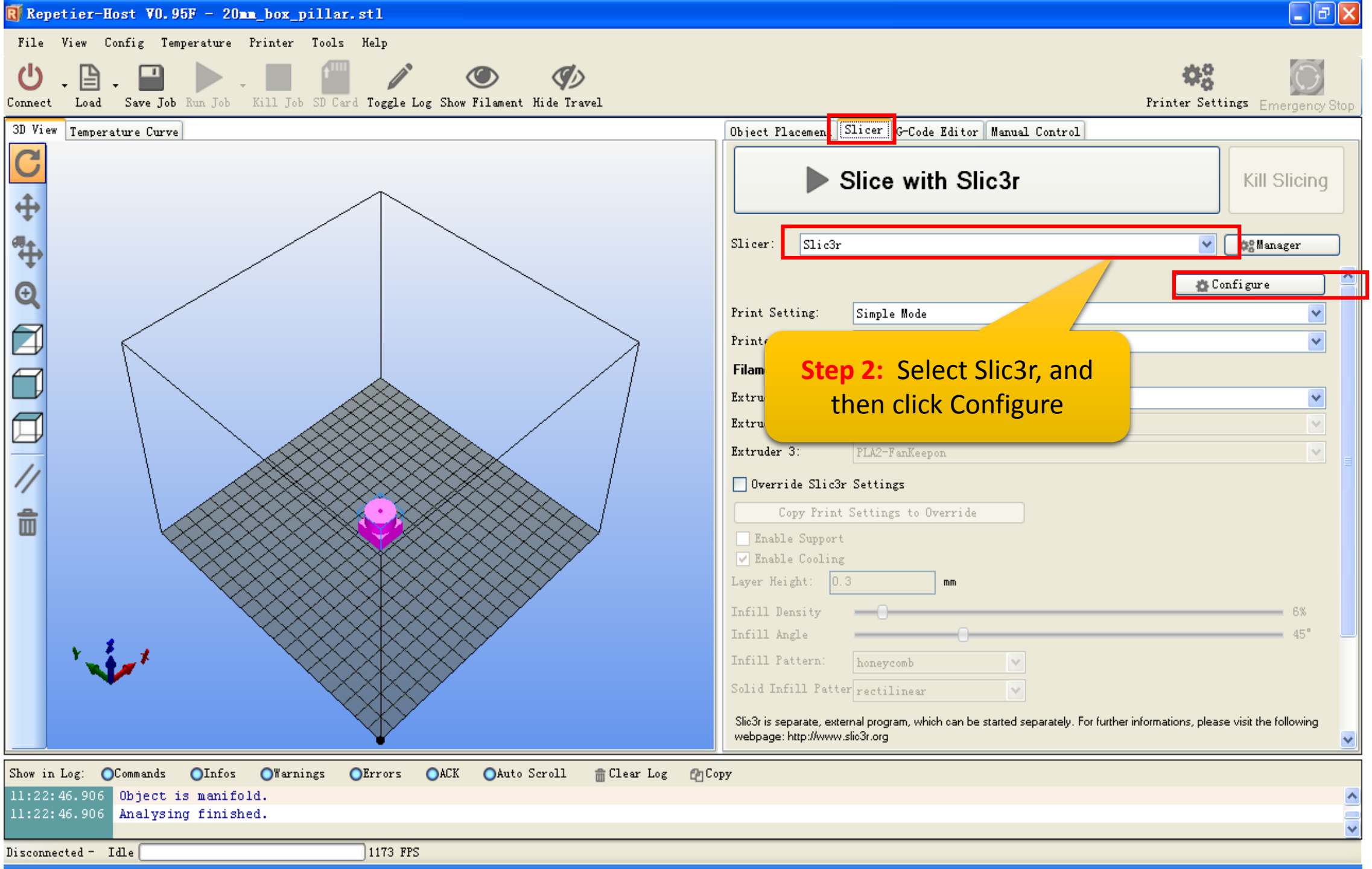
PLA extruder: **200℃**
Heated bed: **55~60℃**

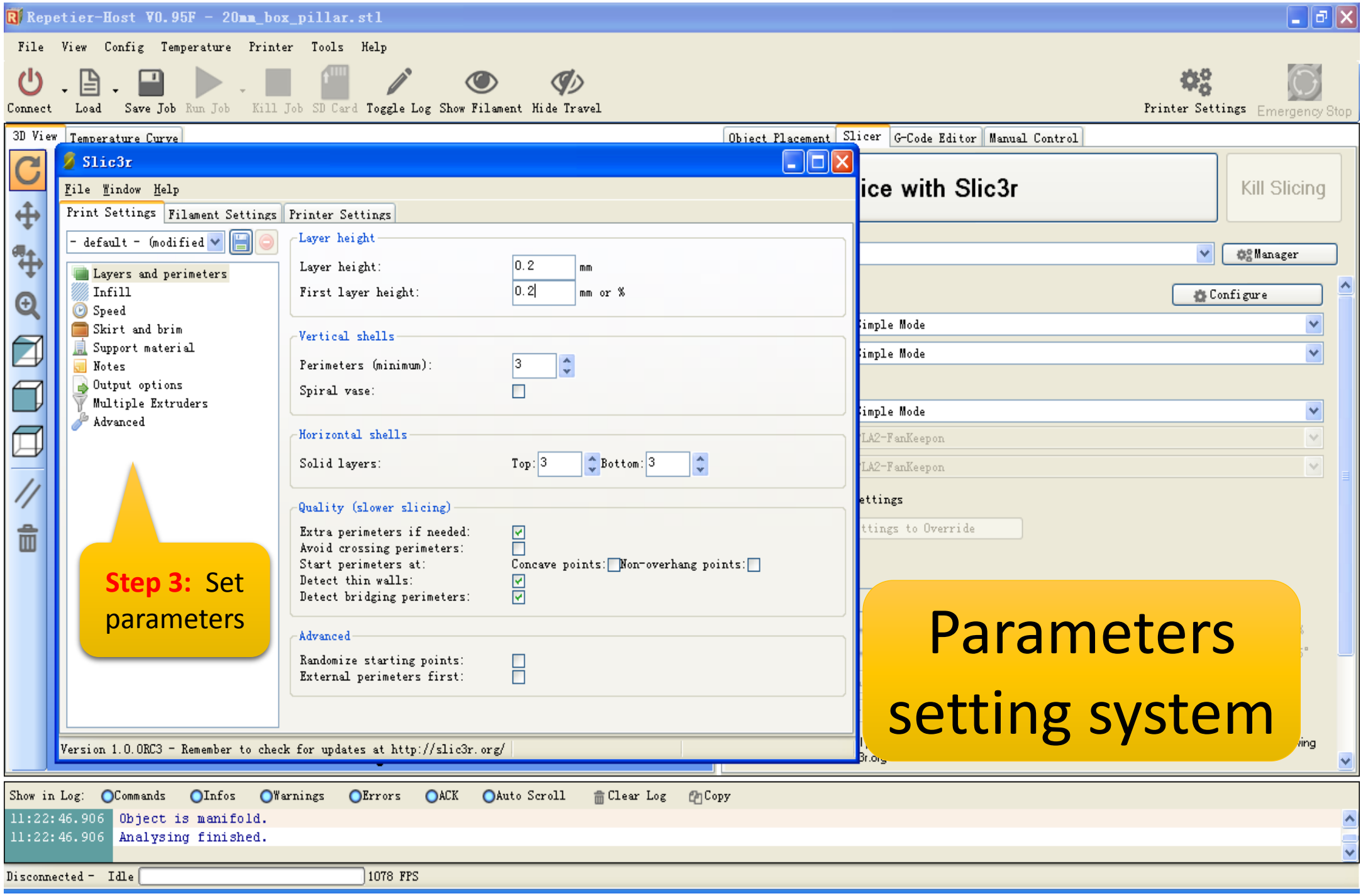


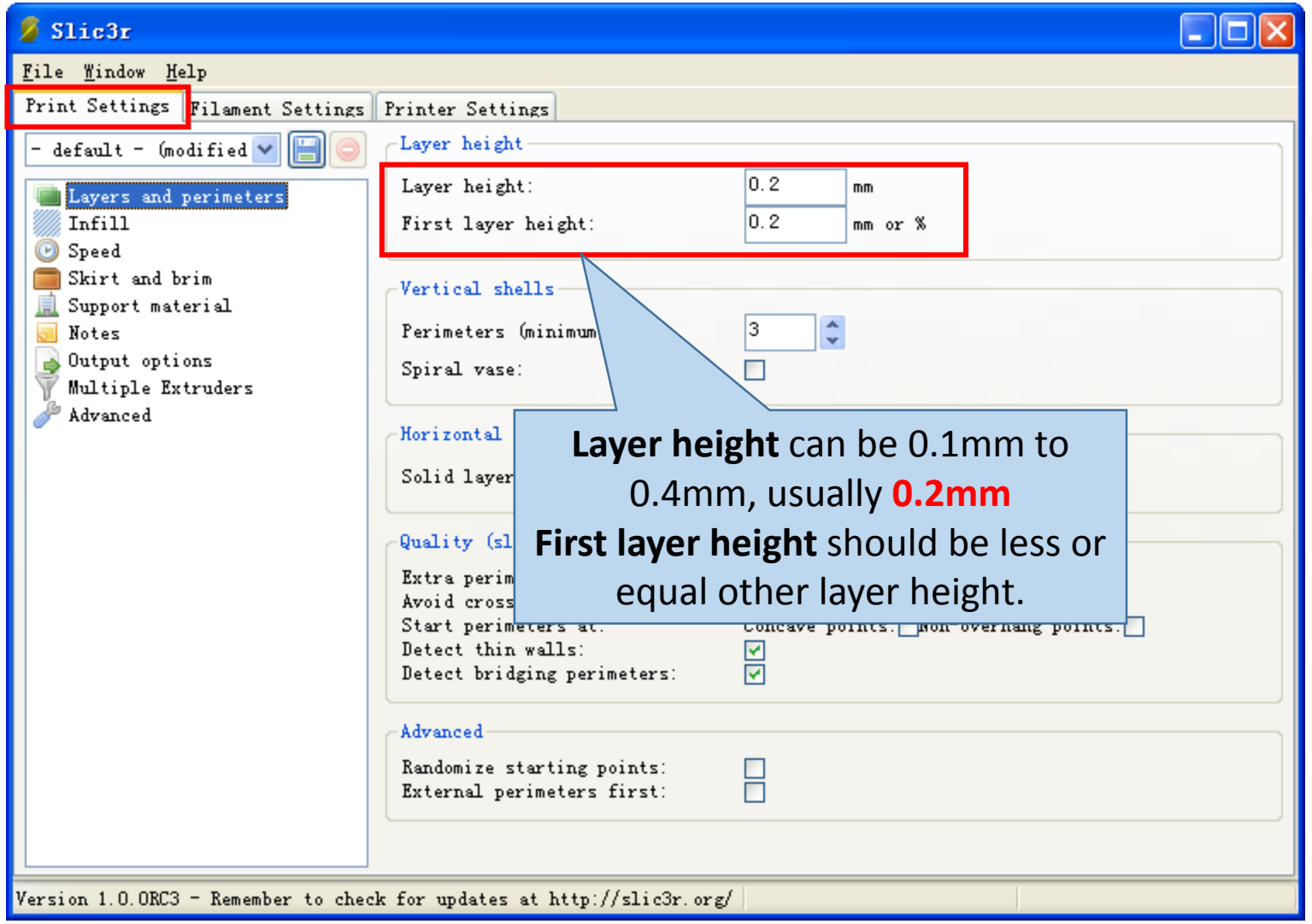
The Zonestar P802 print area set as picture.

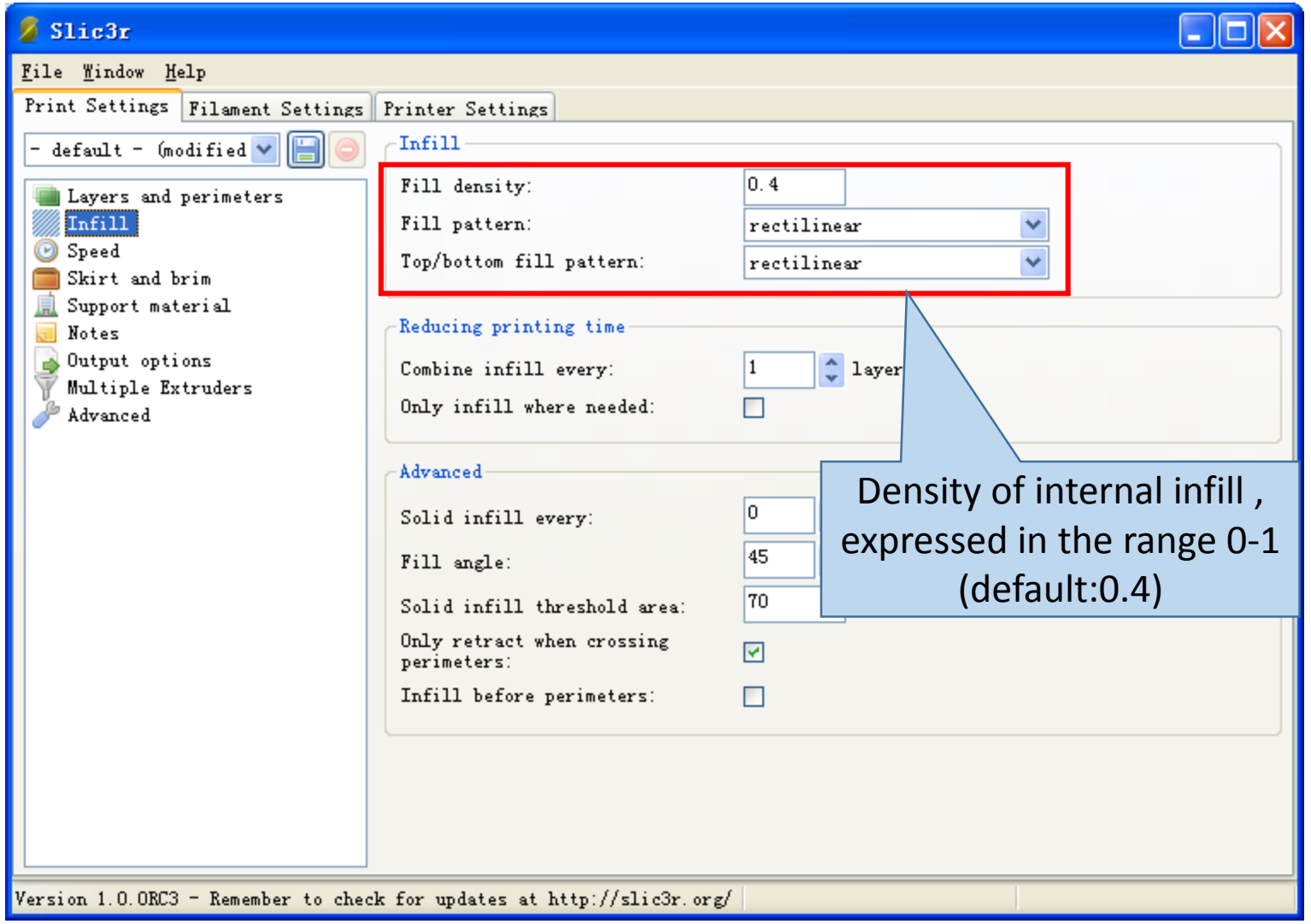
After setting all , click “OK”.

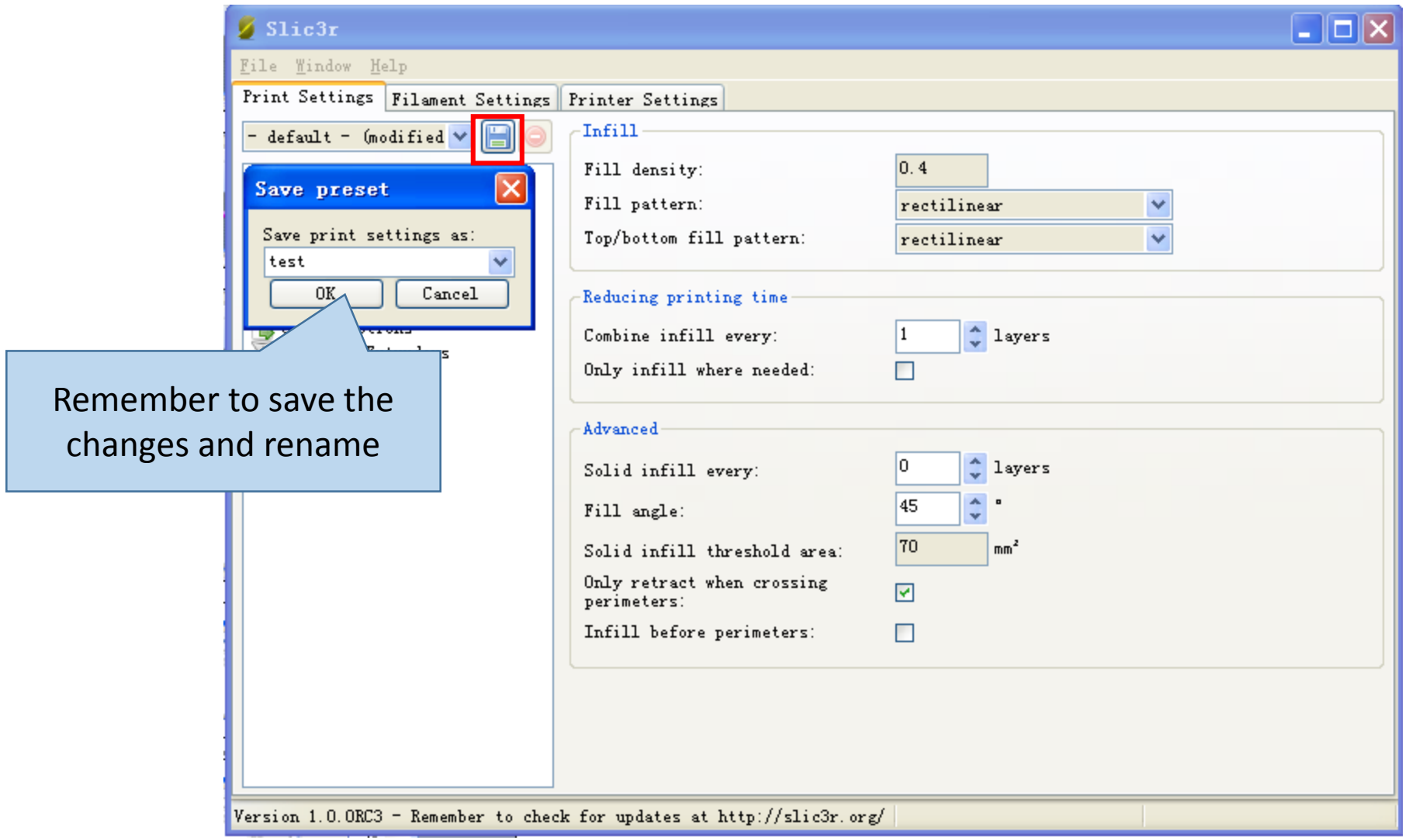

Repetier-Host V0.95F - 20mm_box_pillar.stl
File View Config Temperature Printer Tools Help
Connect Load Save Job Run Job Kill Job SD Card Toggle Log Show Filament Hide Travel
Printer Settings Emergency Stop
Step 1: Load a print object
3D View
Object Placement Slicer G-Code Editor Manual Control
Name Mesh Co...
20mm_box_pillar.stl
Translation X 100 Y 100 Z 0
Scale X 1 Y 1 Z 1
Rotation X 0 Y 0 Z 0
Object Analysis
Deep Analysis
Original - Modified
Modified: No
Manifold: Yes
Intersecting triangles: Not tested
Normals: Oriented
Loop Edges: 0
Highly Connected Edges: 0
Points: 398
Edges: 1188
Faces: 792
Shells: 1
Cut Objects
Position
Show in Log: Commands Infos Warnings Errors ACK Auto Scroll Clear Log Copy
11:22:46.906 Object is manifold.
11:22:46.906 Analysing finished.
Disconnected - Idle
1331 FPS

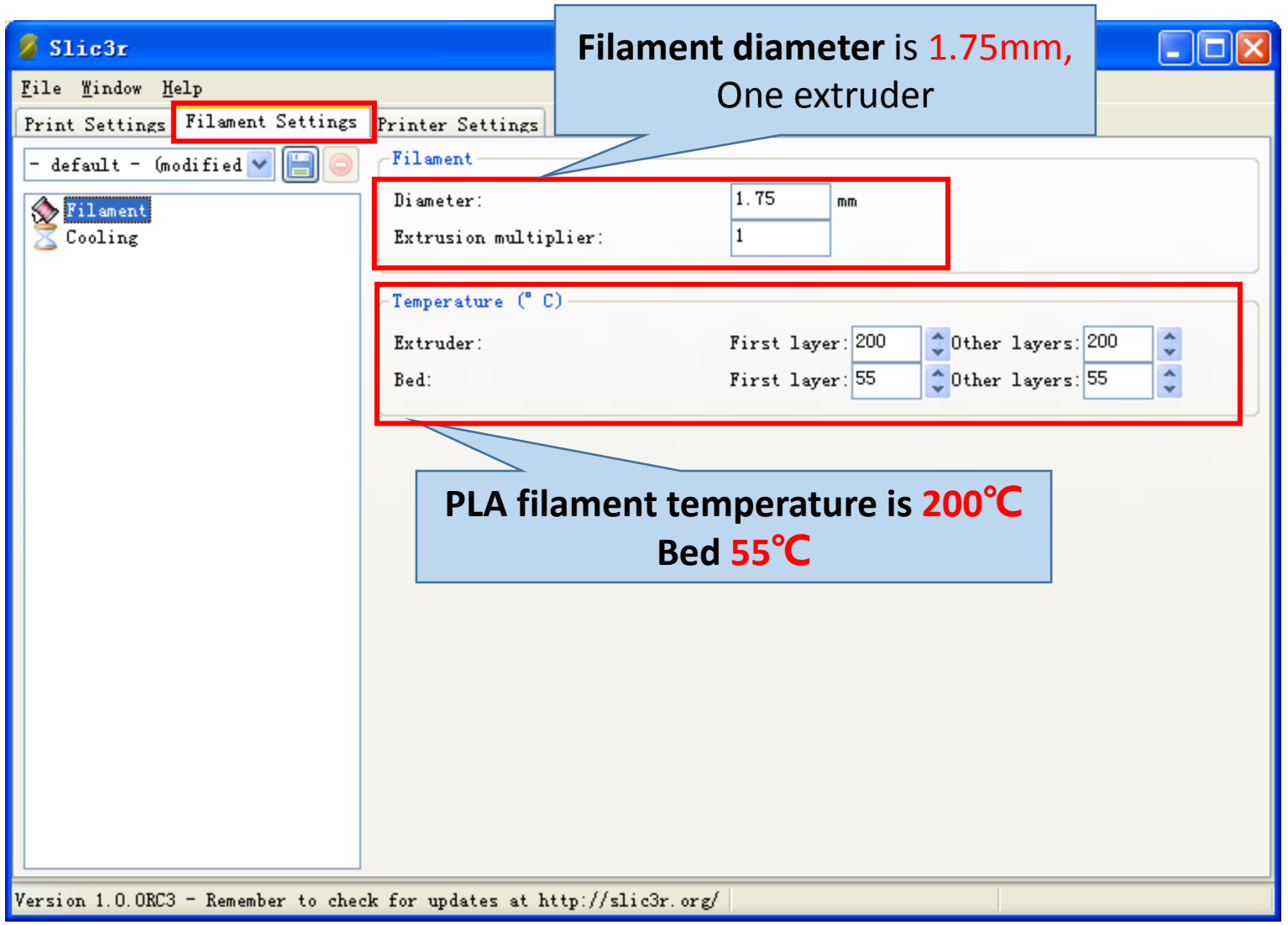

Repetier-Host V0.95F - 20mm_box_pillar.stl
File View Config Temperature Printer Tools Help
Connect Load Save Job Run Job Kill Job SD Card Toggle Log Show Filament Hide Travel
Printer Settings Emergency Stop
3D View Temperature Curve
Object Placemen Slicer G-Code Editor Manual Control
Slice with Slic3r
Kill Slicing
Slicer: Slic3r
Manager
Configure
Print Setting: Simple Mode
Extruder 3: PLA2-FanKeepon
Override Slic3r Settings
Copy Print Settings to Override
Enable Support
Enable Cooling
Layer Height: 0.3 mm
Infill Density 6%
Infill Angle 45°
Infill Pattern: honeycomb
Solid Infill Patter rectilinear
Slic3r is separate, external program, which can be started separately. For further informations, please visit the following webpage: http://www.slic3r.org
Step 2: Select Slic3r, and then click Configure
Show in Log: Commands Infos Warnings Errors ACK Auto Scroll Clear Log Copy
11:22:46.906 Object is manifold.
11:22:46.906 Analysing finished.
Disconnected - Idle 1173 FPS

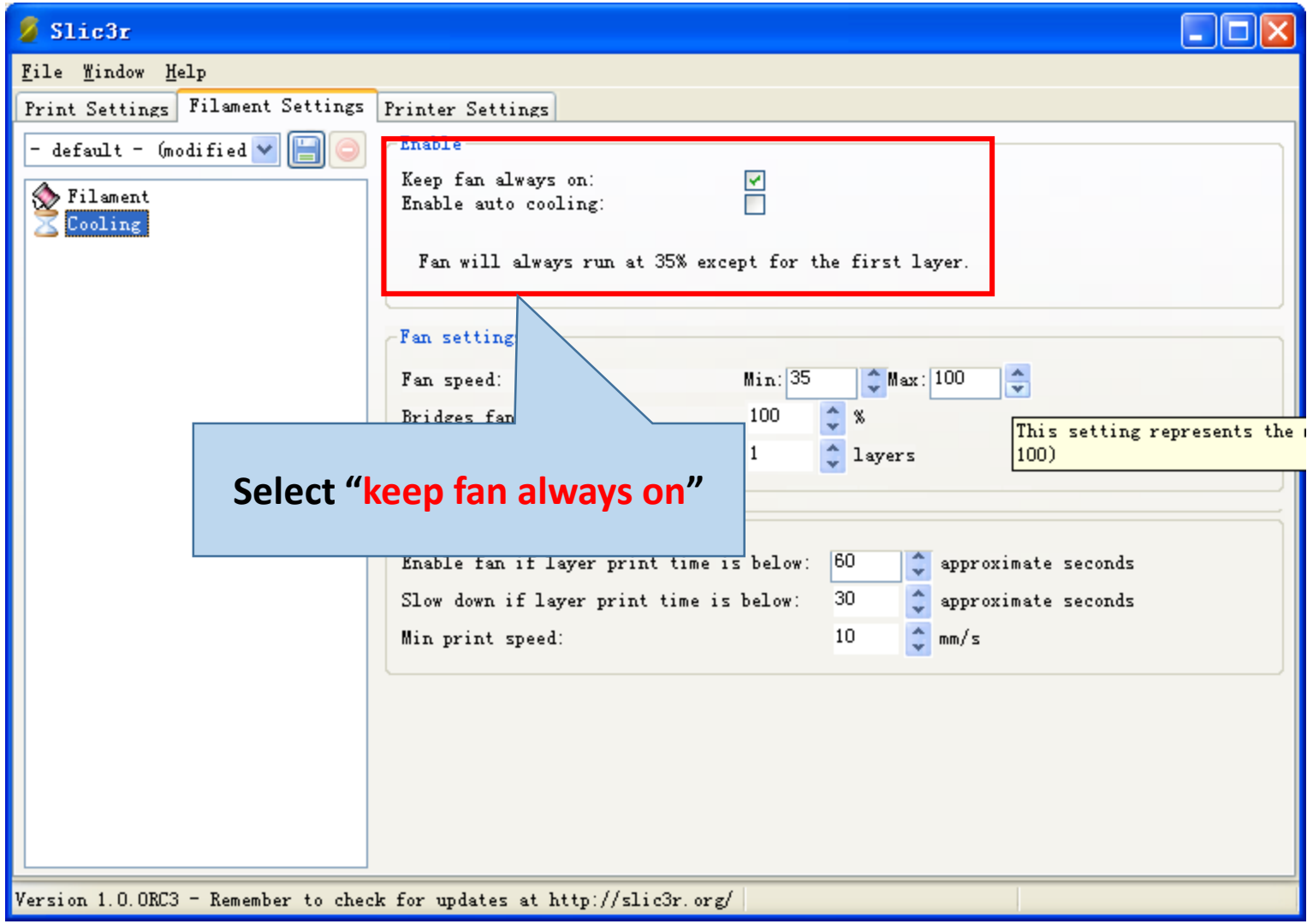

Repetier-Host V0.95F - 20mm_box_pillar.stl
File View Config Temperature Printer Tools Help
Connect Load Save Job Run Job Kill Job SD Card Toggle Log Show Filament Hide Travel
Printer Settings Emergency Stop
3D View Temperature Curve
Object Placement Slicer G-Code Editor Manual Control
Slic3r
File Window Help
Print Settings Filament Settings Printer Settings
- default - (modified
Layers and perimeters
Infill
Speed
Skirt and brim
Support material
Notes
Output options
Multiple Extruders
Advanced
Layer height
Layer height: 0.2 mm
First layer height: 0.2 mm or %
Vertical shells
Perimeters (minimum): 3
Spiral vase:
Horizontal shells
Solid layers: Top: 3 Bottom: 3
Quality (slower slicing)
Extra perimeters if needed:
Avoid crossing perimeters:
Start perimeters at: Concave points: Non-overhang points:
Detect thin walls:
Detect bridging perimeters:
Advanced
Randomize starting points:
External perimeters first:
Version 1.0.0RC3 - Remember to check for updates at http://slic3r.org/
ice with Slic3r
Kill Slicing
Manager
Configure
Simple Mode
Simple Mode
Simple Mode
LA2-FanKeepon
LA2-FanKeepon
ettings
ttings to Override
Step 3: Set parameters
Parameters setting system
Show in Log: Commands Infos Warnings Errors ACK Auto Scroll Clear Log Copy
11:22:46.906 Object is manifold.
11:22:46.906 Analysing finished.
Disconnected - Idle 1078 FPS

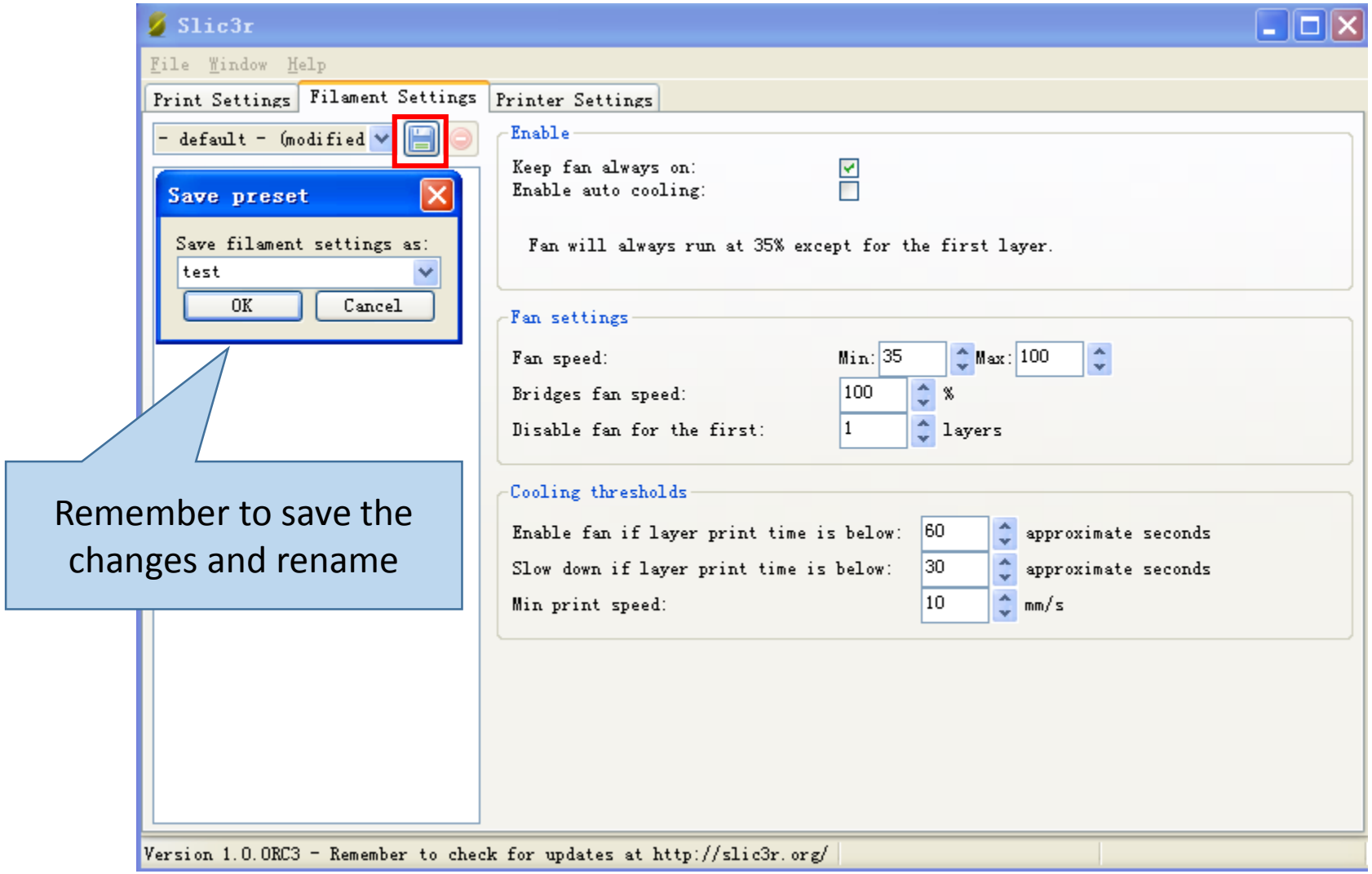

Slic3r
File Window Help
Print Settings
Filament Settings
Printer Settings
- default - (modified
Layers and perimeters
Infill
Speed
Skirt and brim
Support material
Notes
Output options
Multiple Extruders
Advanced
Layer height
Layer height: 0.2 mm
First layer height: 0.2 mm or %
Vertical shells
Perimeters (minimum 3
Spiral vase:
Horizontal
Solid layer
Quality (sl
Extra perim
Avoid cross
Start perimeters at:
Detect thin walls:
Detect bridging perimeters:
Advanced
Randomize starting points:
External perimeters first:
Layer height can be 0.1mm to 0.4mm, usually 0.2mm
First layer height should be less or equal other layer height.
Version 1.0.0RC3 - Remember to check for updates at http://slic3r.org/

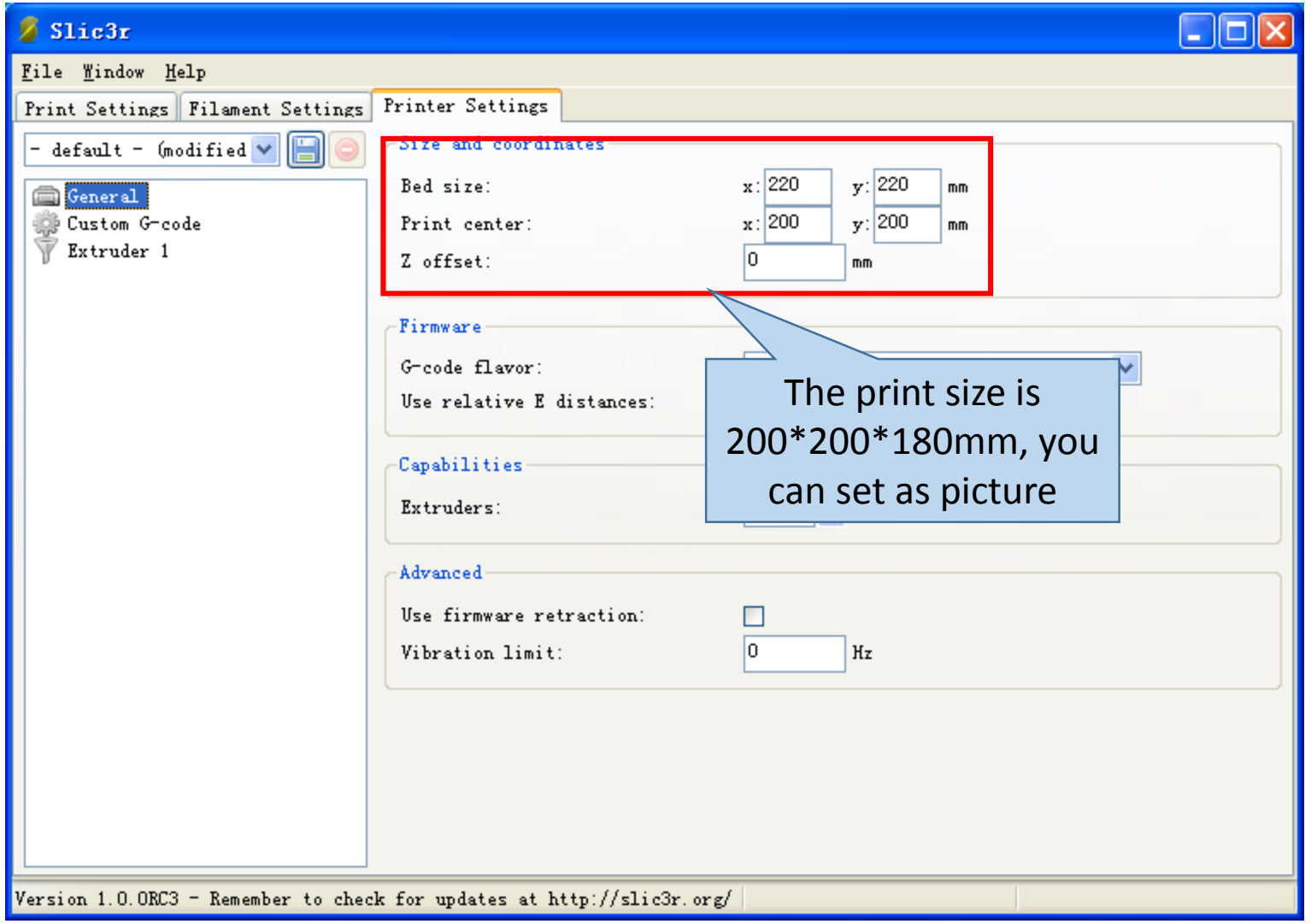

Slic3r
File Window Help
Print Settings
Filament Settings
Printer Settings
- default - (modified
Layers and perimeters
Infill
Speed
Skirt and brim
Support material
Notes
Output options
Multiple Extruders
Advanced
Infill
Fill density: 0.4
Fill pattern: rectilinear
Top/bottom fill pattern: rectilinear
Reducing printing time
Combine infill every: 1 layer
Only infill where needed:
Advanced
Solid infill every: 0
Fill angle: 45
Solid infill threshold area: 70
Only retract when crossing perimeters:
Infill before perimeters:
Density of internal infill , expressed in the range 0-1 (default:0.4)
Version 1.0.0RC3 - Remember to check for updates at http://slic3r.org/

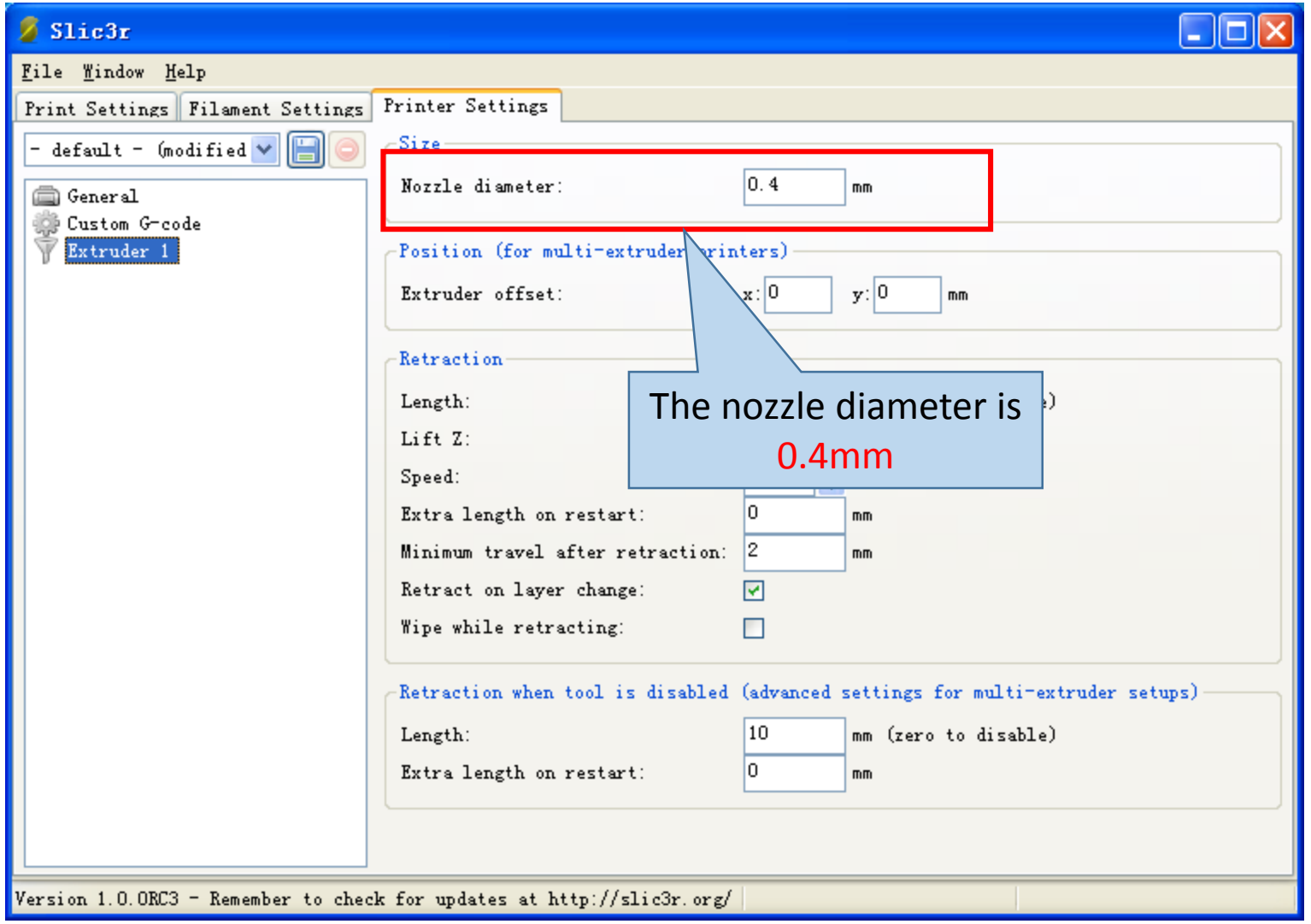

Slic3r
File Window Help
Print Settings
Filament Settings
Printer Settings
- default - (modified
Save preset
Save print settings as:
test
OK
Cancel
Infill
Fill density:
0.4
Fill pattern:
rectilinear
Top/bottom fill pattern:
rectilinear
Reducing printing time
Combine infill every:
1
layers
Only infill where needed:
Advanced
Solid infill every:
0
layers
Fill angle:
45
Solid infill threshold area:
70
mm²
Only retract when crossing perimeters:
Infill before perimeters:
Version 1.0.0RC3 - Remember to check for updates at http://slic3r.org/
Remember to save the changes and rename

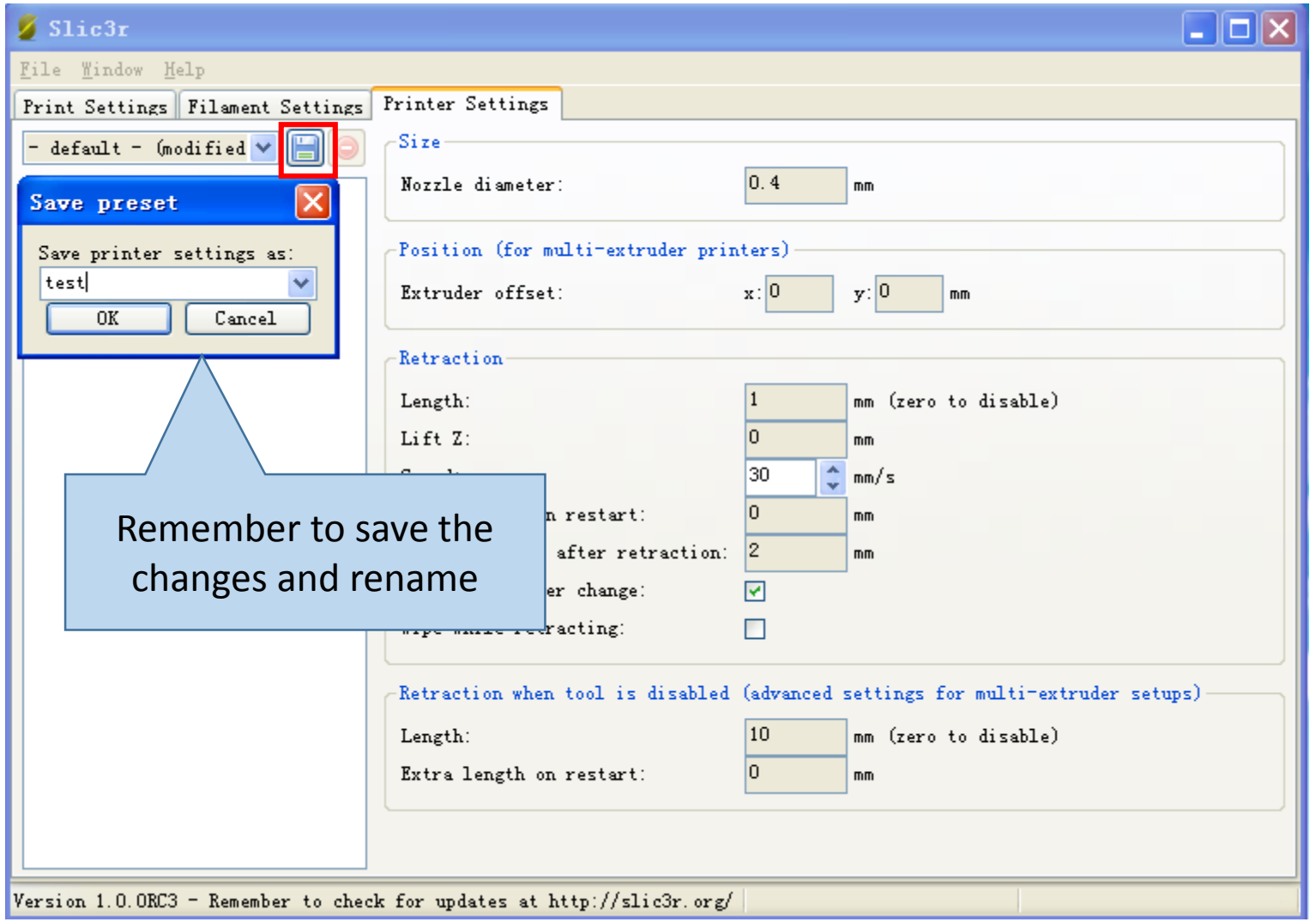

Slic3r
File Window Help
Print Settings
Filament Settings
Printer Settings
- default - (modified
Filament
Cooling
Filament
Diameter: 1.75 mm
Extrusion multiplier: 1
Temperature (° C)
Extruder: First layer: 200 Other layers: 200
Bed: First layer: 55 Other layers: 55
Version 1.0.0RC3 - Remember to check for updates at http://slic3r.org/
Filament diameter is 1.75mm,
One extruder
PLA filament temperature is 200℃
Bed 55℃

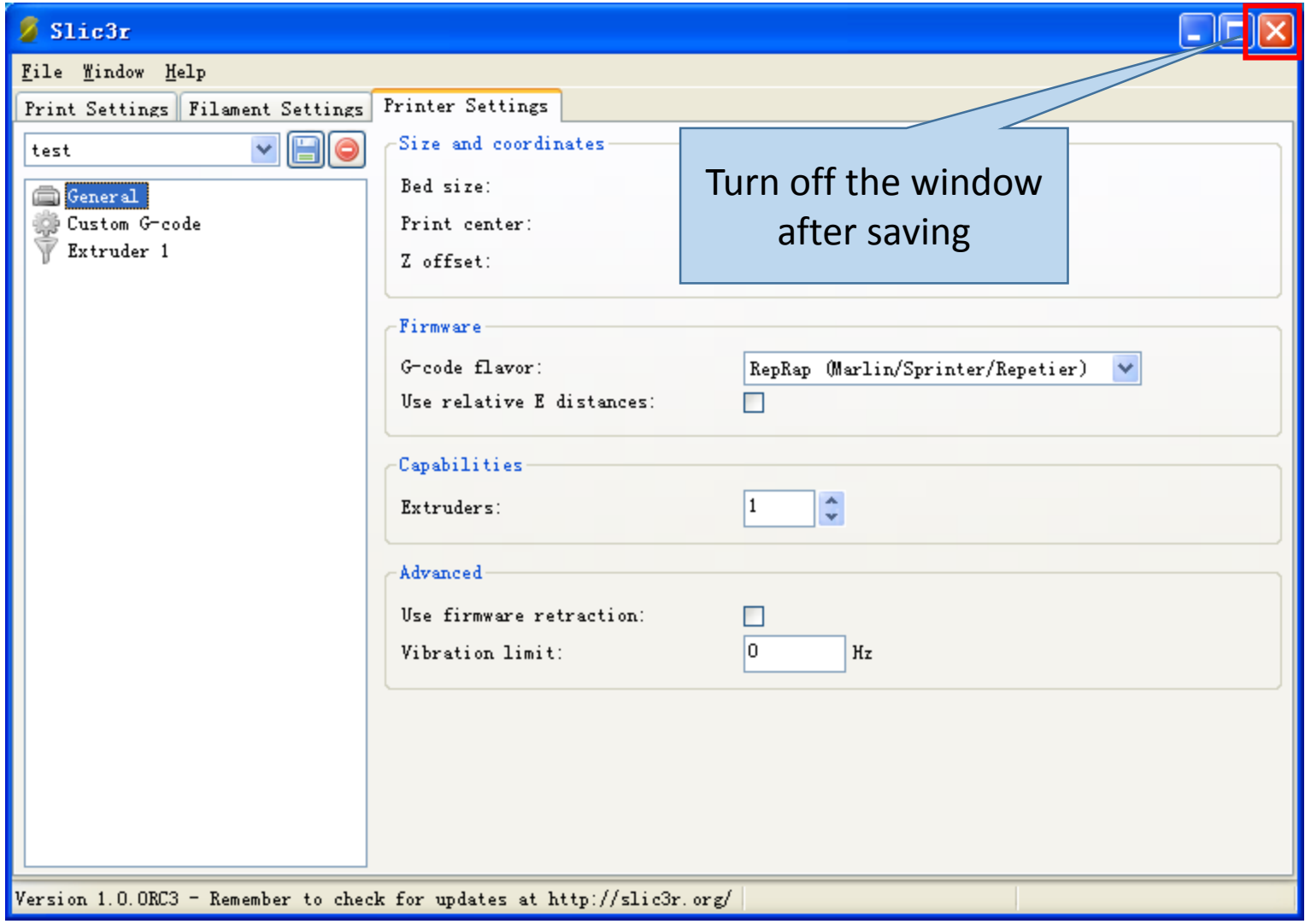

Slic3r
File Window Help
Print Settings
Filament Settings
Printer Settings
- default - (modified
Filament
Cooling
Enable
Keep fan always on:
Enable auto cooling:
Fan will always run at 35% except for the first layer.
Fan setting
Fan speed:
Min: 35
Max: 100
100
%
1
layers
This setting represents the
100)
Enable fan if layer print time is below:
60
approximate seconds
Slow down if layer print time is below:
30
approximate seconds
Min print speed:
10
mm/s
Version 1.0.0RC3 - Remember to check for updates at http://slic3r.org/
Select “keep fan always on”

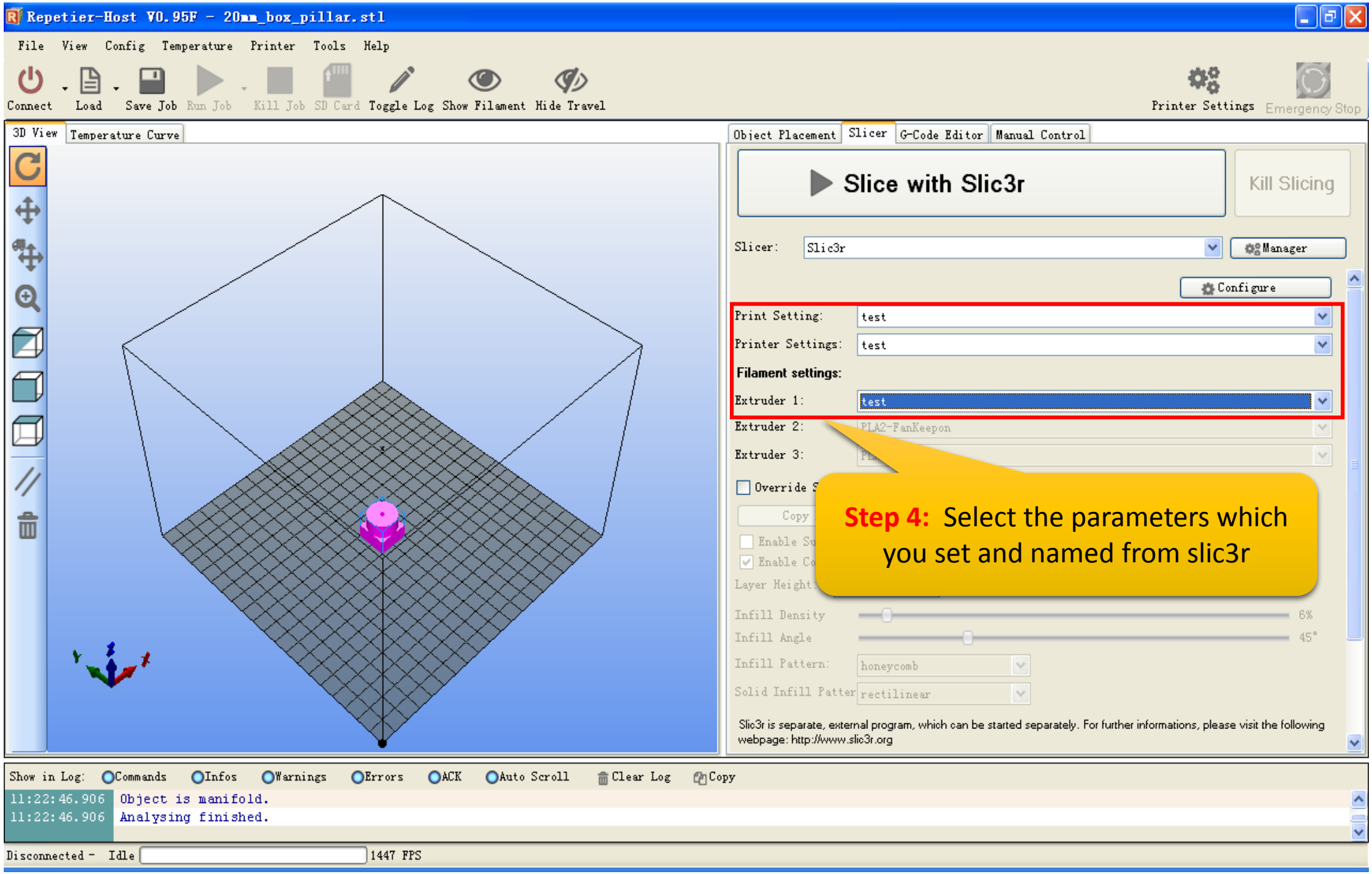

Slic3r
File Window Help
Print Settings
Filament Settings
Printer Settings
- default - (modified
Save preset
Save filament settings as:
test
OK
Cancel
Enable
Keep fan always on:
Enable auto cooling:
Fan will always run at 35% except for the first layer.
Fan settings
Fan speed:
Min: 35
Max: 100
Bridges fan speed:
100
%
Disable fan for the first:
1
layers
Cooling thresholds
Enable fan if layer print time is below:
60
approximate seconds
Slow down if layer print time is below:
30
approximate seconds
Min print speed:
10
mm/s
Version 1.0.0RC3 - Remember to check for updates at http://slic3r.org/
Remember to save the changes and rename

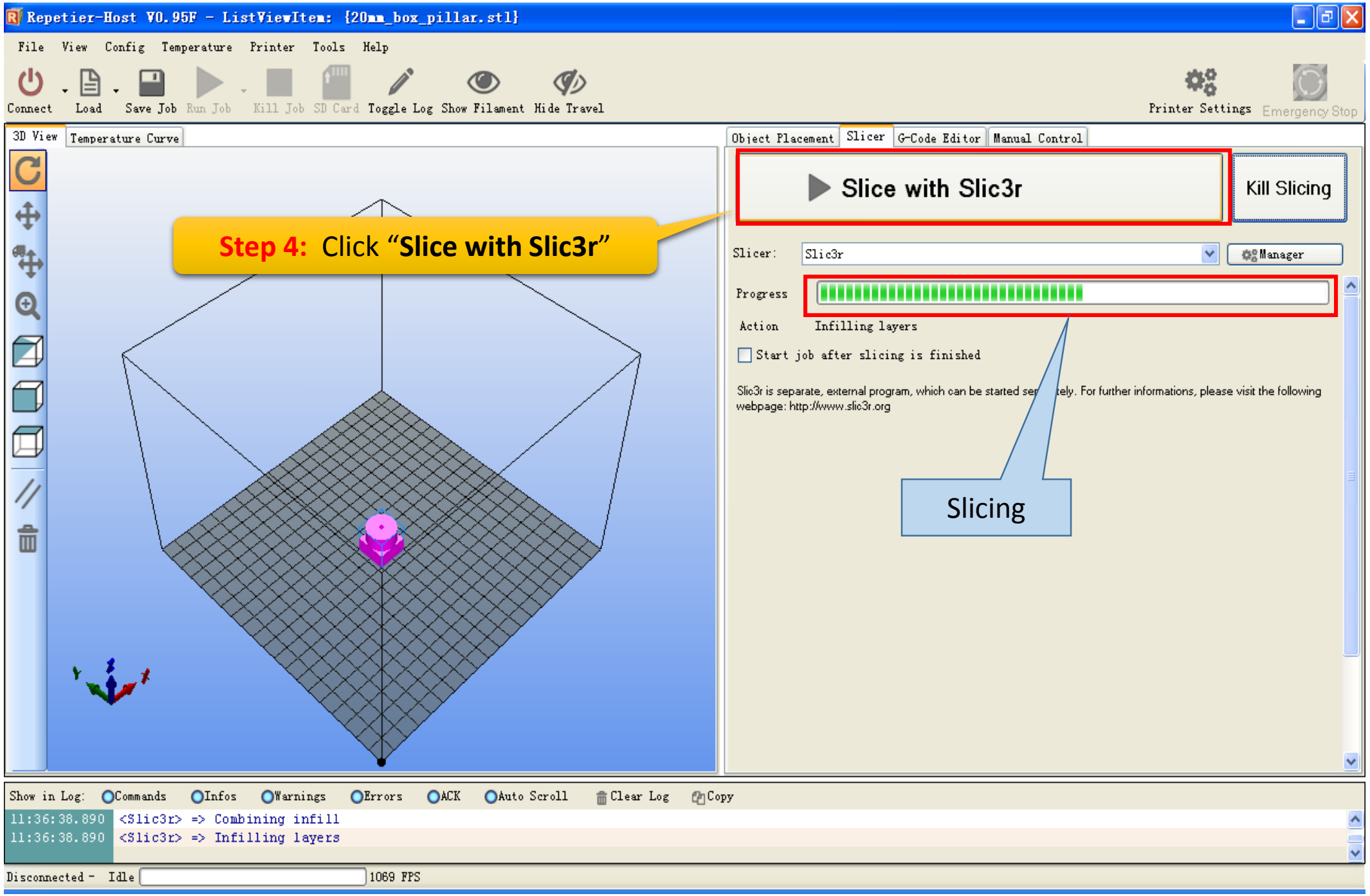

Slic3r
File Window Help
Print Settings
Filament Settings
Printer Settings
- default - (modified
General
Custom G-code
Extruder 1
Size and coordinates
Bed size: x: 220 y: 220 mm
Print center: x: 200 y: 200 mm
Z offset: 0 mm
Firmware
G-code flavor:
Use relative E distances:
Capabilities
Extruders:
Advanced
Use firmware retraction:
Vibration limit: 0 Hz
The print size is 200*200*180mm, you can set as picture
Version 1.0.0RC3 - Remember to check for updates at http://slic3r.org/

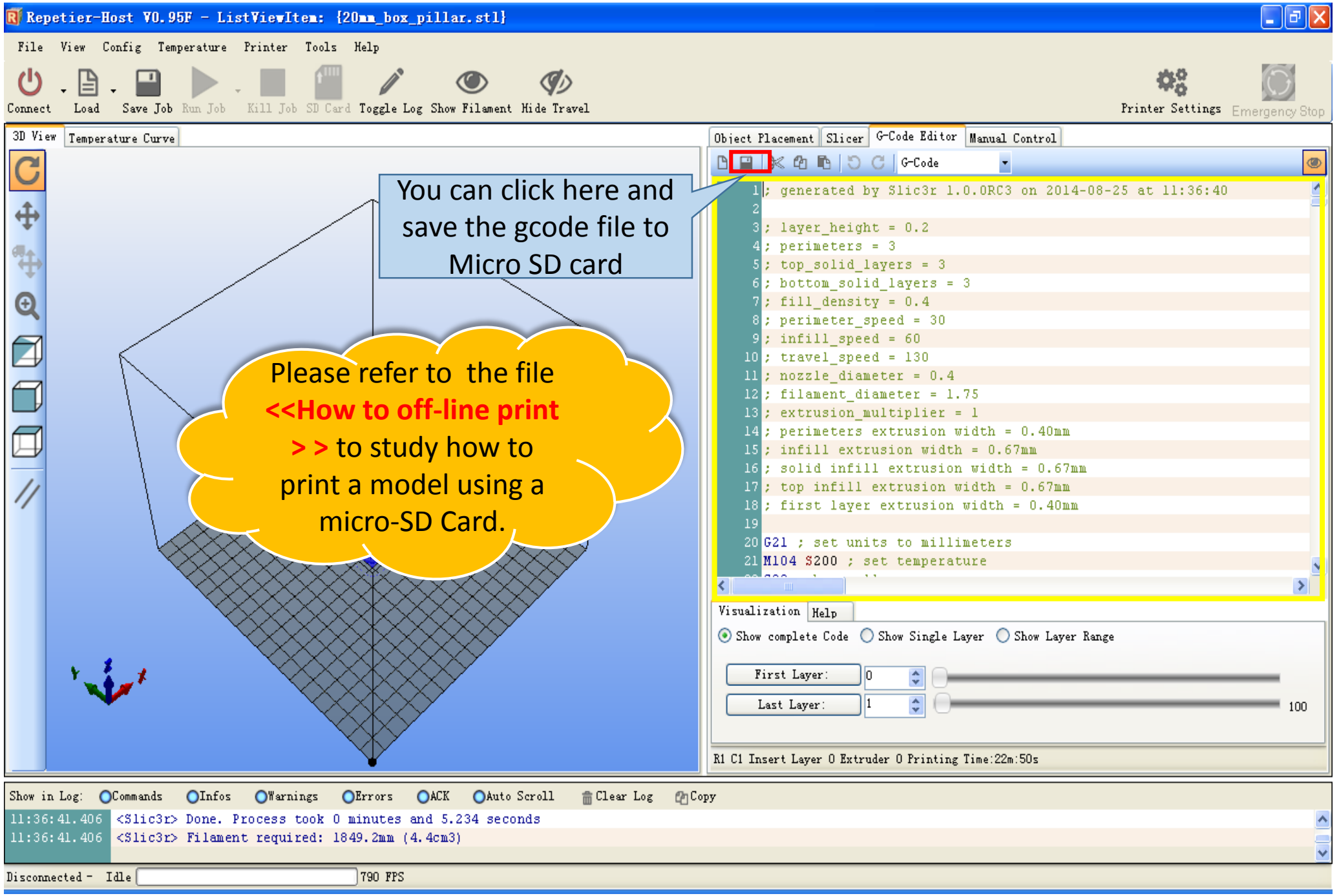

Slic3r
File Window Help
Print Settings
Filament Settings
Printer Settings
- default - (modified
General
Custom G-code
Extruder 1
Size
Nozzle diameter: 0.4 mm
Position (for multi-extruder printers)
Extruder offset: x: 0 y: 0 mm
Retraction
Length:
Lift Z:
Speed:
Extra length on restart: 0 mm
Minimum travel after retraction: 2 mm
Retract on layer change:
Wipe while retracting:
Retraction when tool is disabled (advanced settings for multi-extruder setups)
Length: 10 mm (zero to disable)
Extra length on restart: 0 mm
The nozzle diameter is
0.4mm
Version 1.0.0RC3 - Remember to check for updates at http://slic3r.org/

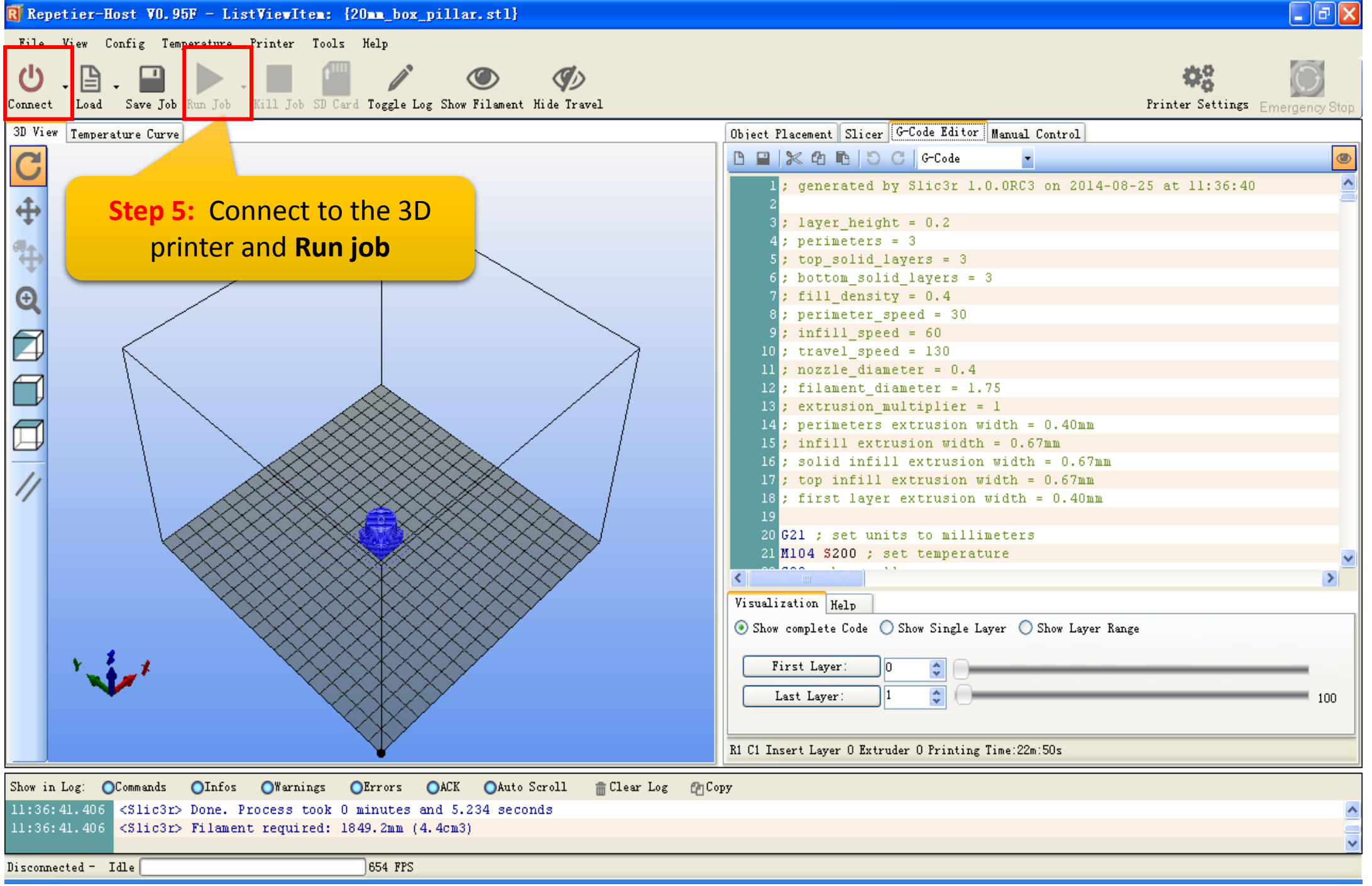

Slic3r
File Window Help
Print Settings
Filament Settings
Printer Settings
- default - (modified
Save preset
Save printer settings as:
test
OK
Cancel
Remember to save the changes and rename
Size
Nozzle diameter:
0.4
mm
Position (for multi-extruder printers)
Extruder offset:
x: 0
y: 0
mm
Retraction
Length:
1
mm (zero to disable)
Lift Z:
0
mm
30
mm/s
0
mm
2
mm
Retraction when tool is disabled (advanced settings for multi-extruder setups)
Length:
10
mm (zero to disable)
Extra length on restart:
0
mm
Version 1.0.0RC3 - Remember to check for updates at http://slic3r.org/


Slic3r
File Window Help
Print Settings
Filament Settings
Printer Settings
test
General
Custom G-code
Extruder 1
Size and coordinates
Bed size:
Print center:
Z offset:
Turn off the window after saving
Firmware
G-code flavor:
RepRap (Marlin/Sprinter/Repetier)
Use relative E distances:
Capabilities
Extruders:
1
Advanced
Use firmware retraction:
Vibration limit:
0
Hz
Version 1.0.0RC3 - Remember to check for updates at http://slic3r.org/


Repetier-Host V0.95F - 20mm_box_pillar.stl
File View Config Temperature Printer Tools Help
Connect Load Save Job Run Job Kill Job SD Card Toggle Log Show Filament Hide Travel
Printer Settings Emergency Stop
3D View Temperature Curve
Object Placement Slicer G-Code Editor Manual Control
Slice with Slic3r
Kill Slicing
Slicer: Slic3r
Manager
Configure
Print Setting: test
Printer Settings: test
Filament settings:
Extruder 1: test
Extruder 2: PLA2-FanKeepon
Extruder 3:
Override S
Copy
Enable Su
Enable Co
Layer Height
Infill Density 6%
Infill Angle 45°
Infill Pattern: honeycomb
Solid Infill Patter rectilinear
Slic3r is separate, external program, which can be started separately. For further informations, please visit the following webpage: http://www.slic3r.org
Step 4: Select the parameters which you set and named from slic3r
Show in Log: Commands Infos Warnings Errors ACK Auto Scroll Clear Log Copy
11:22:46.906 Object is manifold.
11:22:46.906 Analysing finished.
Disconnected - Idle 1447 FPS


Repetier-Host V0.95F - ListViewItem: {20mm_box_pillar.stl}
File View Config Temperature Printer Tools Help
Connect Load Save Job Run Job Kill Job SD Card Toggle Log Show Filament Hide Travel
Printer Settings Emergency Stop
3D View Temperature Curve
Step 4: Click "Slice with Slic3r"
Object Placement Slicer G-Code Editor Manual Control
Slice with Slic3r
Kill Slicing
Slicer: Slic3r
Manager
Progress
Action Infilling layers
Start job after slicing is finished
Slic3r is separate, external program, which can be started ser tely. For further informations, please visit the following webpage: http://www.slic3r.org
Slicing
Show in Log: Commands Infos Warnings Errors ACK Auto Scroll Clear Log Copy
11:36:38.890 <Slic3r> => Combining infill
11:36:38.890 <Slic3r> => Infilling layers
Disconnected - Idle
1069 FPS


Repetier-Host V0.95F - ListViewItem: {20mm_box_pillar.stl}
File View Config Temperature Printer Tools Help
Connect Load Save Job Run Job Kill Job SD Card Toggle Log Show Filament Hide Travel
Printer Settings Emergency Stop
3D View Temperature Curve
You can click here and save the gcode file to Micro SD card
Please refer to the file <<How to off-line print > > to study how to print a model using a micro-SD Card.
Object Placement Slicer G-Code Editor Manual Control
G-Code
1 ; generated by Slic3r 1.0.0RC3 on 2014-08-25 at 11:36:40
2
3 ; layer_height = 0.2
4 ; perimeters = 3
5 ; top_solid_layers = 3
6 ; bottom_solid_layers = 3
7 ; fill_density = 0.4
8 ; perimeter_speed = 30
9 ; infill_speed = 60
10 ; travel_speed = 130
11 ; nozzle_diameter = 0.4
12 ; filament_diameter = 1.75
13 ; extrusion_multiplier = 1
14 ; perimeters extrusion width = 0.40mm
15 ; infill extrusion width = 0.67mm
16 ; solid infill extrusion width = 0.67mm
17 ; top infill extrusion width = 0.67mm
18 ; first layer extrusion width = 0.40mm
19
20 G21 ; set units to millimeters
21 M104 S200 ; set temperature
Visualization Help
Show complete Code Show Single Layer Show Layer Range
First Layer: 0
Last Layer: 1
100
R1 C1 Insert Layer 0 Extruder 0 Printing Time:22m:50s
Show in Log: Commands Infos Warnings Errors ACK Auto Scroll Clear Log Copy
11:36:41.406 <Slic3r> Done. Process took 0 minutes and 5.234 seconds
11:36:41.406 <Slic3r> Filament required: 1849.2mm (4.4cm3)
Disconnected - Idle
790 FPS


Repetier-Host V0.95F - ListViewItem: {20mm_box_pillar.stl}
File View Config Temperature Printer Tools Help
Connect
Load
Save Job
Run Job
Kill Job
SD Card
Toggle Log
Show Filament
Hide Travel
Printer Settings
Emergency Stop
3D View
Temperature Curve
Step 5: Connect to the 3D printer and Run job
Object Placement
Slicer
G-Code Editor
Manual Control
G-Code
; generated by Slic3r 1.0.0RC3 on 2014-08-25 at 11:36:40
; layer_height = 0.2
; perimeters = 3
; top_solid_layers = 3
; bottom_solid_layers = 3
; fill_density = 0.4
; perimeter_speed = 30
; infill_speed = 60
; travel_speed = 130
; nozzle_diameter = 0.4
; filament_diameter = 1.75
; extrusion_multiplier = 1
; perimeters extrusion width = 0.40mm
; infill extrusion width = 0.67mm
; solid infill extrusion width = 0.67mm
; top infill extrusion width = 0.67mm
; first layer extrusion width = 0.40mm
G21 ; set units to millimeters
M104 S200 ; set temperature
Visualization
Help
Show complete Code
Show Single Layer
Show Layer Range
First Layer:
Last Layer:
100
R1 C1 Insert Layer 0 Extruder 0 Printing Time:22m:50s
Show in Log:
Commands
Infos
Warnings
Errors
ACK
Auto Scroll
Clear Log
Copy
11:36:41.406 <Slic3r> Done. Process took 0 minutes and 5.234 seconds
11:36:41.406 <Slic3r> Filament required: 1849.2mm (4.4cm3)
Disconnected - Idle
654 FPS