Easy Print 3D Print Software

User Guide

Content

Content 2
1. Introduction 3
2. Download and Installation 3
    2.1 Download 3
    2.2 Install software 3
    2.3 Start Easy Print 6
3. Meeting Easy Print 6
    3.1 Interface overview 7
    3.2 Control 9
    3.3 View modes 11
    3.4 Change Views 12
        3.4.1 Move 12
        3.4.2 Rotate 12
        3.4.3 Scale 12
        3.4.4 Set 13
        3.4.5 Reset 13
    3.5 Edit Models 13
        3.5.1 Move 13
        3.5.2 Rotate 14
        3.5.3 Scale 14
4. Preparation for 3D Printing 14
    4.1 Connecting a Geeetech 3D Printer 14
    4.2 Add a Third-Party Printer 16
5. Entry-level printing 21
6. Advanced-level printing 25
    6.1 Create a new configuration/setting 25
    6.2 Print settings 26
7. Upgrade Firmware 39
8. Upgrade Easy Print 41

1. Introduction

Easy Print is a3D printing software that is customized for our GEEETECH series printers. It is suitable for both novices and experts. Easy Print converts your 3D design into model for 3D printing with just a few clicks on the simple and intuitive interface.

It has a list of presets for printing quality, so just select what quality you want and Easy Print will take care of the rest. And for experts seeking fine control over the printing process, it has a wide range of parameters that can be modified manually. Once sliced, you can get layer visualization and time and material estimates.

This manual was prepared for Easy Print 1.0.14 beta.

2. Download and Installation

2.1 Download

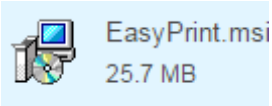
A. For some 3D printers, we have already prepare the latest driver installation package in the SD card. You just need to insert the SD card into a computer.

B. Please download the latest Easy Print free through the link here:

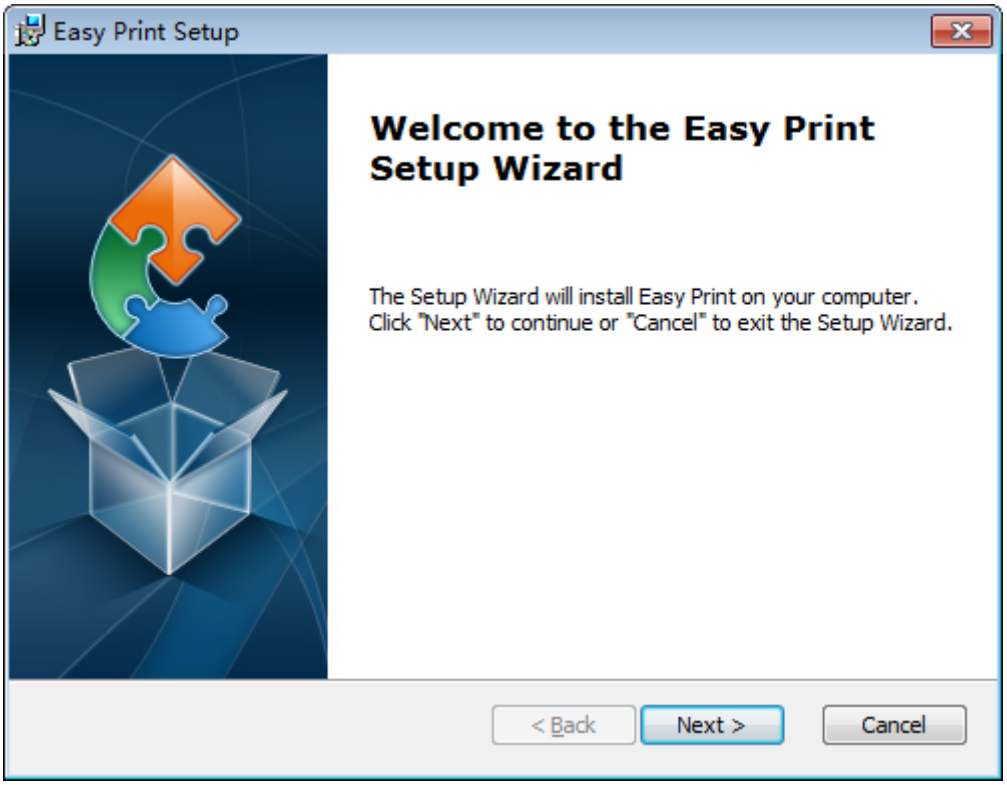
http://162.144.205.148/firmware/EasyPrint.msi

2.2 Install software

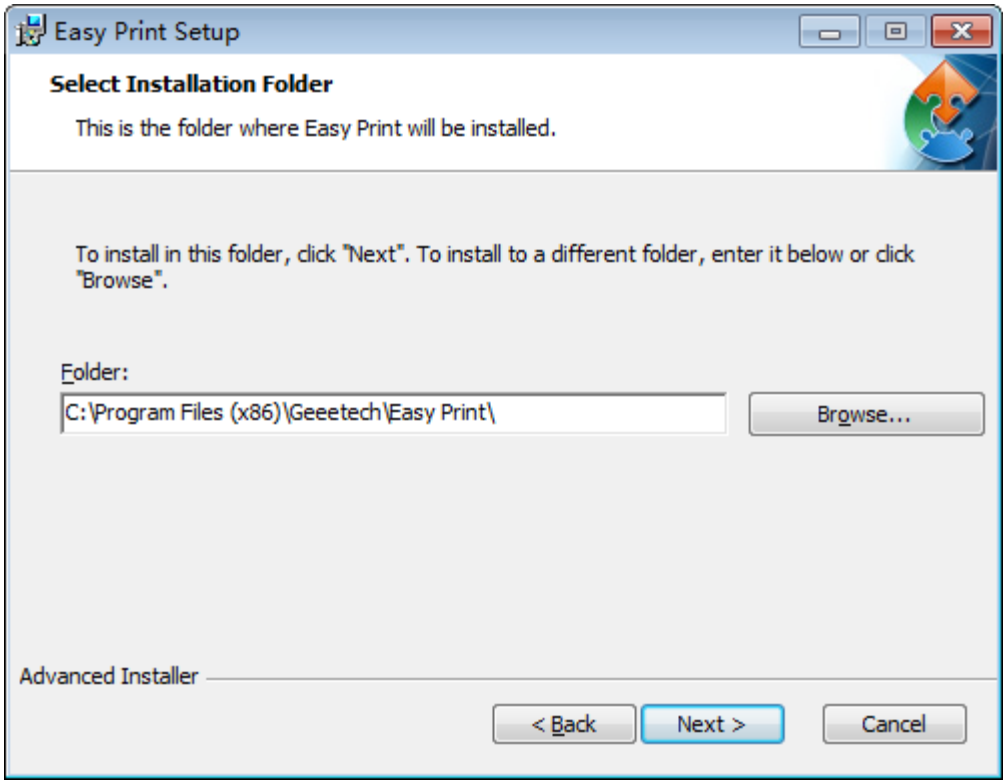
Step1. Upon downloading, Unzip the downloaded file, and complete installation according to the instruction start the installer.



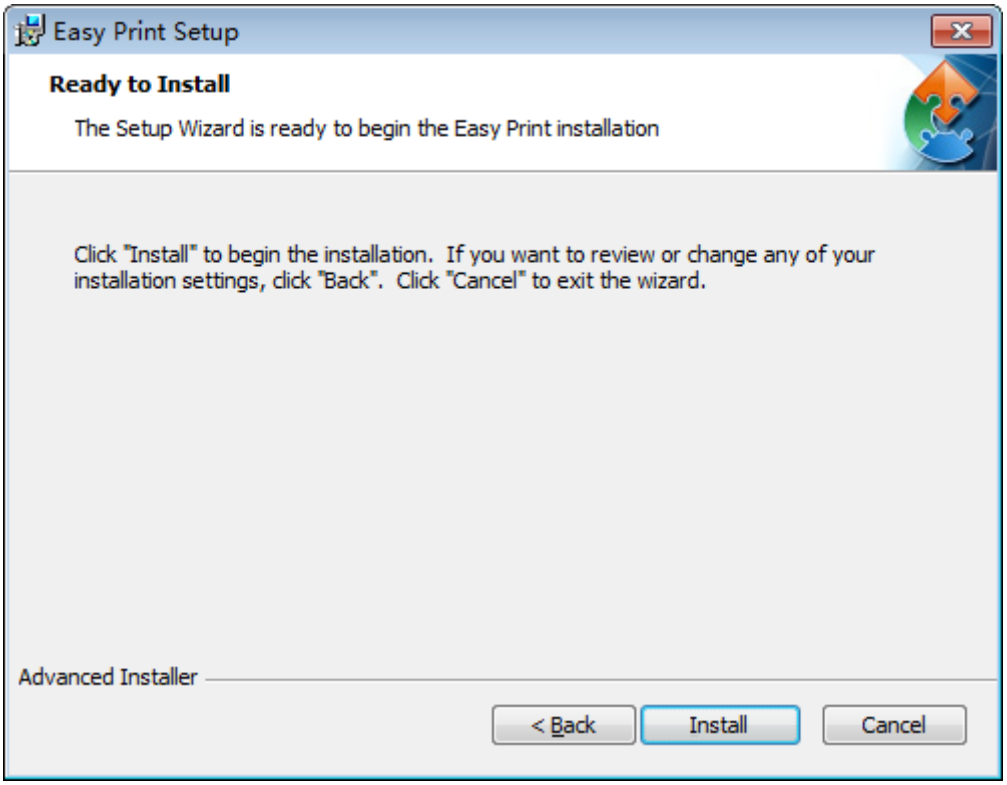
Step2. Click [Next] to continue.



Step3. Select the destination where you would like to install the Easy Print, click [ok] to continue.

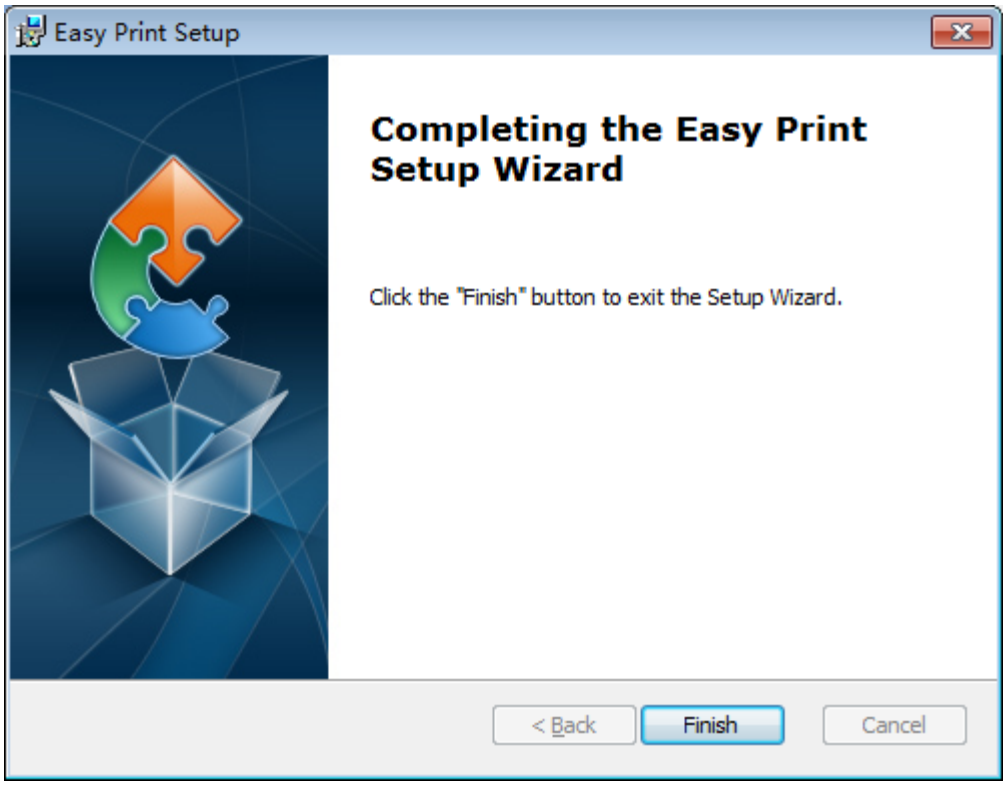


Step4. Get ready to install. Click [Install].



Step5.Wait about half a minute while installing, the progress bar will show you the rate.

Step6. Setup wizard has finished the installing; you can click [finish] to exit out setup wizard.

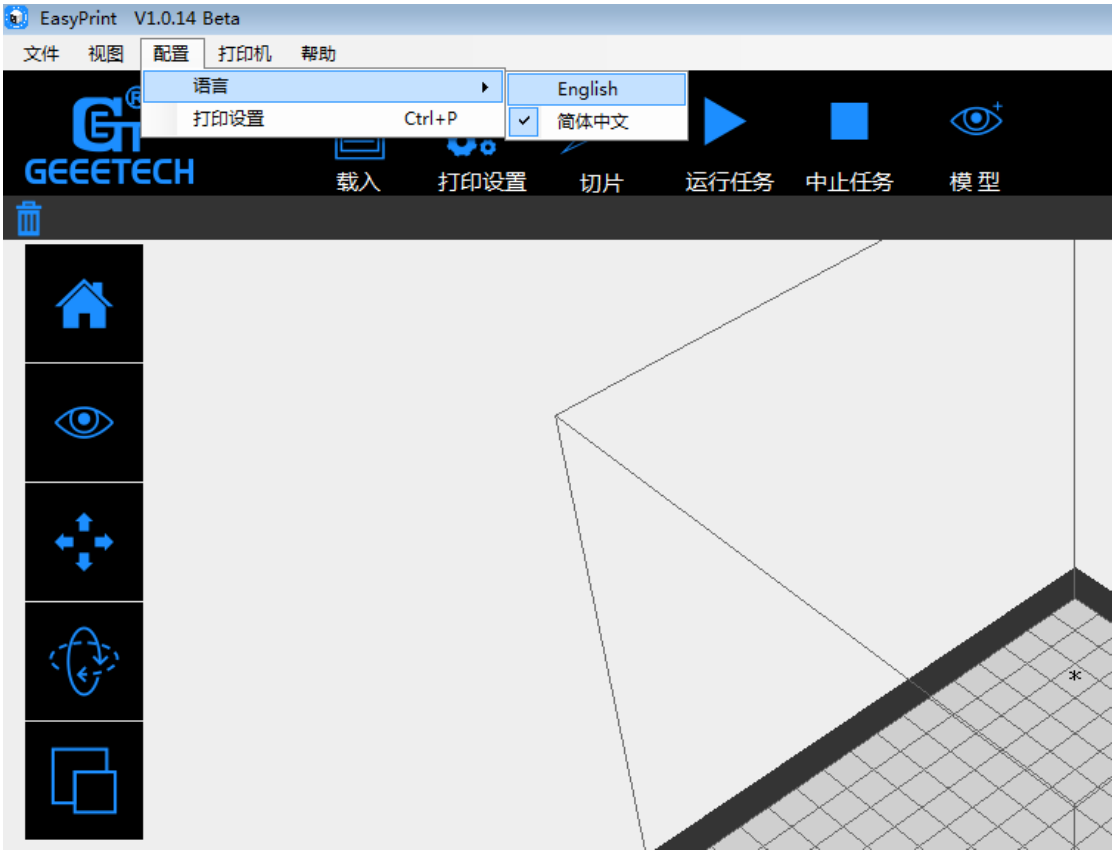


2.3 Start Easy Print

Click the Easy Print icon on the desktop to start the software.

3. Meeting Easy Print

Easy Print is devised with bilingual function. You can click [Config> Language] button on the top right to choose English and Mandarin.



3.1 Interface overview

The overview below shows the main interface of Easy Print with Load, Settings, Slicer, Run job, Kill job etc. as its important features.

Load：You can load a model file into the software；

Settings：Click Settings to open the settings dialog. Use the dialog to adjust printing parameters for your object;

Slicer: Click Slice to Turn 3D models into print files；

Run job：Click run job to send the sliced print file to your printer；

Kill job：click kill job to stop the printing job；

Object：view the object in different ways--- original mode, sliced mode and under printing;

Manual control：to direct control to the printer;

disconnected：click to connect the printer to the software

connected：click to disconnect the printer to the software

Emergency stop: click to stop the printer in the event of an emergency.

The defaulted view

Click this tab and you could change the view of the model in three different ways: top view, front view and left view.

Click [Reset] to adjust the pre-set parameter.

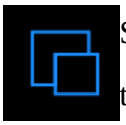
Move: Click the Move button to go into Move mode. In Move mode,

the coordinate system can be used to set the specific place of the model.

The object can also be moved freely by ‘click-and-drag’ anywhere on the object.

Rotate: Click the rotate button to go into rotate mode. In rotate mode, click and set the angle to rotate your model around the x/y/z axis to your likes.

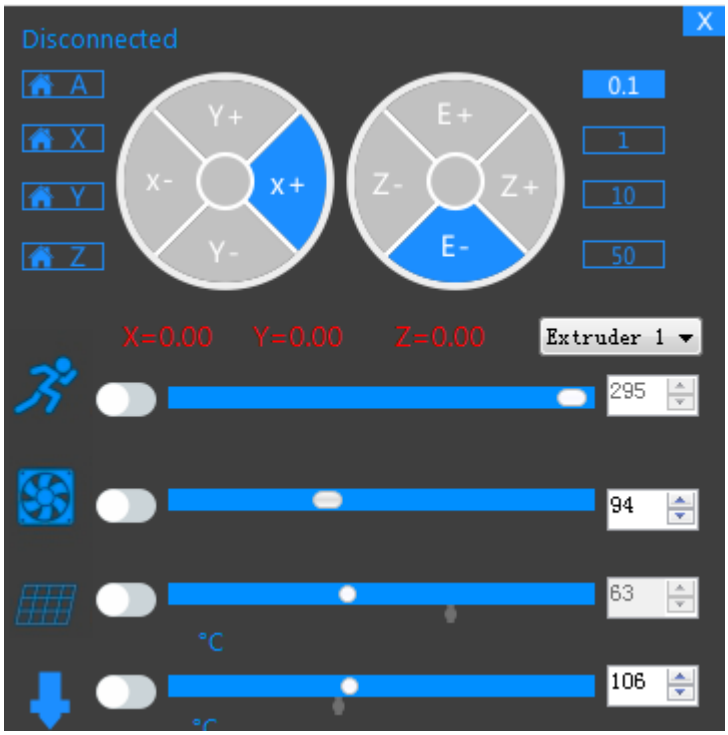
Click [Reset] button to lay the object flat on the build plate.



Scale: Click the Scale button to go into Scale mode. In Scale mode, the model can be zoomed in three different ways:

- Use the ‘lock/unlock’ icon on the panel to enable/disable uniform scaling.
- Set a scale percentage in the tool panel, for the X, Y or Z axis.
- Reset the object to its original dimensions.

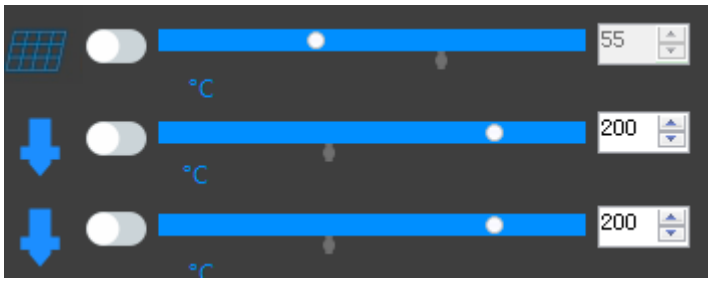
3.2 Control



This is the control panel where you can direct control to your printer. On the panel you see the most important printer status, so you always know what is happening.

The “Speed multiply” slider allows it to change the printing/move speed in relation to the feed rate. Be careful if you increase the speed. If you haven’t defined the real limits in your firmware this can cause lost steps during print!

You can change the fan speed and activate/deactivate it by clicking the fan icon.

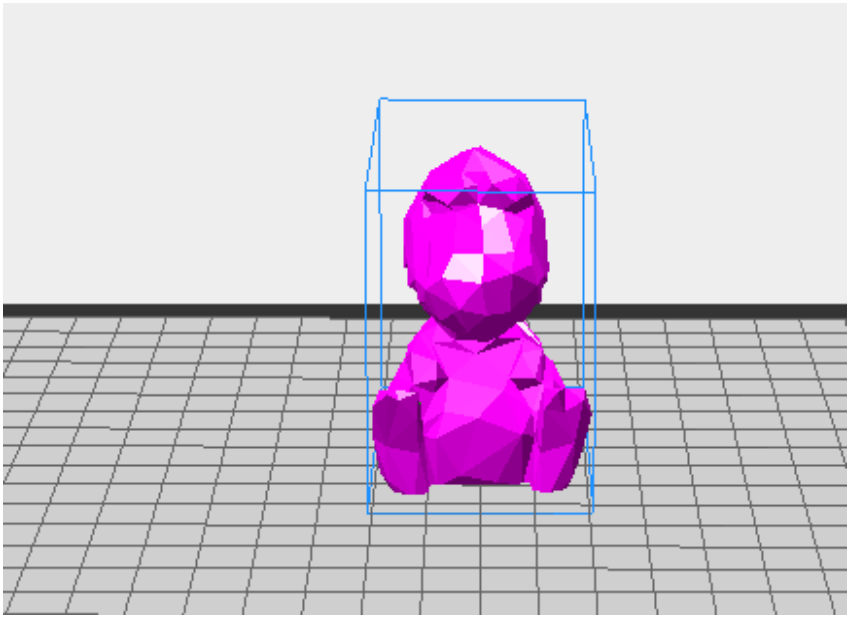


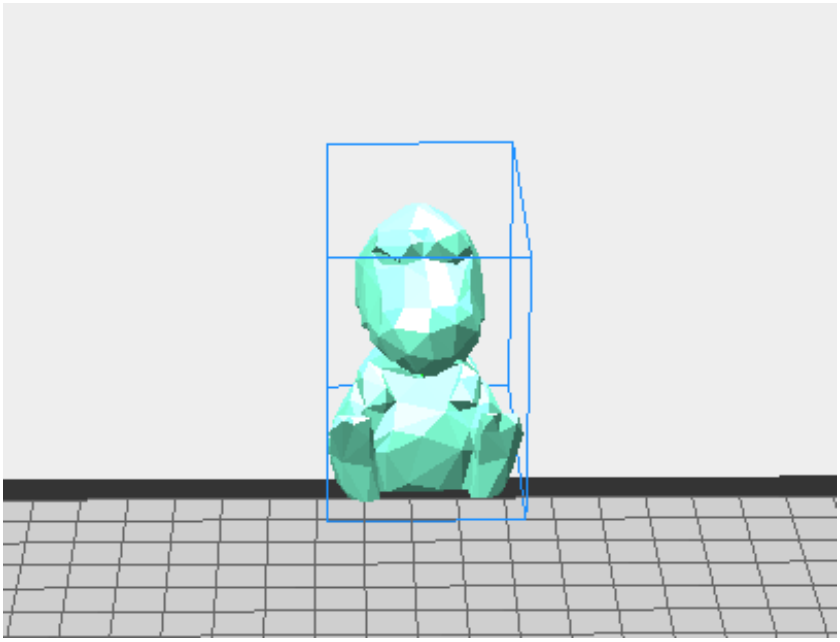
The extruder and print bed blocks allow it to change the temperature. By clicking the extruder or bed icon, the extruder or the bed is deactivated or reactivated.

3.3 View modes

Easy Print offers three views for the model. The different views allow you to check the model on specific details. There are three different views available.

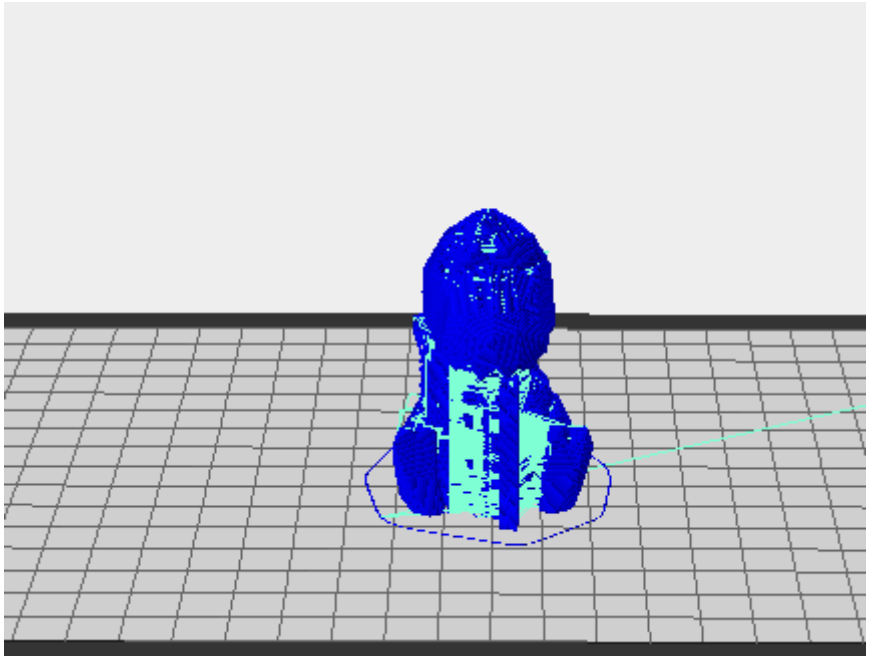
- Select the **original model** in the object view mode menu. You will see the solid object. The model is colored rose red when placed correctly and green when it is located out of the build plate





- After sliced, select the **sliced model** in the object view mode menu to observe the model in layers. The layers view represents the print head moves of the printer.

When the model is **under printing**, you can see the moving trace of the print head.



3.4 Change Views

Change your view of the model by moving, rotating, scaling the view and so on.

3.4.1 Move

You can move the view of printing frame in the software interface by following three ways:

a. Press and hold the middle button of mouse, and then drag the mouse.

b. Hold the **shift** key, long-press the right button of the mouse and drag.

c. Hold the **shift** key, long-press the left button of the mouse and drag.

3.4.2 Rotate

You can rotate the view of printing frame in the software interface by long-pressing the left button of the mouse and scrolling the mouse.

3.4.3 Scale

You can zoom in or out the view of printing frame in the software interface by

scrolling the mouse wheel in any condition.

3.4.4 Set

You can observe the model from six angles by following two ways. (**Top/ Bottom/ Front/Back/ Left/Right View**)

a. Click the **View** menu, and then select the view you need to observe the model.

b. Click the **View** button on the left of the software interface, and select the view you need.

3.4.5 Reset

a. Click the **View** menu, and then select [Isometric View].

b. Click the **Home view** on the left of the software interface.

c. Click the **View** button on the left of the software interface, and then select **Reset**.

3.5 Edit Models

You can edit the models by moving, rotating, scaling the model and so on.

3.5.1 Move

When the model is selected, you can change its position on the build platform by following two ways:

a. Click **Move** button on the left of the software interface. Long-press the left button of the mouse and drag and you could adjust the position of the model in X/Y direction.

b. Click **Move** button on the left of the software interface and then enter the distance value you want to move on X/Y/Z axes positioning. Click **Reset** button to reset distance values.

Note: Usually you need to click **Center** after finishing setting, so that the model is located on and closely related to the build platform. Once the position of the model

goes beyond the build platform, it will assume green.

3.5.2 Rotate

When the model is selected, you can change the orientation of your model on the build platform by the following step:

Click **Rotate** button on the left of the software interface, and then enter into rotating angel values in X/Y/Z axes positioning. Or click **Reset** button to reset rotating angle values.

3.5.3 Scale

When the model is selected, you can change the size of the model on the build platform by clicking the **Scale** button on the left of the software interface. Then enter into scale values in X/Y/Z axes positioning. Or click **Reset** button to reset scale values.

Note: Use the 'lock/unlock' icon on the panel to enable/disable uniform scaling.

4. Preparation for 3D Printing

4.1 Connecting a Geeetech 3D Printer

Connect the printer with your computer by using a USB cable. Turn on the power switch.

Step1. Start Easy Print.

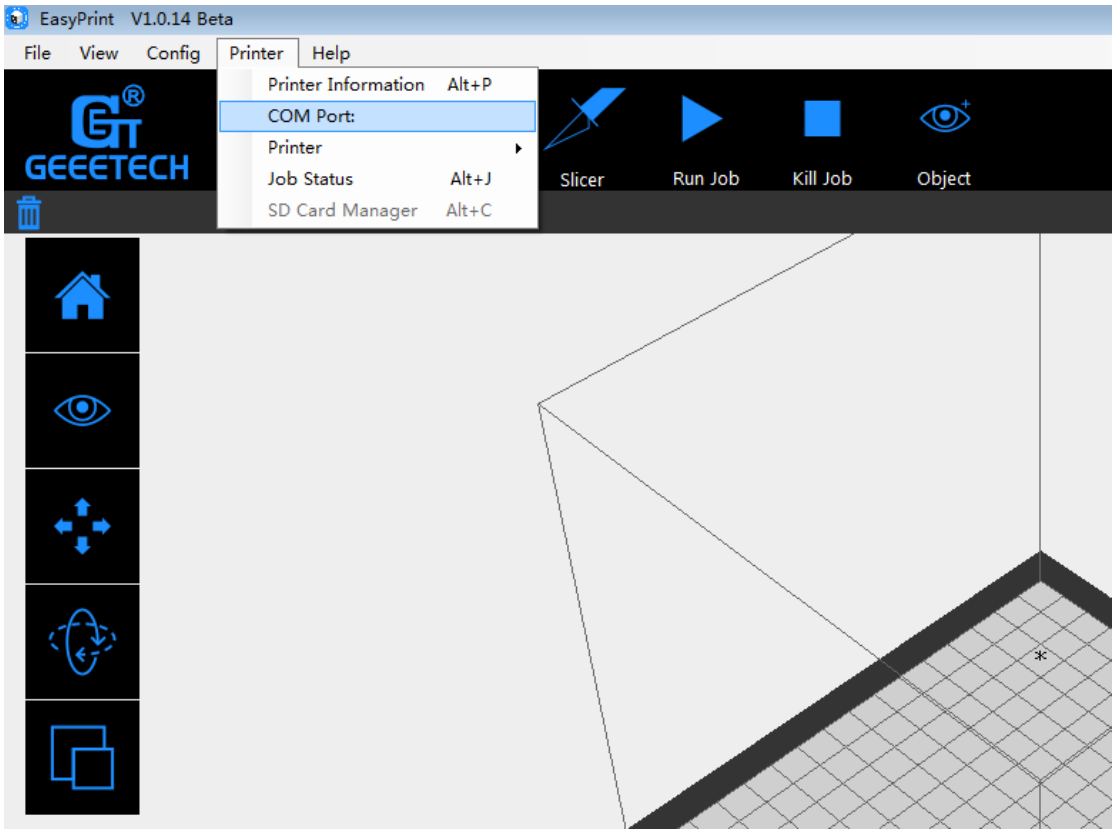
If it is the first time for you to start Easy Print, you need to configure your printer before using the USB cable. Please select the corresponding COM Port and the Printer type to ensure smooth printing process.

Step2. Select the COM Port

Click the **Printer** menu, and then select the corresponding COM Port.

* The COM Port refers to the port that can be used for connecting the printer and your computer, identical to the USB serial in the device manager.

In case you could not find the COM Port, make sure that the power switch of your printer is turned on and the USB cable is well connected with the computer.

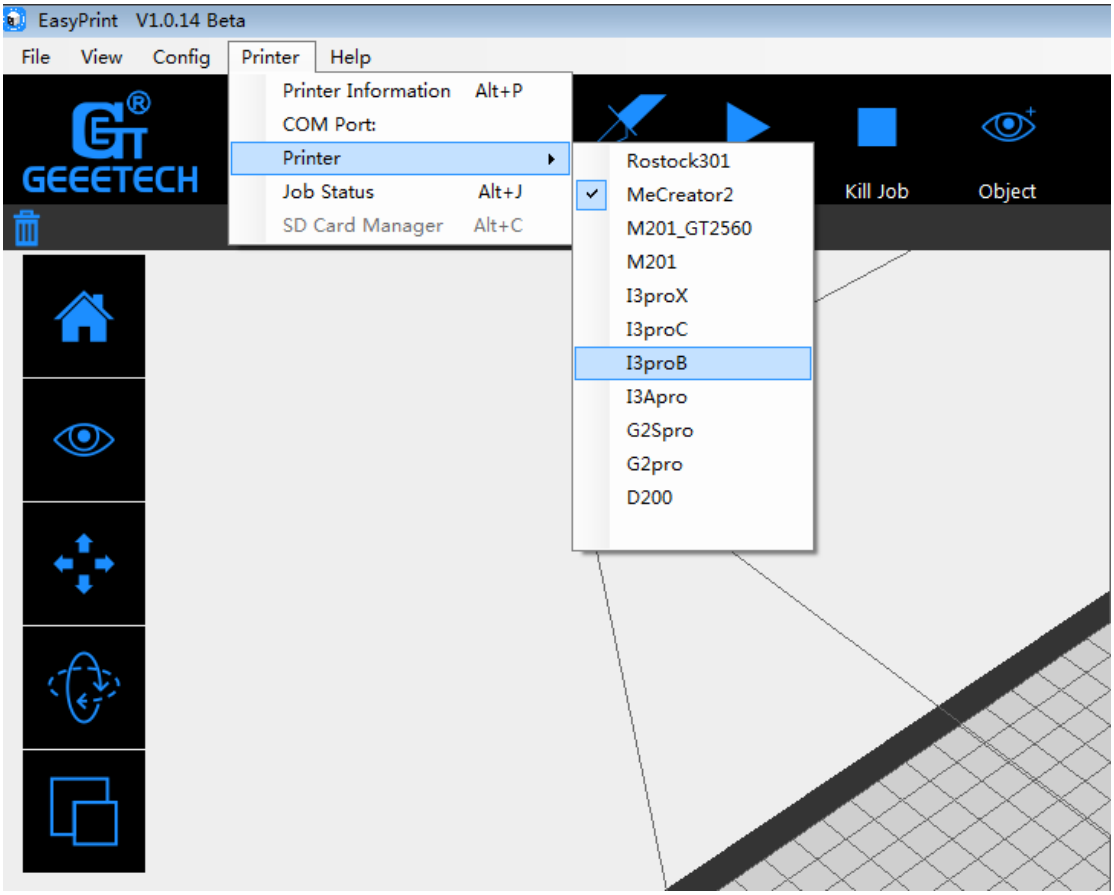


*Generally speaking, when the printer is connected with your computer by a USB cable, the computer will autonomously install the device driver. After successful installation, please search the USB Serial Port (COM) in the device manager.

If you are confronted with autonomous installation failure, please install the device driver manually.

Step3. Select the Printer

Click the **Printer** menu, and then choose the proper printer type.

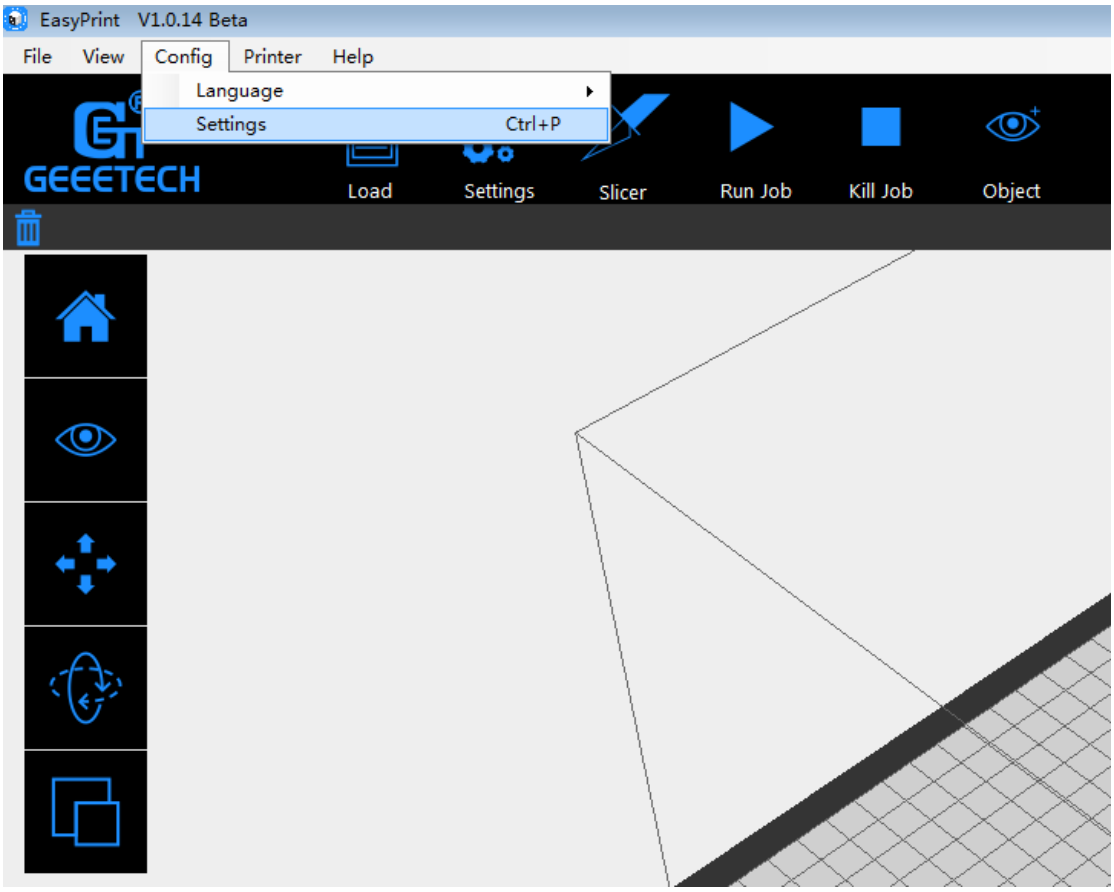


Click the **Connect** button on the top right. You could observe the real-time status of the printer on the bottom of the software interface.

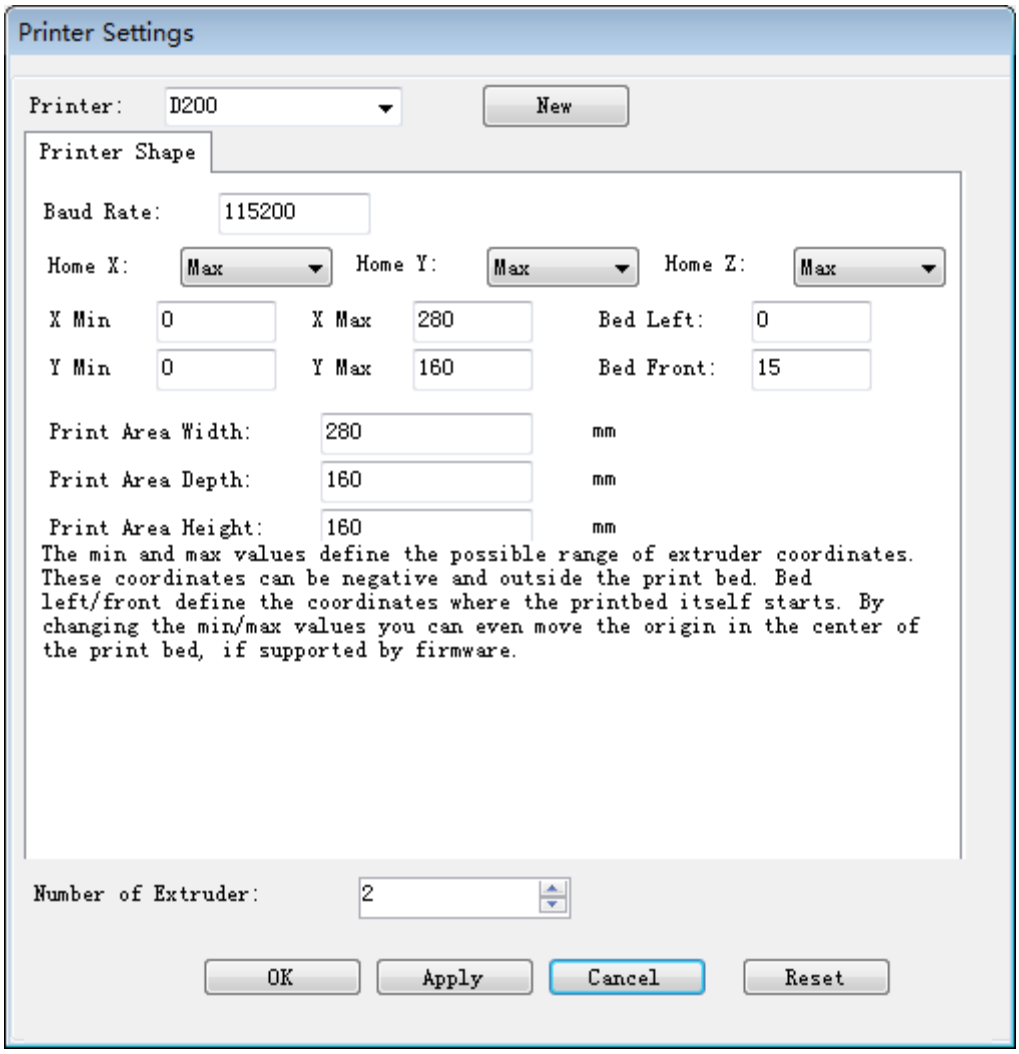
4.2 Add a Third-Party Printer

Although Easy Print is customized for GEEETECH 3D printers, you can also add more printers later on. Therefore, take the following steps:

1. Go to **Config** in the menu bar, and then select **Settings**.



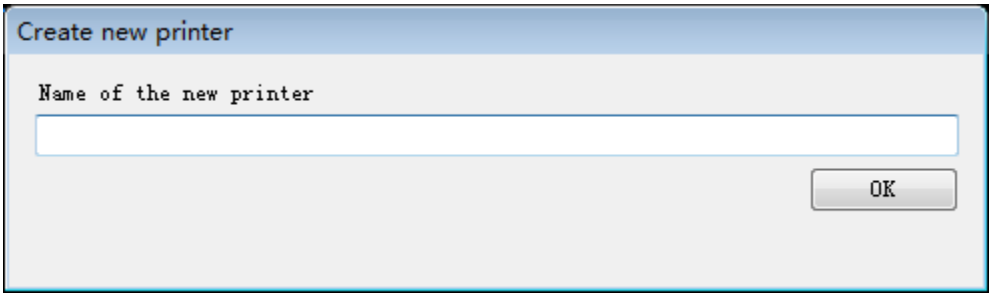
2. You will be presented with the following setting panel; it provides all the options necessary to configure the custom machine. Change the printer settings, including the name of the Printer, Printer Shape, Baud Rate etc.



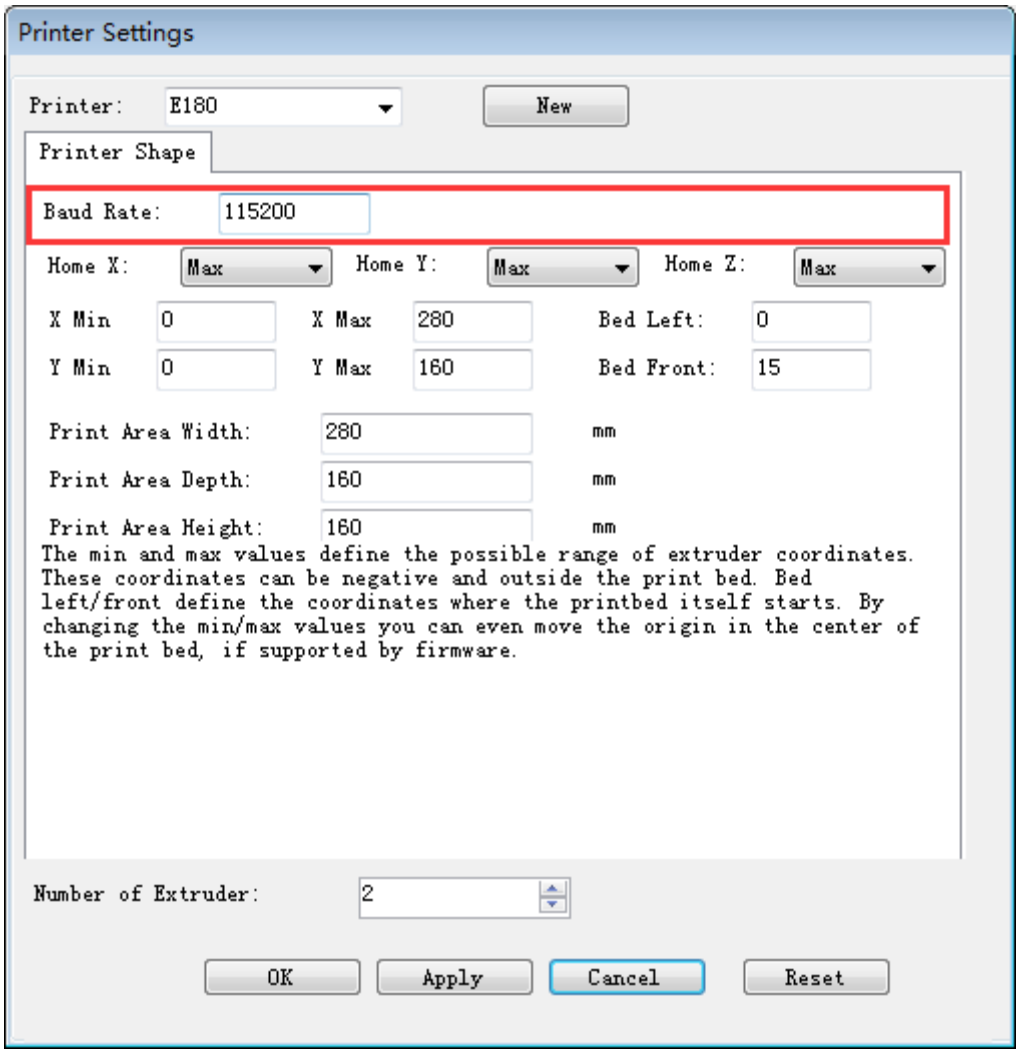
3. Create a New Printer

Especially when you're using multiple printers it can be useful to name them. At the top you see a drop down box, with the currently selected printer. At the start you have only the default printer.

a. Click “New” and input the printer name in the dialog box.

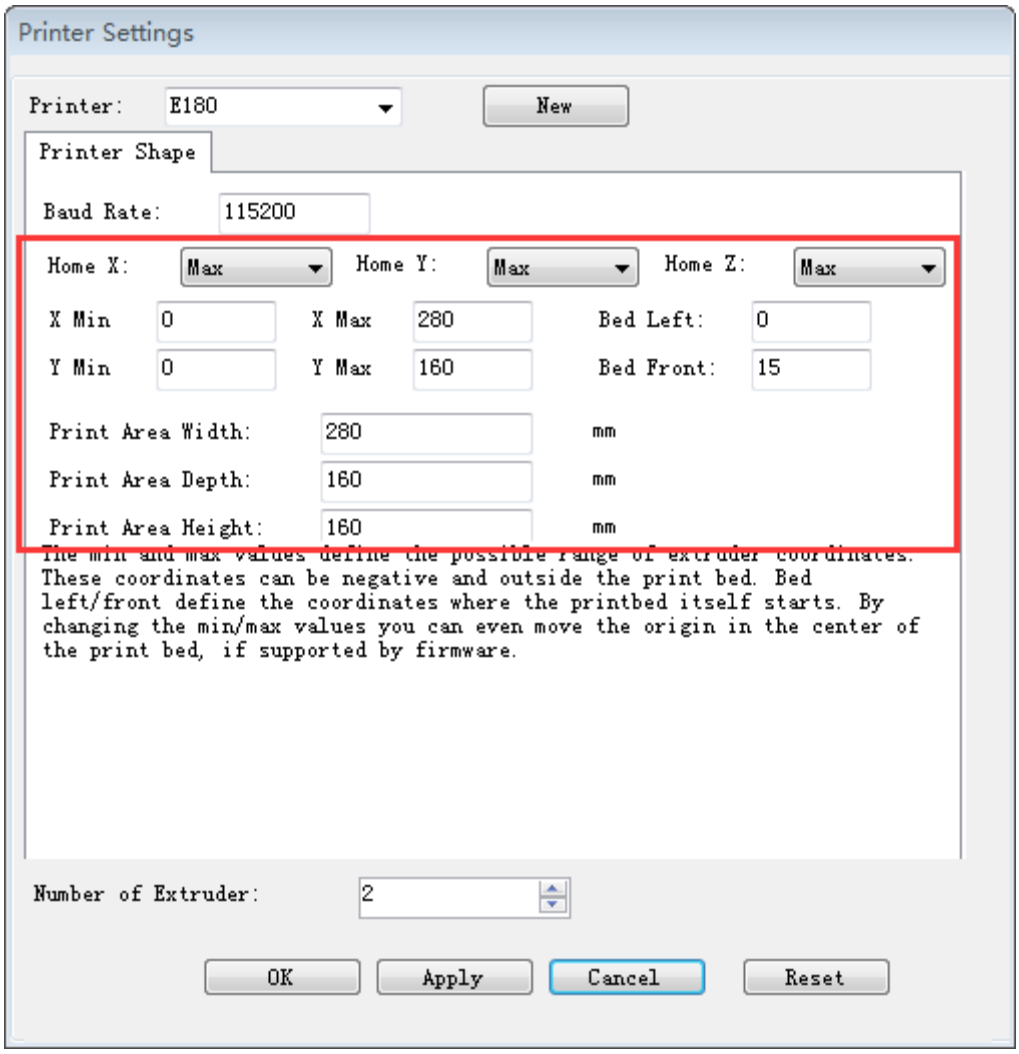


b. **Baud Rate**: Corresponding to the Baud Rate in firmware. It is 115200 by default.

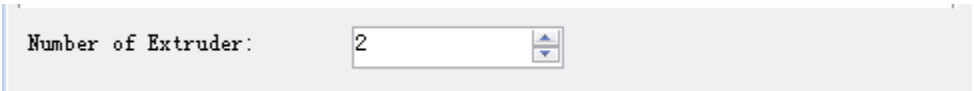


- Printer shape

The “Printer Shape” tab defines your printer shape, or, to be more exact, the shape of your build area. EasyPrint will use this to limit your moves and to check, if your models fit onto the print bed.

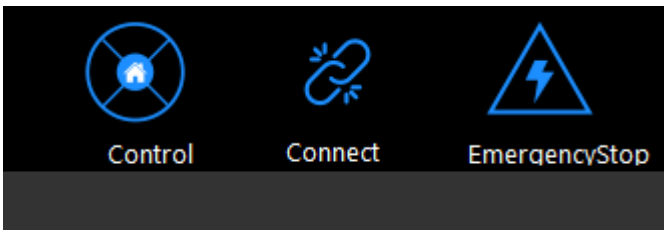


- Home: the position of the limit switch.
- Max/Min: defining the moving values of nozzle. If the value is negative, the nozzle goes beyond the heatbed. The values of bed left and bed front refer to the position of the heatbed at the beginning of the printing process. You could also set the positioning values of the nozzle in the X/Y direction.
- Print Area: the build volume of the printer.
- Extruder



Choose the number of the extruder. After all settings have been finished, click [Apply] to save it.

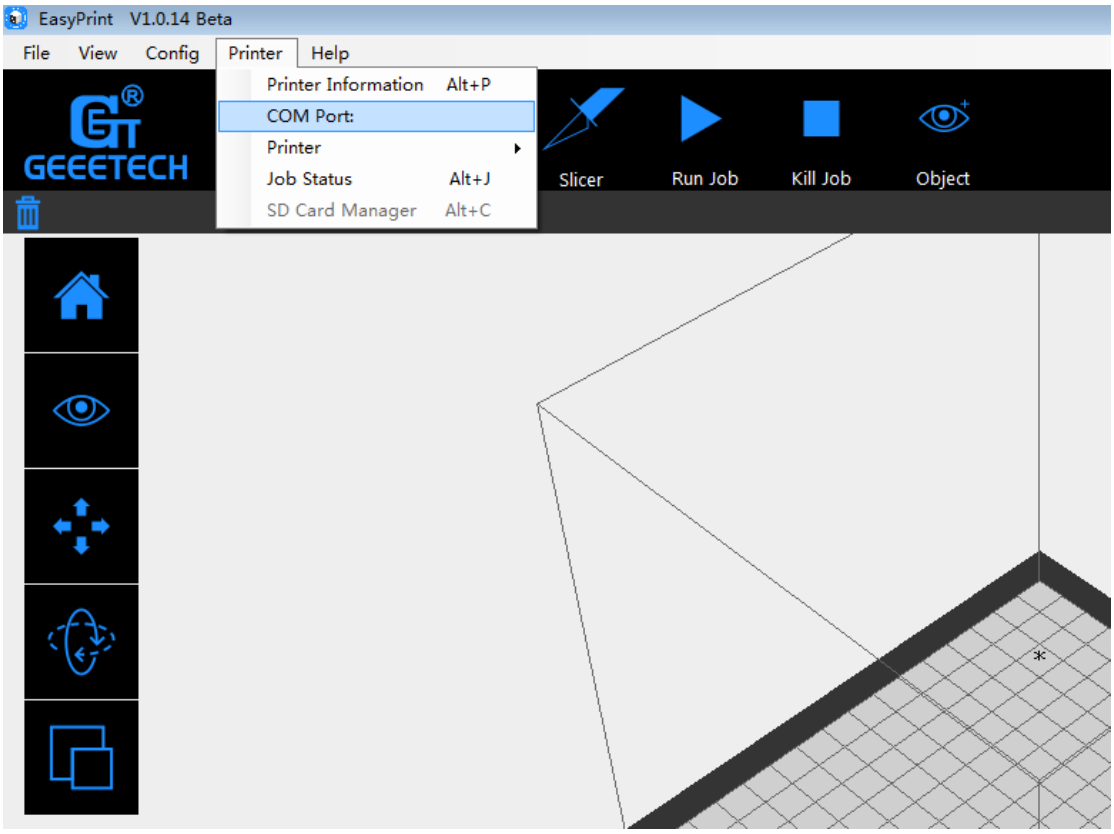
4. Click [connect] to connect the printer to the software.



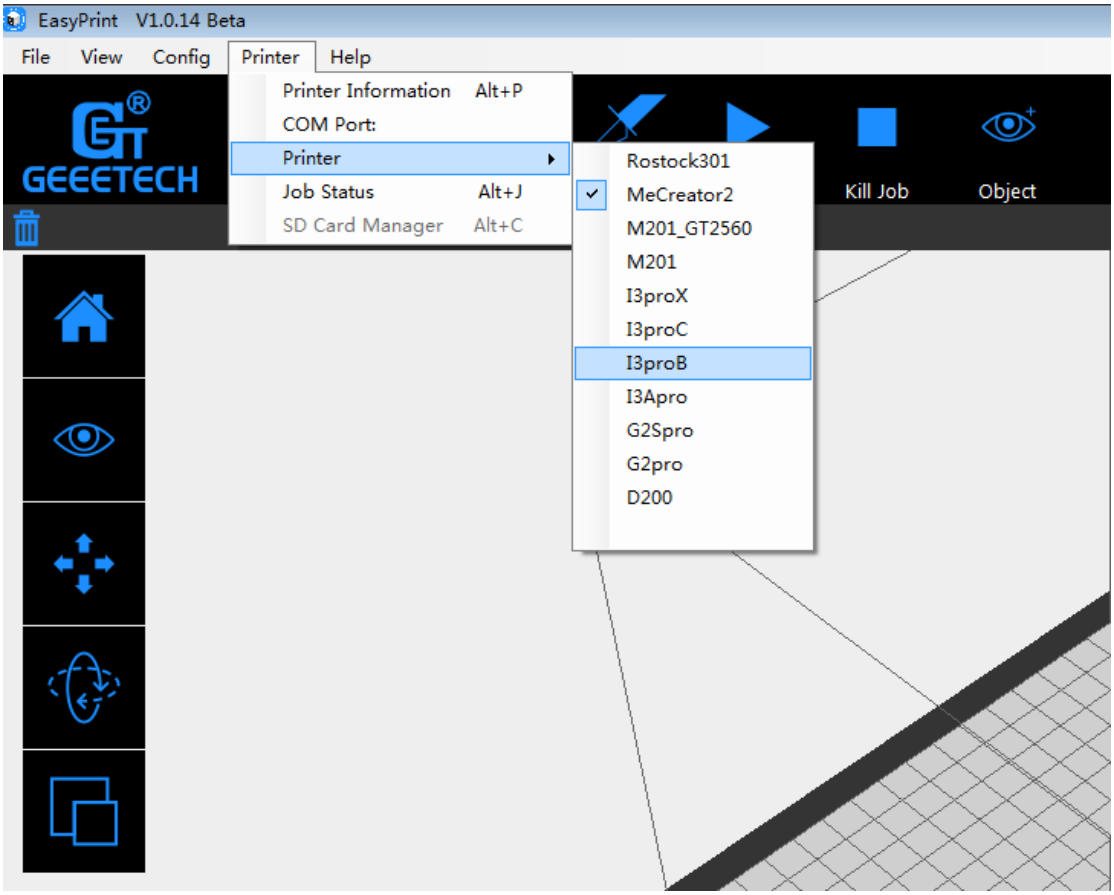
5. Entry-level printing

After learning the software and the leveling, you are ready to print your first model now, in this chapter; we will guide you through the steps to turn your 3D models into .gcode file the printer can recognize.

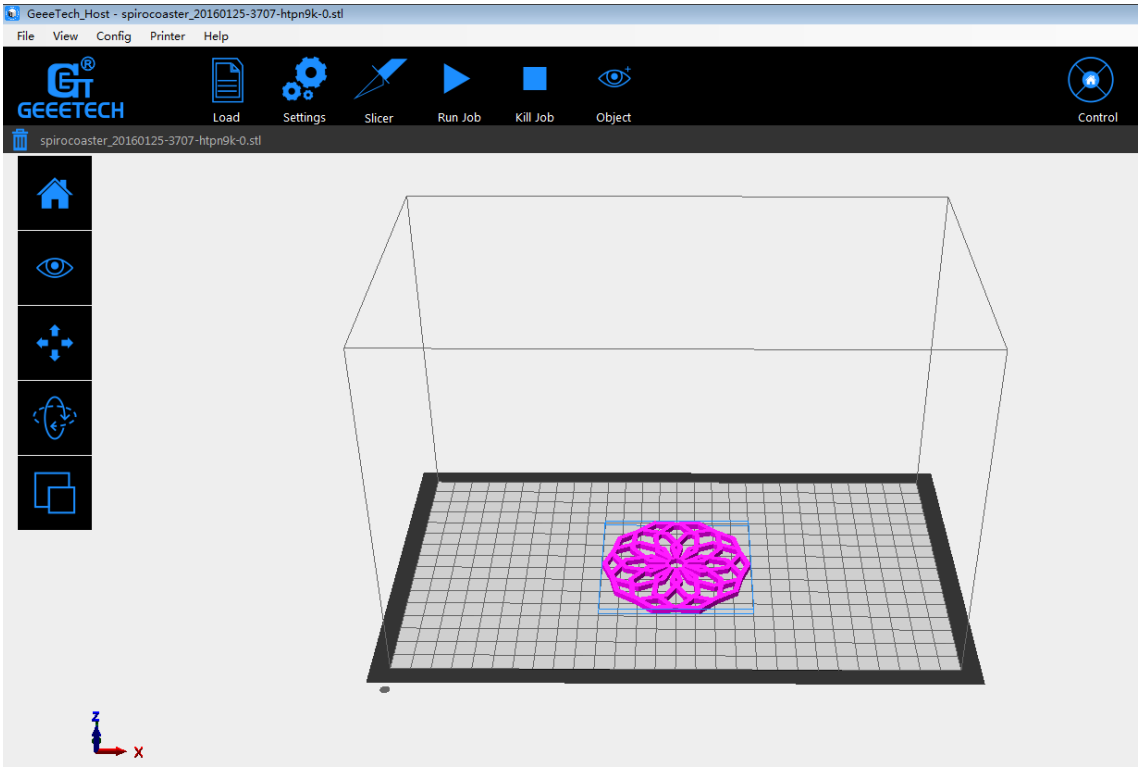
1. Connect the printer, choose [printer]-[COM Port].



2. Choose printer type as GiantArm D200.

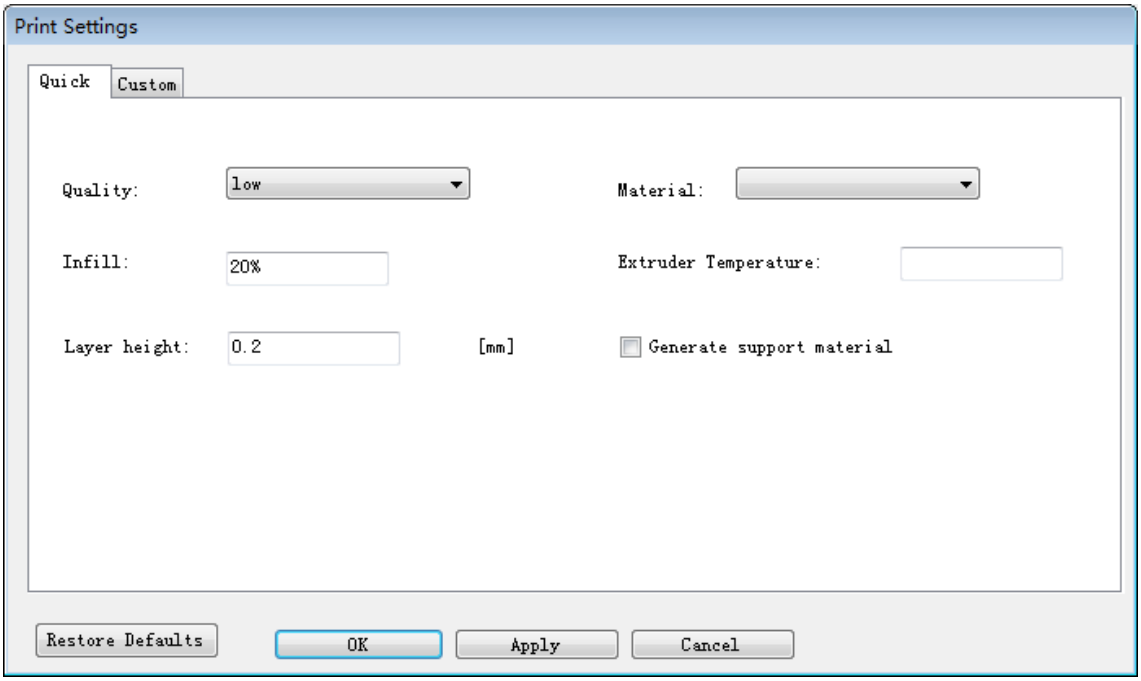


3. Click [load] to load a.stl from your computer, or you can drag it to the 3D view window. Now you would see your model at the center of a gray rectangle, the representation of your 3D printer's build plate. Then you can use the tools on the left to edit your model.



4. Slicing

Continue to set the print settings. Choose [quick] tab. Here, we have prepared 3 printing quality settings for you.



- Quality

Choose Low, Standard, or High quality to specify the surface quality of your 3D

print.

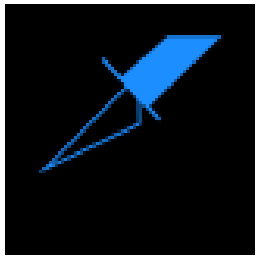
- Objects sliced with the High quality profile will have finer layers and will print more slowly.
- Objects sliced with the Standard quality profile will be printed using the default settings.
- Objects sliced using the Low quality profile will be printed with thicker layers and will print faster.

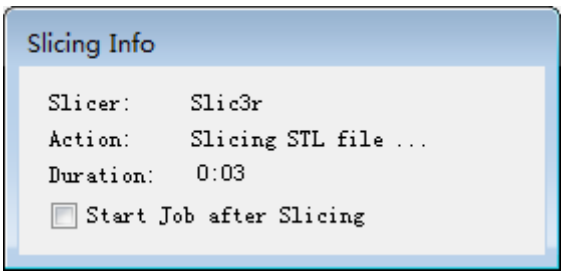
- Infill: The density of the infilling structure.
- Material: Please select PLA/ ABS.
- Extruder temperature: Different filament requires different temperature for extruders; please refer to the suggested temperature.
- Layer height: the thickness of each layer. You can leave it as default.
- Supports

Select this checkbox to have your object printed with support structures. Easy Print will automatically generate supports for any overhanging sections of your object. Supports will be easily removable once you remove your finished object from the build plate.

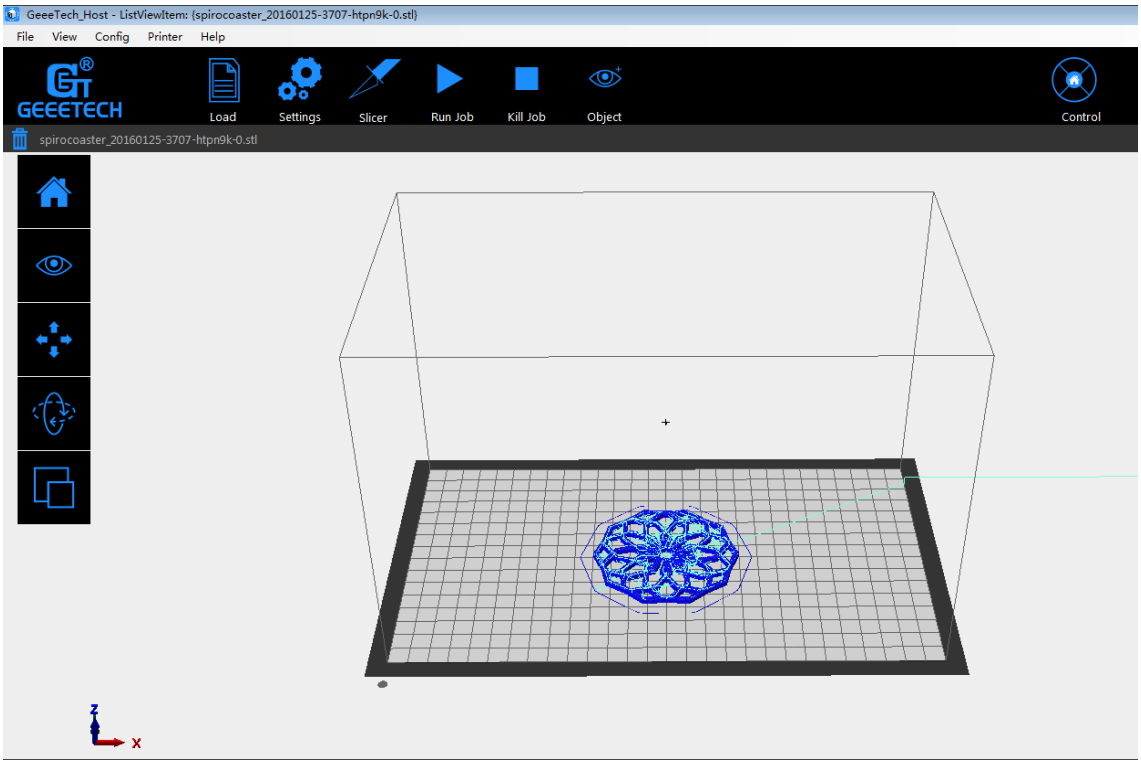
When you have finished the setting process, click [Apply]. The current settings will be used to slice your model next time you print or export a print file.

5. Click the [slicer] icon to slice. It will take a while; the slicing time depends on the size of the model.





6. Upon slicing, you can see the sliced model.



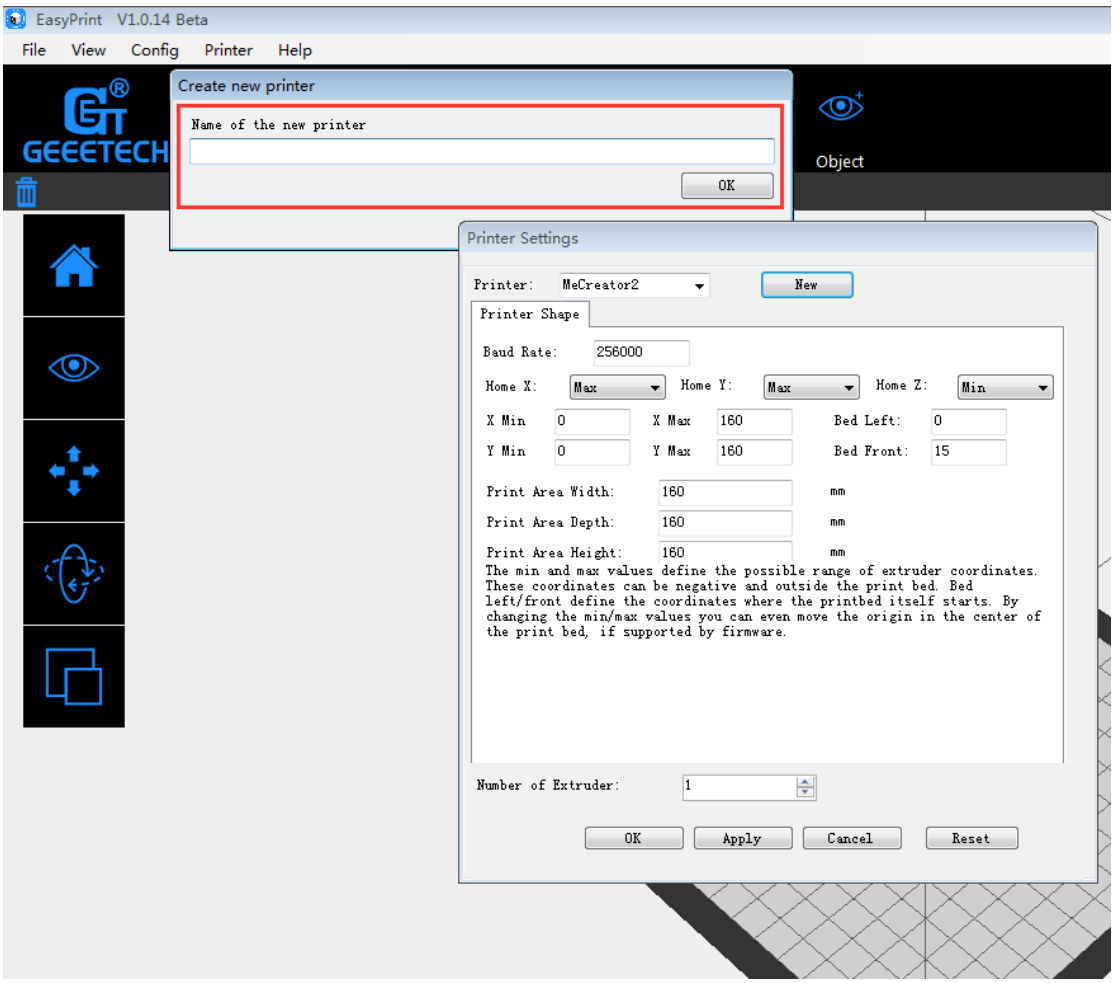
6. When you are ready to print your model, click [Run job] to start the printer.

6. Advanced-level printing

For advanced users who want to exert more sophisticated control over the whole printing process, Easy Print provides a series of parameters for you to engineer the printing configuration.

6.1 Create a new configuration/setting

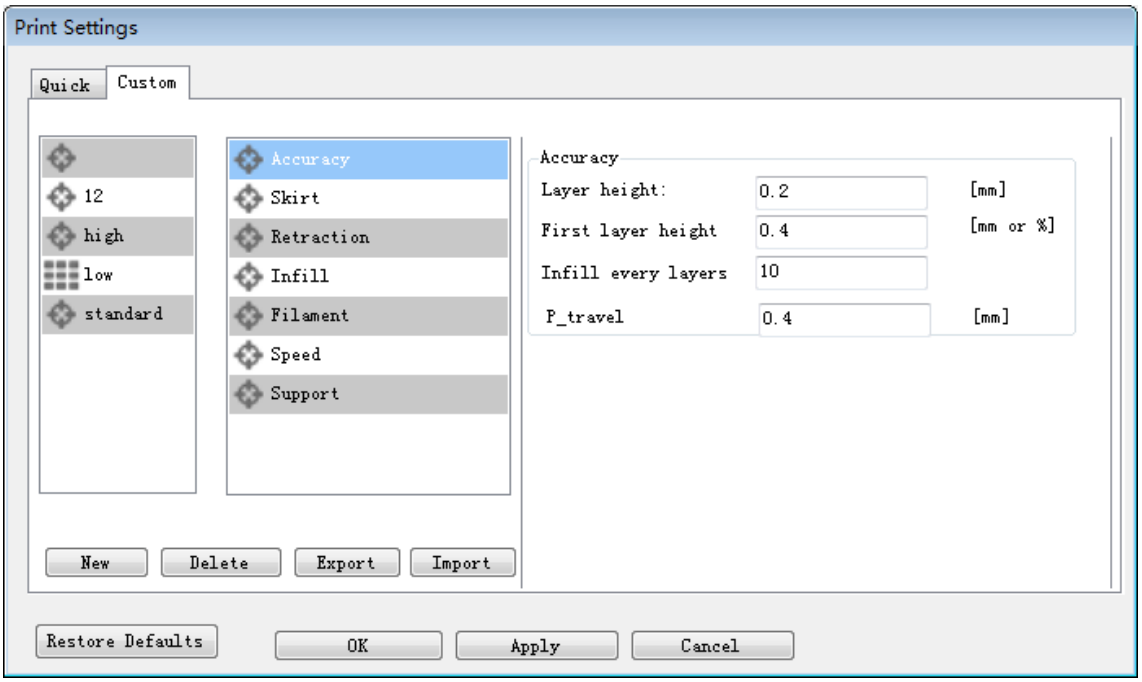
To customize settings, you need to create a new setting first. Click the [create new conf.] tab and enter the name of new configuration in the textbox. Click [Ok].



6.2 Print settings

Choose the [custom] tab to enjoy the sophisticated print settings.

1) Accuracy：

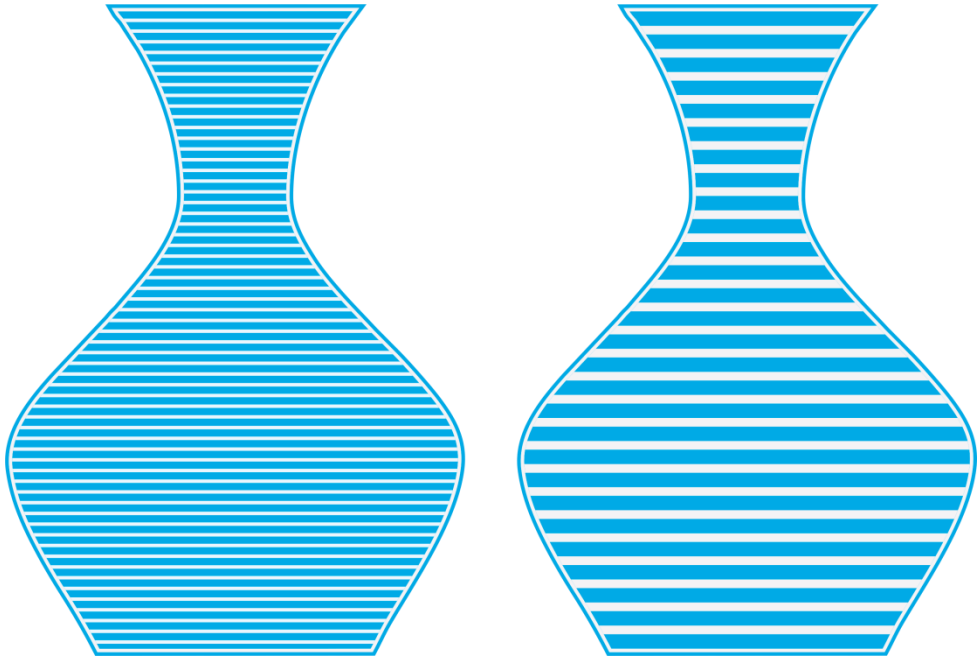


a. Layer Height:

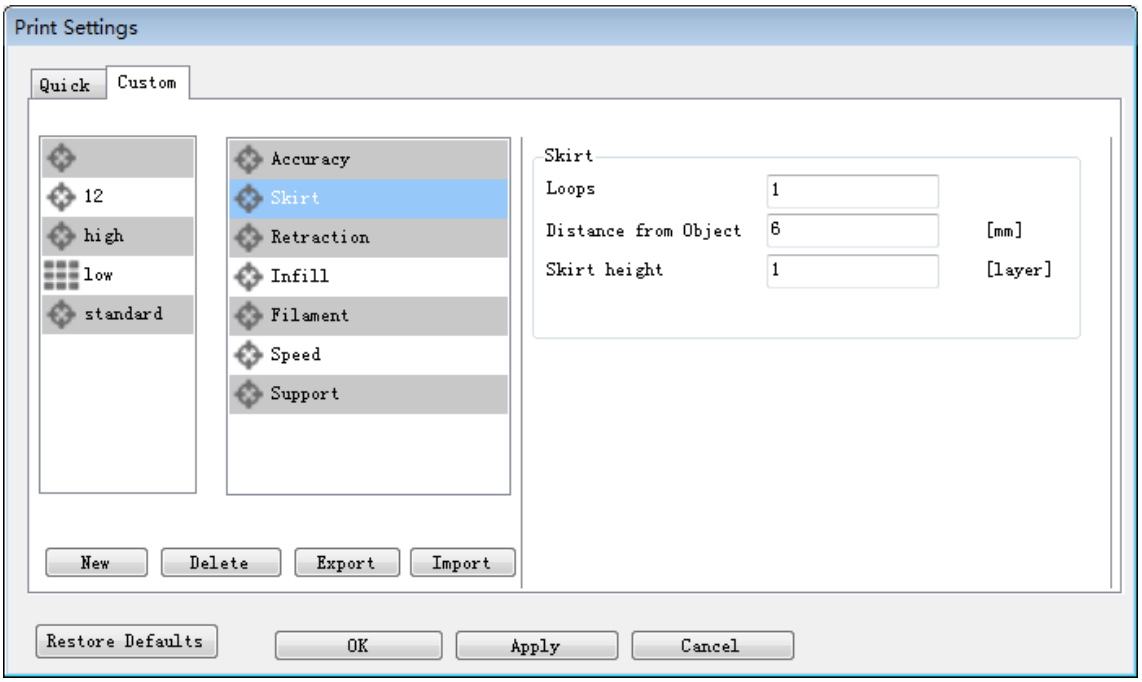
The layer height is one of the most often adjusted settings. It is the thickness of one printed layer (in mm). With a thinner layer height you will increase the quality of the print, leading to a smoother surface and more detail visible in the Z-direction (height) of the model. On the other hand, by using thicker layers you can decrease the print time substantially.

b. First Layer Height.

This setting defines the height of the first layer of your print. The First layer height will influence the adhesion between the model and the build platform. The maximum value is 0.4mm. Usually, it is suggested to use the default value.



2) Skirt:

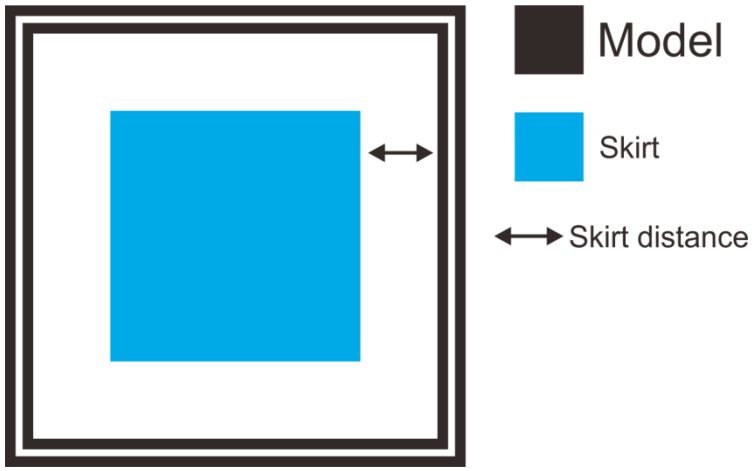


A skirt is a line printed around the object on the first layer, but not connected to the object. This helps prime the extrusion and can also be a good check for bed leveling before the print starts and it can ensure that the material is flowing smoothly from the extruder before it starts on the model proper.

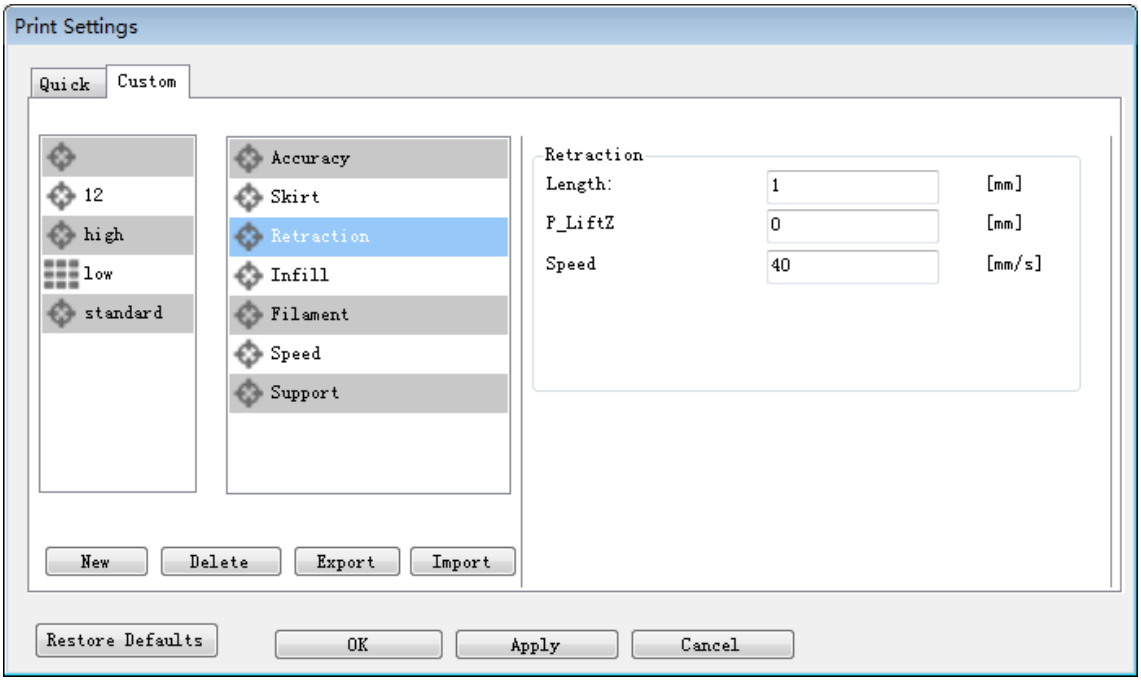
a. Loops: The number of skirt lines printed around the model. One loop is usually sufficient.

b. Distance from object: The distance between the object and the skirt. The default of 6mm is usually sufficient.

c. Skirt height: The number of layers to lay down a skirt for. For ensuring the material is flowing smoothly, one layer is sufficient, however the skirt function can also be used to build walls around the object in case it should be protected from draughts.



3) Retraction:



Retraction is used at the places in a print where the printer has to do a travel move between two printed parts. Without retraction extruded material will hang between the parts. This means that the filament is pulled back by the feeder, so that it doesn't leak from the nozzle during the travel moves. By using retraction, "stringing" (thin threads of plastic in between the printed parts) is prevented, resulting in a much cleaner final model. You have to be careful with flexible materials or models that require a lot of retractions, as that might lead to grinding of the filament.

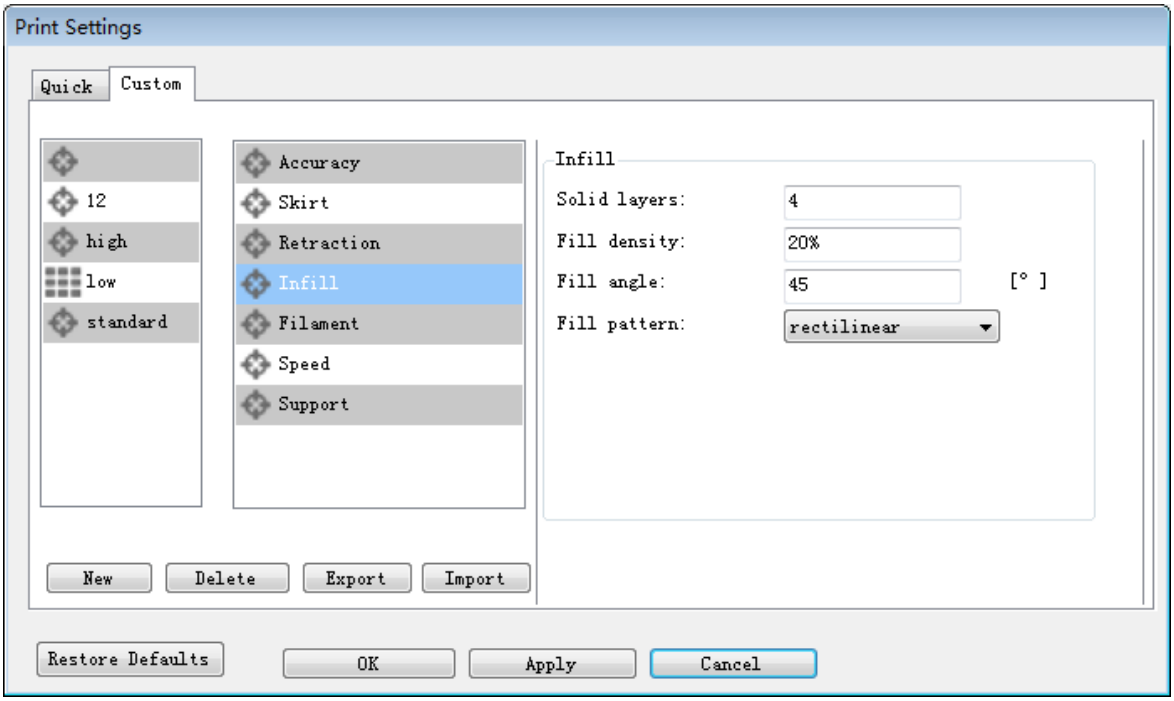
a. Length: The number of millimeters to retract. Note that the measurement is taken from the raw filament entering the extruder. A value of between 1 and 2mm is

usually recommended. Bowden extruders may need up to 4 or 5mm due to the hysteresis introduced by the tube.

b. Lift Z: Raises the entire extruder on the Z axis by that many millimeters during each travel. This can be useful to ensure the nozzle will not catch on any already laid filament; however it is usually not necessary and will slow the print speed. A value of 0.1mm is usually sufficient.

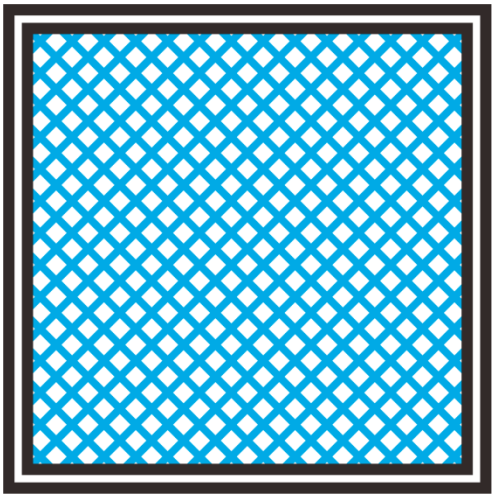
c. Speed: The speed at which the extruder motor will pull back the filament. The value should be set to as quick as the extruder can handle without skipping steps, and it is worth experimenting with this value to find the quickest retraction possible.

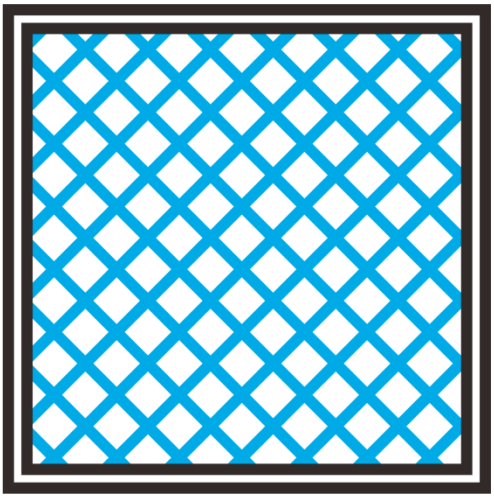
4) Infill:



a. solid layers: Forces a solid fill pattern on the specified layers. Zero will disable this option.

b. Fill Density: The infill density defines the amount of plastic used inside the print. A higher infill density means that there is more plastic on the inside of your print, leading to a stronger object. An infill density around 20% is used for models with a visual purpose; higher densities can be used for end-use parts.



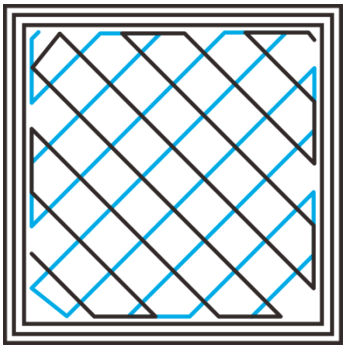


c. Fill angle: By default the infill pattern runs at 45 °to the model to provide the best adhesion to wall structures. Infill extrusions that run adjacent to perimeters are liable to delaminate under stress. Some models may benefit from rotating the fill angle to ensure the optimal direction of the extrusion.

d. Fill Pattern: It is the pattern of filling shape which effects printing duration.

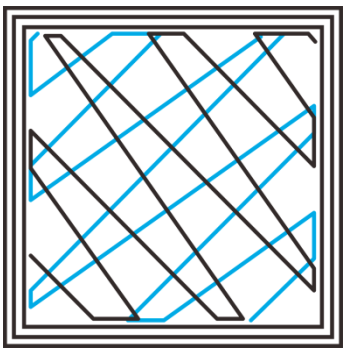
Easy Print offers seven infill patterns, including four regular flavors, and three more exotic ones. The numbers on the right of each figure are a rough estimate of material used and time taken for a simple 20mm cube model. Note that this is only indicative, as model complexity and other factors will affect time and material.

1. Line ：



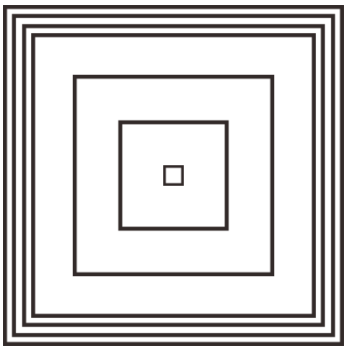
350.57mm/5m: 23s

2. Rectilinear



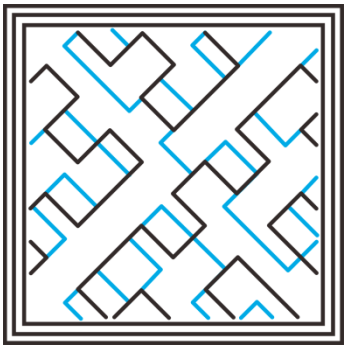
344.51mm/5m: 20s

3. Concentric



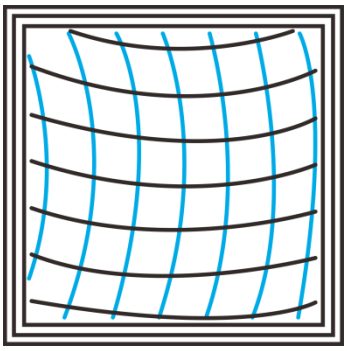
344.51mm/5m: 20s

4. Hilbert Curve



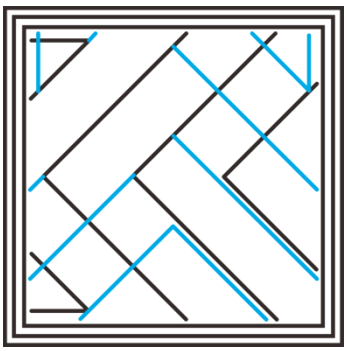
332.82mm /5m: 28s

5. Archimedean Chords



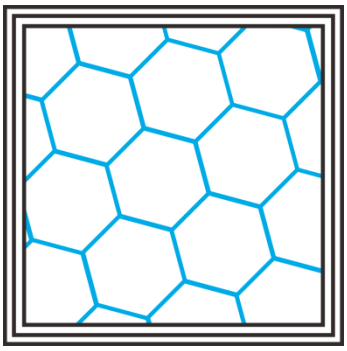
333.66mm /5m: 27s

6. Octagram Spiral



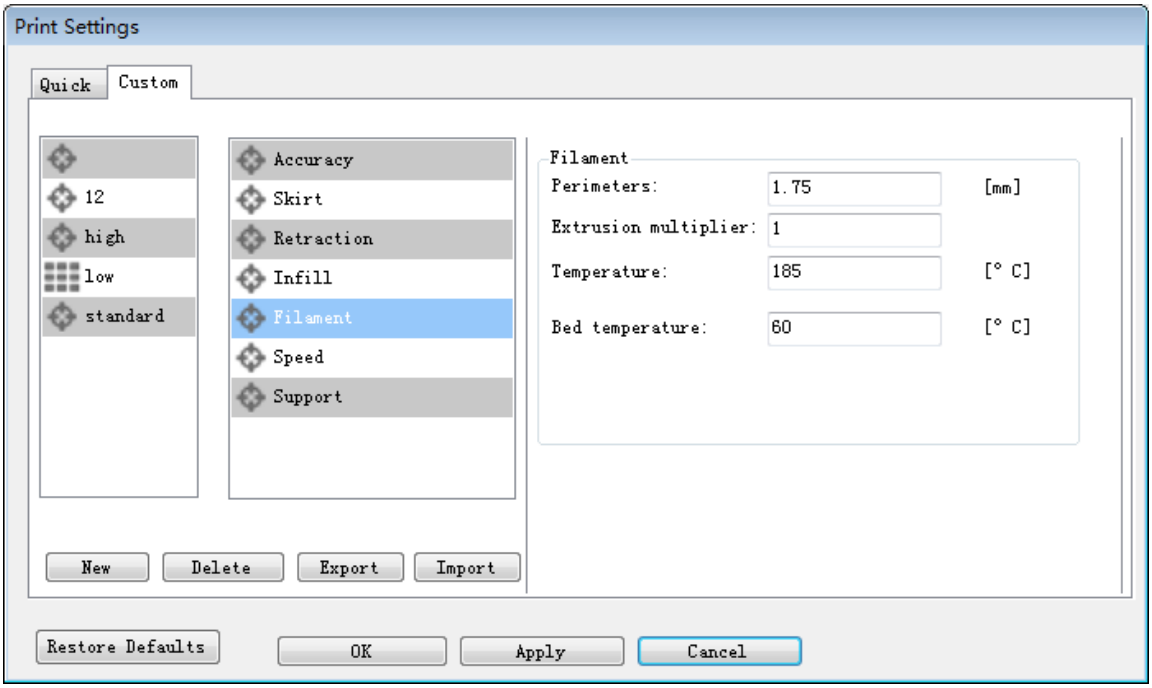
318.63mm/5m: 15 s

7. Honeycomb



362.73mm /5m: 39s

5) **Filament**:



a. Diameter:

In this field you can enter the diameter of the filament, so that Easy Print could calculate the extrusion rate correctly. The most commonly used filament diameters are

1.75mm and 3mm, but it might be necessary to change this setting when using filament from other suppliers.

b. Extrusion multiplier:

The extrusion multiplier setting allows the fine tuning of the extrusion flow rate, and it is given as a factor, e.g. 1 means 100%, 1.5 would mean 150%. Whilst the value should ideally be set in the firmware, it can be useful to test slight changes to the rate by altering this value. It varies the amount of plastic proportionally and should be changed in very small steps (e.g. +/- 0.05) as the effects are very visible.

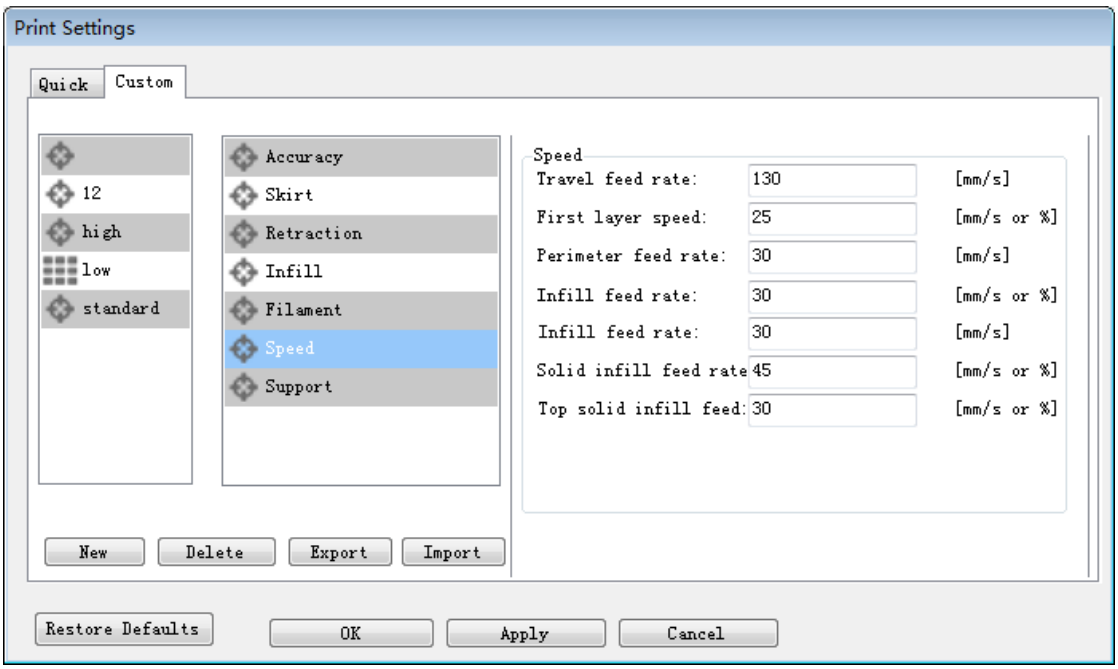
c. Extrusion temperature:

This is the temperature of the nozzle that is used during printing. The extrusion temperature can differ per material. PLA, for example, needs a printing temperature of approximately 190 -210 ℃, while ABS needs a higher temperature of 230 - 250 ℃ to properly extrude the plastic.

d. Heatbed temperature

For printers with a heated bed, the temperature of the heated bed can also be set here. We recommend using 60 ℃ for PLA and 90 ℃ for ABS to ensure that the plastic sticks well to the build plate.

6) Speed:



The print speed defines at which speed (in mm/s) the print head moves while

printing. Based on this setting, Easy Print will also calculate how fast the filament must be extruded. A higher print speed will lead to a shorter print time. But keep in mind that increasing the print speed means that you might have to increase the temperature as well, to ensure the plastic is properly melted.

Although you can choose one overall print speed for the complete print, it's also possible to use different print speeds for specific parts of the print:

a. Travel feed rate: This is the speed at which the print head moves when it's not extruding, that is, when the print head is moving from one point to another. A higher travel speed decreases the chance of filament leaking from the nozzle, producing a cleaner object. On the other hand, a very high speed means that the nozzle can hit an already printed part so fast that it might be damaged or deformed by the hot nozzle. This can be prevented by using lift z when retracting though.

b. First layer speed: With this setting you can specifically change the speed for the first layer of the print. By default a low speed is used for the bottom layer, so that the material sticks well to the build plate on the first layer. As mentioned in the above section, the first layer is important to lay down correctly, and a slower pace helps enormously. Setting a value of 50%, or even less can really help.

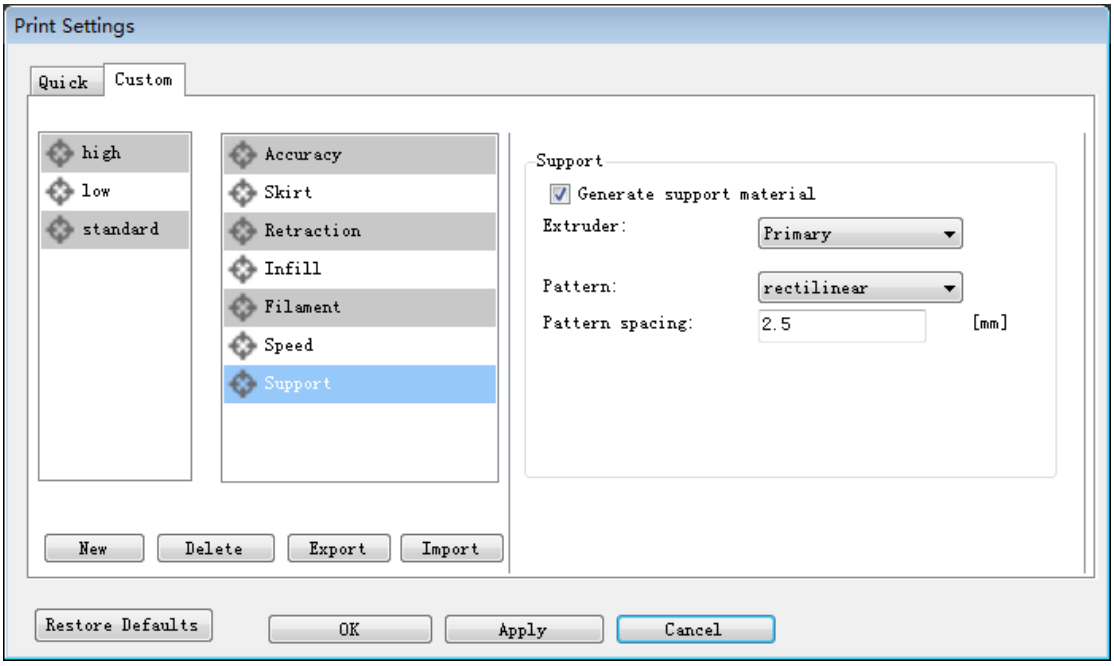
c. Perimeter feed rate: The speed at which the walls are printed. You can also set the speed for the outer and inner wall separately. Printing the outer wall a bit slower usually results in a better surface finish.

d. Perimeter feed rate: The speed of printing the wall. You can set the speed of printing the exterior wall and the interior wall respectively. Usually, printing the exterior wall at a low speed will bring smooth surface for your final product.

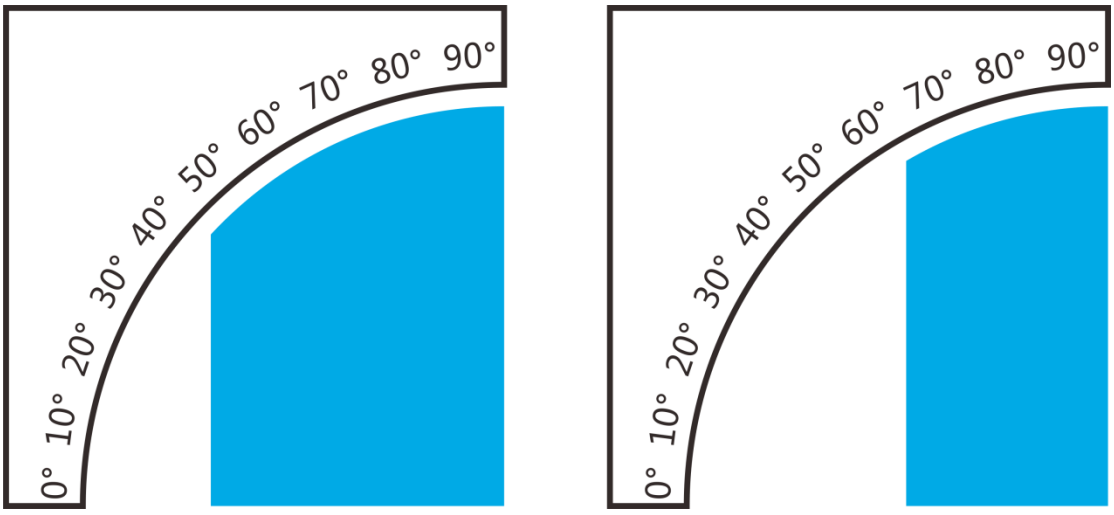
e. Infill feed rate: The speed at which the infill material is printed. Since (visual) quality of the infill is not important, you could use a higher speed for the infill. But keep in mind that this might affect the strength of your print.

f. Top solid infill speed: The speed at which the top layers are printed. A lower speed increases the reliability of closure of the top layers, especially on large area prints.

7) Support:



Generally, most 3D models will print with overhanging parts by up to a certain degree. The angle is determined by several factors, most notably layer height and extrusion width, and is usually around 45°. For models with larger overhangs, a support structure may have to be printed below it. This incurs the use of more material, longer print times, and post-printing clean-up.



Support (the blue part)

a. Generate support material

Some models have overhanging parts, which means that parts of the model float mid-air when you would print the model. In this case you must use a support structure

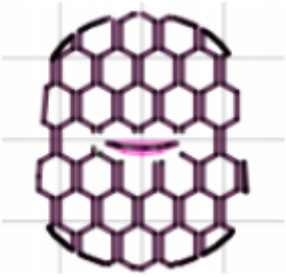
under the model to prevent the plastic from falling down. This can be achieved by generating support material. Select this checkbox to have your object printed with support structures.

b. Extruder

If you are printing with dual extruder, you can select one of them to print the support material.

c. Pattern

There are different patterns available for printing support structures, resulting in sturdiness or the easiness to remove the support. You can choose from the following patterns.



(1) Honeycomb

(2) Rectilinear

d. Pattern Spacing

Pattern Spacing determines the distance between support lines, and is akin to infill density apart from being defined only in mm. The adjustment of this attribute by taking into account the width of the support extrusion and the amount of support material will adhere to the object.

Care should be taken to choose a support pattern which matches the model, where the support material attaches perpendicularly to the wall of the object, rather than in parallel, so it will be easy to remove. If the support structure does run along the length of a wall, the Pattern Angle option allows the direction of the support lines to be rotated.

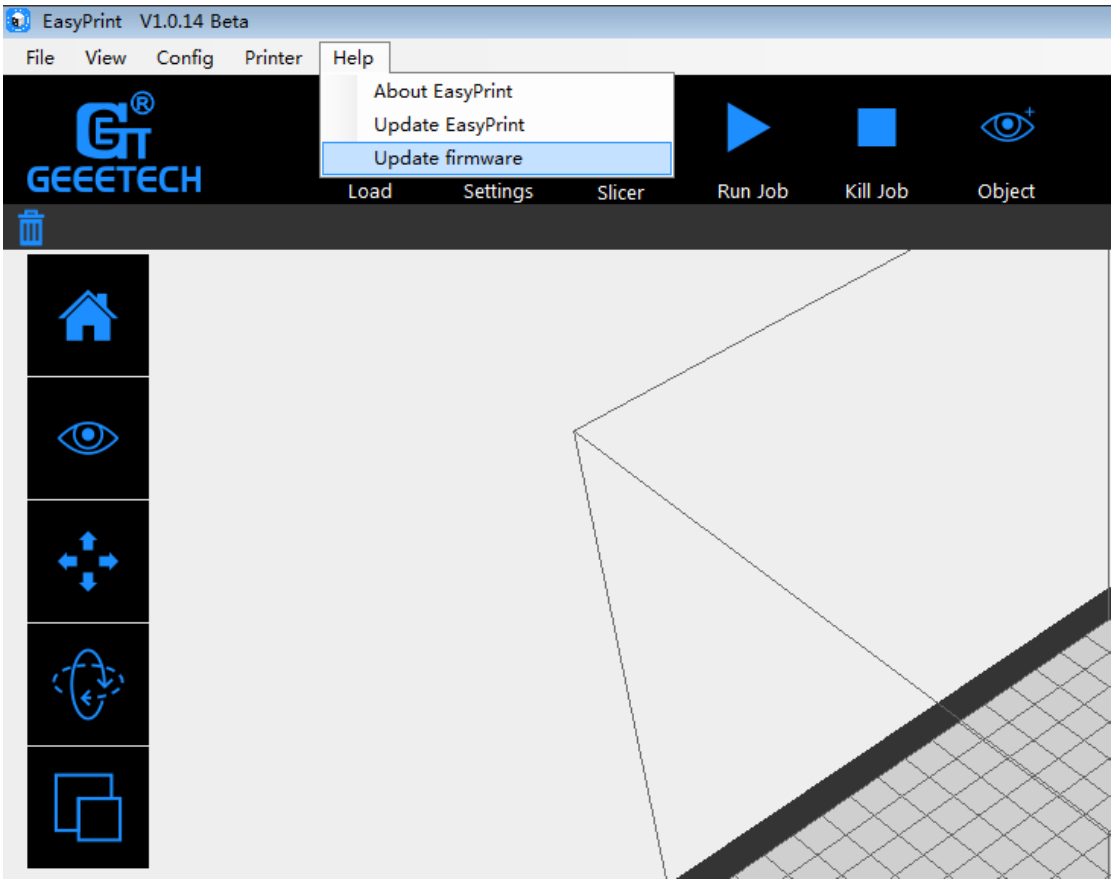
Now all the parameters are set. Click [Apply] and [OK].

7. Upgrade Firmware

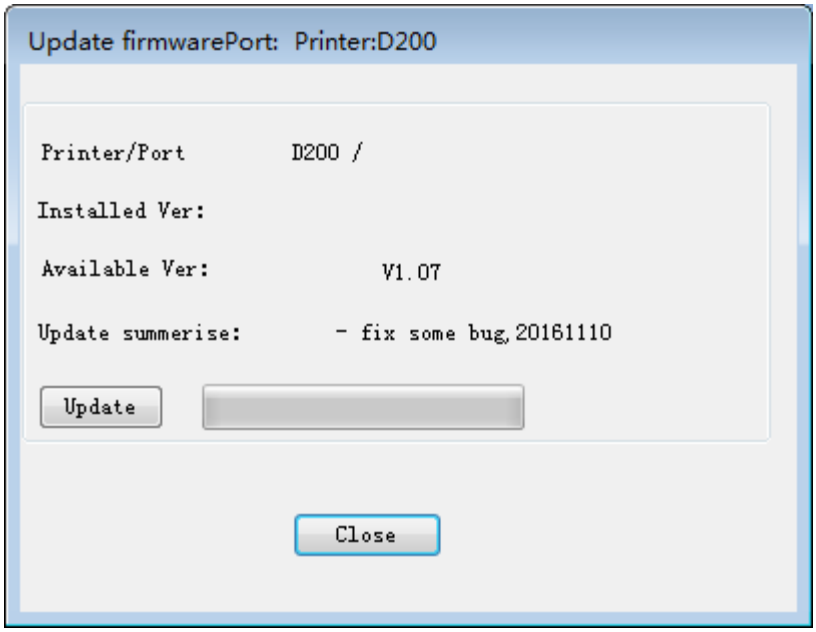
Easy Print provides all the users with online service of upgrading firmware to guarantee perfect printing performance.

The specific method is as follows:

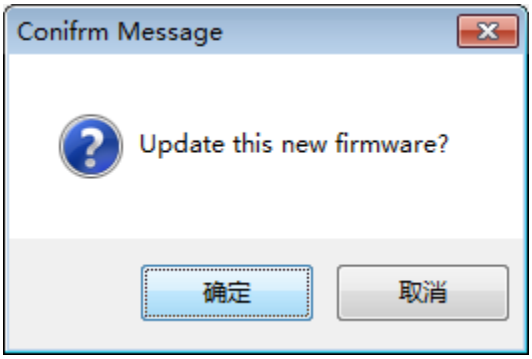
1. Connect your GEEETECH 3D printer to the computer with the USB cable.
2. Turn the printer on.
3. In Easy Print, choose the COM port and the printer type.
4. Click ’help’> 'Update firmware'



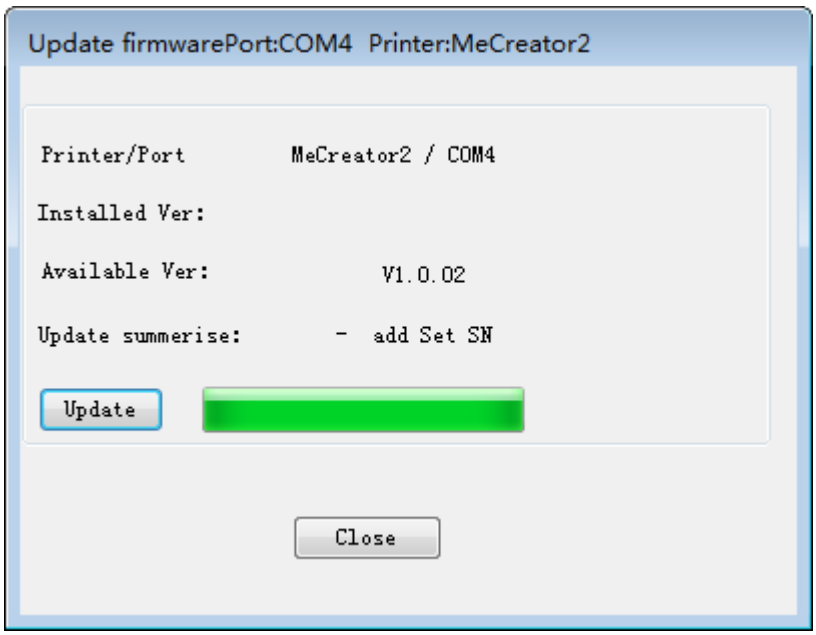
After clicking the [upgrade firmware] tab, a window will pop up with the information of printer type and port, the current firmware version installed, the available version and the update summary of the latest firmware.

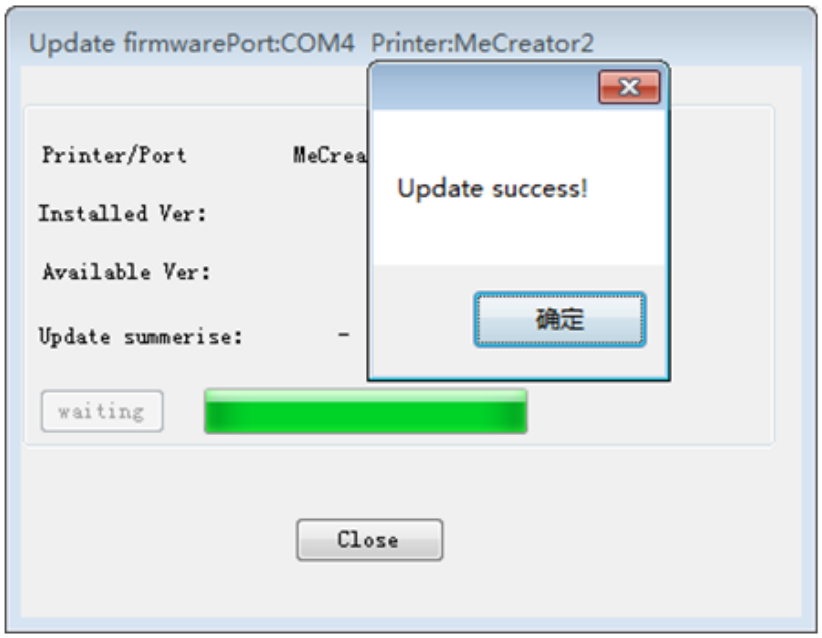


Click [**update]** and confirm updating, the progress bar will turn green; the firmware will be updated automatically. This process may take some time.



When the upload is complete the message ‘update successes will be displayed.



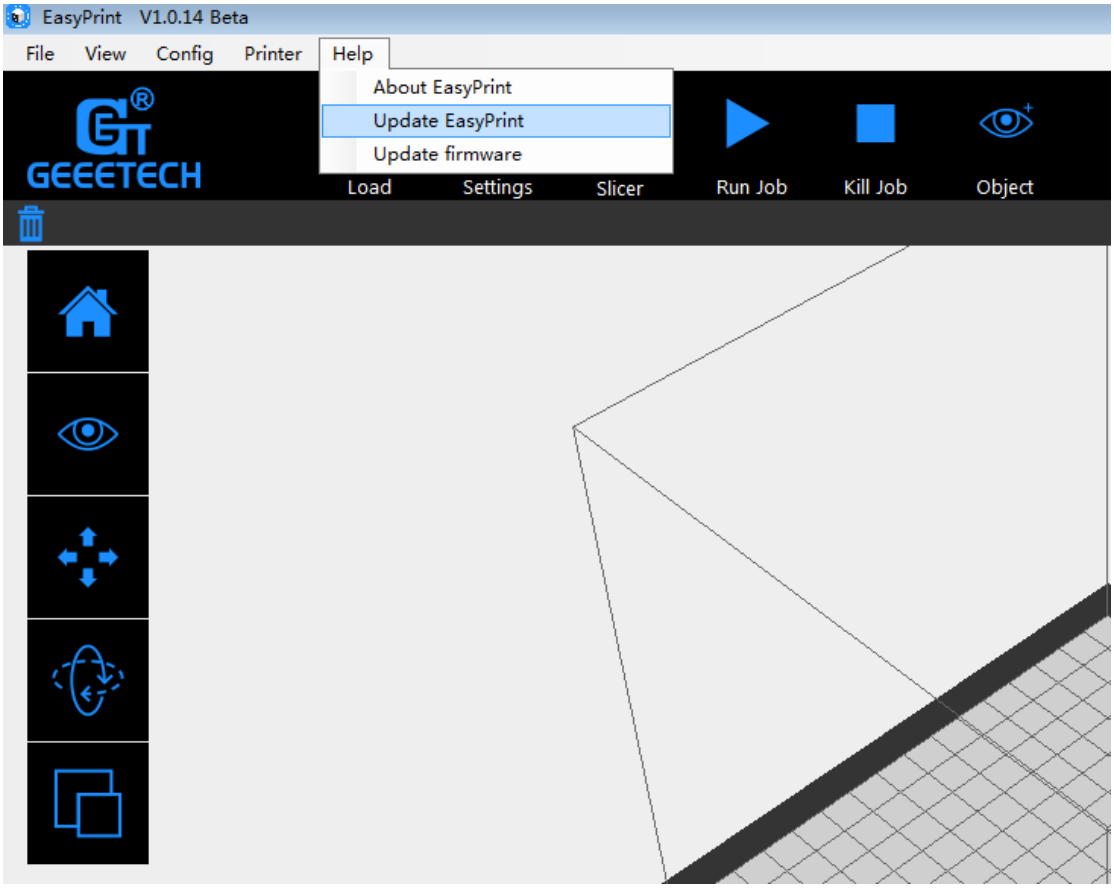


Click ‘ok’ and ‘Close’ on the windows to complete the firmware upgrade.

8. Upgrade Easy Print

When a new version of Easy Print is available, we will provide all the users with online upgrading service so that you could upgrade the software immediately. When you start Easy Print, a dialog box will appear as follows.

You can also check for update at [help]-[update easy print]



If there is a new version available, just click the Download button and launch the upgrade Setup Wizard.

Note: Please keep your Easy Print in the newest version to enjoy additional functions and excellent printing performance.

9. Feedback

This is beta software and it is our first 3D printing software, there might be a lot of bugs and problems, if you meet any bugs when using it, please feel free to report it to us through our forum. Your feedback is highly appreciated.

www.geeetech.com