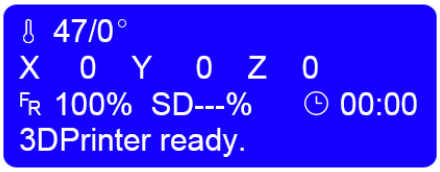
3DPrinter User Manual

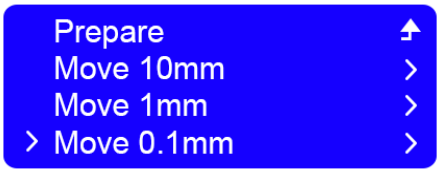
First Setup

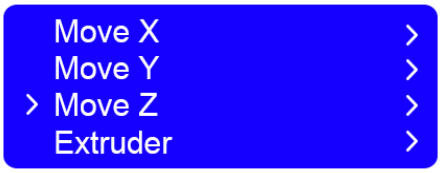
Set Z-Probe offset

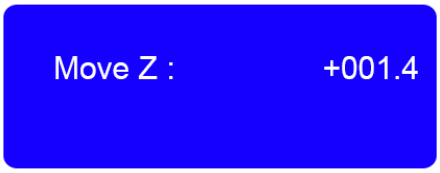
1. On the main screen, press button to enter the main menu, then go to Prepare>Level bed, press button that will perform Automatic Bed Leveling.



2. When the nozzle stops at the center of the bed, go to Main Menu>Prepare>Move axis>Move 0.1mm, select “Move Z” and press button, adjust the parameter to move the bed up and down, when nozzle just touches the bed, remember the current value and press button.







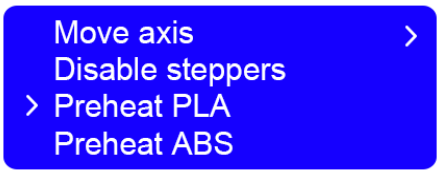
3. Return to the main menu, then go to Control>Motion, select “Z Offset” and set the value that you remember in the previous step.

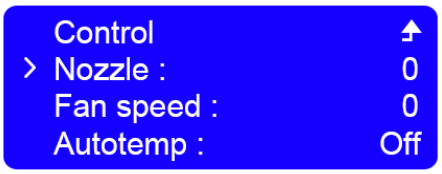
4. Return to the previous menu, go to Main Menu>Control, select “Store memory” and press button.

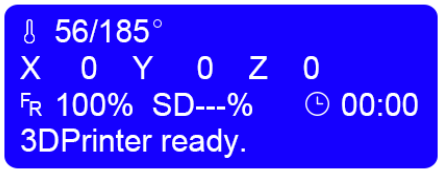
Note: Z Probe Offset is necessary. But it need not be changed frequently unless the slope of bed changes obviously or you reassemble the extruder module. If you want to calibrate the Z Probe Offset, you should set the value to 0 and store it in EEPROM first.

Load/Unload Filament

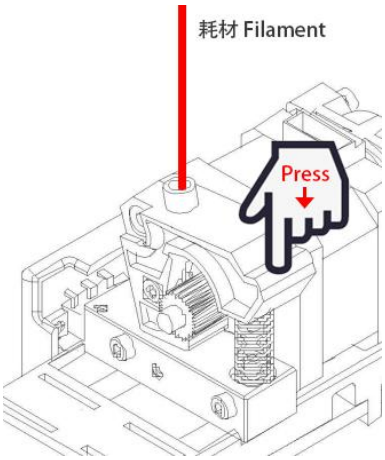
1. Go to Main Menu>Prepare, select “Preheat PLA” or “Preheat ABS”, the nozzle starts to heat up, you can also go to Main Menu>Control>Temperature>Nozzle, set the preheat temperature directly (PLA 185~200C°, ABS 220~240C°). Nozzle temperature can be displayed on the main screen in real time.



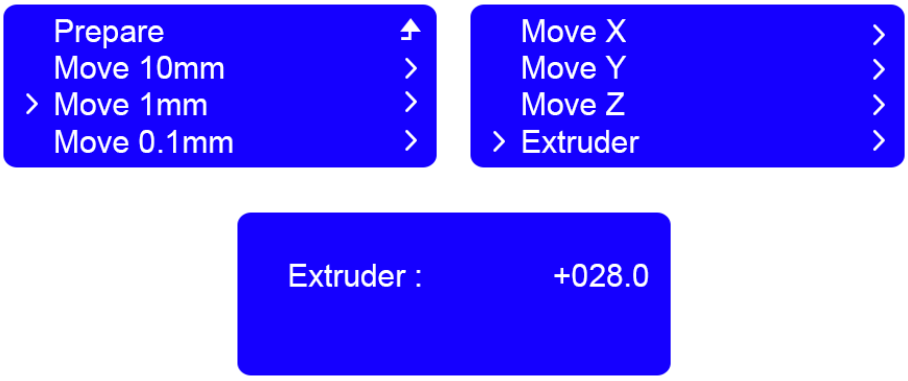




2. Insert filament as shown below once nozzle reaches the target temperature.

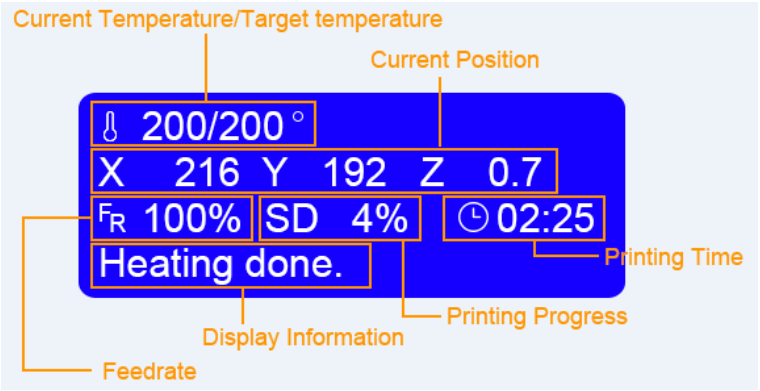


3. Go to Main Menu>Prepare>Move axis>Move 1mm, select “Extruder” and adjust the value, it will load filament when increasing the value and unload filament when decreasing value.



SD Card Printing

Insert the SD card and go to Main Menu>Print from SD, select the gcode file you want to print and press the button. Then your 3D printer starts printing, the status information including temperature, coordinate position, printing progress and time will be displayed on the main screen. Before your first print, make sure you have already set the Z Probe Offset.

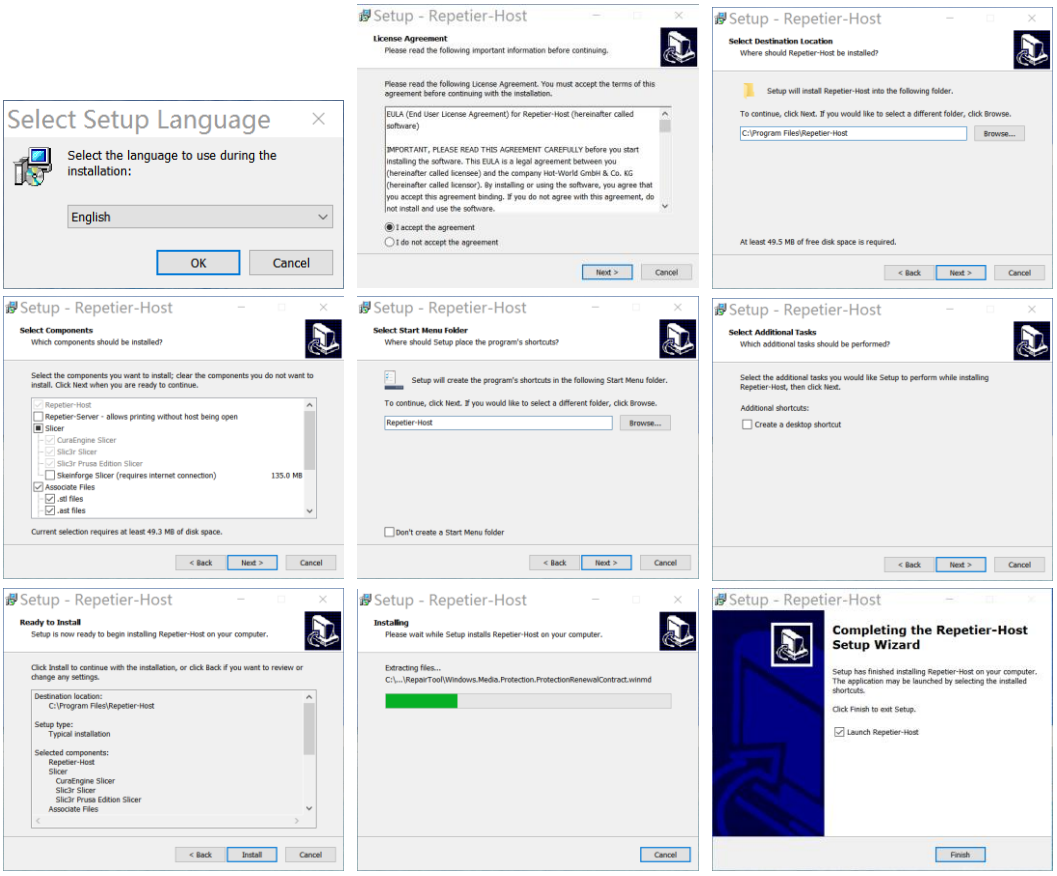


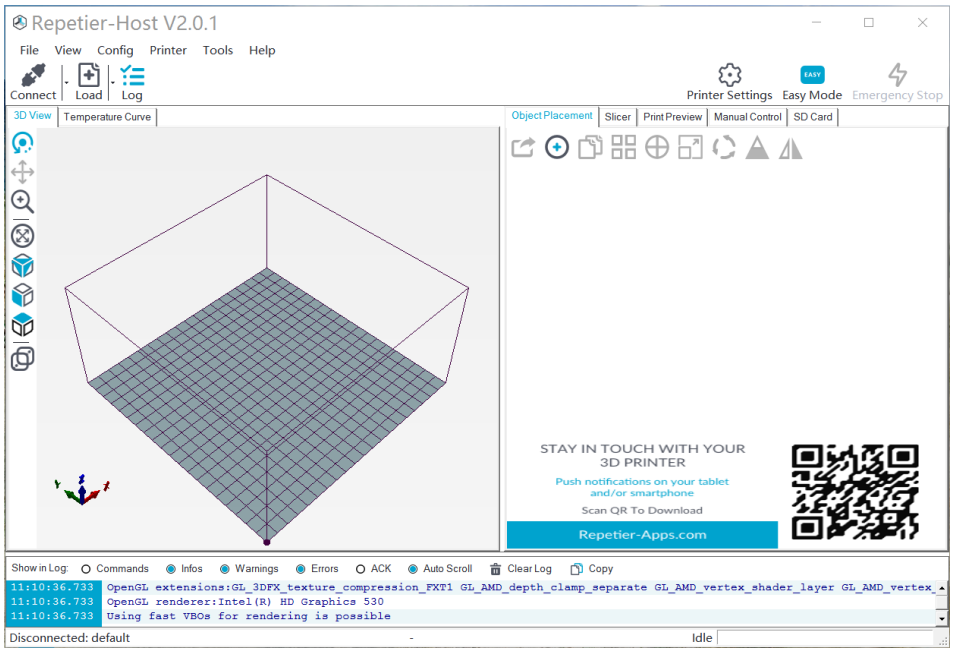
Guide to Repetier-Host

Download & Installation

Go to the [download page] and fetch the latest version for your computer operating system.

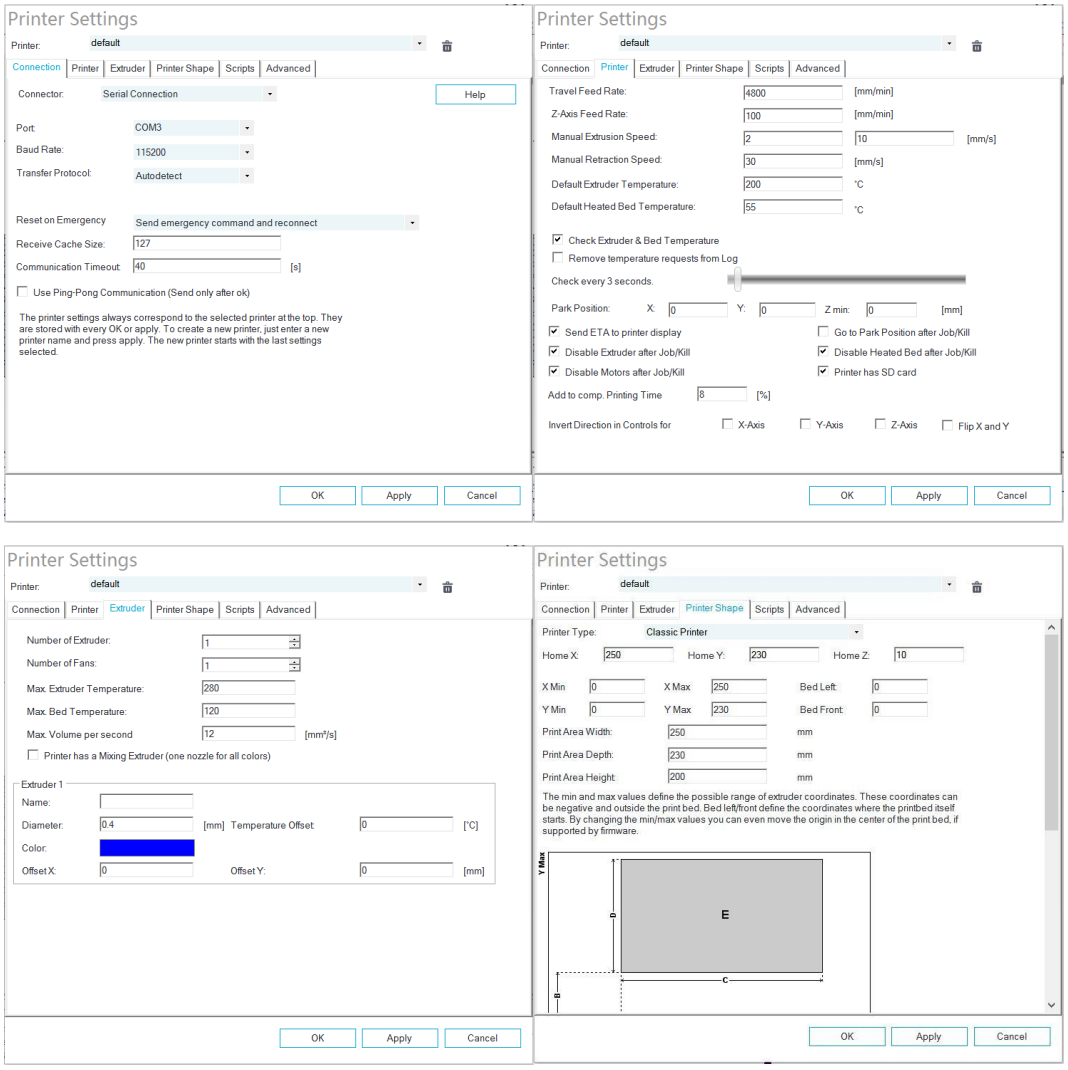
Double click the Package Installer to install the software. The installation process is as follows:





Configuring your 3D Printer

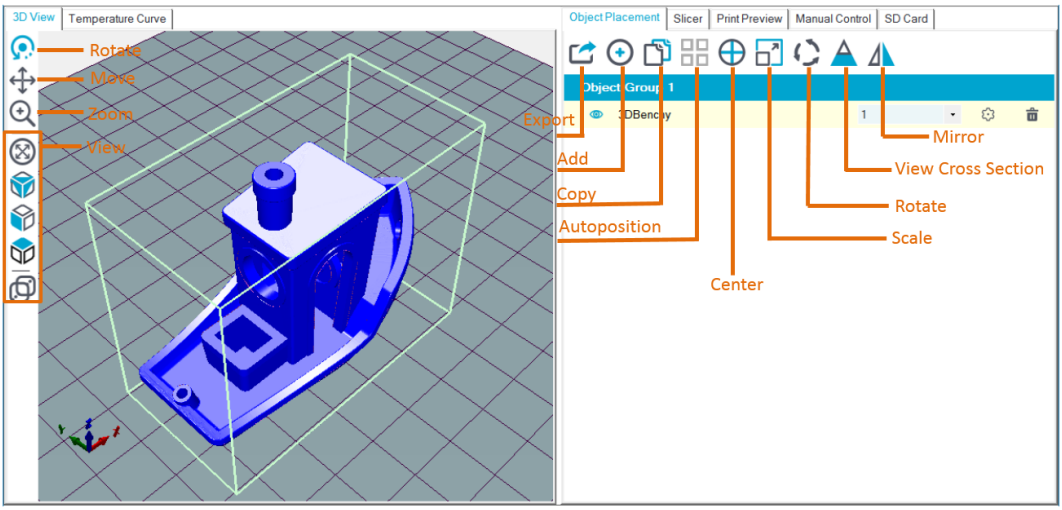
When you run Repetier-Host for the first time, you should configure your printer so you can connect your computer to the host. Make sure your printer is connected and enabled.
Klick the button “Printer Settings” in the top right corner of the interface, and you will see a window which contains several configuration tabs like this:



There are some key parameters in Printer Settings, such as Port, Baud Rate (115200) and Printer Area. You can configure your printer referring to the pictures above.

Object Placement

Prepare the objects you want to print, and slice the models into gcode files recognized by the 3D printer. Here you can arrange them on your print bed, rotate and scale them to your likes.

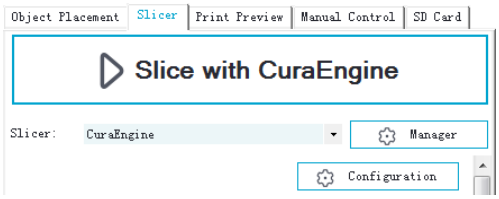


Slicer

Several slicers are bundled with Repetier-Host, we will take Cura and Slic3r for examples, mainly introduce the import and application of the predefined configuration files.

Cura

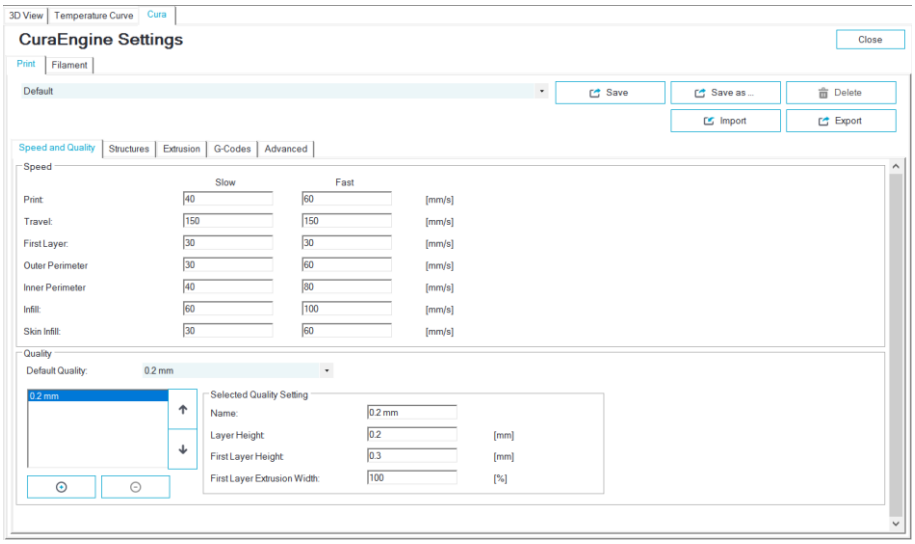
CuraEngine is an external slicer software which is bundled with the host. To use it, you normally only need the right tab with the quick settings. Here you select your predefined configurations to use plus some important settings. To define the parameters of your configuration, click the “Configuration” button to enter the setup.

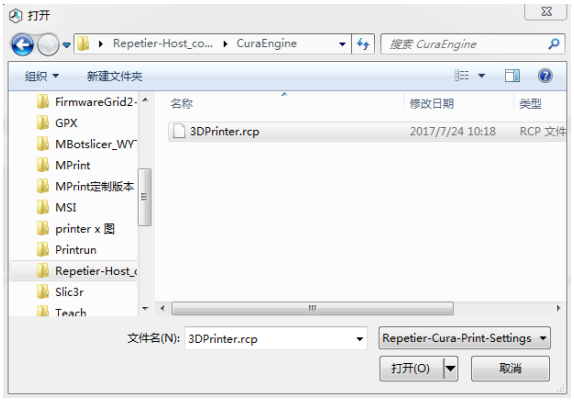


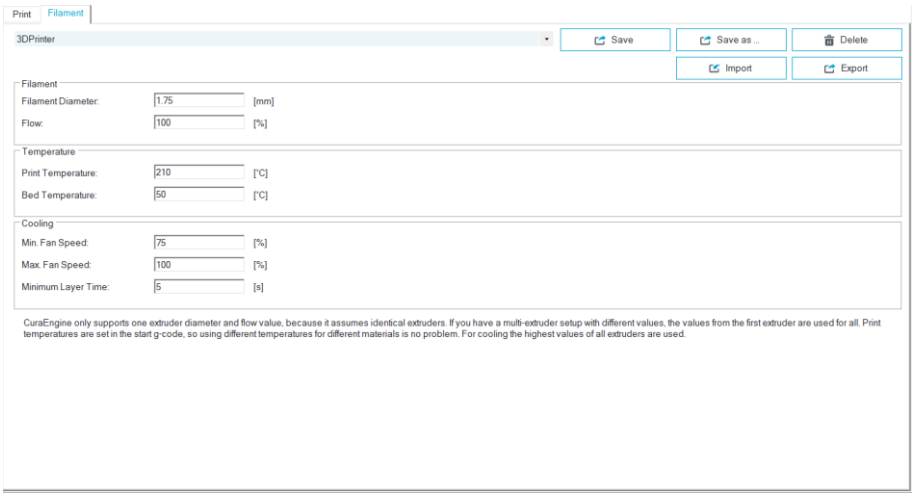
The configuration is split into two main parts. The biggest part is the print settings, these are subdivided into five tabs. The second part is filament settings, which contain parameters influenced by the filament selection.

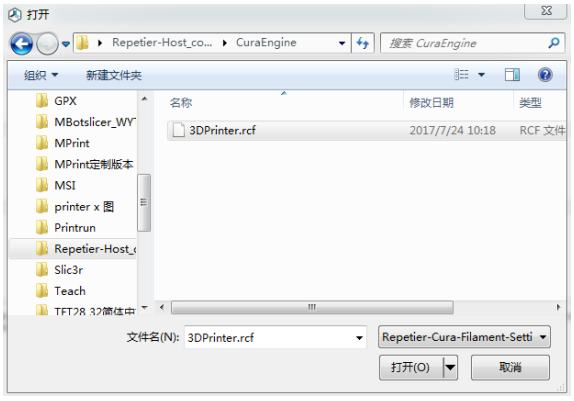
To import a predefined configuration, click the “Import” button, and select the configuration files. Both print and filament settings should import configuration.

By clicking into a parameter field or by hovering a field, a help bubble will show up with detailed informations about this parameter.

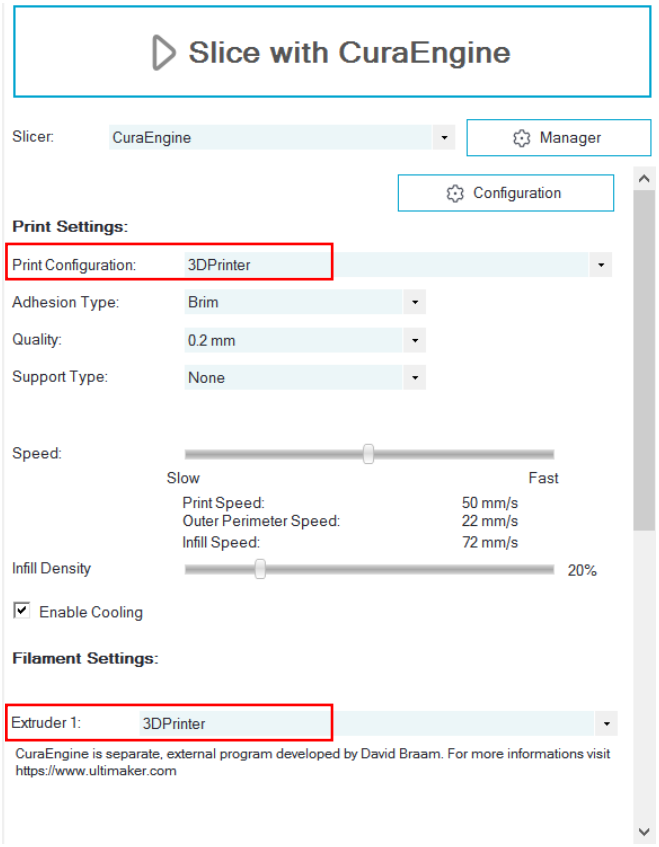

3D View
Temperature Curve
Cura
CuraEngine Settings
Close
Print
Filament
Default
Save
Save as ...
Delete
Import
Export
Speed and Quality
Structures
Extrusion
G-Codes
Advanced
Speed
Slow
Fast
Print:
Travel:
First Layer:
Outer Perimeter
Inner Perimeter
Infill:
Skin Infill:
[mm/s]
Quality
Default Quality:
0.2 mm
Selected Quality Setting
Name:
Layer Height:
First Layer Height:
First Layer Extrusion Width:
[mm]
[%]


打开
Repetier-Host_co... ▸ CuraEngine
搜索 CuraEngine
组织
新建文件夹
FirmwareGrid2-
GPX
MBotslicer_WY
MPrint
MPrint定制版本
MSI
printer x 图
Printrun
Repetier-Host_
Slic3r
Teach
名称
修改日期
类型
3DPrinter.rcp
2017/7/24 10:18
RCP 文件
文件名(N): 3DPrinter.rcp
Repetier-Cura-Print-Settings
打开(O)
取消


Print
Filament
3DPrinter
Save
Save as ...
Delete
Import
Export
Filament
Filament Diameter:
Flow:
[mm]
[%]
Temperature
Print Temperature:
Bed Temperature:
[°C]
Cooling
Min. Fan Speed:
Max. Fan Speed:
Minimum Layer Time:
[s]
CuraEngine only supports one extruder diameter and flow value, because it assumes identical extruders. If you have a multi-extruder setup with different values, the values from the first extruder are used for all. Print temperatures are set in the start g-code, so using different temperatures for different materials is no problem. For cooling the highest values of all extruders are used.

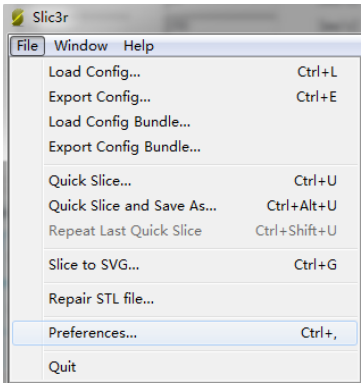

打开
Repetier-Host_co... ▸ CuraEngine
搜索 CuraEngine
组织
新建文件夹
GPX
MBotslicer_WY
MPrint
MPrint定制版本
MSI
printer x 图
Printrun
Repetier-Host_
Slic3r
Teach
名称
修改日期
类型
3DPrinter.rcf
2017/7/24 10:18
RCF 文件
文件名(N): 3DPrinter.rcf
Repetier-Cura-Filament-Setti
打开(O)
取消

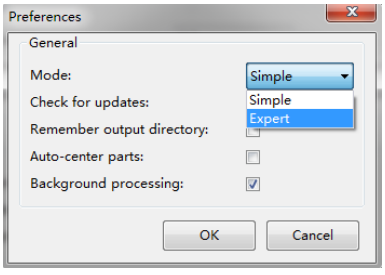
The predefined configuration will be displayed and chosen in the basic setting tab after import configurations successfully. Simply set up the quality, support, speed and infill density, so that you can slice the objects easily.



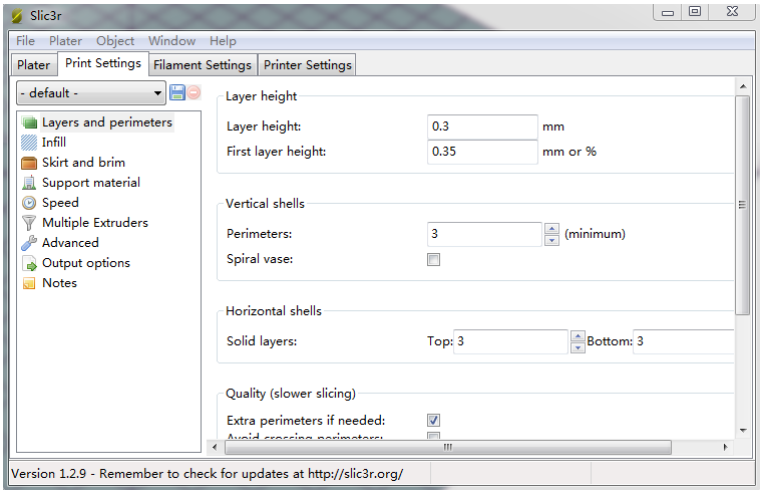
Slic3r

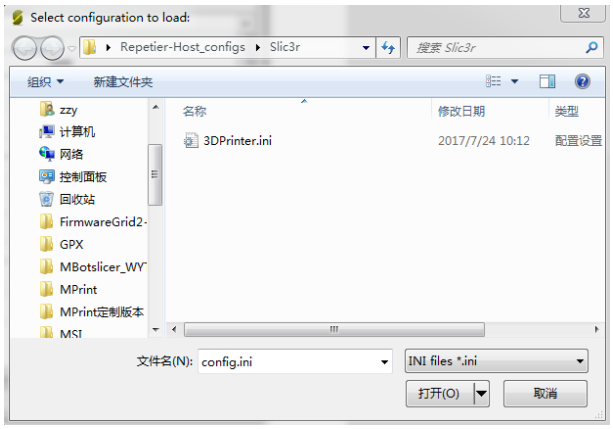
Slic3r is an external slicer software which is bundled with the host. You can start it directly from the host with the "Configuration" button. To import the configuration, you need to change the mode to "Expert" first, select File menu and click Preferences, change the mode, save and exit Slic3r.

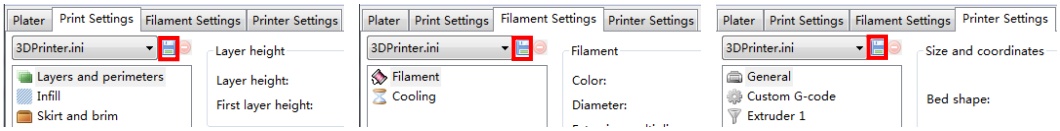




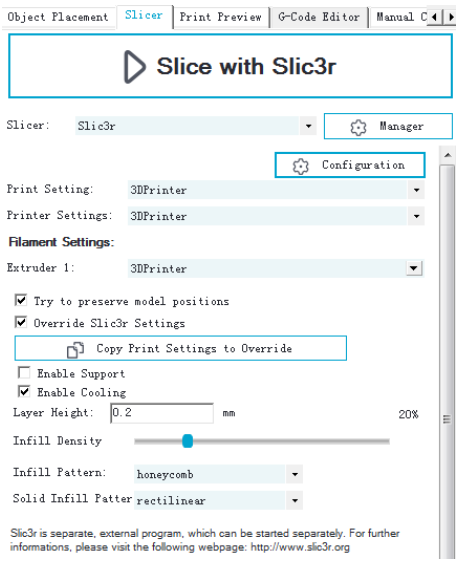
Click the "Configuration" to rerun Slic3r, and you will see the configuration is split into three parts: Print Settings, Filament Settings and Printer Settings. Click File>Load Config…, select and import a predefined configuration. Finally click "Save" in three tabs respectively.







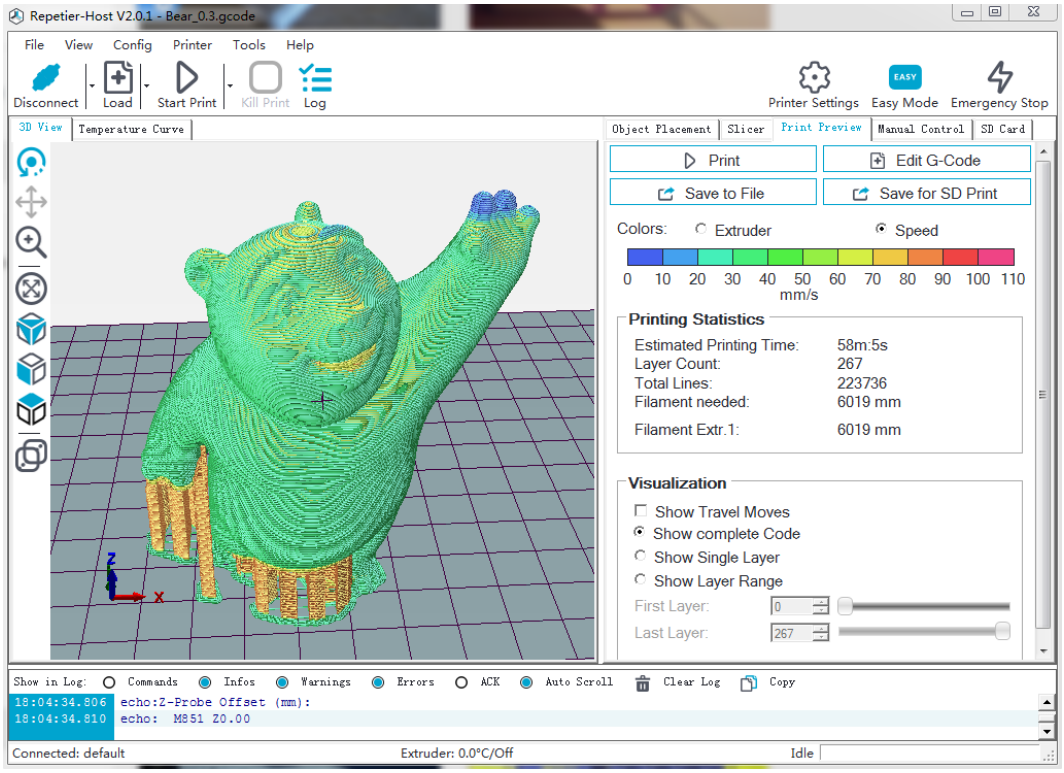
When you switch back to the host, you will see this configuration in the combo boxes right to “Print Settings”, “Printer Settings” and “Extruder 1/2/3”. To slice objects loaded into the host, you select the profiles you want to use for your slice and hit “Slice with Slic3r”.



One cool feature is the “Override Slic3r Settings”. If you enable it, the values in the selected profiles get overwritten with values set below. As a first action, you will probably click “Copy Print Settings to Override”. This will copy the values from the selected profiles to the controls below. Now you can change the most often changed values directly without the need to create/modify the profile in Slic3r.

Print Preview

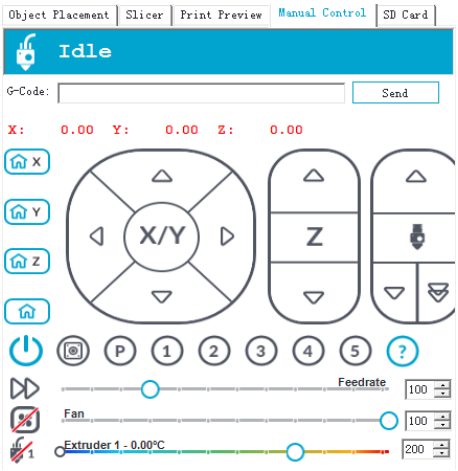
After slicing you can see the slicing result. You can rotate and inspect the whole model, a layer range or just single layers and you can visualize the travel moves. So, you can check if everything is correct before printing, what can save a lot of time and money.
If everything is OK, you can save the G-Code or start the print job with USB cable.



Manual Control

If you want to control your 3D printer manually, make sure your 3D printer is connected to PC. Again, we remind you to set communication mode, Port and Baud Rate correctly.

At the top, you see the most important printer status, so you always know what is happening. The next row allows it to send any g-code command you like. Enter it and press return or hit the send button.



The next block controls the positioning of the extruder. With the arrow keys, you move the extruder in any direction. When you hover over the arrow, you will see a distance appear in mm, telling you how large the move will be. At the top, you see the current extruder position. After the connection, they are red. Red means the host has no idea where the extruder really is. Press the home button to move the extruder to its defined position. After that the color turns black, telling you the position is known.

Below the arrows you have the following buttons:

Stop motor: Will disable the stepper motors.

Park: Moves the extruder head into parking position defined in printer settings.

The “Speed multiply” slider allows it to change the printing/move speed in relation to the send feed rate.

If you have a fan attached, you can change the fan speed and activate/deactivate it the same way as the temperatures.

The extruder block allows it to change the temperature. The temperature can be set either on the right in the text field as well as by clicking on the temperature curve. If you change it in the text field, you need to press return or leave the field to set the value. By clicking the extruder or bed icon the extruder or the bed is deactivated or reactivated. In the right area of the temperature curve you see the last read temperature.

That’s the Guide for 3DPrinter and Repetier-Host. Now it’s time to enjoy the fun with 3D printing. For further information about the software, please refer to the following website:

http://marlinfw.org/

https://www.repetier.com/

http://manual.slic3r.org/

https://ultimaker.com/en/resources/manuals/software