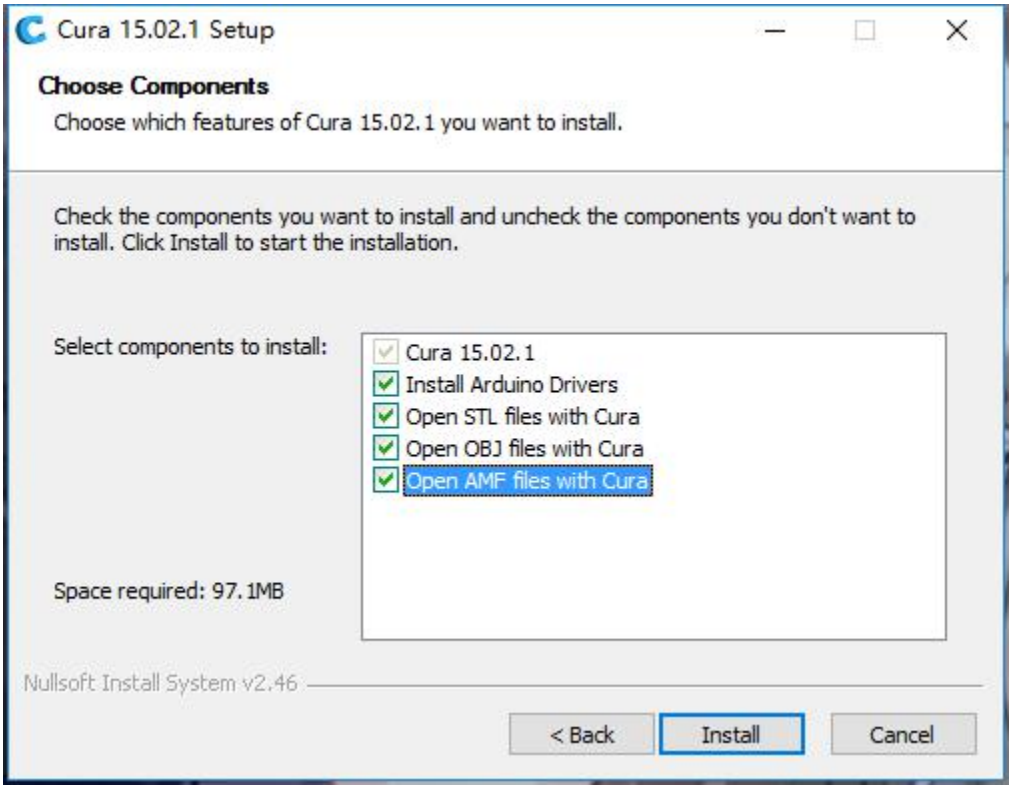
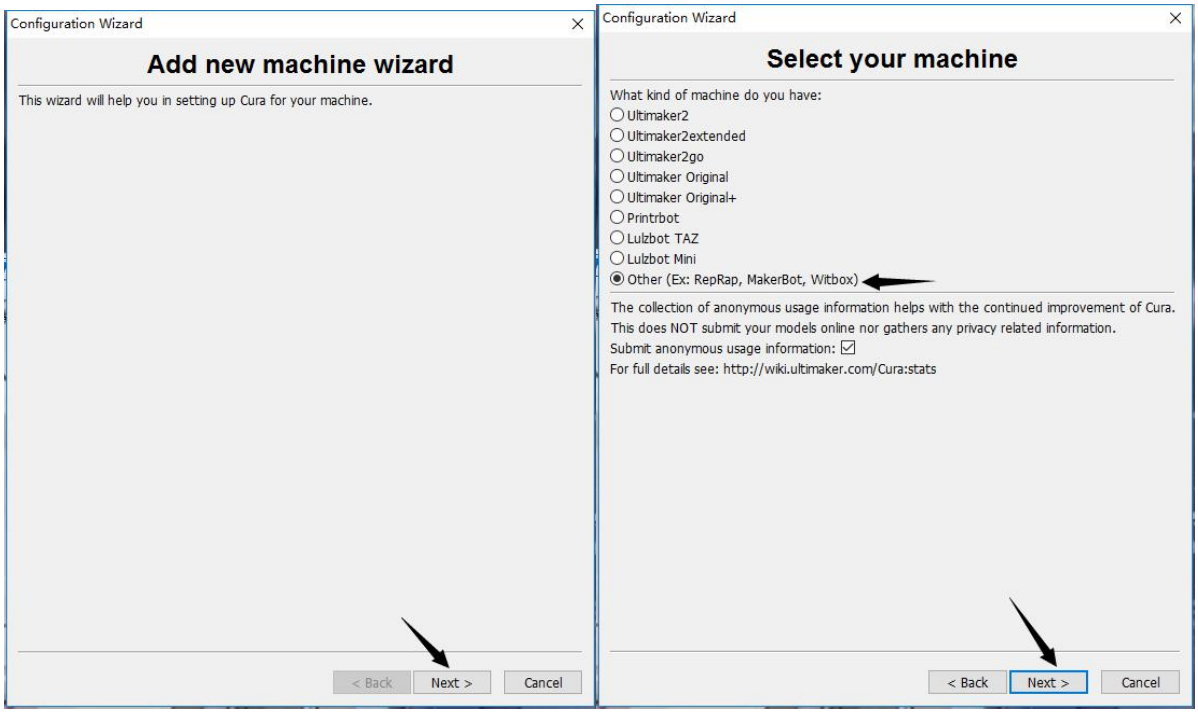
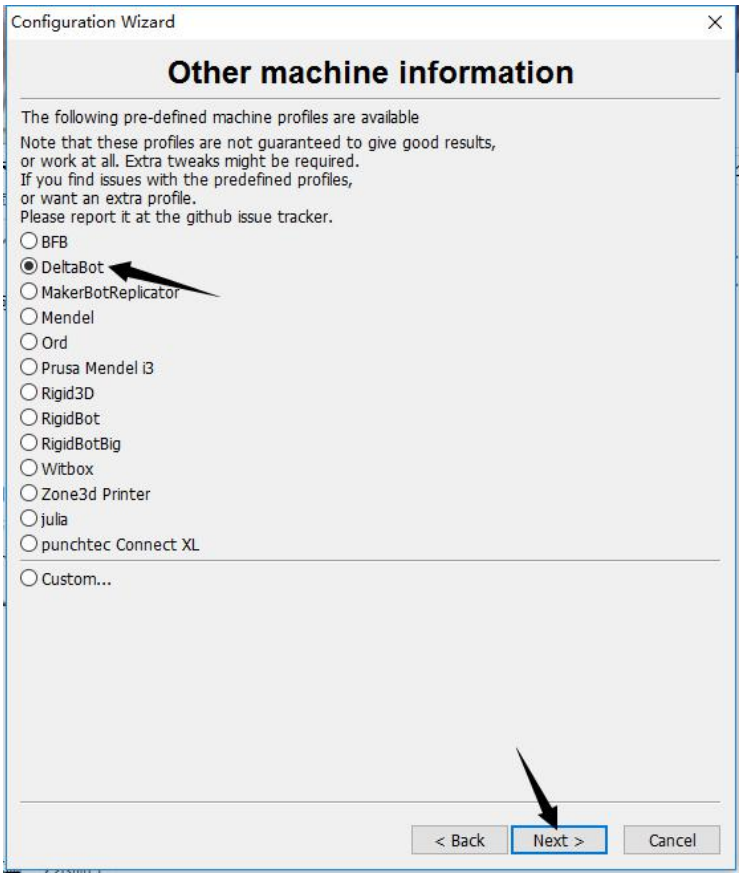
1. Printing Models

1) Installing slicer. Taking Cura for example, double click “Cura_15.04.exe” to install it. Open Cura after installation, and the startup guide dialogue box appears, as figure 6.16 shows.

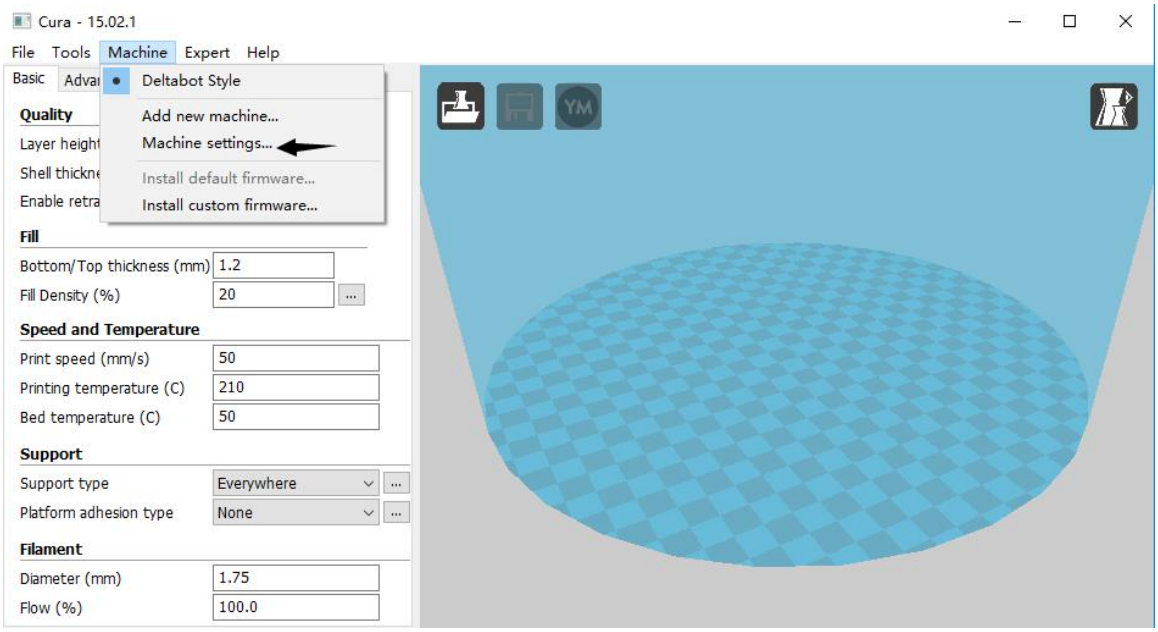


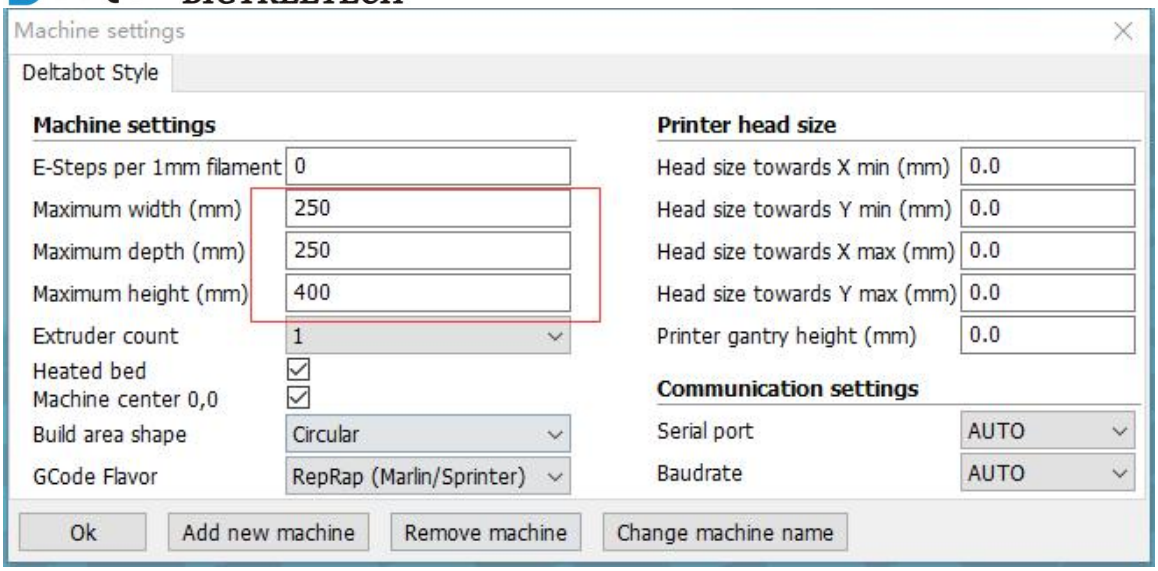

2) Select the language and click “ Next”, select the kind of machine “Other(Ex: RepRap, Makerbot, Witbox)” and click “Next”, then choose “DeltaBot” and click “Next”, “Finsh”, and appears the main interface.



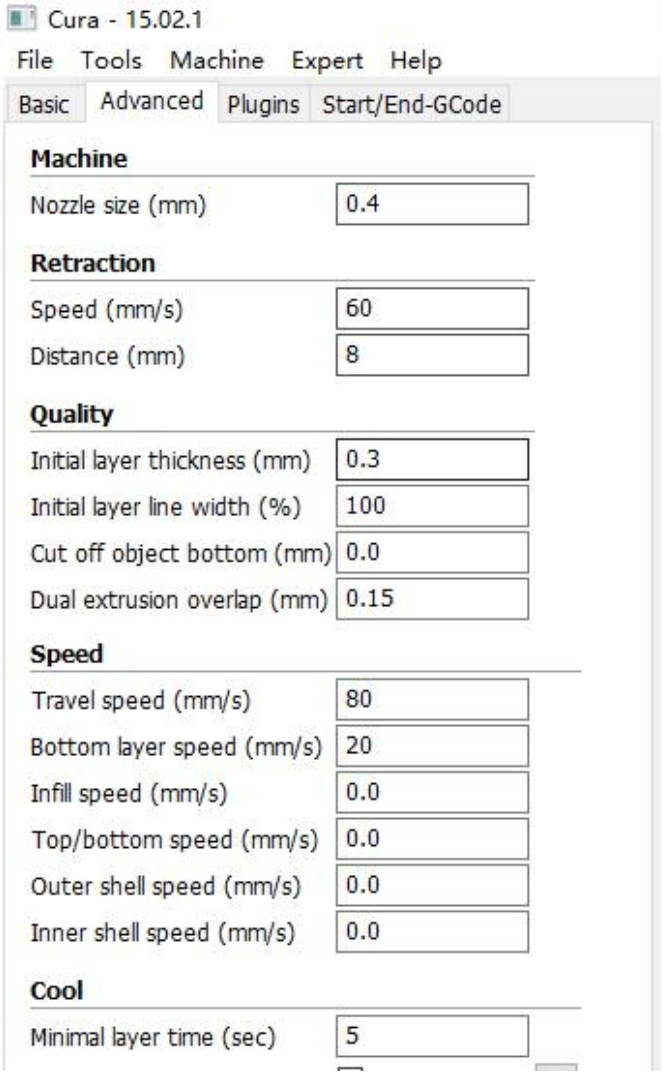


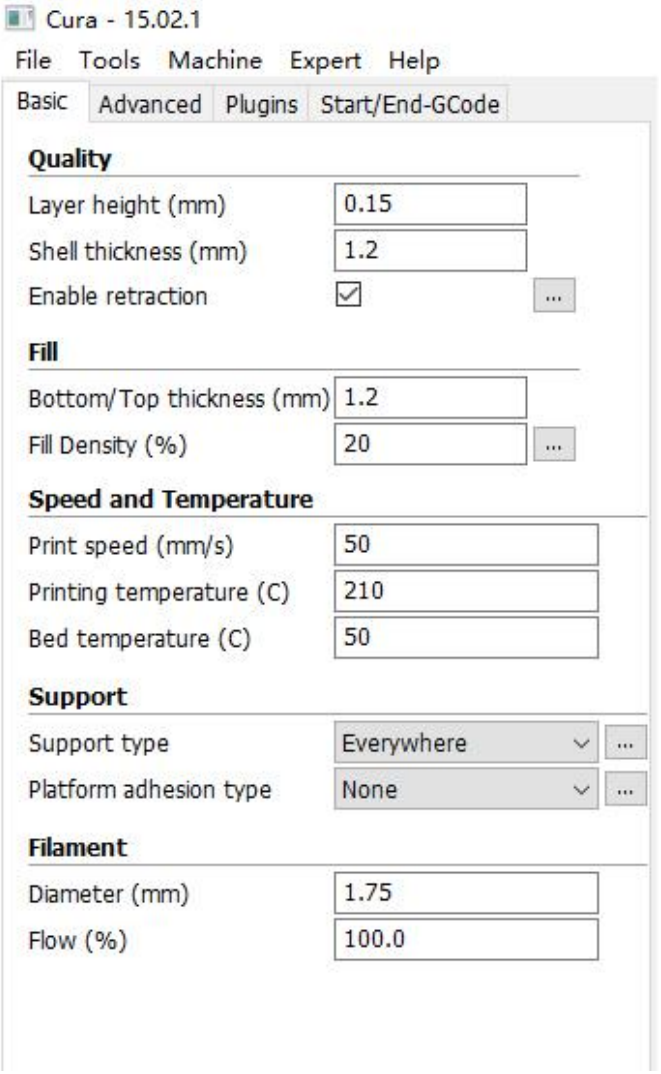

Click “ Machine” on the menu bar, and “machine settings”, change the maximum width and depth to 200, maximum height to 300, "Hot bed" need to be checked, machines have added hot bed. The serial port and baud rate can just be AUTO, or change serial port to your device, and baud rate to 250000.

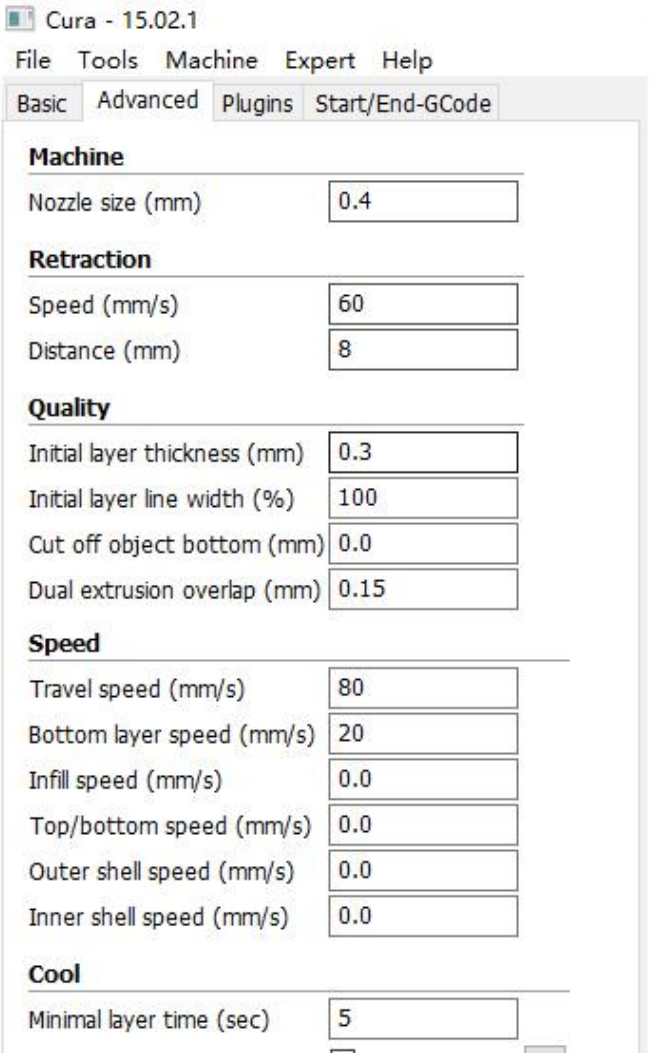


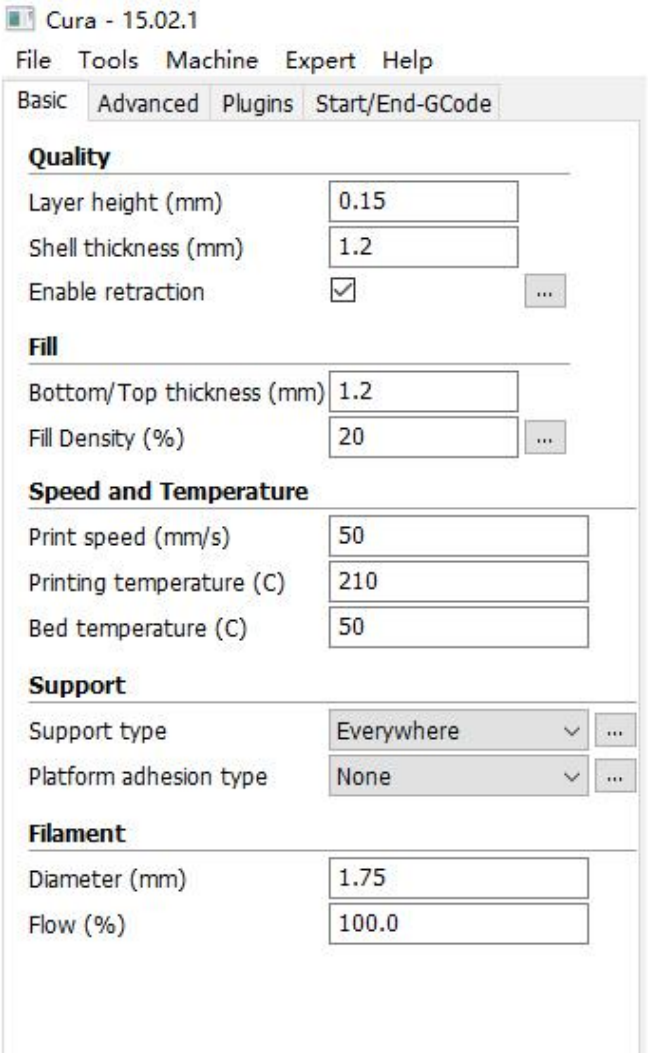



Click “OK” to back to main interface, then we should change the parameters in option card “ basic” as figure 6.21,it will show the meaning of every parameter when the mouse stays beside. Then switch to option card “Advanced”, as figure 6.22 shows, change nozzle size to 0.4, retraction speed to 40~60, distance to 3~4, and infill speed to 40. The other parameters can keep original or change them as figure6.22.

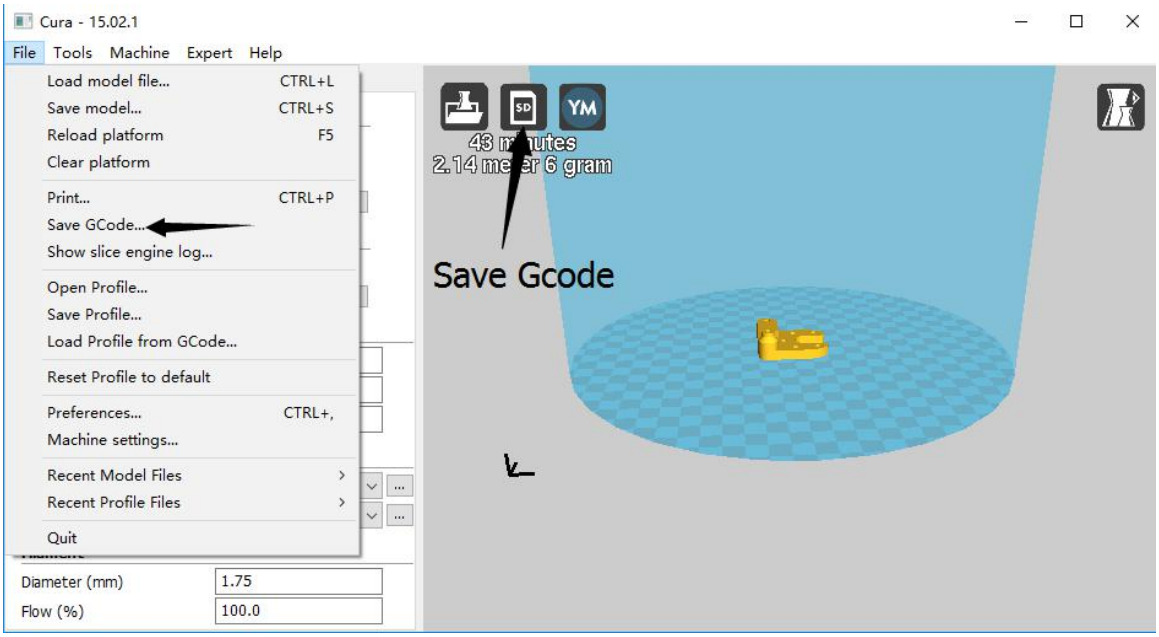





Cura loads a robot model when you open it, you can print it directly. Click " File"-"Print", and the printing dialog box appears, the click "print" to star to print. Or you can load the model file we provide, 20*20*10.stl (as figure 6.23), to print and measure the size to adjust the parameter.


Cura - 15.02.1
File
Tools
Machine
Expert
Help
Load model file... CTRL+L
Save model... CTRL+S
Reload platform F5
Clear platform
Print... CTRL+P
Save GCode...
Show slice engine log...
Open Profile...
Save Profile...
Load Profile from GCode...
Reset Profile to default
Preferences... CTRL+,
Machine settings...
Recent Model Files
Recent Profile Files
Quit
Diameter (mm) 1.75
Flow (%) 100.0
SD
YM
Save Gcode