# Print Settings

## Layer Height

Layer Height – This describes the height of each layer in millimeters (mm).

First Layer Height – This describes the height of the first layer specifically, in millimeters (mm).

## Vertical Shells

Perimeters (minimum) – The vertical shell is the outer most layer on the sides of the printed object. The perimeter determines the size of the outer layer. If you set this to 0, then no vertical shell will be printed.

Spiral vase – When this box is checked, the shell will be printed in a spiral vase, which makes the appearance of the object look clean. You will not as easily see lines where each layer was printed.

## Horizontal Shells

Solid Layers (Top) – determines the size of the top outer layer of the printed object. If you set this to 0, then no Top layer will be printed.

Solid Layers(Bottom) – determines the size of the bottom layer of the printed object. If you set this to 0, then no bottom layer will be printed.

## Quality

These options allow the user to instruct the system to perform a slower printing algorithm in order to increase the quality of the printed object. The object will be printed at a slower pace, but the quality of the object will be more precise.

Extra perimeters if needed –

Avoid crossing perimeters –

Start perimeters at:

Concave points – Starts the layer at the lowest point in the object.

Non-overhang points – Reminds the system not to start a layer at an overhang. Instead, it will find an area that is sturdy with support underneath.

Detect thin walls – The system will detect when a wall is thin and better handle them appropriately.

Detect bridging perimeters –

## Advanced

Randomized starting points – The printer will pick a point at random to start. This can help reduce the markings in the final product and make the layer better blend together visually.

External perimeters first – Always start each layer on the outside, then fill inwards.

## Infill

These options determine how the inside of the object will be printed. Typically, the object is not completely solid and features a special infill pattern to make it lighter and less costly to produce.

Fill Density – determines how dense you want the object’s inside to be. A higher fill density results in using more material and a heavier object.

Fill Pattern – determines which pattern you want to use for the object. You can choose from rectilinear, line, concentric, honeycomb, Hilbert curve, Archimedean chords, or octagram spiral.

Fill Pattern (Top/Bottom) – determines the fill pattern for the bottom and top layers specifically. You can’t use linear or honeycomb for these layers.

## Reducing Printing Time

These options, when used, can help reduce the time it takes the print the object, but may sacrifice the precision of the print.

Combine infill every \_\_\_ layers – Instead of printing each layer individually, the printer can print \_\_\_ layers at once.

Only infill where needed – Checking this option will stop the printer from printing infill except where it will affect the reliability of the structure.

## Advanced Infill

Solid infill every \_\_ layers – Checking this option will cause the printer to print one solid layer at a chosen increment instead of using the selected fill pattern.

Fill angle –

Solid infill threshold area (mm2) –

Only retract when crossing perimeters – Instead of the usual retraction pattern for the head, the head will only retract when crossing perimeters.

Infill before perimeters – Instead of printing the outside perimeter, the printer will print the infill first.

## Speed for Print Moves

The options here will determine the max speed the printer will move when printing under those conditions.

Perimeters – Determines the speed at which the printer will print perimeters

Small Perimeters – Determines the speed at which the printer will print small perimeters

External Perimeters – Determines the speed at which the printer will print external perimeters.

Infill – Determines the speed at which the printer will print the infill.

Solid Infill - Determines the speed at which the printer will print solid infill.

Top Solid Infill - Determines the speed at which the printer will print the top solid infill

Support Material - Determines the speed at which the printer will print the support material.

Bridges - Determines the speed at which the printer will print bridges in the object.

Gap Fill - Determines the speed at which the printer will fill in gaps.

## Speed for Non-Print Moves

Travel – This determines how fast the printer head will move when it is not extruding material.

## Speed Modifiers

First Layer Speed – This affects how fast the printer prints the first layer compared to the other layers.

## Acceleration Control

These options determine how fast the print head can accelerate how performing these tasks.

Perimeters, Infill, Bridge, First Layer, Default

## Skirt

The skirt is a routine the printer performs where it extrudes material away from the object itself to ensure that only good material is used in the actual print job.

Loops – This option determines the number of loops the print head takes when printing the skirt

Distance from Object – This determines how far from the object the skirt should be printed.

Skirt Height – This determines the height of the skirt (measured in units of layers)

Minimum Extrusion Length – The skirt must extrude at least this much material (in mm) before printing the object.

## Brim

Brim Width – This determines the width of the brim in mm.

## Support Material

Sometimes an object that you try to print will not be stable while printing. In cases such as these, you should use support material to ensure that the object will not deform or fall apart while drying.

Generate Support Material – This check box determines if you wish to use support material at all for this print.

Overhang threshold – This is the allowed distance of overhang this material can handle without need for support material underneath.

Enforce support for the first \_\_\_ layers – You can require that a set number of beginning layers use material support.

## Raft

Raft layers –

## Options for Support Materials and Rafts

Pattern – Choose an infill pattern for the support material or raft.

Pattern Spacing – choose how much space between the patterns should be used.

Pattern Angle –

Interface Layers –

Interface Pattern Spacing-

## Output File

Verbose G-Code – Checking this option will cause the system to output all of the G-Codes with the output file. Unchecking this option will cause the system to only output the minimum amount of G-Codes to print the object.

# Material

## Filament

Diameter – This number describes how much diameter is needed in the size of the extrusion for this given material.

Extrusion Multiplier –

## Temperature

First Layer – This is how hot the extruder must be at for the material to print at the first layer.

Other Layers – This is how hot the extruder must be at for the material to print the remaining layers.

## Fan Settings

Keep Fan Always On – Having this option on will keep the fan on during the entire print run. Having it off will change it so that the fan is only on when necessary.

Enable Auto-Cooling – Having this option on will let the system decide when to cool the head.

Fan Speed (Min) – This is the slowest the fan is allowed to turn while it is running.

Fan Speed (Max) – This is the fastest the fan is allowed to turn while it is running.

Bridges Fan Speed – The fan might need to slow down when the system is printing the bridge. This option lets you set how much the fan will slow down when printing the bridge.

Disable Fan for the First \_\_\_\_ Layers – For the first \_\_\_\_ layers, the fan will not be turned on.

## Cooling Thresholds

Enable Fan if Layer Print Time is Below \_\_\_ Seconds – The fan will turn on when the print time is below the threshold given.

Slow Down if Layer Print Time is Below \_\_\_\_ Seconds – The fan will slow down when the print time is below the threshold given.

Min Print Speed – This it he minimum print speed while….

## Retraction

Length – How far away the extruder will move when retracting.

Lift Z – This is how high the extruder will move when retracting.

Speed – This is how fast he extruder will move when retracting

Extra Length on Restart – This is how much extra length the extruder will extruder before returning to printing.

Minimum Travel After Retraction – This is how far the head must move after retraction before it can return to printing.

Retract on Layer Change – Having this setting on will require the head to retract after printing each layer

Wipe while retracting – Having this setting on will require the head to be wiped each time it is retracted.

## Retraction When Tool is Disabled

Length – How far the extruder will retract when disabled.

Extra Length on Restart – This is how much extra length the extruder will extruder when it is disabled

## Custom G-Code

These are special instructions that you can add to the beginning or end of the print when using this material.

Start G-Code – Here you can type your own G-Code instructions to be added to the beginning of the G-Code generated

End G-Code – Here you can type your own G-Code instructions to be added to the end of the G-Code generated.

# Extruders

## Size

Nozzle Diameter – This describes the diameter of the nozzle on the extruder. This is important to know to determine the compatibility between the extruder and the material.

## Position

Extruder offset (x,y) – These coordinates determine how much the system should compensate for the position of the head being off from its expected position.

## Custom G-Code

These are special instructions that you can add to the beginning or end of the print when using this extruder.

Start G-Code – Here you can type your own G-Code instructions to be added to the beginning of the G-Code generated

End G-Code – Here you can type your own G-Code instructions to be added to the end of the G-Code generated.

## Printer

Bed (X,Y) – These coordinates describe the size of the print bed. This will let the system know when it is in danger of going out of bounds

Z Offset – This coordinate describes how much elevation the bed has.

Print Center (X, Y) – These coordinates describe where the center of the print job is.

Bed Temperature First Layer – This determines how hot the bed needs to be when the first layer is being printed

Bed Temperature – This determines how hot the bed needs to be when the other layers are being printed.

Printer Port – ***This is an important option to establish for your Printer.*** This drop down will show the list of current printers that the system detects are connected to the printer. Pick the printer that you wish to describe in this Printer Setting.

Baud Rate –

Line End –

## Section 2

G-Code Flavor – G-Code comes in slightly different variations that have a few different commands. It is important to specify what type of G-Code you are using.

Relative E Distance – Having this option on will let the printer determine position using calculated distance.

Number of Extruders – This number displays how many extruders you have attached to this printer. You can configure what extruders you wish to add with the drop down and the “Add Extruder” button. If you wish to remove an extruder, click the extruder from the list of extruders attached and then click the “Remove Extruder” button.

Vibration Limit –

Firmware Restract –

## Custom G-Code

These are special instructions that you can add to the beginning or end of the print when using this printer.

Start G-Code – Here you can type your own G-Code instructions to be added to the beginning of the G-Code generated

End G-Code – Here you can type your own G-Code instructions to be added to the end of the G-Code generated.

# Print Job

## Printer

Use this drop down menu to select the name of the printer you are using for this print job. You can only use 1 printer per print job.

## Extruder – Material

Use the drop to select from the list of materials you have saved to the system, then click the “Add Material” button to select that material to be used in the print. The material is assigned to an extruder in order of which you select the material. That is, the first material you add will be extruded by the first extruder attached to the printer. To change which extruder is attached to the printer, go to the Printer Menu.

To remove a material, simply click the name of the material from the list of material being used and then click “Remove Material”.

## Subsections

You need at least 1 subsection in order to start printing. First, determine the range of layers the subsection will be (In example, the first subsection could start at layer 0 and end at layer 21). Next, select what Print Configuration you would like this subsection to follow. Then, choose from the list of models that you would like to print. For each model that you are printing, you need to choose an Extruder to print it with. Use the Extruder drop down and the “Add Extruder” button to do so.

When you are ready to create another subsection, use the “Add Subsection” button in the top right hand corner, then fill out the new subsection the way you filled out the first one. Make sure that the subsections are not overlapping in any layer and that each new subsection begins where the last left on (In our example before, if the first subsection ends at layer 21, then the second subsection should start on layer 22.)

You can add as many subsections as needed. If you change your mind and wish to take out a subsection, click the “Remove” button next to where you describe the layer range.

## Start Print

After you have completely defined your settings, click the Start Print button to begin the physical printing process.