

Cura settings and functions

Click [Cura configuration](#) to enter Github to download the configuration file of cura.

V1.0

**You can search for the operation you need based on keywords.
(Search the entire document)**

- Since Cura's slicing function is roughly the same as that of other slicing software, you can also refer to this document when you use other slicing software.
- Mainly introduce some commonly used functions. Users who are already proficient in using Cura do not need to read this document.
- Each setting of Cura has notes. We only make some recommended settings.

Keywords

Quality

- Layer Height
- Initial Layer Height
- Initial Layer Line Width

Shell

- Wall Line Count
- Top Layers
- Bottom Layers
- Optimize All Printing Order
- Outer Befort Inner Walls
- Skin Overlap Percentage

Infill

- Infill Density
- Infill Pattern
- Connect Infill Lines
- Infill Line Directions
- Randomize Infill Start
- Extra Infill Wall Count
- Infill Overlap Percentage
- Infill Before Walls

Material

- Printing Temperature
- Printing Temperature Initial Layer
- Initial Printing Temperature
- Build Plate Temperature
- Build Plate Temperature Initial Layer
- Flow
- Infill Flow
- Skirt/Brim Flow
- Initial Layer Flow
- Standby Temperature

Speed

- Print Speed
- Infill Speed
- Outer Wall Speed
- Inner Wall Speed
- Top/Bottom Speed
- Travel Speed
- Initial Layer Print Speed
- Initial Layer Travel Speed
- Skirt/Brim Speed
- Number of Slower Layers

Travel

- Enable Retraction
- Retraction Distance
- Retraction Speed
- Combing Mode
- Z Hop When Retracted

Layer Height

The height of each layer, but not including the first layer. Because the height setting of the first layer is separate.

The higher the layer height, the shorter the printing time. For large models, a large layer thickness is recommended. But the layer thickness is limited by the diameter of the nozzle, usually the layer thickness is half of the nozzle diameter.

For example, the relationship between nozzle diameter and layer thickness is,

Nozzle diameter(mm)	layer thickness range(mm)
0.3	0.1~ 0.2
0.4	0.15 ~ 0.25

Nozzle diameter(mm)	layer thickness range(mm)
0.5	0.2 ~ 0.3
0.6	0.2 ~0.35
0.8	0.2 ~ 0.4
1.0	0.2 ~ 0.5
...	...

The greater the layer thickness, the shorter the printing time. The greater the layer thickness, the more layered the side of the printed model. Conversely, the smaller the layer thickness, the finer the side of the model, but a lot of time will be wasted. Please set it according to your actual needs.

Obviously, the small model is suitable for small layer thickness. For large models, you can use a small layer thickness or a large layer thickness.

Initial Layer Height

Usually, it should be a little bit higher than **Layer Height**, or the same. It is the first layer of the printed model.

Layer Height mainly affects the side of the model.

Initial Layer Height is to make this model safe to print on the platform. In order to prevent the model from being moved during the entire printing process, we need to use more material for the first layer of the model to make it more securely pasted to the platform. Therefore, it is usually a little bit higher than **Layer Height**.

Initial Layer Line Width

Its function is similar to that of **Initial Layer Height**. But it represents the width of each line of the first layer.

Usually its value is not less than the diameter of the nozzle, that is to say it is not less than 100%.

In individual cases, its value will be set to less than 100%. For example, the gap between area A and area B is 0.8 mm, and the diameter of the nozzle is 1.0 mm. We need to fill the area between A and B instead of leaving a gap, then we need to set its value to less than 80%.

The larger its value, the more material will be extruded.

Wall Line Count

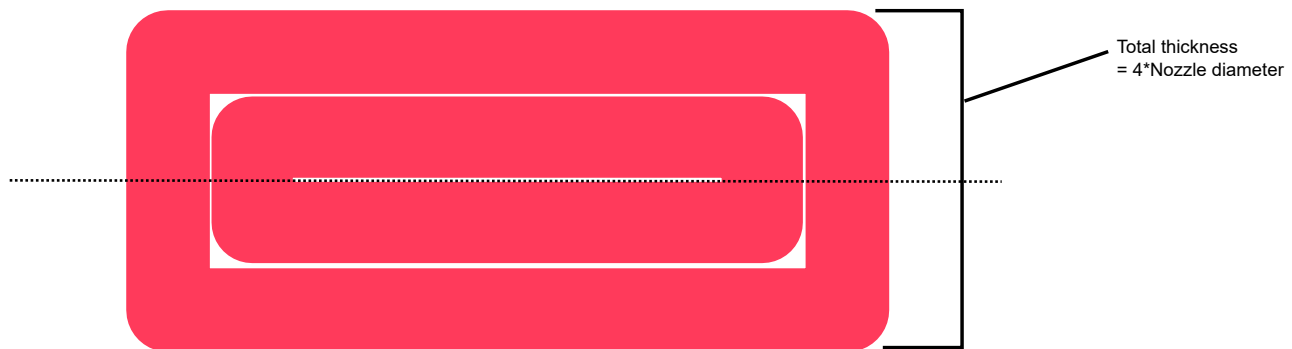
The thickness of the wall. Usually set to 2.

If the model is a vertical thin slice, and the thickness is less than 4 times that of the nozzle, you can consider setting it to 1.

Why is it 4 times? Because we assume its value is 2. Then it is a closed circle with two circles (2 times the nozzle diameter), then the total thickness is 4 times.

Each line is the width of a nozzle diameter

Wall Line Count = 2



Top Layers & Bottom Layers

Because **Infill** is used, we need to set its number according to the size of the **Infill Density**.

If the gap between fills is relatively large, **Top Layers** 's value should increase accordingly.

If the gap between fillings is relatively small, **Top Layers** 's value should be reduced accordingly.

Usually the **Infill Density** is 20%. Then it is set to 3 layers to completely cover the gap between the fillings.

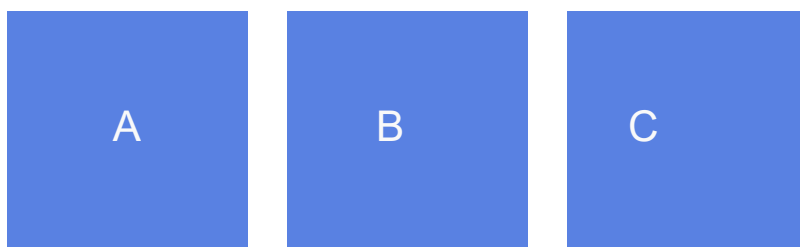
And **Bottom Layers** is the number of solid layers below the filling. Similar to the role of **Top Layers**.

Optimize All Printing Order

This can be seen in cura's introduction to it.

For its function description, the most intuitive is the embodiment of simplify3D.

For example, there are three separate areas to be printed on the same layer,



The printing order is ABC. When printing the next layer, it is CBA. In other words,
ABC -> CBA -> ABC -> CBA...

The function of this option is similar to it. Otherwise, it will be printed in the following order,
ABC -> ABC -> ABC -> ABC...

Obviously, `ABC -> CBA` is a better model. Because reduce the distance of travel. It also reduces the possibility of hanging silk.

Outer Befort Inner Walls

It is recommended to turn off this feature. Usually the value of `Wall Line Count` we print is 2 or more. Then it is divided into inner ring and outer ring. The advantage of printing the inner ring first is that when the extruder starts printing, a bump or no extruded material will be extruded, which is a defect. We need to hide this flaw inside the model without affecting the model. Therefore, it is recommended not to enable this feature.

But for geeks, it has a special purpose.

Skin Overlap Percentage

Please see Cura's introduction to it. It's already very clear. No special suggestions.

Infill Density

Fill rate size.

Infill Pattern & Infill Line Directions

Choose a filling according to your liking.

Its role is related to the overall strength of the model. If the model is not very important, it is recommended to use the `Lines` pattern. Because this pattern saves printing time. If you have requirements for the strength of the model, then you can choose other patterns.

In fact, you can judge the strength of the printed model based on the pattern.

Connect Infill Lines

It is recommended to turn on this feature. Because it can reduce the `Retraction` action.

It is not a good thing to `Retraction` too frequently(it will cause more extrusion points in the model and reduce the printing efficiency).

Randomize Infill Start

It is recommended to turn on this feature. It will select the nearest position and start printing the filler.

Extra Infill Wall Count

No special requirements. Equivalent to the value of `Wall Line Count` + 1, but it is connected to the filler.

Infill Overlap Percentage

Please see Cura's introduction to it. It's already very clear. No special suggestions.

Infill Before Walls

This is a question of order. For the final printed model, there is almost no difference.

Printing Temperatrue

Does not include the temperature of the first layer. Because the printing temperature of the first layer is set separately.

Please set the printing temperature according to the characteristics of the material, or according to your experience.

Printing Temperatrue Initial Layer

Print the temperature of the first layer.

Initial Printing Temperature

It can be understood as the starting temperature.

for example,

Printing Temperatrue = 215

Printing Temperatrue Initial Layer = 230

Initial Printing Temperature = 180

When you start printing, the extruder starts to heat up. When the temperature reaches 180 degrees, the printer starts to return to zero, and the target temperature is set to 230 degrees. When printing the second layer, the temperature was set to 215 degrees again.

So why is the temperature of the first layer(230) higher than the printing temperature(215)?

Because we want to ensure that the first layer of material has enough viscosity. Another reason is that the extrusion rate of the first layer is relatively large, and we need to ensure the melting rate of the material.

There is only one purpose for this, which is to achieve a higher success rate for the first layer of printing. Only if the first layer is successfully printed can subsequent printing be successful.

Build Plate Temperature & Build Plate Temperatrue Initial Layer

The temperature of the heating plate is generally set to a higher temperature of the first layer, and the temperature of the other layers gradually decrease.

Flow

The extrusion rate of the print. The default is 100%. No special settings are required. Because it contains more printing positions.

For example, Solid, Infill, Inner, Outer, Skirt and so on.

Infill Flow

Set its extrusion rate separately.

Skirt/Brim Flow

Set its extrusion rate separately.

Initial Layer Flow

Set its extrusion rate separately.

Standby Temperature

If there are two extruders. When one extruder is working, the other extruder is in standby state. In this state, the temperature maintained by the extruder is it.

Its purpose is to reduce hanging wire. If the printer has an extruder brush, you can ignore this problem. (MT3X and MT2X both have brushes)

But we recommend that its temperature and `Printing Temperature` should not be too different. If the difference is too large, the temperature of the extruder will wait for heating. Because it has to wait for the temperature to rise from `Standby Temperature` to `Printing Temperature`.

Print Speed**Infill Speed****Outer Wall Speed****Inner Wall Speed****Top/Bottom Speed****Travel Speed****Initial Layer Print Speed****Initial Layer Travel Speed****Skirt/Brim Speed****Number of Slower Layers**

There is no special explanation for speed, only the following suggestions,

- `Outer Wall Speed` is smaller than `Inner Wall Speed`. Because `Outer Wall Speed` is the surface of the model, use this method to improve the perfection of the surface.

- The speed of `Initial Layer Print Speed` should be lower. If the printing speed of the first layer is too fast, it may cause the model to fail to paste onto the platform.
- The speed of `Travel` should be set faster.
- `Number of Slower Layers`, the number is generally set from 1 to 3. It is meaningless to set other values.
- Don't rush to increase the printing speed. After you have accumulated a certain amount of printing experience, you can try to increase the printing speed.

Enable Retraction

It is best to turn on this feature.

Retraction Distance

This distance can be set based on experience. Because of different nozzle diameters, the set length is also different. If it is a 0.4mm nozzle, set it to 2mm. If it is a 0.8mm nozzle, set it to 5mm.

Its function is to reduce the phenomenon of material hanging when the extruder is not working.

Retraction Speed

The speed is generally between 20 and 30.

Z Hop When Retracted

It will increase the printing time. But you can set it according to your needs. Its function is also to prevent the material from hanging onto the model.

Combing Mode

Please see Cura's introduction to it.

Regarding support, it is very complicated. Its introduction will be added in the next update.

MULTOO

Yiwu Mantuo Intelligent Technology Co.,Ltd