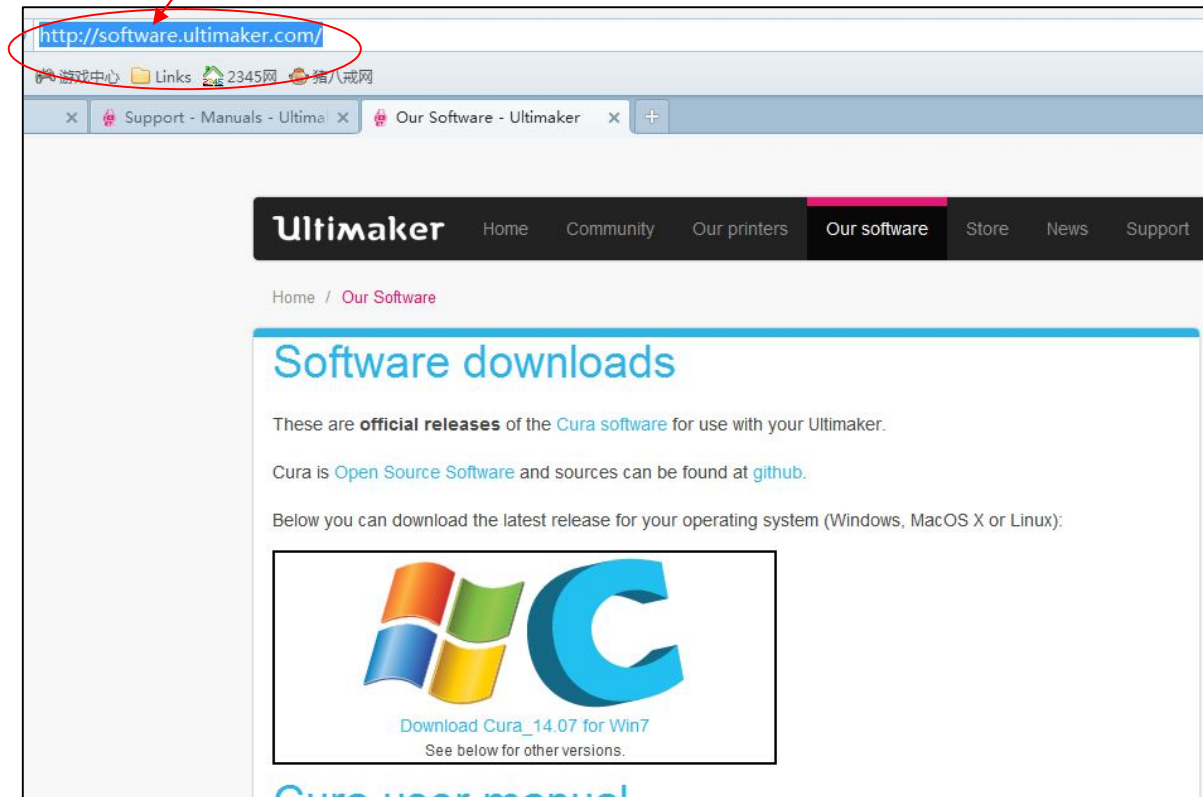


# Cura14.07使用说明

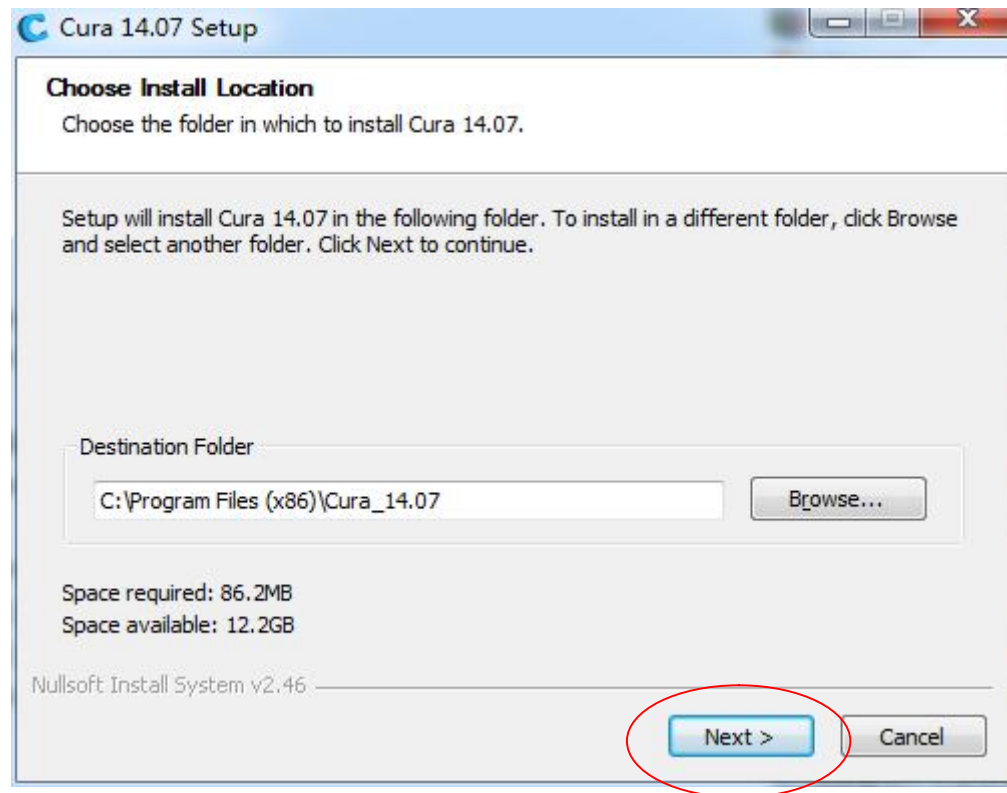
# 软件下载

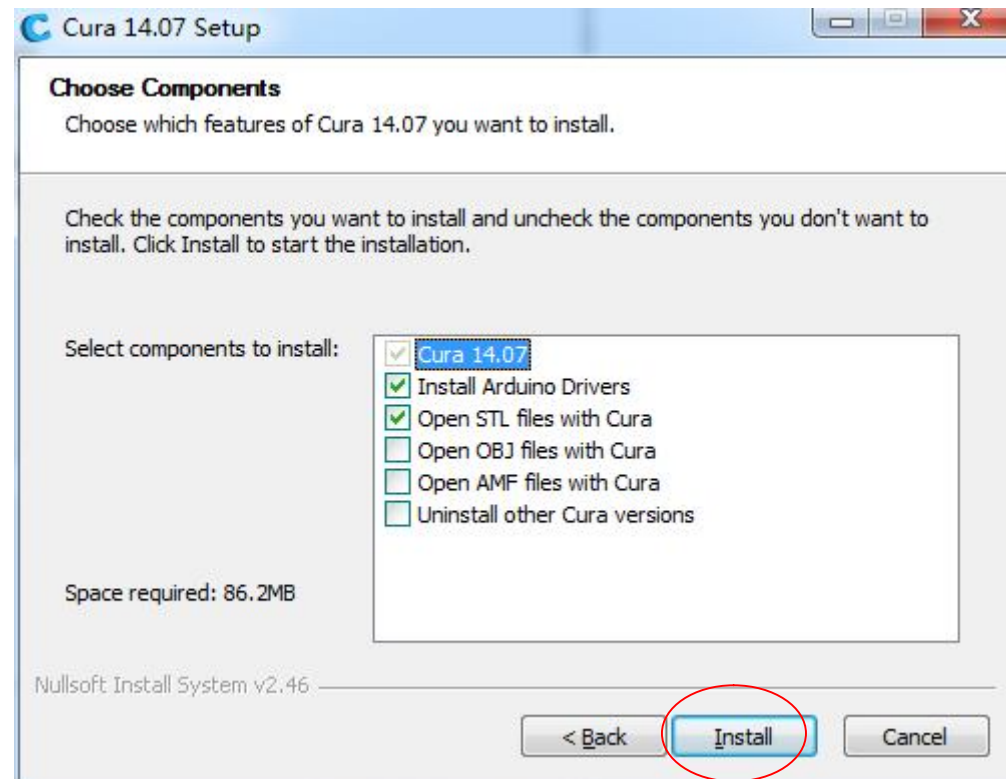
官方下载网站<http://software.ultimaker.com>

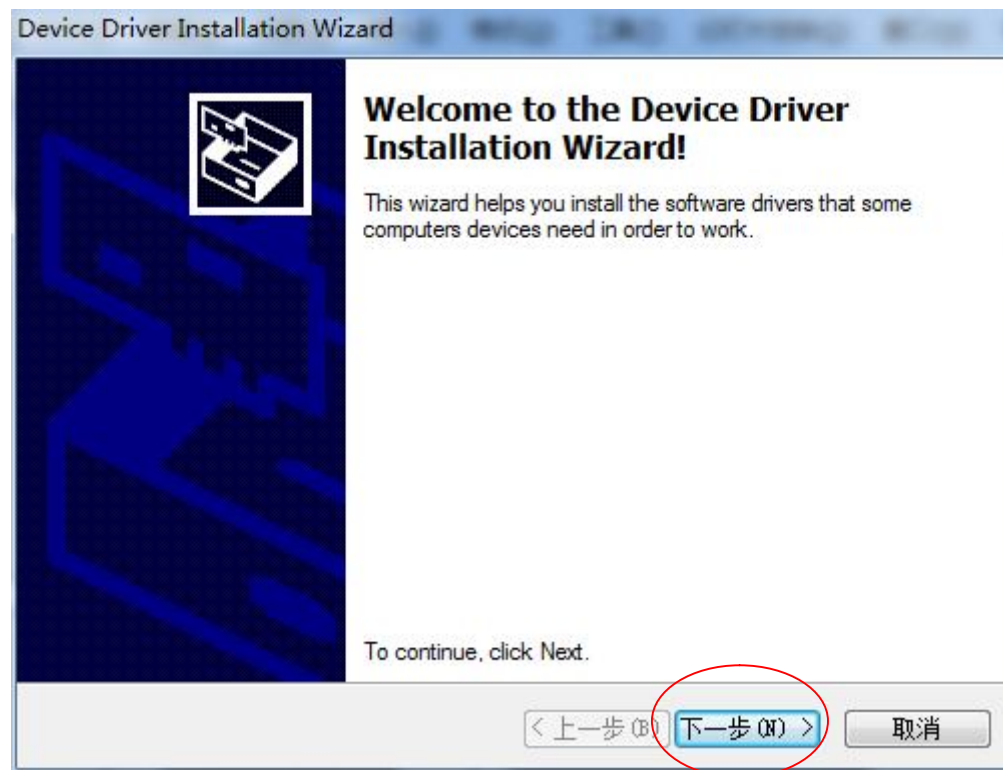


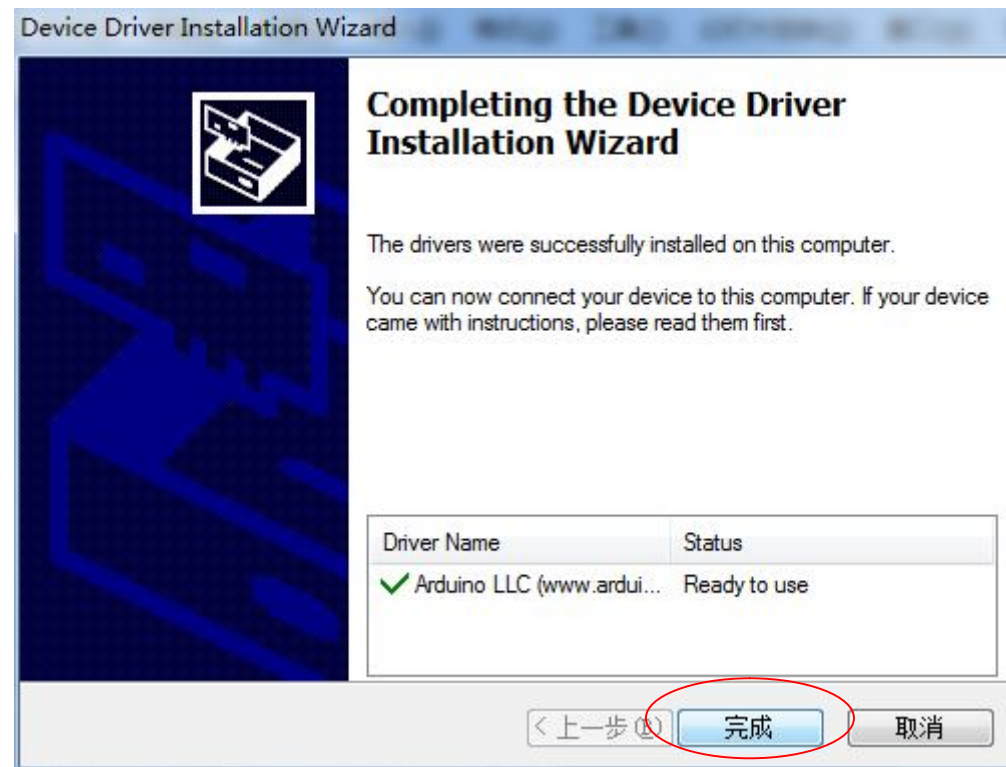
深圳市创必得科技有限公司  
[www.cbd-3d.com](http://www.cbd-3d.com)

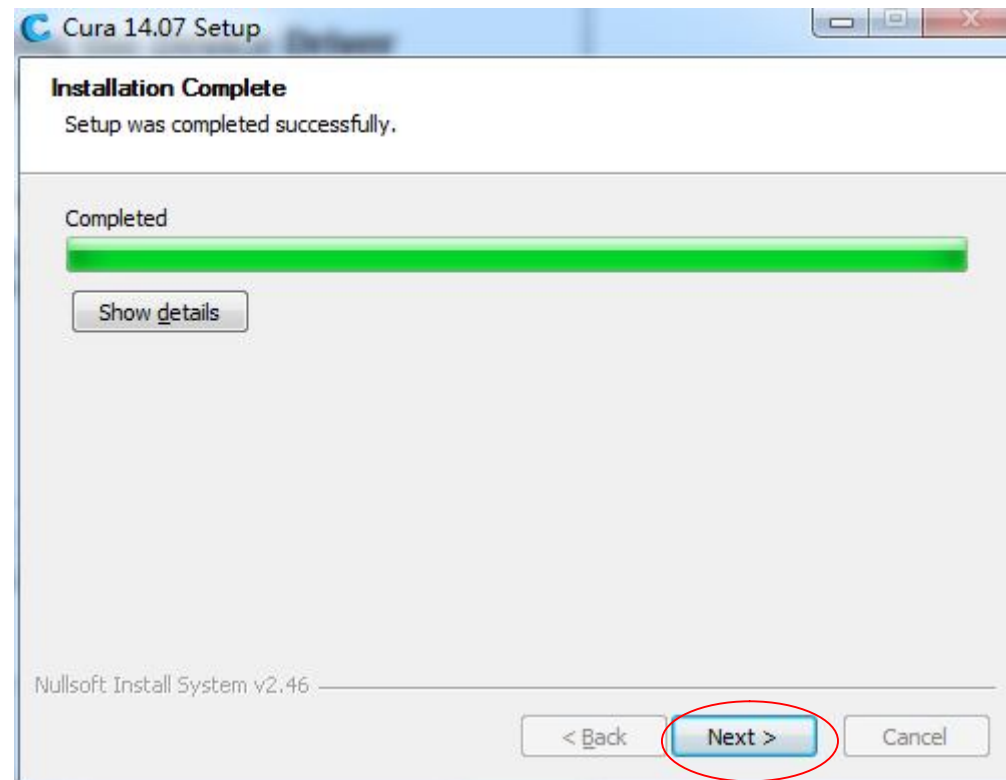
# 软件安装

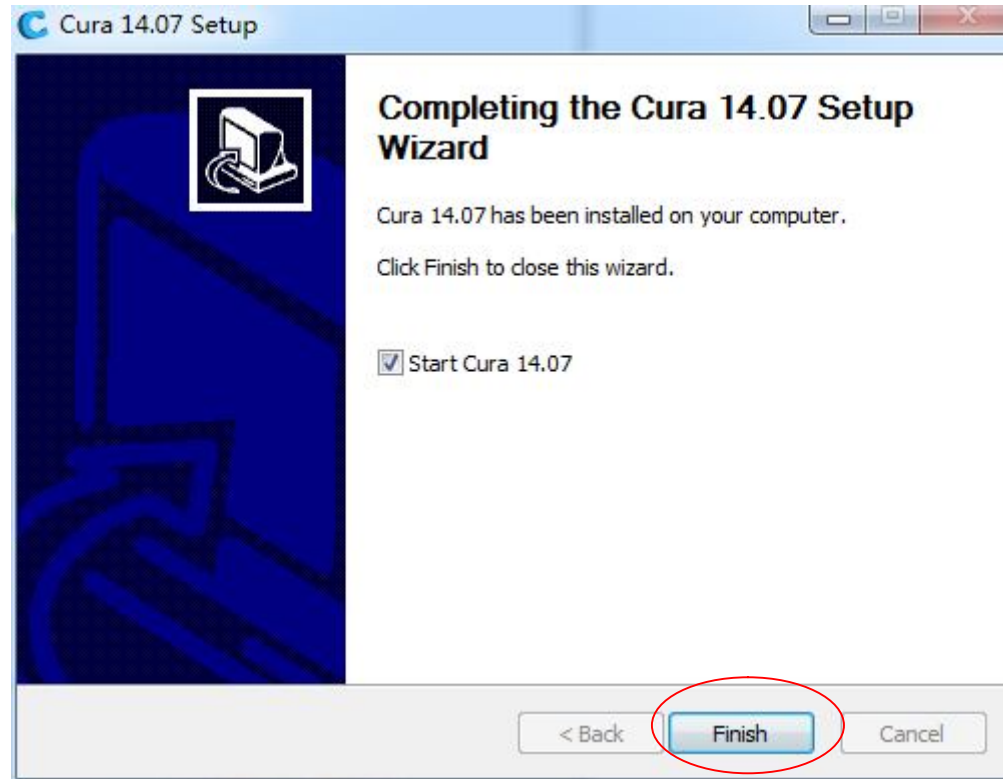






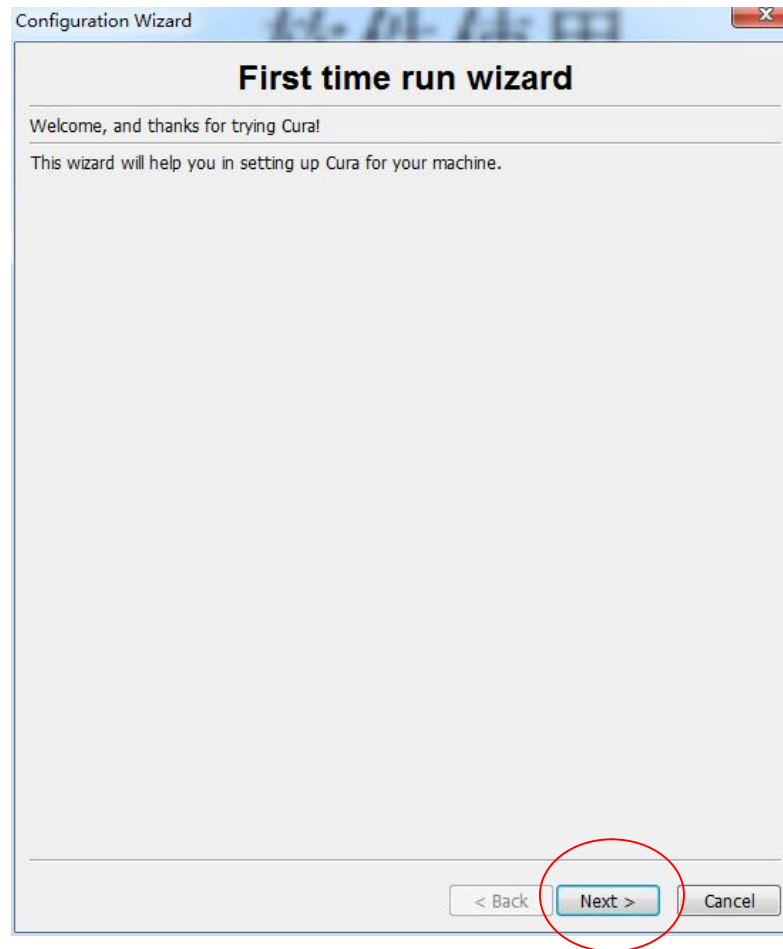








# 软件第一次使用



深圳市创必得科技有限公司  
[www.cbd-3d.com](http://www.cbd-3d.com)

Configuration Wizard

## Select your machine

What kind of machine do you have:

☐ Ultimaker2

☒ Ultimaker Original

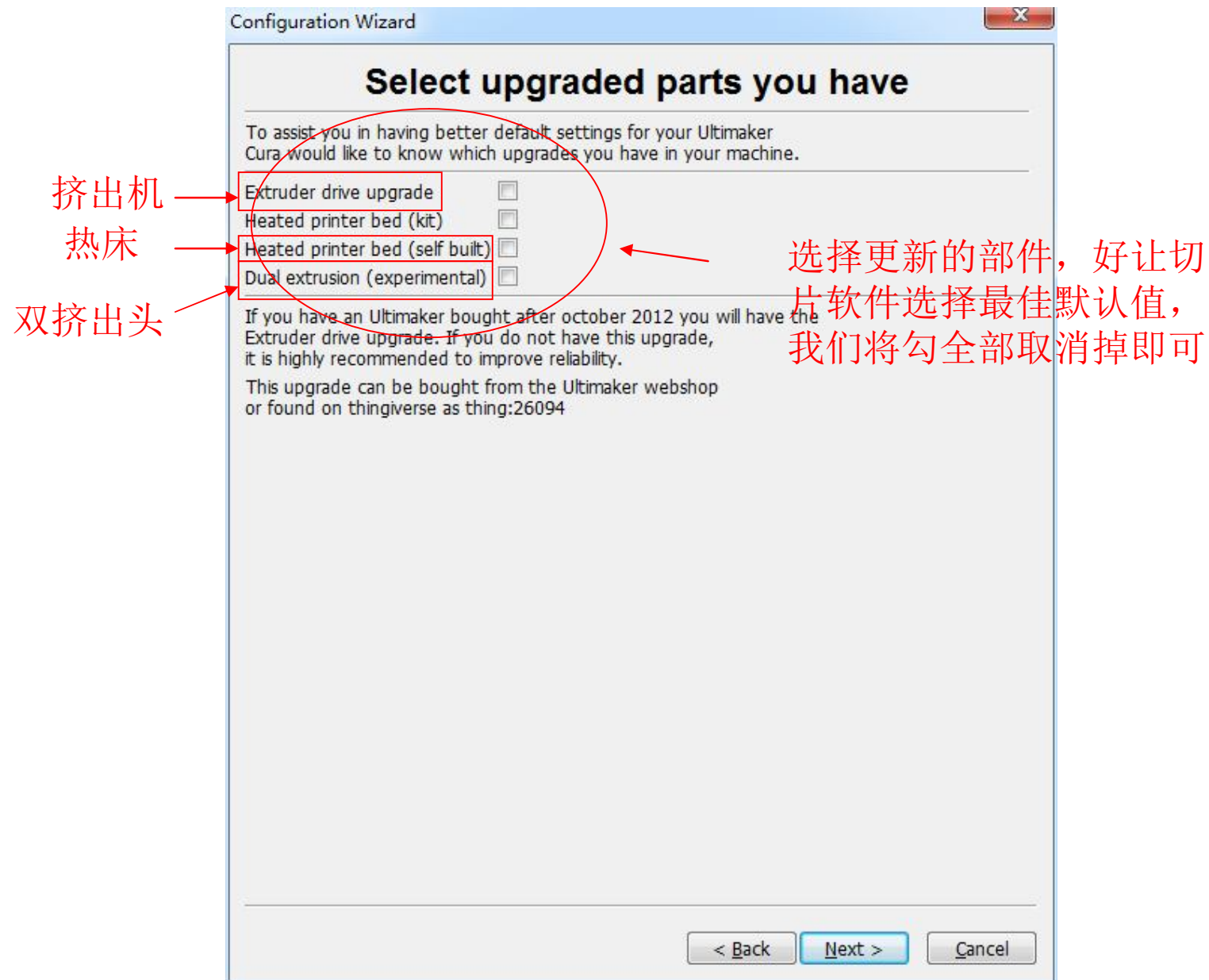
☐ Printbot

☐ Other (Ex: RepRap, MakerBot)

The collection of anonymous usage information helps with the continued improvement of Cura.  
This does NOT submit your models online nor gathers any privacy related information.  
Submit anonymous usage information: ☒  
For full details see: <http://wiki.ultimaker.com/Cura:stats>

< Back Next > Cancel

根据机器选择合适的机型，我们选择Ultimaker Original





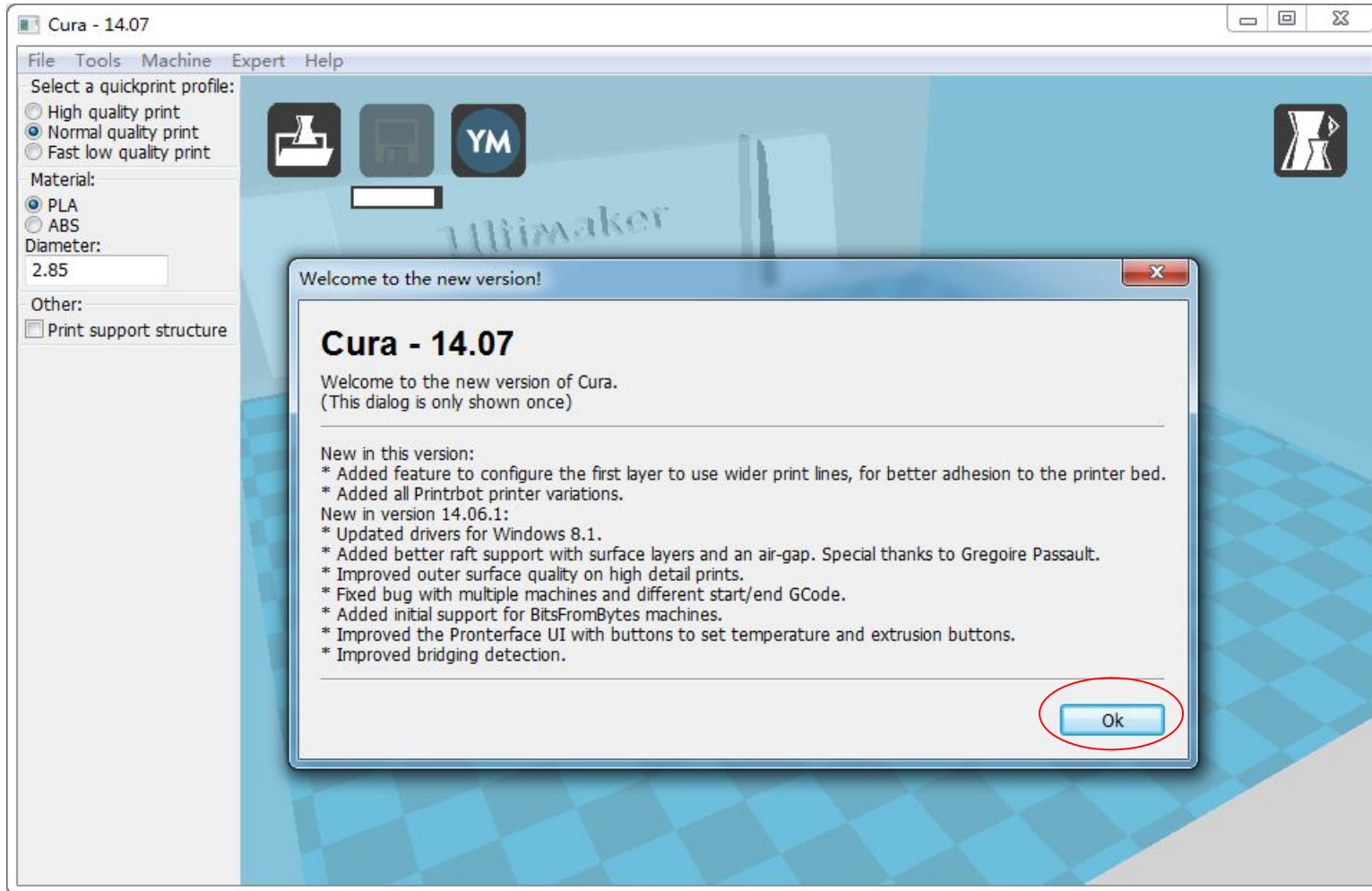
更新固件，我们不需要更新





机器初始化调平，我们也不需要，  
直接结束

初始化界面，从14.06开始，cura对raft进行了许多改善，所以我们强烈建议使用新版的软件



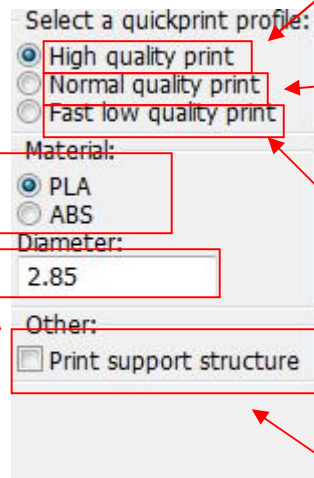


# 简易设置

不同材料的选择直接  
影响热床温度和  
挤出头温度

PLA:70/220

ABS:100/245



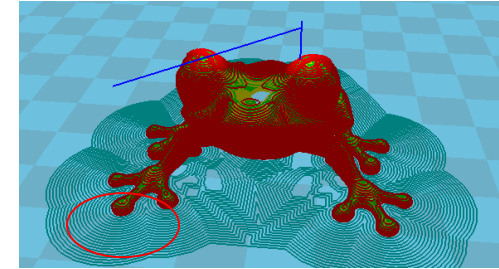
层厚0.06mm,大部分的模型  
在这个层厚情况下都打不  
好,除非模型非常简单

层厚0.1mm,大部分模型在  
这个层厚情况下都能打成  
功,表面比较细腻

层厚0.2mm,这个厚层打印模型成功率最  
高,表面能看到一层层的纹路

只要是悬空角度大于一定值的模型最好加上支撑,  
否则打印很可能会失败,但也有例外。如果不太确  
定,加上支撑结构。

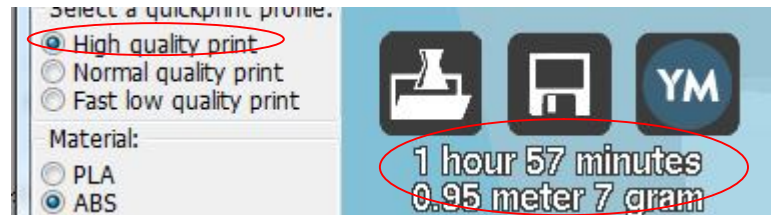
线材直径,因为耗材误  
差,3mm直径用默认值  
2.85即可



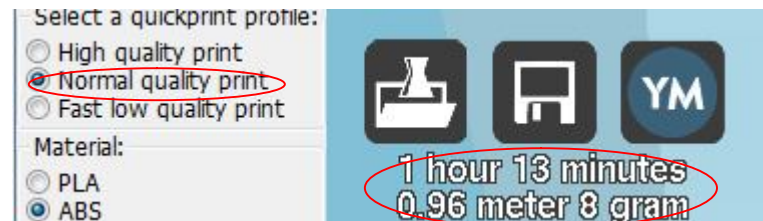
从模型上看,  
默认是平台依  
附类型是brim



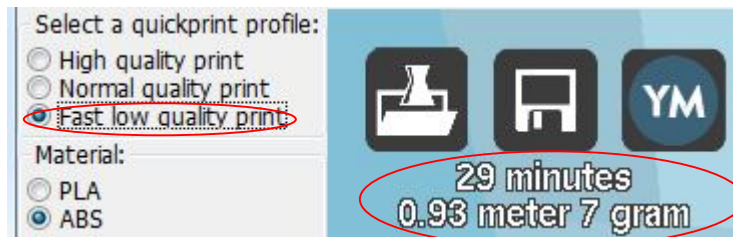
# 打印时间的差异



0.06mm层厚



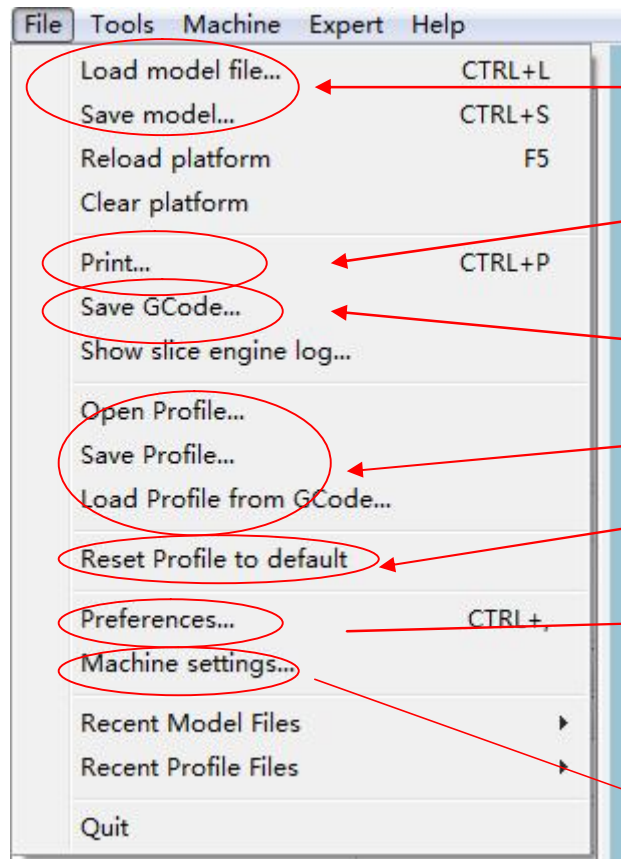
0.1mm层厚



0.2mm层厚

打印时间和层厚几乎成正比例关系

# 软件菜单



加载模型，保存模型

在线打印，为了打印的稳定性，我们的机器不支持在线打印，只支持SD卡打印

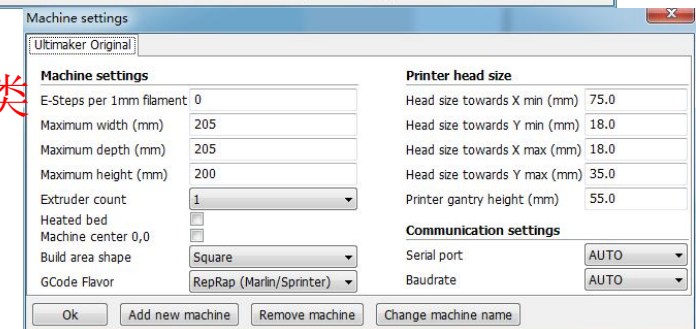
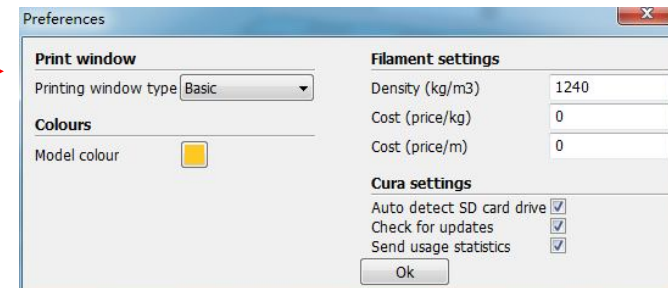
保存G代码到文件

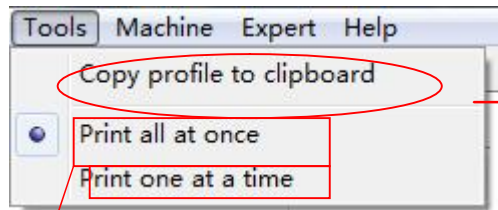
打开，保存打印配置

恢复参数到默认设置

偏好

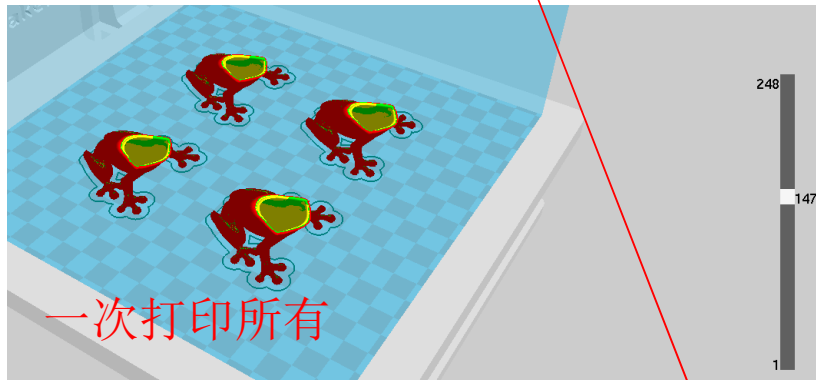
机器尺寸及gcode类型参数等



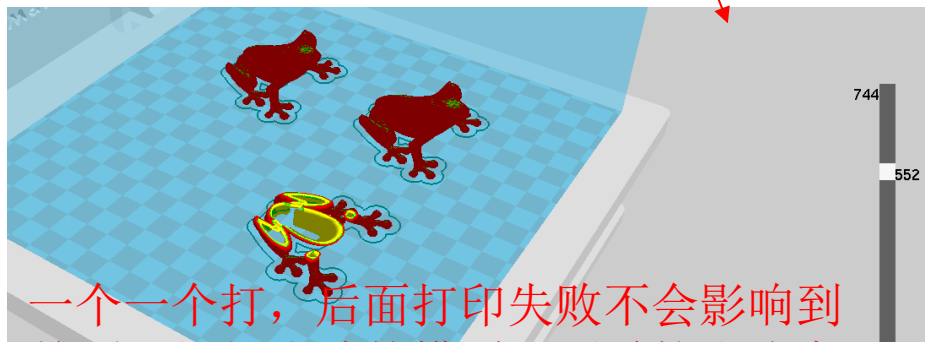


```
CURA_PROFILE_STRING:eNrtWktv20YQvhJGf8QeEzRWSUqKnQi8JLV9aYoAdtF  
F2JFrsStSS6xu5QsG/xy/Xb5ECXLrdMIzYs62OBwZnfmm28ehpXSFZNhwvg80YF  
8JwlTdNqJzy6yZ1SEJ06kmlJI81FHRKcTlMWnNNUmUeJlMdhak/oWrx0ZhyHxC  
XK8C33VycXeXs1DxO4b3I6eQPNeKhilG7FbP2qWFUxSXUoW+P4eqR/sEQ73CUe  
cMrirYNPXEEVRSGkDn4XOXOKlOqZkFlI44QphFiJa50wLmkaslstS/vujdCJs+C  
C7VYMlnjsBGEC5GWGQu8sSPEHQJOOEvjWg2g0IzBp5jtt4a5PzgdPxSbMB8Ih/v  
o65wlop14LnuwO0mrMJ4tCOlmShzHYwG467UBlu/815uv8t4HuJhwdLA234TiWz
```

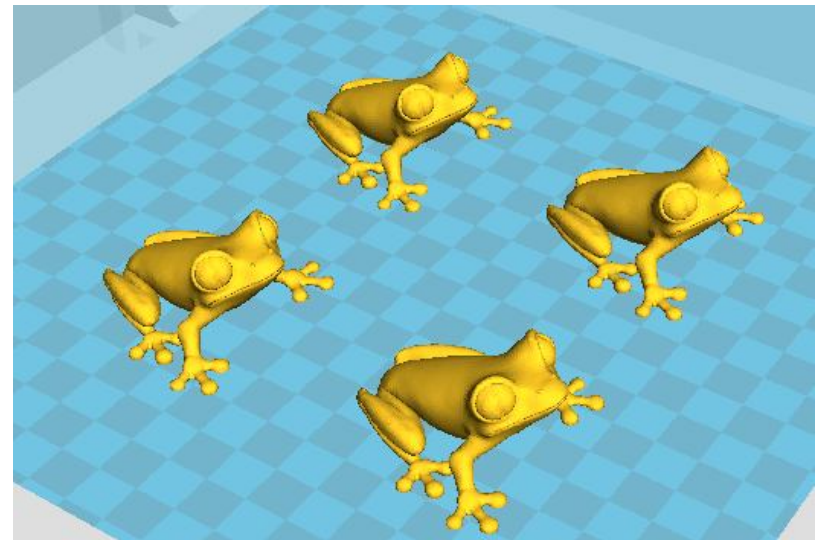
配置参数被运算成看上去无意义的字符，这个配置也会在导出gcode的时候保存到文件结尾



一次打印所有

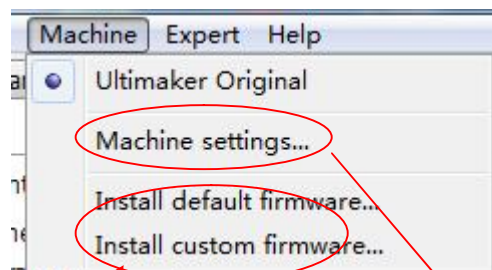


一个一个打，后面打印失败不会影响到前面已经打成功的模型，不过挤出头容易碰到其他已经打完的模型

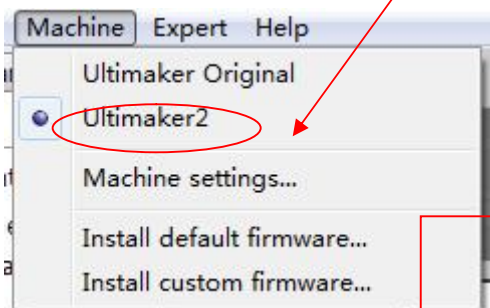
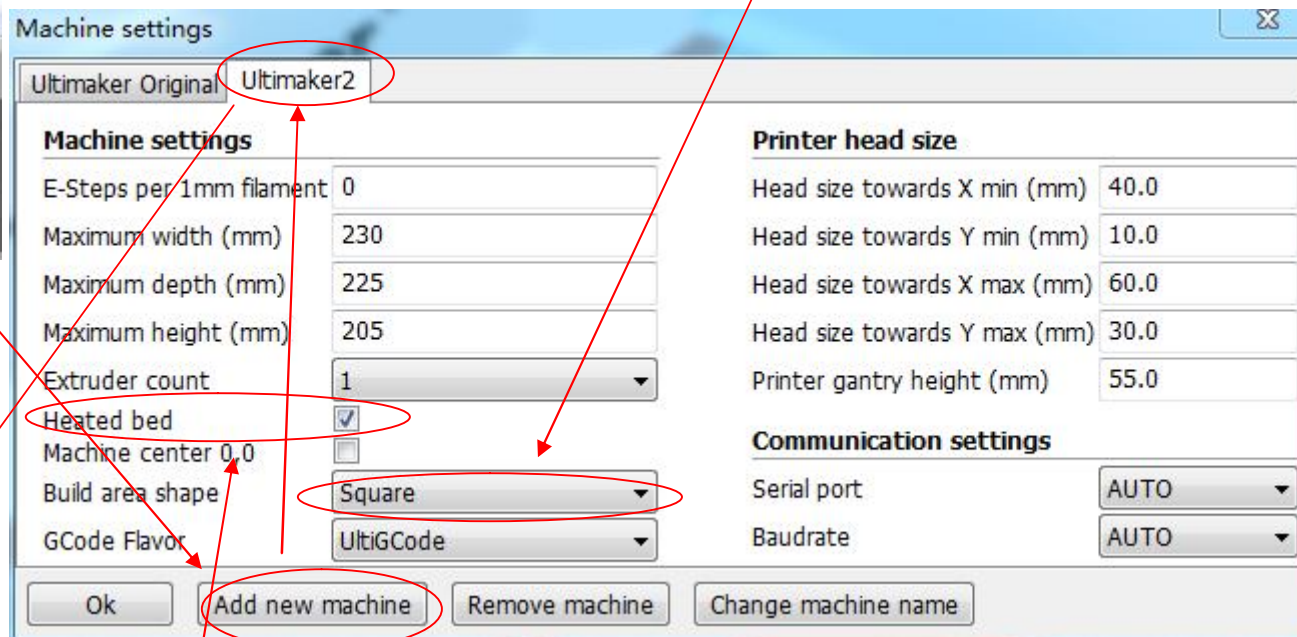


## 通过Cura软件控制多种不同类型的机器

此处可选择方形平台或是圆形平台,delta机型修改这里



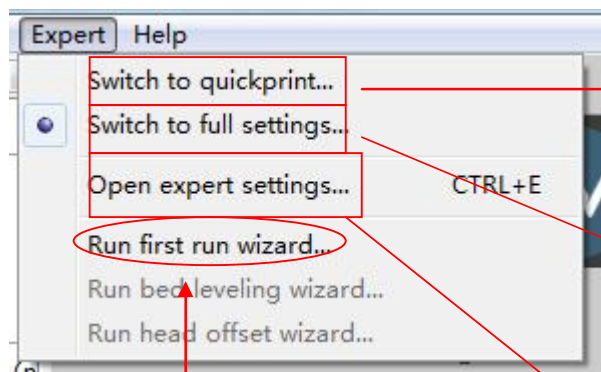
升级电路板固件,这只对开源硬件arduino有用



如果要打印ABS, 一定要勾选此选项, 否则没有热床的温度设置

Print speed (mm/s)	60
Printing temperature (C)	230
Bed temperature (C)	70



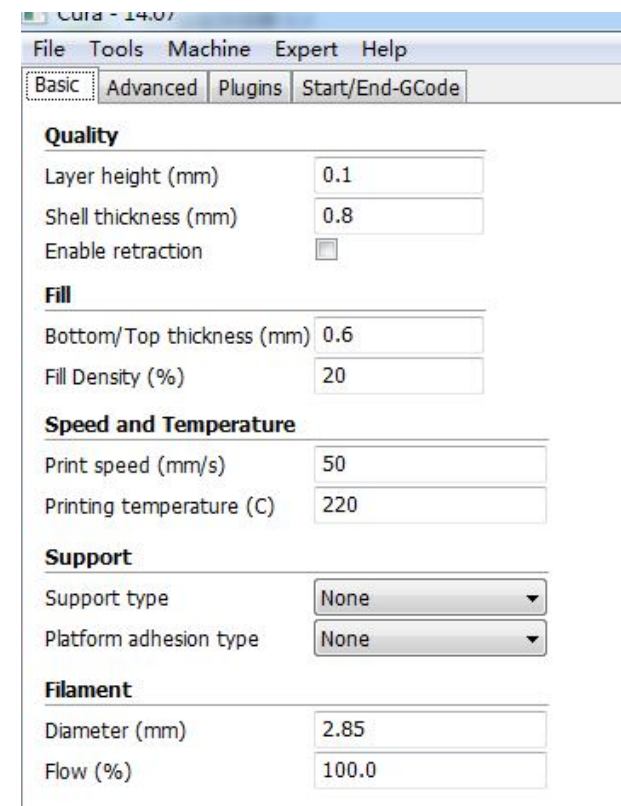
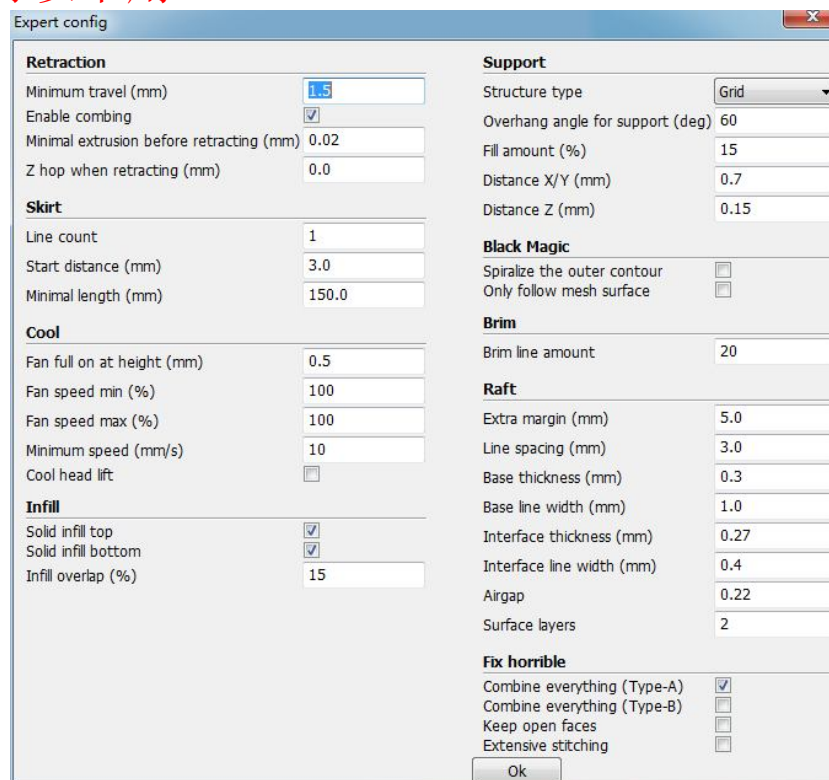
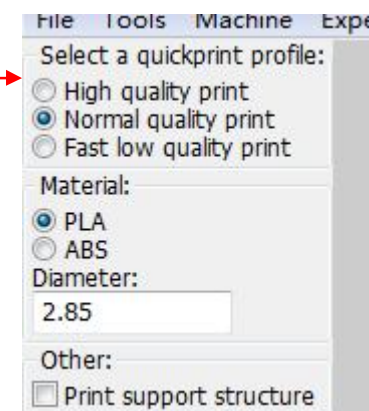


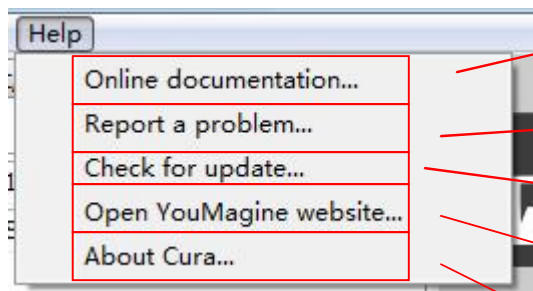
快速设置模式

完整设置模式

初次运行向导,基本上可以不用

专家模式





在线帮助

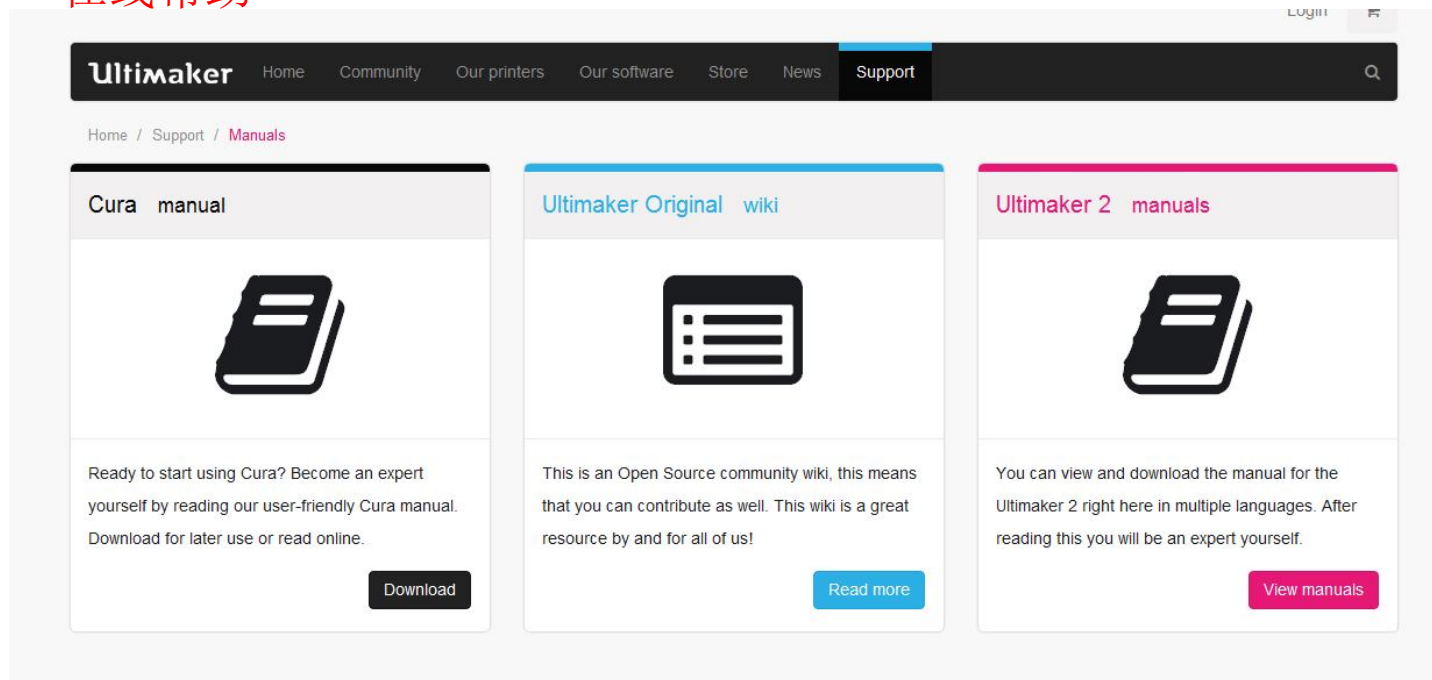
报告问题

检查更新

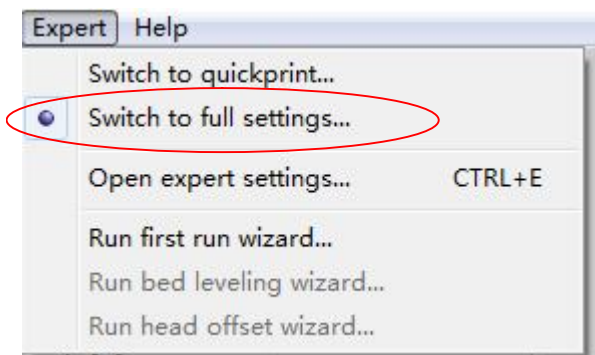
打开YouMagine网站

关于Cura

## 在线帮助

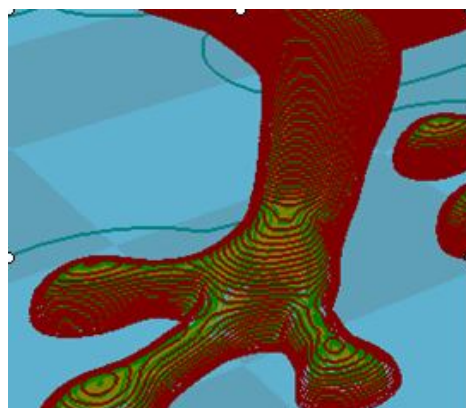


深圳市创必得科技有限公司  
www.cbd-3d.com

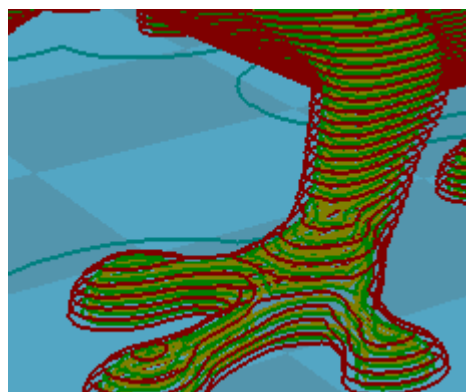


# 完整打印设置

层厚:0.1打印精度比较高, 一般用0.2, 如果要节省打印时间, 用0.3也可, 层厚越大, 打印时间越短

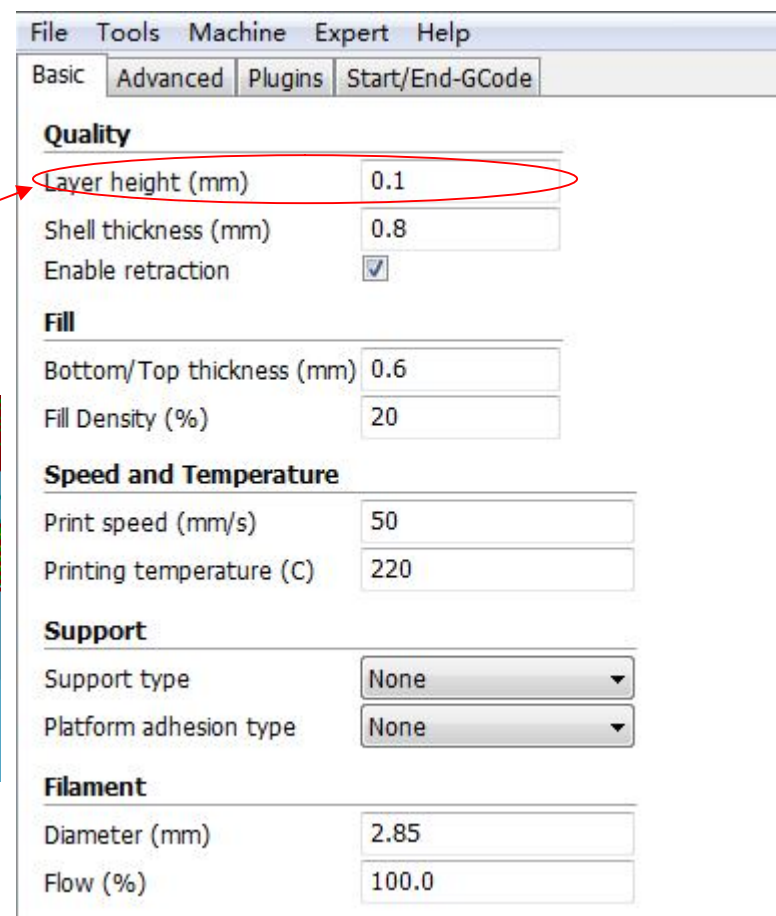


左:0.1mm层厚

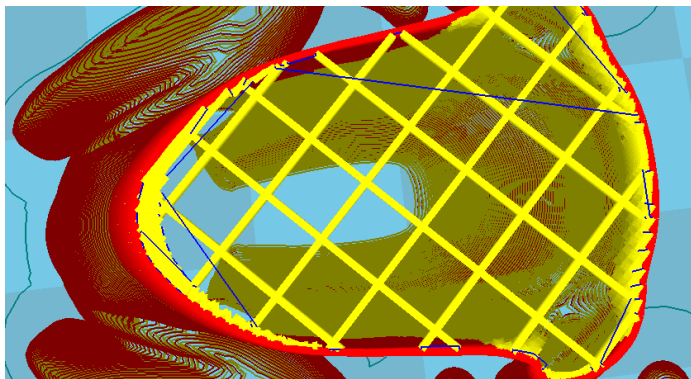


右:0.3mm层厚

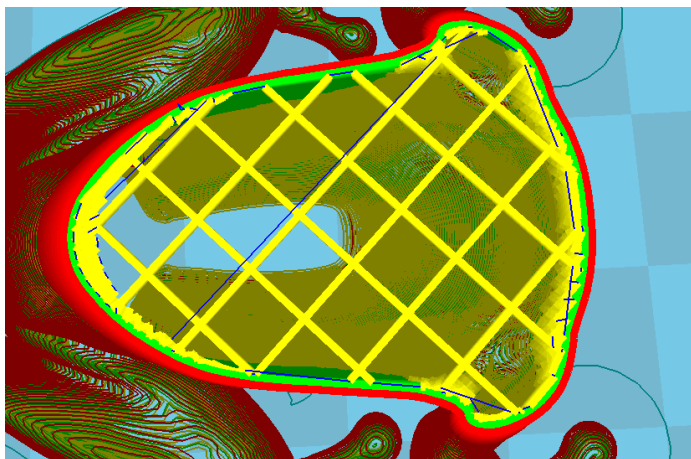
层厚小, 容易虚丝, 不建议使用低于0.1的层厚



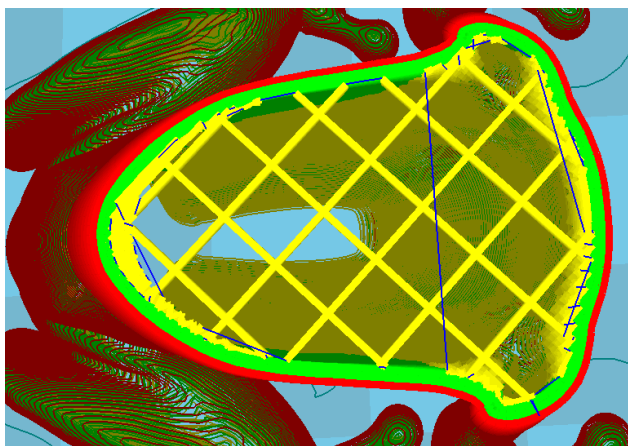
0.4



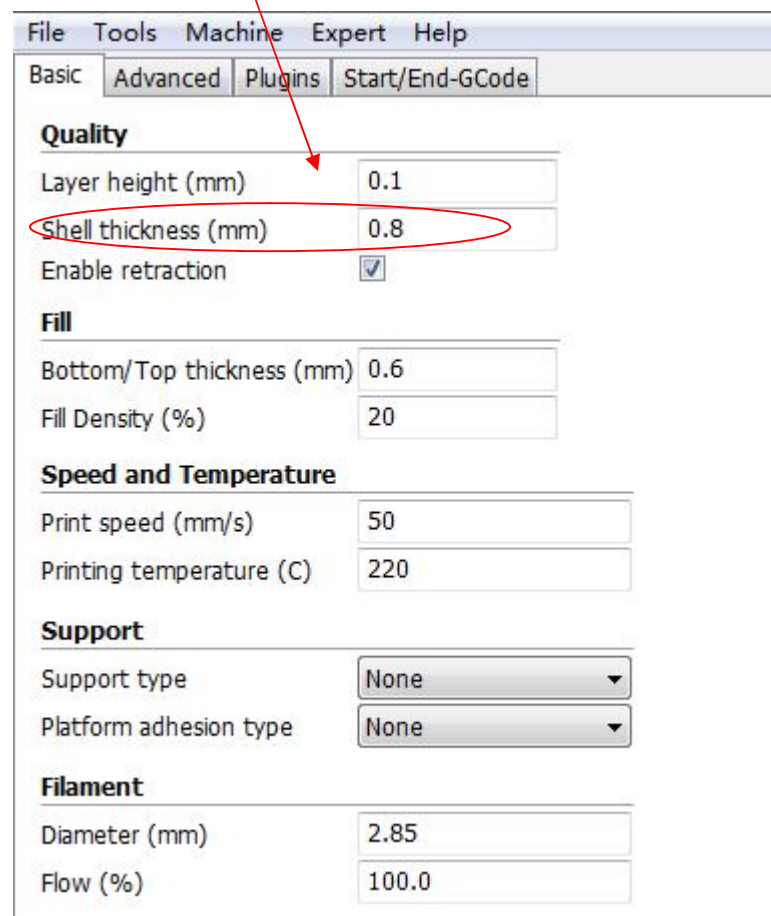
0.8



1.2



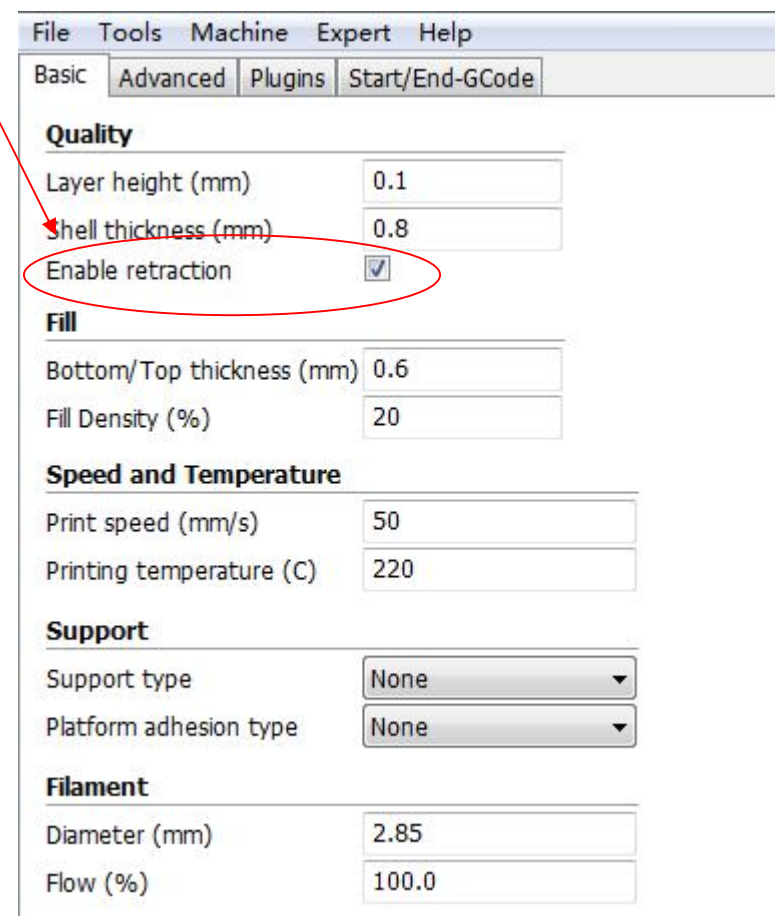
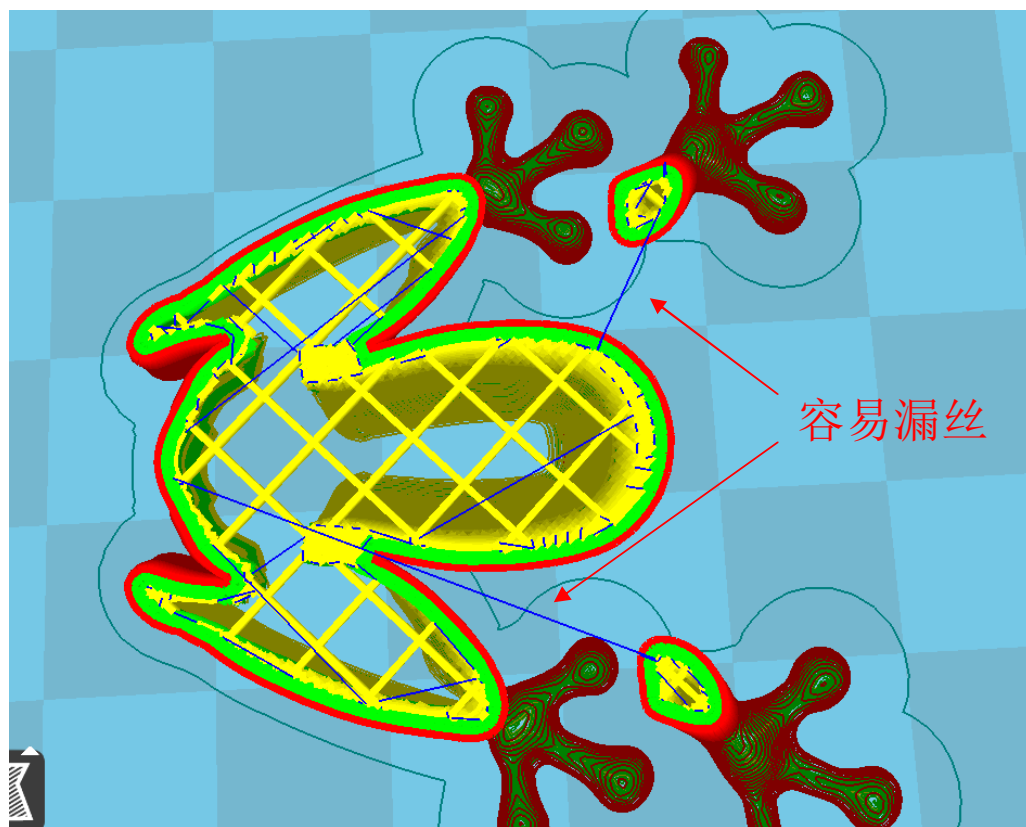
外壳的厚度

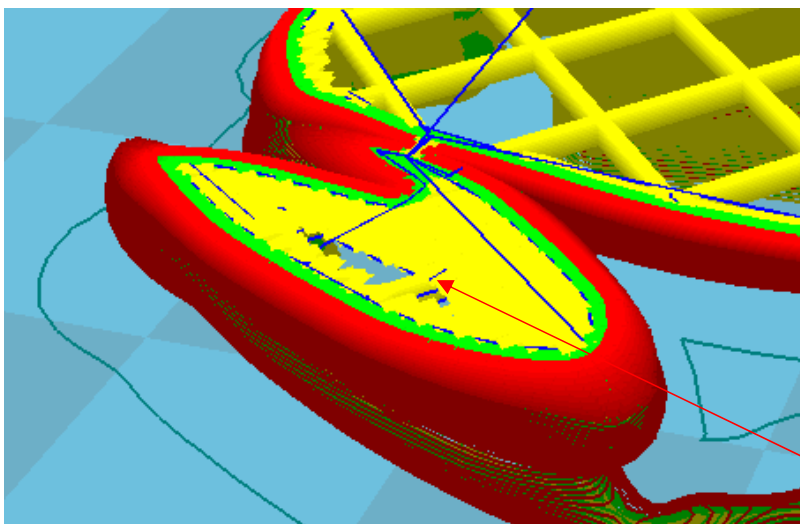


0.4mm的壁太薄，1.2mm的壁打印时间长，一般而言0.8mm刚刚好,尽量使用挤出头直径的整数倍

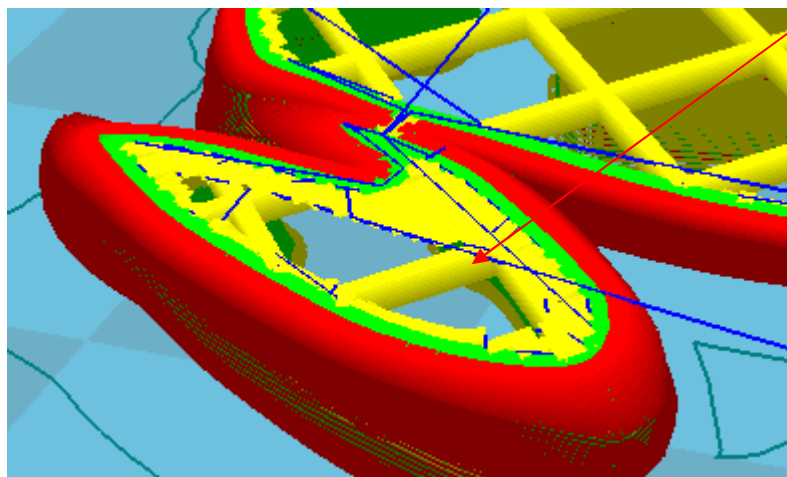


退丝是为了在快速移动的时候不要让丝漏出来,否则影响外观





Bottom/Top thickness:1mm

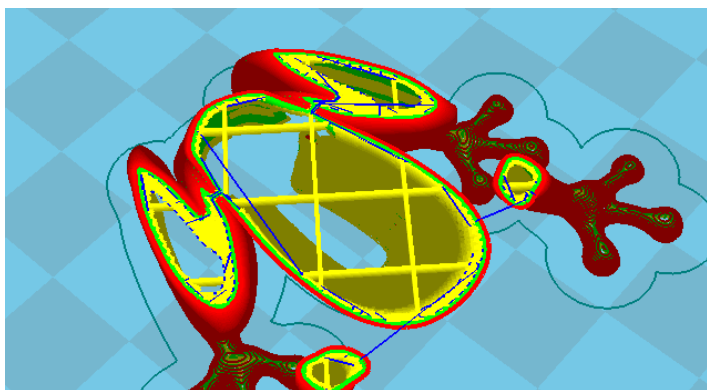


Bottom/Top thickness:0.6mm

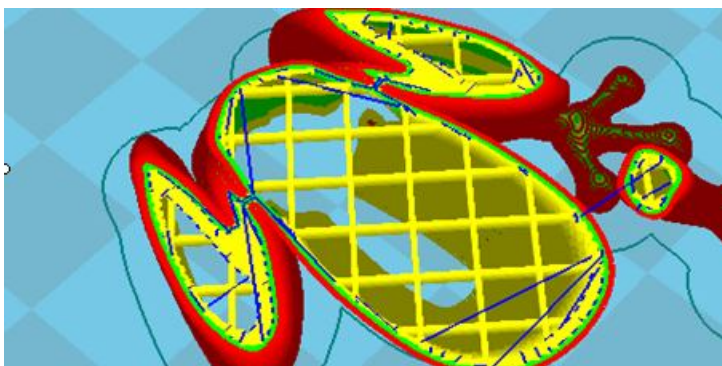
顶层和底层厚度，如果填充密度小于20%，0.6的厚度非常容易造成顶部有空洞，1mm的值一般都比较好的

在同样的层数

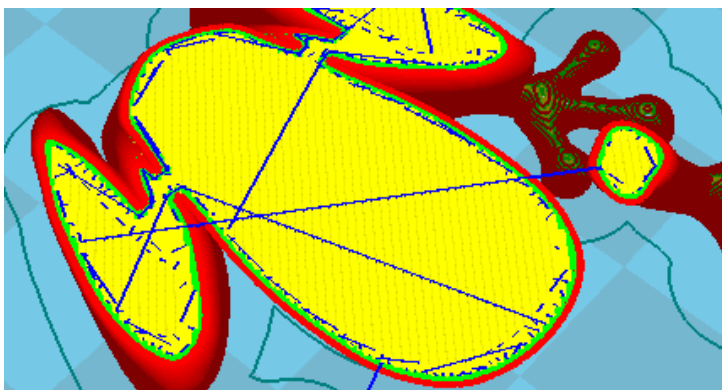
Quality	
Layer height (mm)	0.1
Shell thickness (mm)	0.8
Enable retraction	<input checked="" type="checkbox"/>
Fill	
Bottom/Top thickness (mm)	0.6
Fill Density (%)	10
Speed and Temperature	
Print speed (mm/s)	50
Printing temperature (C)	220
Support	
Support type	None
Platform adhesion type	None
Filament	
Diameter (mm)	2.85
Flow (%)	100.0



10%



20%



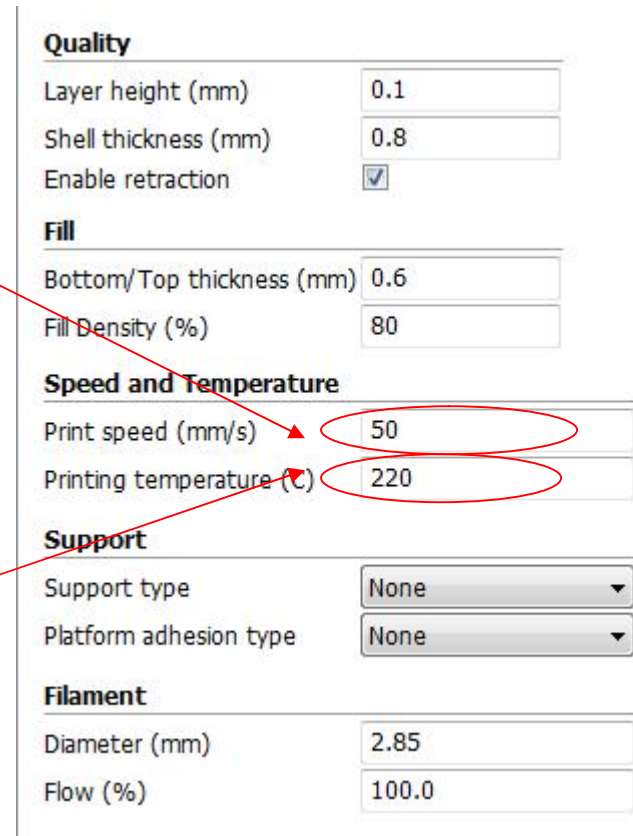
80%

填充密度：如果不要求强度很高，10%即可，高强度的话，提高填充比例，但是打印时间会增加，如果有斜坡顶，

Quality	
Layer height (mm)	0.1
Shell thickness (mm)	0.8
Enable retraction	<input checked="" type="checkbox"/>
Fill	
Bottom/Top thickness (mm)	0.6
Fill Density (%)	10
Speed and Temperature	
Print speed (mm/s)	50
Printing temperature (C)	220
Support	
Support type	None
Platform adhesion type	None
Filament	
Diameter (mm)	2.85
Flow (%)	100.0

这个是默认的全局速度，如果外壳，填充速度没有另外设置的话，就用这个速度,打印时间并不是与速度成直接正比例，超过90的速度打印省不很多，打印很容易出质量问题,一般而言，60是一个比较好的速度

PLA在190度开始熔融，但是粘度比较大，很难挤动，建议把温度设成210度以上，特别是打印速度快，层厚比较大时，把温度设置高一点



The image shows a settings panel for a 3D printer, likely Cura. It is divided into several sections: Quality, Fill, Speed and Temperature, Support, and Filament. Red lines and circles highlight specific settings. A red line points from the text '超过90的速度' to the 'Print speed (mm/s)' field, which is set to 50 and circled in red. Another red line points from the text '把温度设置高一点' to the 'Printing temperature (°C)' field, which is set to 220 and circled in red.

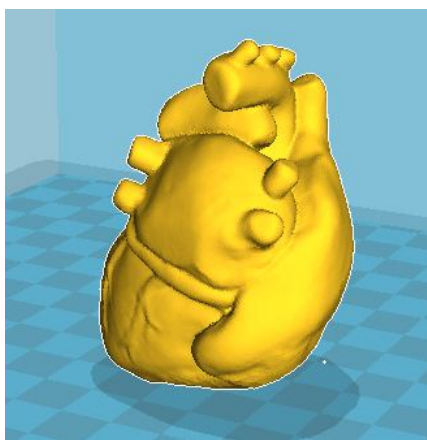
Quality	
Layer height (mm)	0.1
Shell thickness (mm)	0.8
Enable retraction	<input checked="" type="checkbox"/>

Fill	
Bottom/Top thickness (mm)	0.6
Fill Density (%)	80

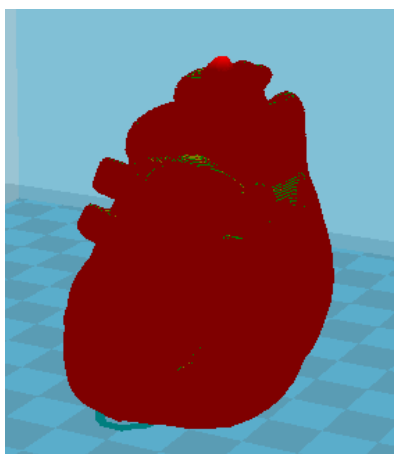
Speed and Temperature	
Print speed (mm/s)	50
Printing temperature (°C)	220

Support	
Support type	None
Platform adhesion type	None

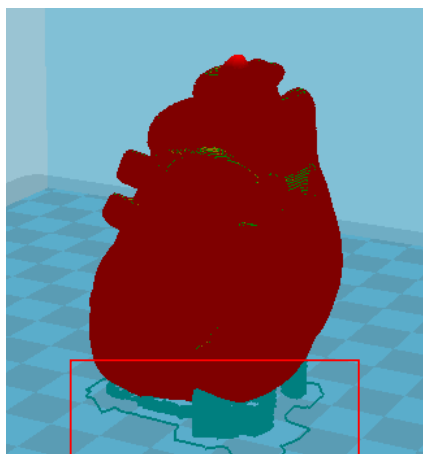
Filament	
Diameter (mm)	2.85
Flow (%)	100.0



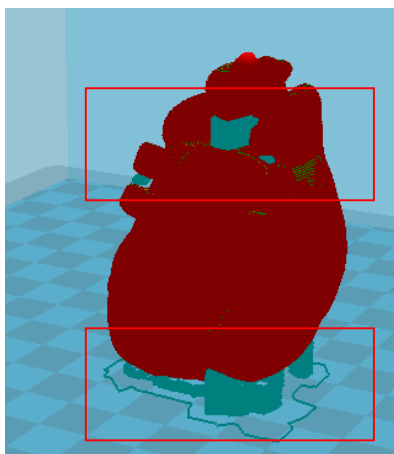
原始图



支撑:none



支撑:touching buildplate



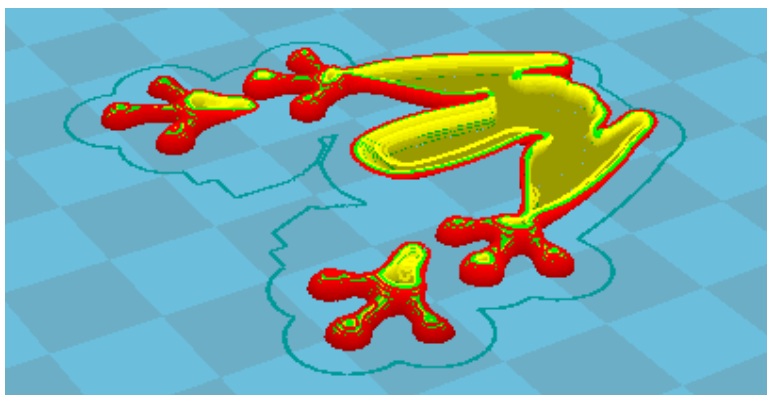
支撑:everywhere

支撑类型:无, 接触底板, 所有地方

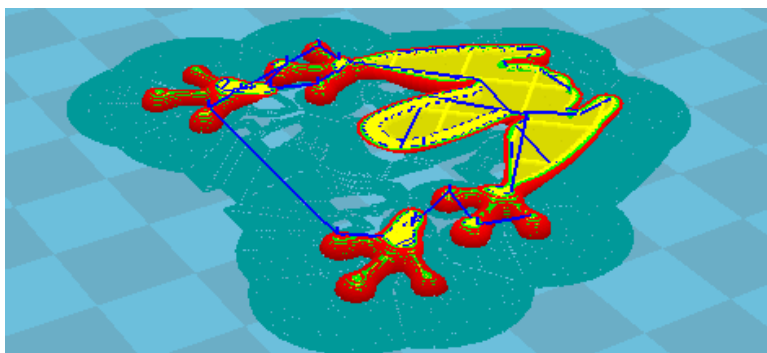
Quality	
Layer height (mm)	0.2
Shell thickness (mm)	0.8
Enable retraction	<input checked="" type="checkbox"/>
Fill	
Bottom/Top thickness (mm)	1
Fill Density (%)	10
Speed and Temperature	
Print speed (mm/s)	50
Printing temperature (C)	220
Support	
Support type	Everywhere
Platform adhesion type	None Touching buildplate Everywhere
Filament	
Diameter (mm)	2.85
Flow (%)	100.0

对于结构复杂的模型, 通常都会需要加支撑, **Everywhere**的支撑有可能会落在模型上, 造成表面不好看, 通常的做法是旋转模型到某一个方位, 尽量避免产生支撑

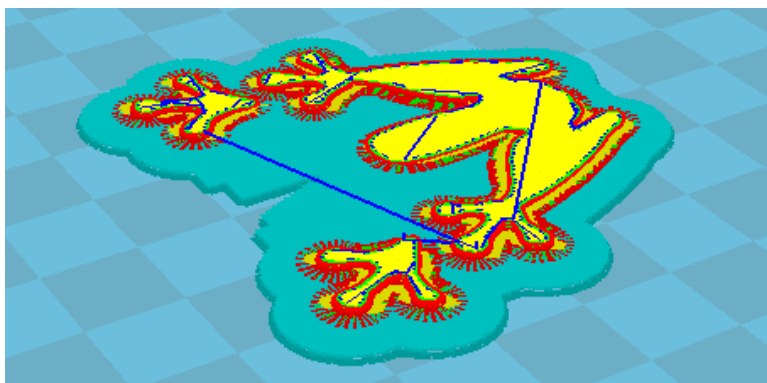




none



brim



raft

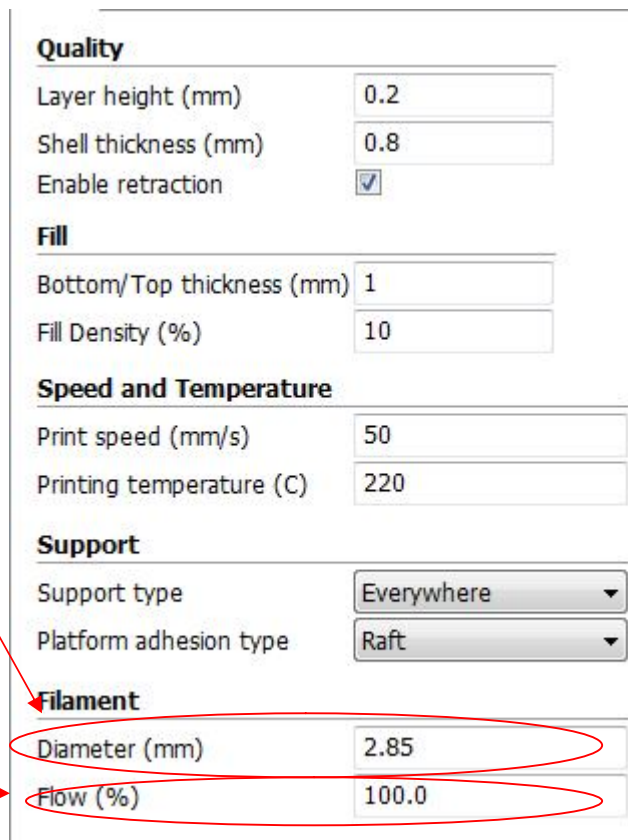
平台附着类型:无, 边, 筏

<b>Quality</b>	
Layer height (mm)	0.2
Shell thickness (mm)	0.8
Enable retraction	<input checked="" type="checkbox"/>
<b>Fill</b>	
Bottom/Top thickness (mm)	1
Fill Density (%)	10
<b>Speed and Temperature</b>	
Print speed (mm/s)	50
Printing temperature (C)	220
<b>Support</b>	
Support type	Everywhere
Platform adhesion type	<div> None  None  Brim  Raft </div>
<b>Filament</b>	
Diameter (mm)	2.00
Flow (%)	100.0

一般而言, 如果你的平台调得很平, 并且底板的蓝色胶带纸没被撕坏, 用 **none** 也是可以的, 否则最好是用的 **raft**, 但是有时 **raft** 很难撕, **makerware** 的 **raft** 则非常容易撕, 这和切片软件有非常大的关系

ultimaker默认是2.85,选择小的直径会让挤出丝增多,不易虚丝,但是出丝过多,会让模型变“胖”,2.85是个不错的值

Flow是出丝比例,增加出丝比例和减少丝直径的效果是一样的,不过这个值更直观



The image shows a portion of the Ultimaker Cura software interface, specifically the 'Settings' panel for a print job. The settings are organized into several sections: 'Quality', 'Fill', 'Speed and Temperature', 'Support', and 'Filament'. The 'Filament' section is highlighted with a red oval, and its sub-sections 'Diameter (mm)' and 'Flow (%)' are also circled in red. A red arrow points from the text '2.85是个不错的值' to the 'Diameter (mm)' field, which is set to 2.85. Another red arrow points from the text 'Flow是出丝比例...' to the 'Flow (%)' field, which is set to 100.0. The 'Support' section shows 'Support type' set to 'Everywhere' and 'Platform adhesion type' set to 'Raft'. The 'Speed and Temperature' section shows 'Print speed (mm/s)' set to 50 and 'Printing temperature (C)' set to 220. The 'Fill' section shows 'Bottom/Top thickness (mm)' set to 1 and 'Fill Density (%)' set to 10. The 'Quality' section shows 'Layer height (mm)' set to 0.2, 'Shell thickness (mm)' set to 0.8, and 'Enable retraction' checked.

Quality	
Layer height (mm)	0.2
Shell thickness (mm)	0.8
Enable retraction	<input checked="" type="checkbox"/>

Fill	
Bottom/Top thickness (mm)	1
Fill Density (%)	10

Speed and Temperature	
Print speed (mm/s)	50
Printing temperature (C)	220

Support	
Support type	Everywhere
Platform adhesion type	Raft

Filament	
Diameter (mm)	2.85
Flow (%)	100.0

挤出头大小,根据实际尺寸设计,但是将该值变大  
可以缩小打印时间

回退速度和回退时间,  
根据机型进行配置

最底层的厚度, 0.3是为了  
让模型更容易剥离

最底层的出丝比例, 默认100%

双挤出头设置,可不用

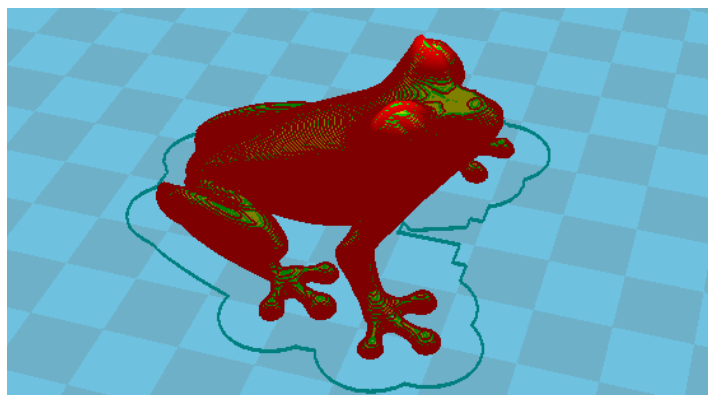
填充, 壳内部, 壳外部速度设置,  
0.0表示使用全局默认速度, 如果全  
局速度>60,为了保证打印质量,  
shell的速度最好小于60

每层最小时间, 当打印时间小于此值时,  
会使实际速度小于设置的速度当打印细长  
的物体时, 减少每层时间可能会更好

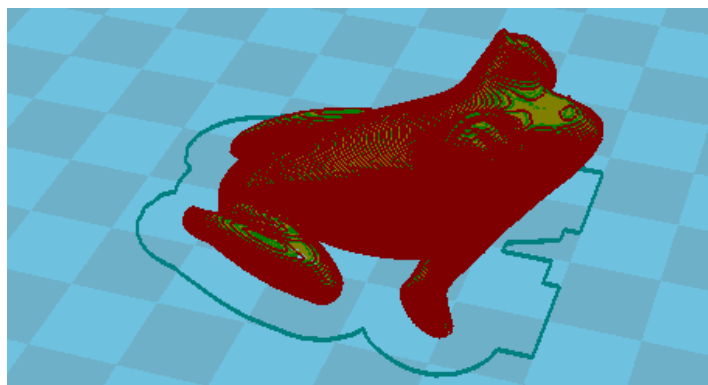
使能风扇冷却

Basic	Advanced	Plugins	Start/End-GCode
<b>Machine</b>			
Nozzle size (mm)		0.4	
<b>Retraction</b>			
Speed (mm/s)		40.0	
Distance (mm)		4	
<b>Quality</b>			
Initial layer thickness (mm)		0.3	
Initial layer line width (%)		100	
Cut off object bottom (mm)		0.0	
Dual extrusion overlap (mm)		0.15	
<b>Speed</b>			
Travel speed (mm/s)		150.0	
Bottom layer speed (mm/s)		20	
Infill speed (mm/s)		0.0	
Outer shell speed (mm/s)		0.0	
Inner shell speed (mm/s)		0.0	
<b>Cool</b>			
Minimal layer time (sec)		5	
Enable cooling fan		<input checked="" type="checkbox"/>	

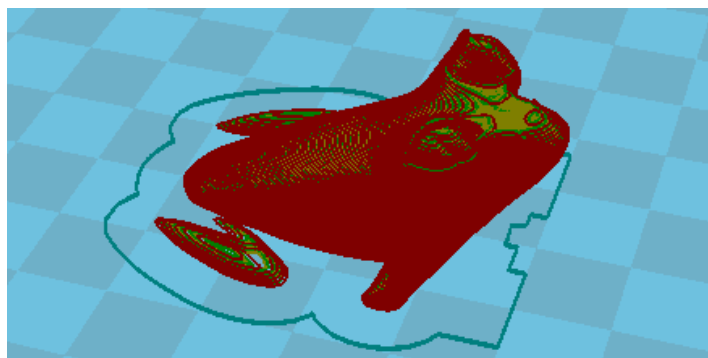




Cut off:0mm



Cut off:3mm



Cut off:6mm

模型底部裁剪

#### Machine

Nozzle size (mm) 0.4

#### Retraction

Speed (mm/s) 40.0

Distance (mm) 4

#### Quality

Initial layer thickness (mm) 0.3

Initial layer line with (%) 100

Cut off object bottom (mm) 0

Dual extrusion overlap (mm) 0.15

#### Speed

Travel speed (mm/s) 150.0

Bottom layer speed (mm/s) 20

Infill speed (mm/s) 0.0

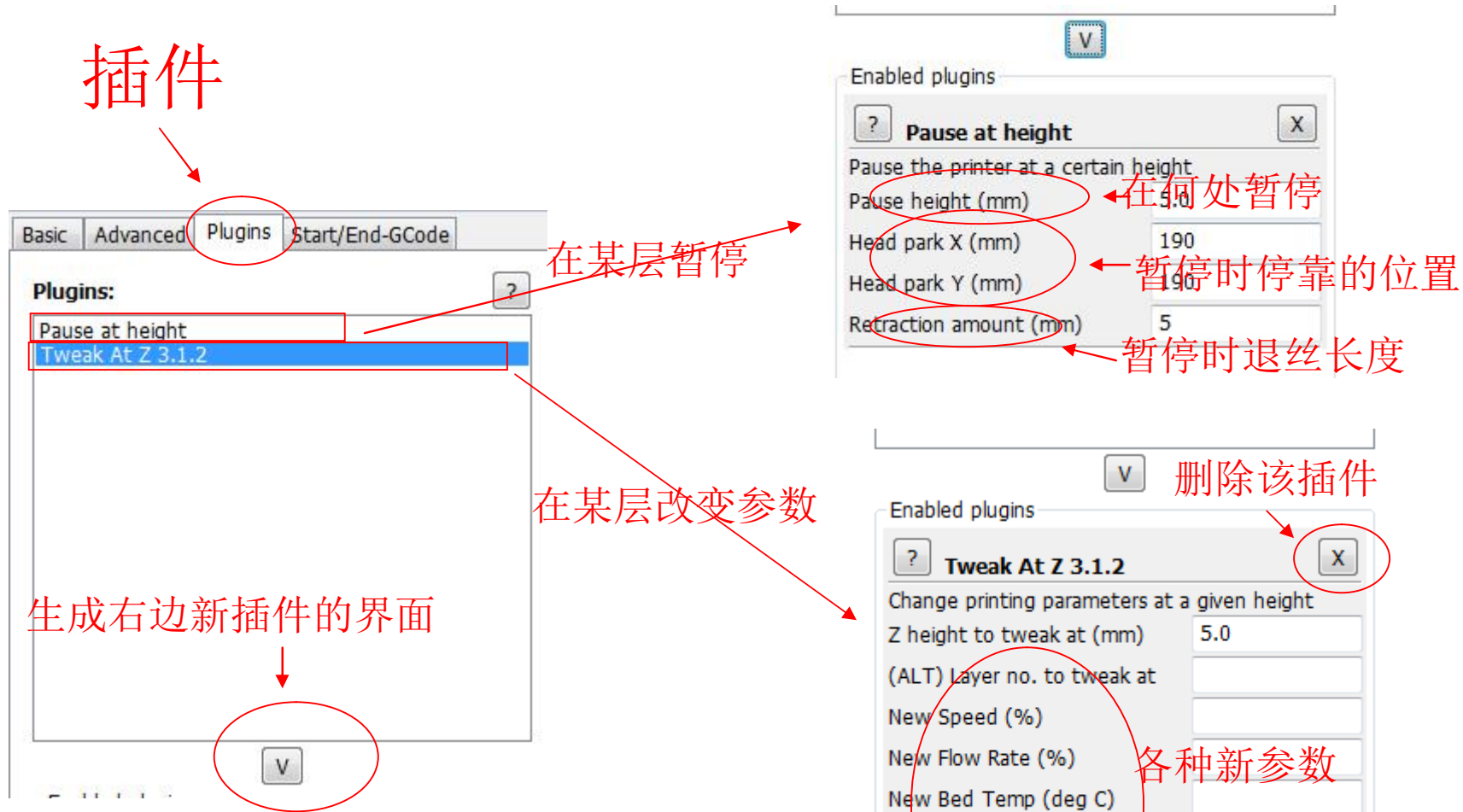
Outer shell speed (mm/s) 0.0

Inner shell speed (mm/s) 0.0

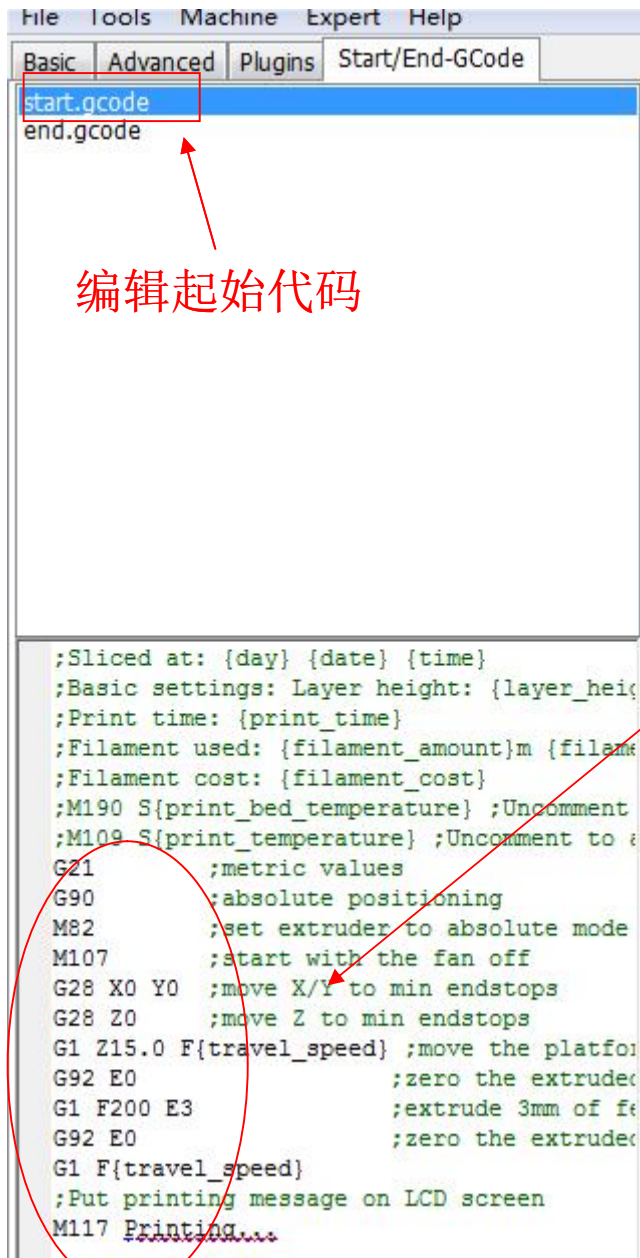
#### Cool

Minimal layer time (sec) 5

Enable cooling fan ☒



在某种特定的情况下，可能会用到插件功能，默认情况下可以不用管



gcode,用来给  
电路板执行的  
指令

不同的固件可能会需要一些定制指令，比如delta  
机器需要在起始化码中加入调平代码等，一般情  
况下不需要去修改

G代码的含义在<http://www.reprap.org/wiki/Gcode>  
中可以找到

打开文件

将gcode保存到文件

连接到youmagine网站

左键:移动物体

中键: 缩放视角

右键: 旋转视角

shift+右键:平移视角

模型显示方式

先左键选中物体才  
会出现这此功能

旋转

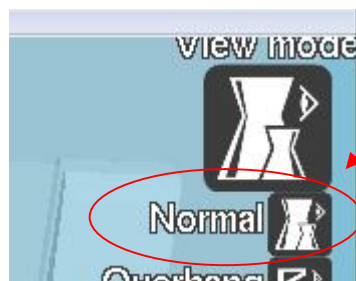
缩放

镜像



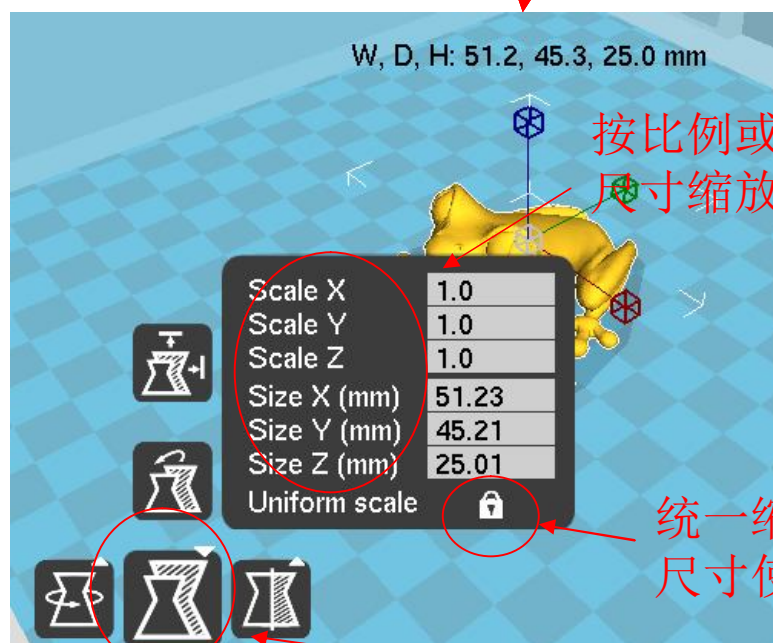
# 物体操作

物体的这些操作仅当显示模式处于Normal下才有效



1, 点击旋转

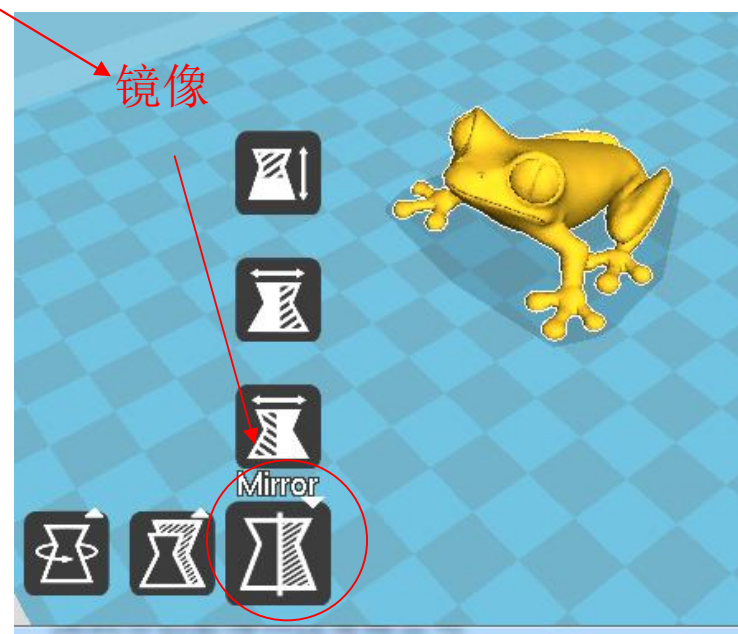
2 拉动旋转控制圆, 默认以15度为单位, 按住shift的时候抖动旋转控制圆, 可以以1度为单位进行旋转



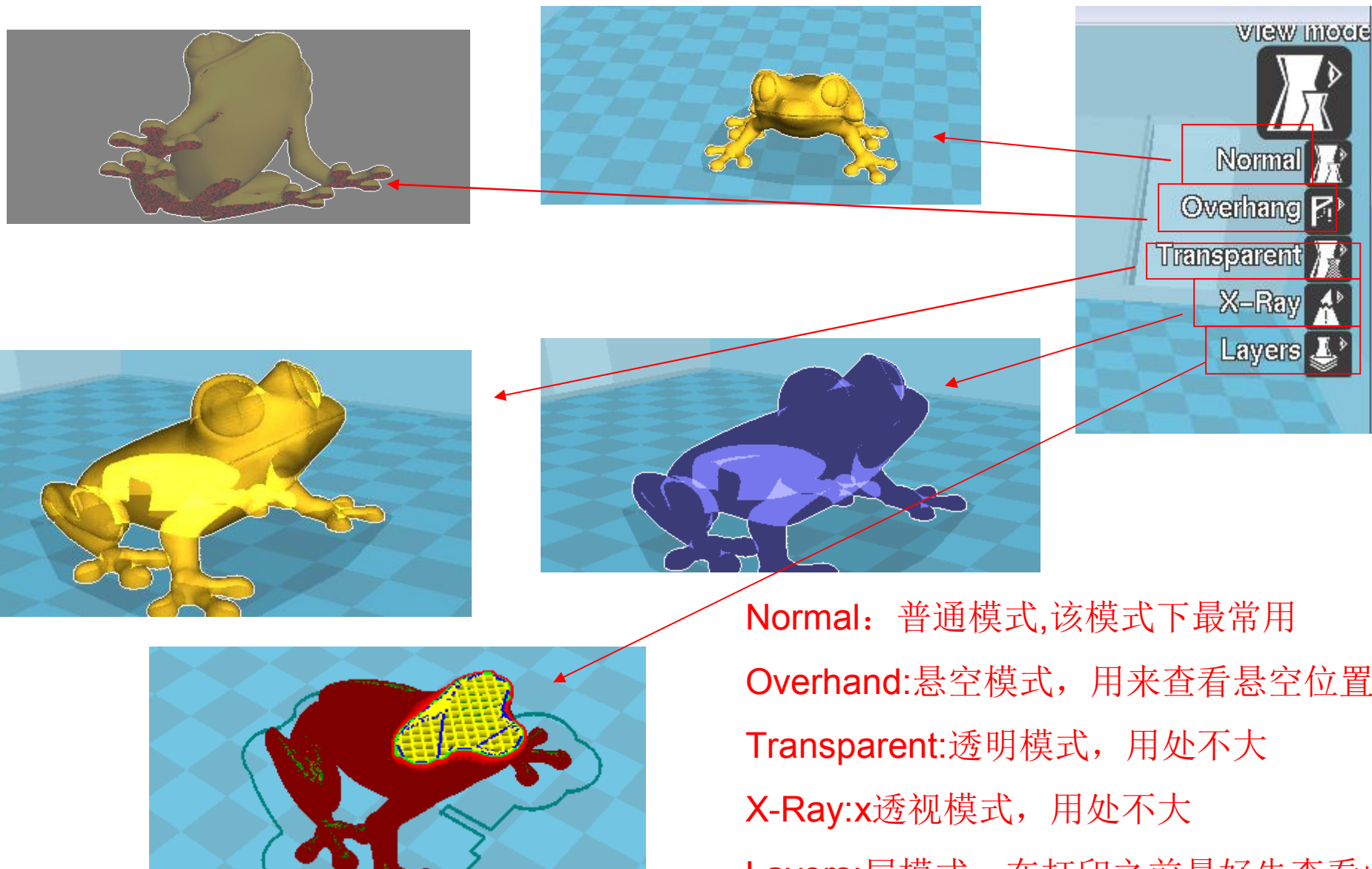
按比例或按尺寸缩放

统一缩放尺寸使能

缩放按钮



镜像



**Normal:** 普通模式,该模式下最常用

**Overhand:**悬空模式，用来查看悬空位置

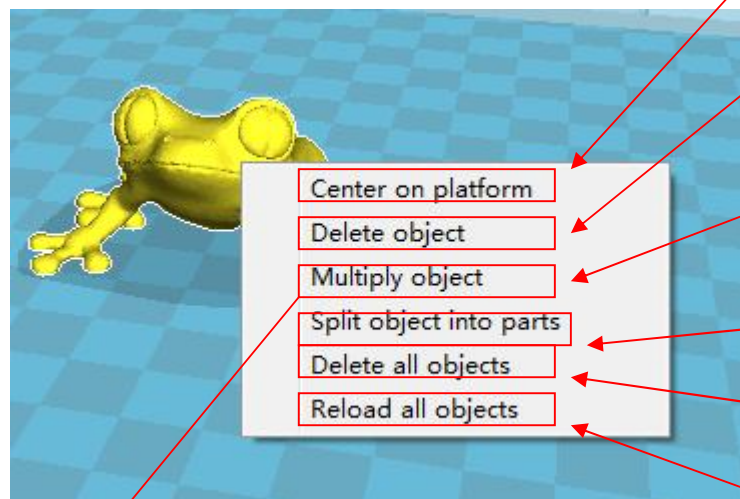
**Transparent:**透明模式，用处不大

**X-Ray:**x透视模式，用处不大

**Layers:**层模式，在打印之前最好先查看此模式，以确认模式被正确切片

显示模式

# 鼠标右键功能



移动到平台中央

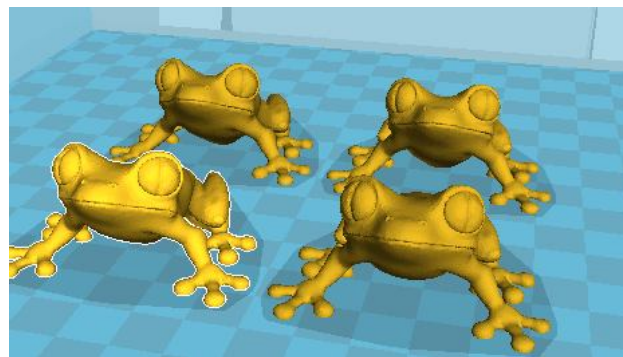
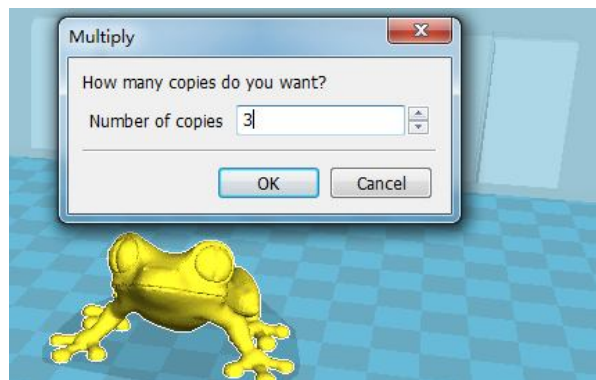
删除模型

模型复制

模型分解

删除所有模型

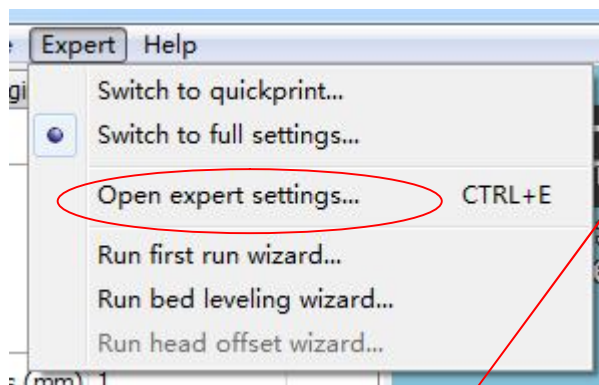
重载所有模型





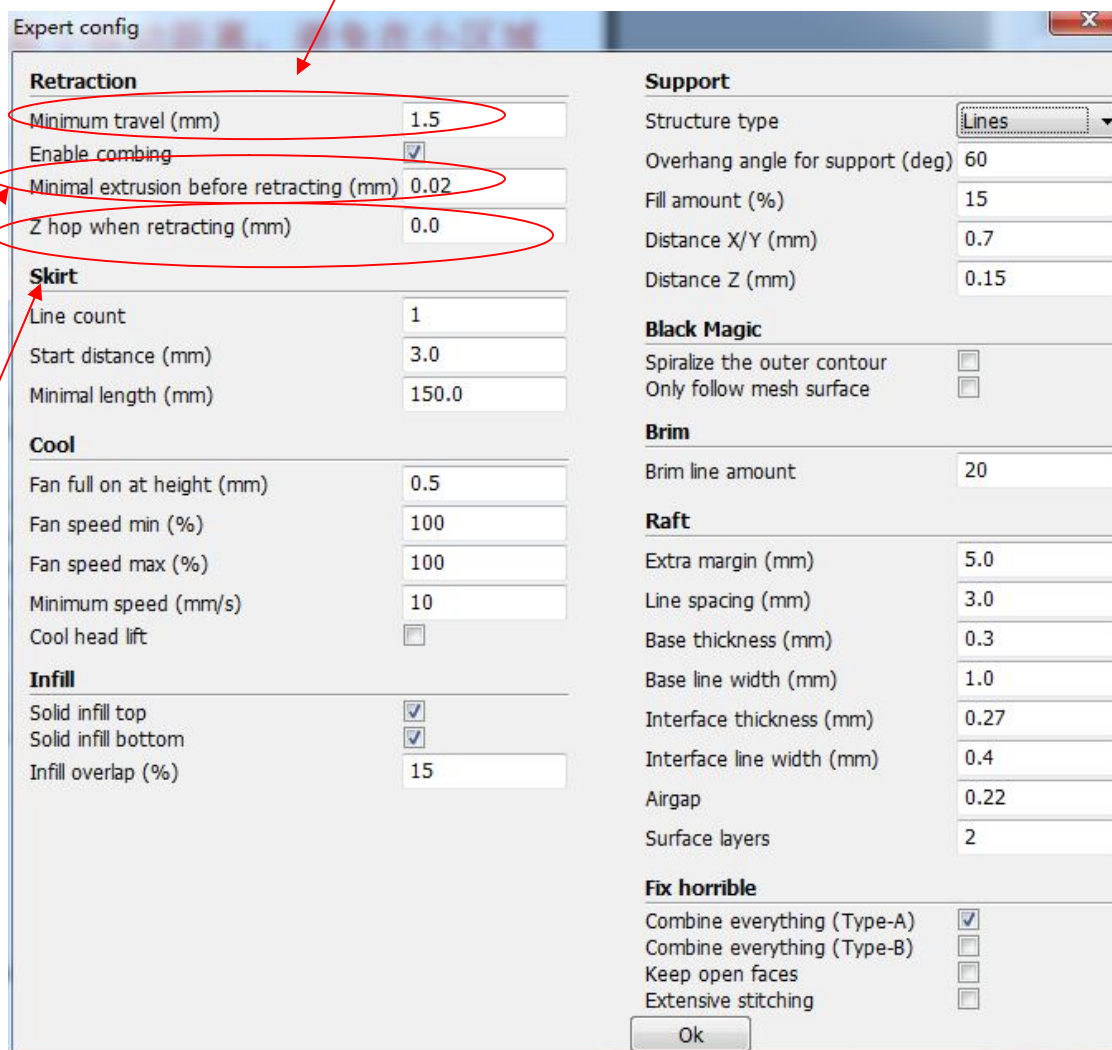
# 专家模式

一次新的回退发生前的最小移动距离，避免在小区域频繁的回退



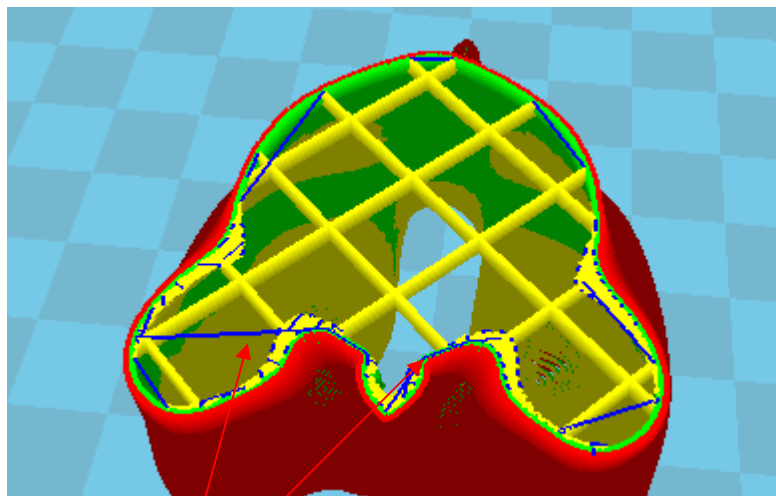
一次新的挤出前的最少挤出长度，防止挤出过于频率

回退时，Z平台下降（挤出头上升）的高度，如果要设置此值，2mm是比较合适的，“跳”一下会使壁上的“泪滴”减少，但物体间会有很多很细的丝



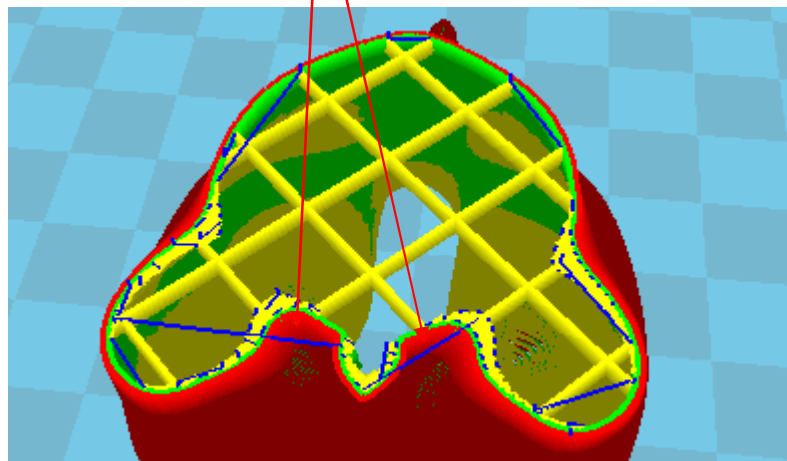


Enable Combing对表面质量非常重要,这也是cura优于slic3r的一个方面,使能组合的话,所有的路径尽量不穿过外壁,哪怕是走弯路!

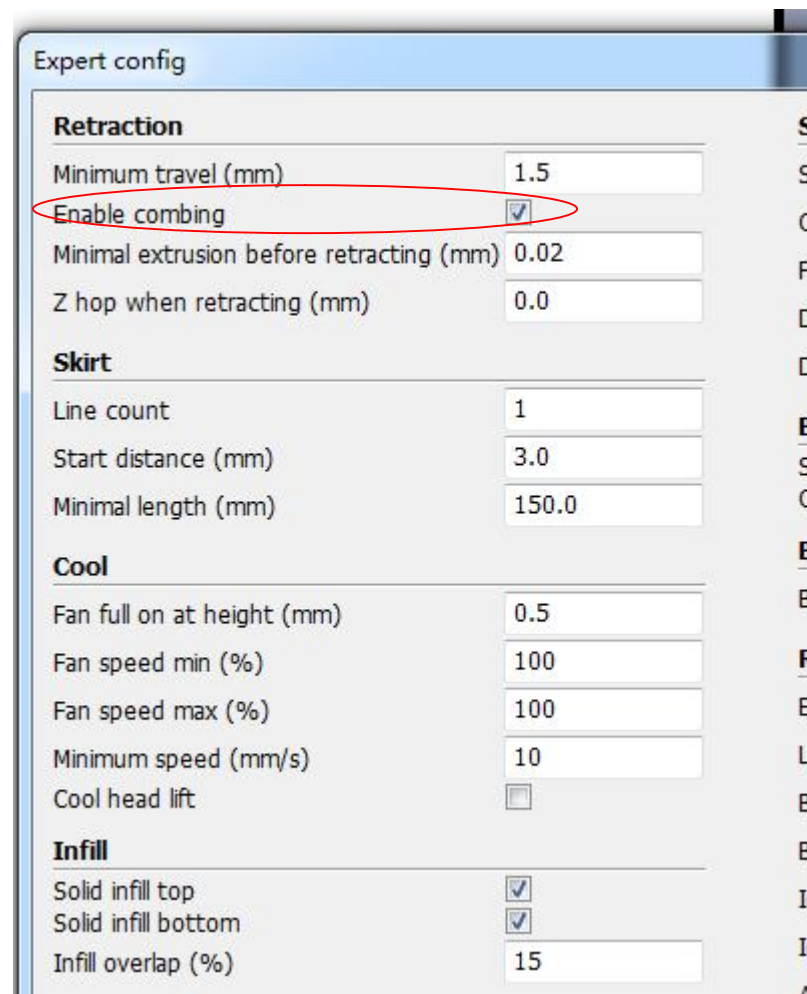


绕过去的路径  
直接从外壁穿过的路径(蓝线)

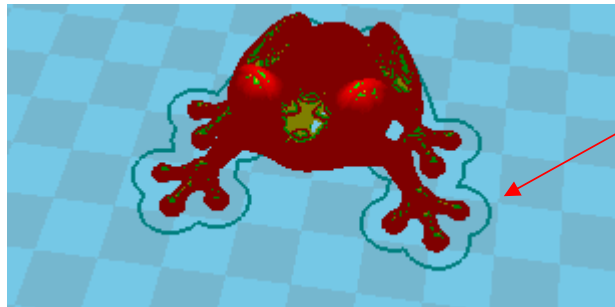
Enable  
后的切片  
路径



未Enable  
后的切片  
路径



衬底(skirt)通常是为了防止挤出头在打印前处于未充满状态，而且只有当 **adhesion type** 处于 **None** 的时候才有，一般为1就可，但当你的模型尺寸达到打印的极限尺寸时，最好将其设为0，否则很有可能因为多出的这个**skirt**使打印尺寸过大



Line count:1  
Start distance:3



Line count:3  
Start distance:6

<b>Retraction</b>	
Minimum travel (mm)	1.5
Enable combing	<input type="checkbox"/>
Minimal extrusion before retracting (mm)	0.02
Z hop when retracting (mm)	0.0
<b>Skirt</b>	
Line count	1
Start distance (mm)	3.0
Minimal length (mm)	150.0
<b>Cool</b>	
Fan full on at height (mm)	0.5
Fan speed min (%)	100
Fan speed max (%)	100
Minimum speed (mm/s)	10
Cool head lift	<input type="checkbox"/>
<b>Infill</b>	
Solid infill top	<input checked="" type="checkbox"/>
Solid infill bottom	<input checked="" type="checkbox"/>
Infill overlap (%)	15

# 冷却设置

为了增加对底板的粘附能力，通过不会一开始打印的时候就用风扇冷却

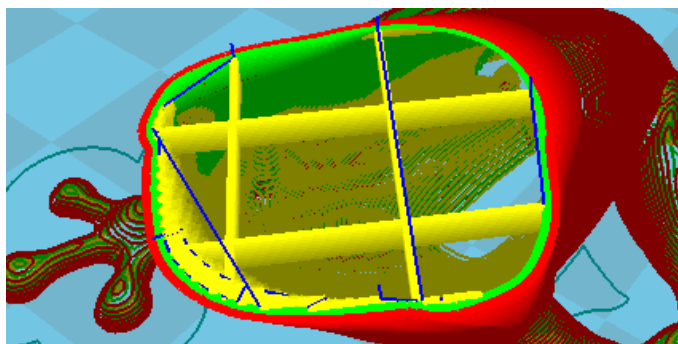
最大风扇速度和最小风扇速度，如果两者不相等，切片软件会在每一层的打印时在最大和最小之间选择一个合适的风扇速度

当因为cool time的条件降低打印速度时，最小允许的最低打印速度

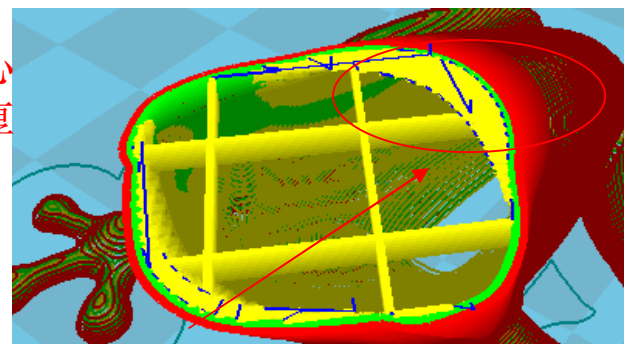
使能后，当cool time因为最小打印速度限制而无法及时满足时，会通过打印完一层后移开挤出头并且Z轴下降的方式“拖延时间”，以满足cool time,但这种方式会千万挤出头漏夜

Expert config

<b>Retraction</b>	
Minimum travel (mm)	1.5
Enable combing	<input type="checkbox"/>
Minimal extrusion before retracting (mm)	0.02
Z hop when retracting (mm)	0.0
<b>Skirt</b>	
Line count	3
Start distance (mm)	6.0
Minimal length (mm)	150.0
<b>Cool</b>	
Fan full on at height (mm)	0.5
Fan speed min (%)	100
Fan speed max (%)	100
Minimum speed (mm/s)	10
Cool head lift	<input type="checkbox"/>
<b>Infill</b>	
Solid infill top	<input checked="" type="checkbox"/>
Solid infill bottom	<input checked="" type="checkbox"/>
Infill overlap (%)	15



如果没有顶层实心填充，则只有壁厚一个约束



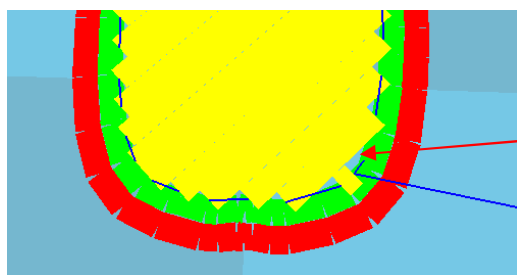
去掉使能

勾选使能



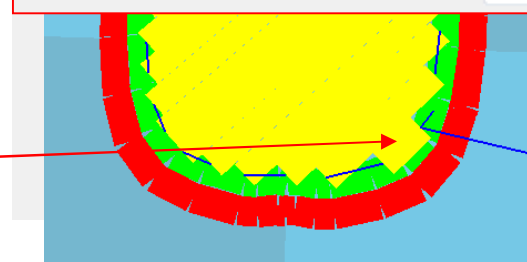
去掉实心底填充后

Start distance (mm)	3.0
Minimal length (mm)	150.0
<b>Cool</b>	
Fan full on at height (mm)	0.5
Fan speed min (%)	100
Fan speed max (%)	100
Minimum speed (mm/s)	10
Cool head lift	<input type="checkbox"/>
<b>Infill</b>	
Solid infill top	<input checked="" type="checkbox"/>
Solid infill bottom	<input checked="" type="checkbox"/>
Infill overlap (%)	15



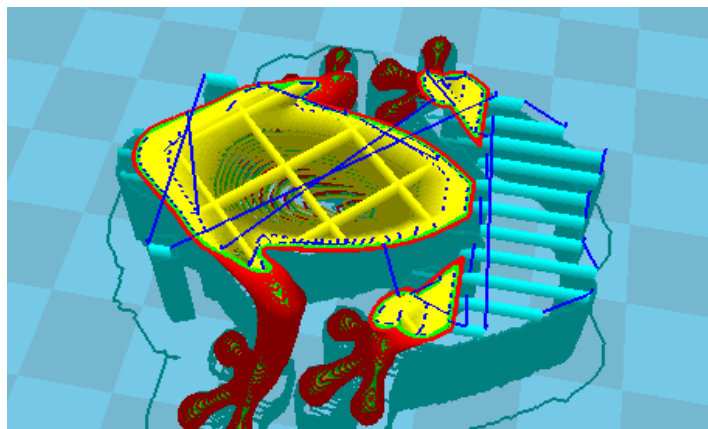
左图infill overlap 15%

右图infill overlap 30%



填充与shell的重叠比例

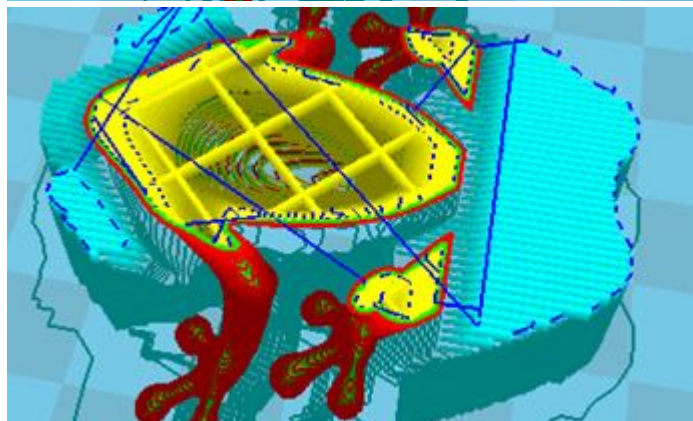




类型:lines

比例:15%

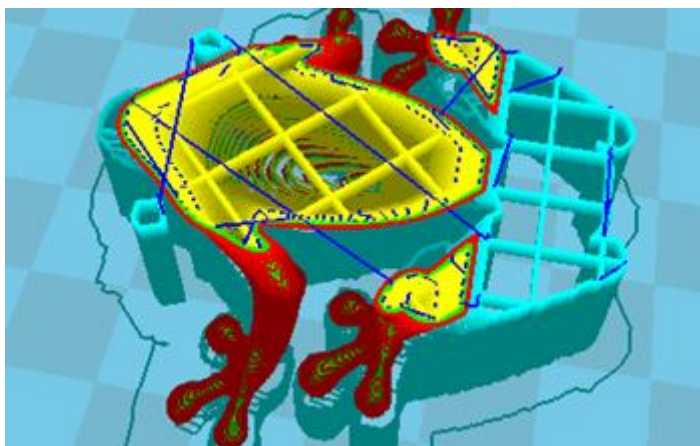
X/Y距离:0.7mm



类型:lines

比例:50%

X/Y距离:3mm

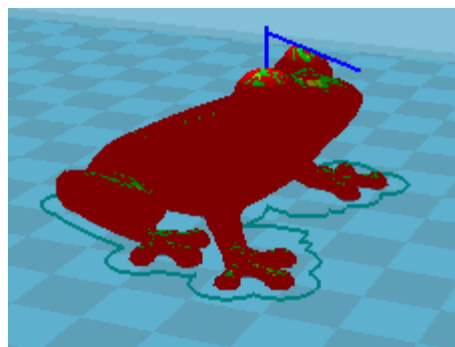


类型:grids

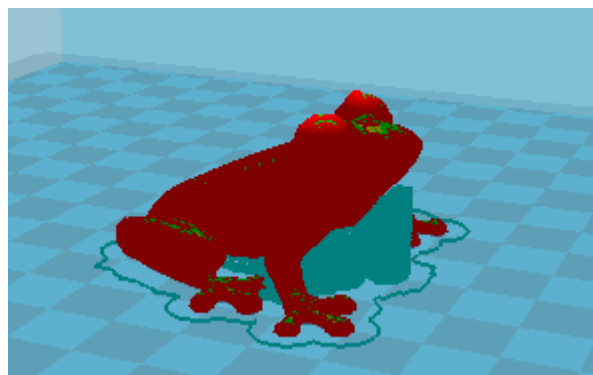
比例:15%

X/Y距离:0.7mm

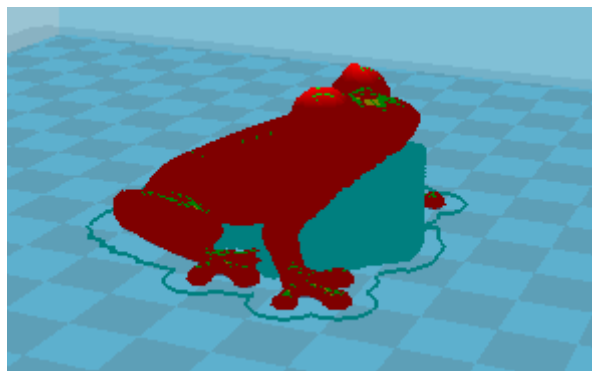
<b>Support</b>	
Structure type	Lines
Overhang angle for support (deg)	60
Fill amount (%)	15
Distance X/Y (mm)	0.7
Distance Z (mm)	0.15
<b>Black Magic</b>	
Spiralize the outer contour	<input type="checkbox"/>
Only follow mesh surface	<input type="checkbox"/>
<b>Brim</b>	
Brim line amount	20
<b>Raft</b>	
Extra margin (mm)	5.0
Line spacing (mm)	3.0
Base thickness (mm)	0.3
Base line width (mm)	1.0
Interface thickness (mm)	0.27
Interface line width (mm)	0.4
Airgap	0.22
Surface layers	2
<b>Fix horrible</b>	
Combine everything (Type-A)	<input checked="" type="checkbox"/>
Combine everything (Type-B)	<input type="checkbox"/>
Keep open faces	<input type="checkbox"/>
Extensive stitching	<input type="checkbox"/>
Ok	



60度



40度



20度

Support

Structure type: Lines

Overhang angle for support (deg): 60

Fill amount (%): 15

Distance X/Y (mm): 0.7

Distance Z (mm): 0.15

**Black Magic**

Spiralize the outer contour: ☐

Only follow mesh surface: ☐

**Brim**

Brim line amount: 20

**Raft**

Extra margin (mm): 5.0

Line spacing (mm): 3.0

Base thickness (mm): 0.3

Base line width (mm): 1.0

Interface thickness (mm): 0.27

Interface line width (mm): 0.4

Airgap: 0.22

Surface layers: 2

**Fix horrible**

Combine everything (Type-A): ☒

Combine everything (Type-B): ☐

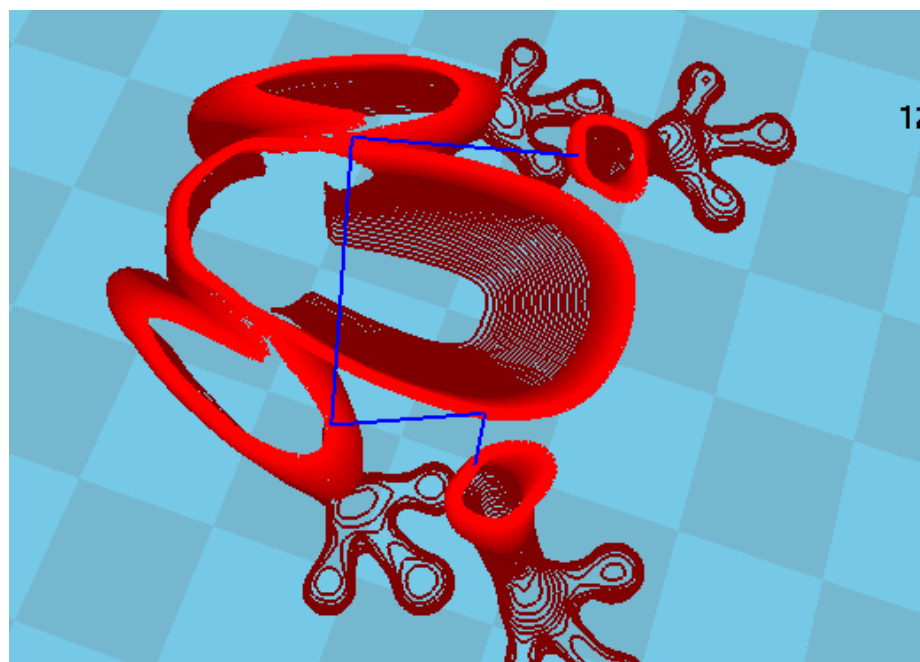
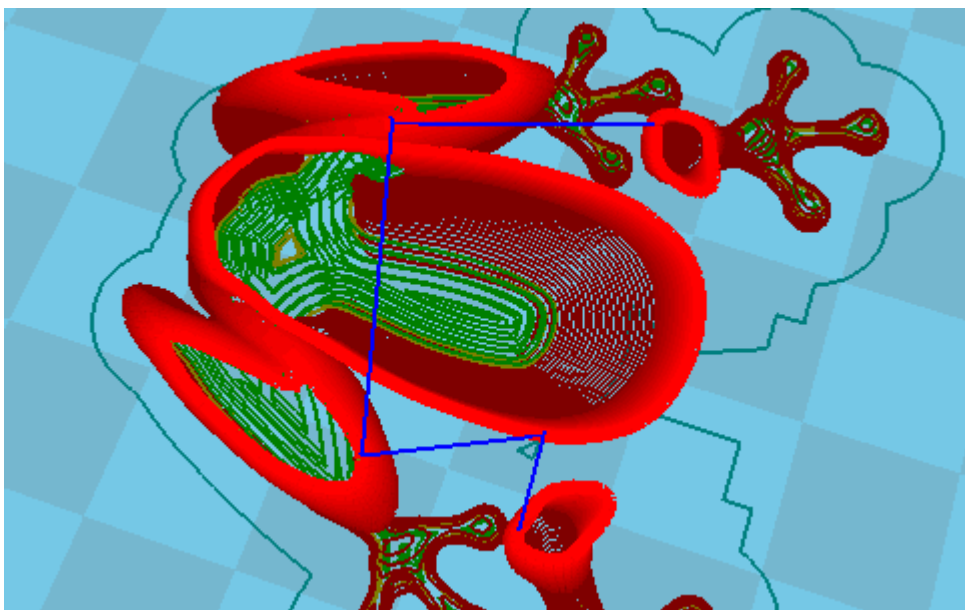
Keep open faces: ☐

Extensive stitching: ☐

Ok

支撑与被支撑物体的距离，太近难剥离，太远下悬丝会影响外观

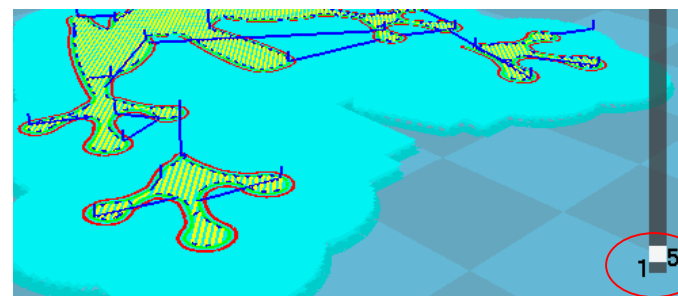
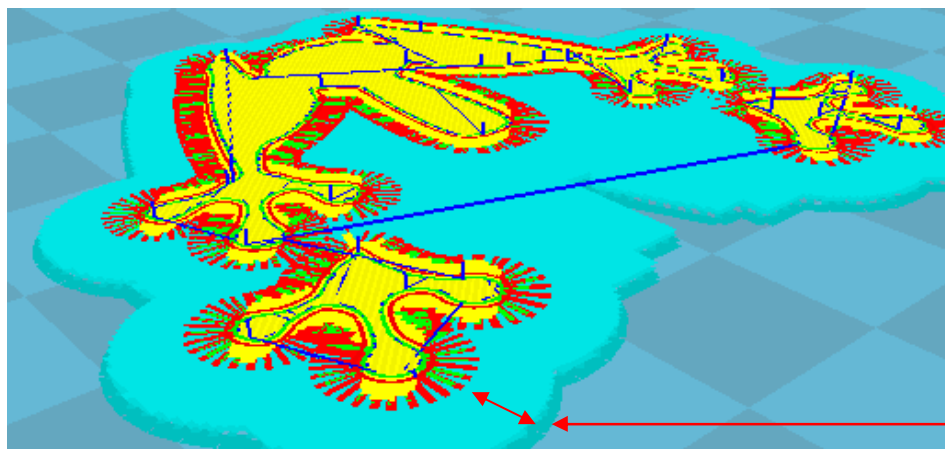




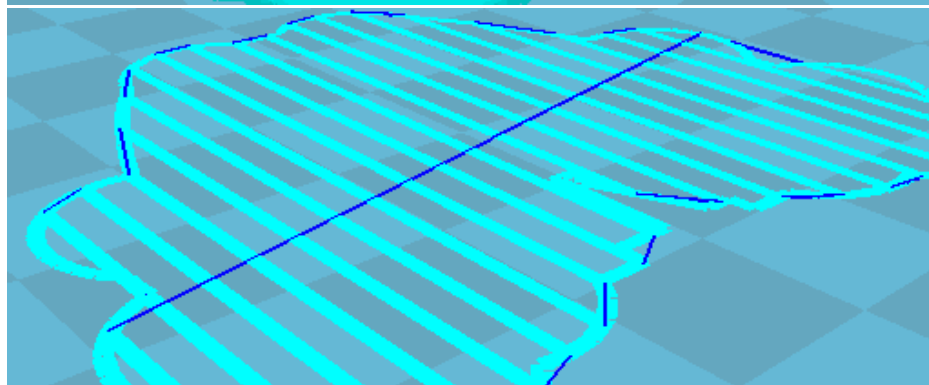
XY一边移动Z轴一边上升，只有一个实心底和单层壁

Support	
Structure type	Grid
Overhang angle for support (deg)	60
Fill amount (%)	15
Distance X/Y (mm)	0.7
Distance Z (mm)	0.15
<b>Black Magic</b>	
Spiralize the outer contour	<input type="checkbox"/>
Only follow mesh surface	<input type="checkbox"/>
<b>Brim</b>	
Brim line amount	20
<b>Raft</b>	
Extra margin (mm)	5.0
Line spacing (mm)	3.0
Base thickness (mm)	0.3
Base line width (mm)	1.0
Interface thickness (mm)	0.27
Interface line width (mm)	0.4

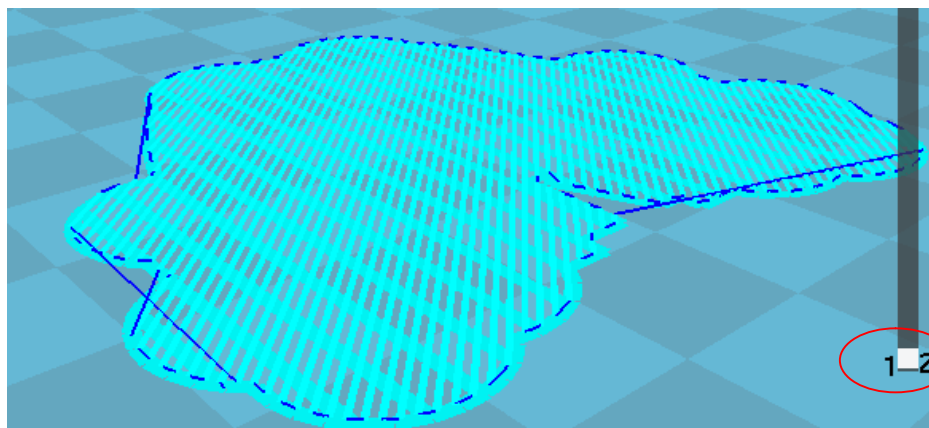
打印路径只沿着外壁走，没有底也没有填充



第一层与raft的间隙



base line

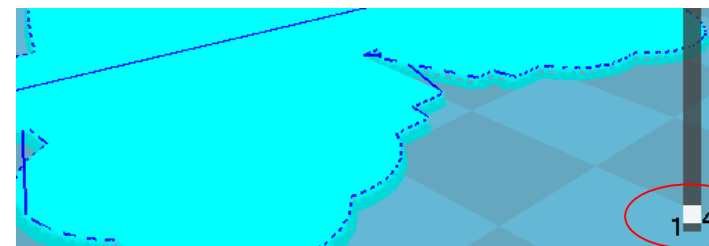


1-2

Brim line amount	20
<b>Raft</b>	
Extra margin (mm)	5.0
Line spacing (mm)	3.0
Base thickness (mm)	0.3
Base line width (mm)	1.0
Interface thickness (mm)	0.27
Interface line width (mm)	0.4
Airgap	0.22
Surface layers	2
<b>Fix horrible</b>	
Combine everything (Type-A)	<input checked="" type="checkbox"/>
Combine everything (Type-B)	<input type="checkbox"/>
Keep open faces	<input type="checkbox"/>
Extensive stitching	<input type="checkbox"/>
Ok	

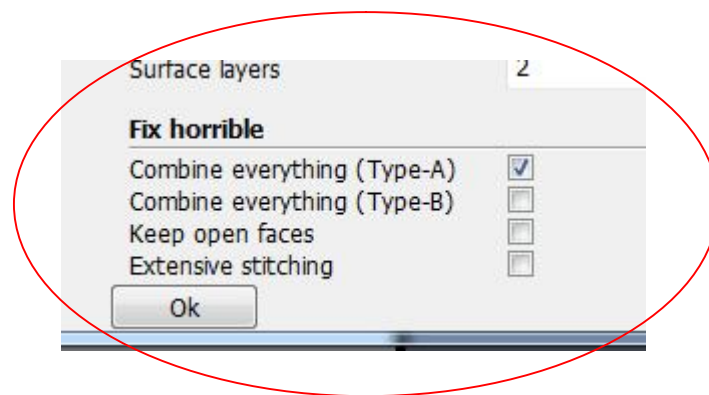
Interface line

surface



1-4

这些参数主要用来修改模型上未封闭的空洞，一般使用默认值即可



至此，本教程全部结束!!!