SQUID BLOX

INSTRUCTIONS

BY JAMES CANN DESIGN

Open-source printing instructions

What you will need to get started ...

1. 3D Filament Printer
2. Mac or PC Computer
3. 1.5mm Nozzle
4. PLA Filament
5. PrusaSlicer Program

Step 1. Set up the 3D printer

Change or screw on the 1.5mm printer nozzle and tighten it with a wench. (drill a accurate 1.5 hole in an existing nozzle if you want)

(Top tip - If the previous nozzle is stuck, set the printer to heat up the filament and the nozzle will unscrew. but be careful the nozzle maybe up to 225° Degrees.)

Then change or install the plastic filament to your desired colour, we recommend using recycled PLA for its environmental qualities, strength and durability.

Check the printer bed is level and clear for use.

Step 2. Downloading...

On the JamesCann.design website select and download the desired dimensioned squidblox file which will look like this:
Squidblox_0mm+00mm.stl

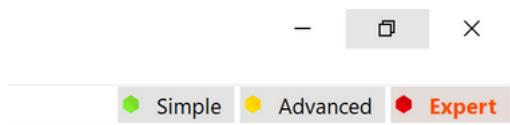Also download a free slicer program called:
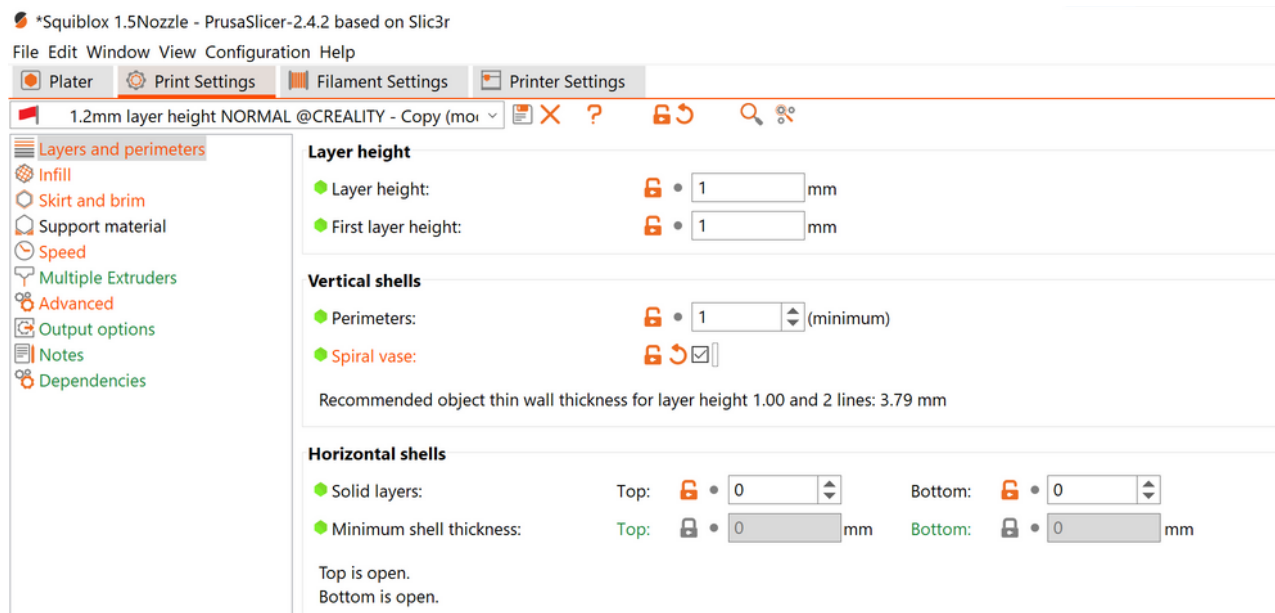**PrusaSlicer**

Step 3. Set up the Slicer program

1.Open up PrusaSlicer, **select your model of printer** and create a new project named **Squidblox 1.5Nozzle**

2. On the right hand side of the screen select **Expert**

Simple ● Advanced ● **Expert**

3.Go to **Print Settings** and **select Layers and perimeters.**

*Squiblox 1.5Nozzle - PrusaSlicer-2.4.2 based on Slic3r

File Edit Window View Configuration Help

Plater | Print Settings | Filament Settings | Printer Settings

1.2mm layer height NORMAL @CREALITY - Copy (mo

Layers and perimeters
Infill
Skirt and brim
Support material
Speed
Multiple Extruders
Advanced
Output options
Notes
Dependencies

Layer height

● Layer height: 1 mm

● First layer height: 1 mm

Vertical shells

● Perimeters: 1 (minimum)

● Spiral vase:

Recommended object thin wall thickness for layer height 1.00 and 2 lines: 3.79 mm

Horizontal shells

● Solid layers: Top: 0   Bottom: 0

● Minimum shell thickness: Top: 0 mm Bottom: 0 mm

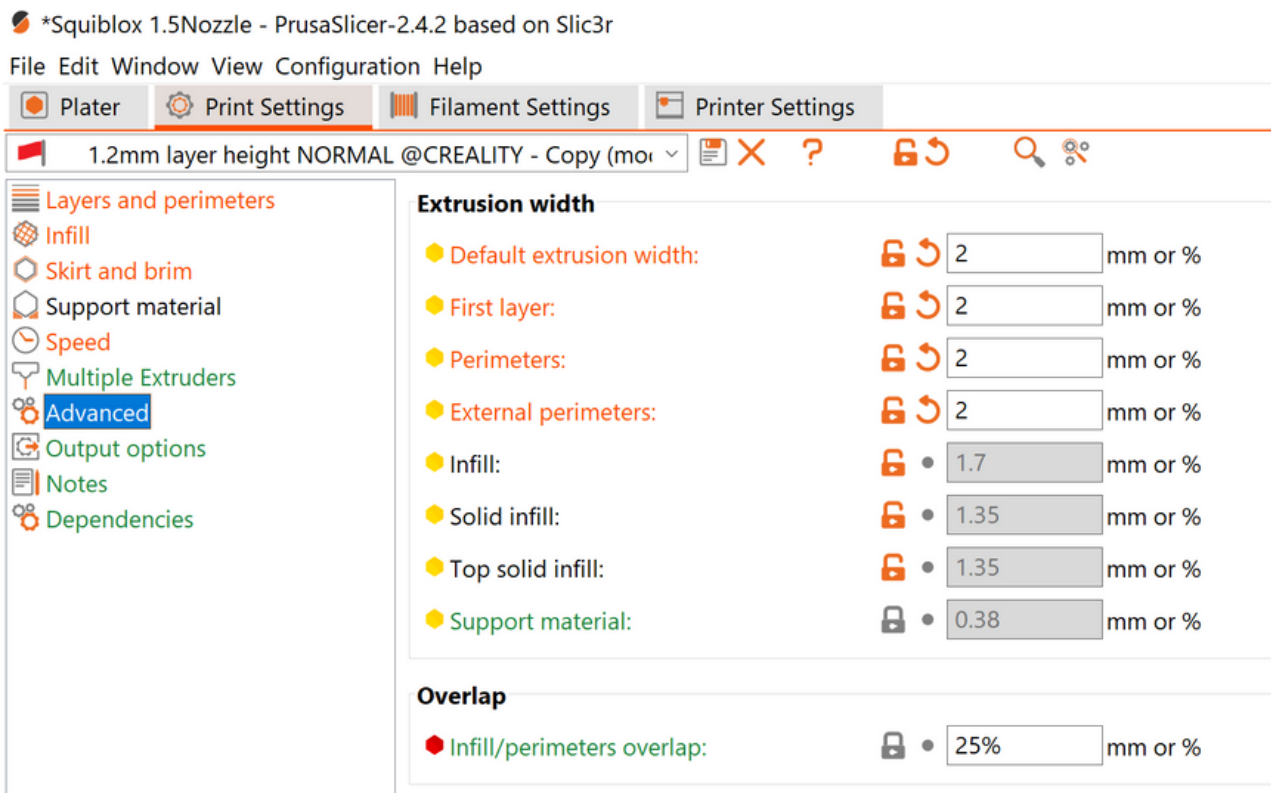Top is open.
Bottom is open.

4.Chnage the **layer height to 1mm**
5.Chnage the **First layer height to 1mm**

6. In the **Verticale shells** section tick **spiral vase mode and click okay.**

7. In the **Horizontal Shells** section turn **Solid layers values all to O**
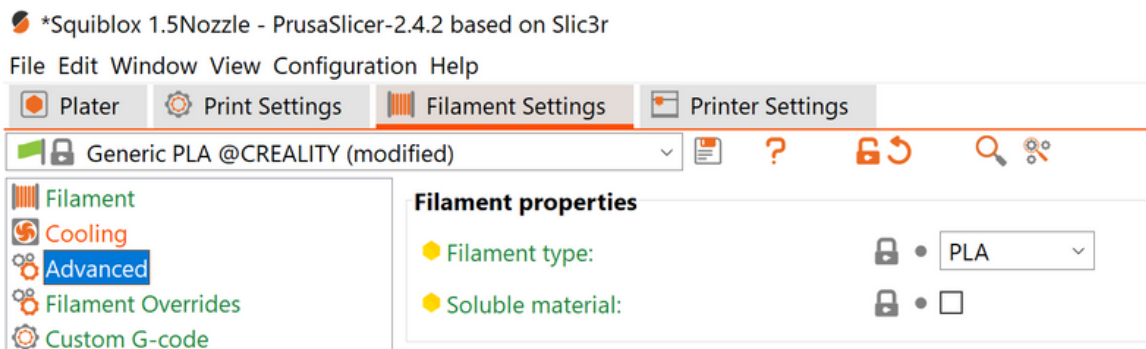
8. In the printer settings still on the left hand side column click **advanced**
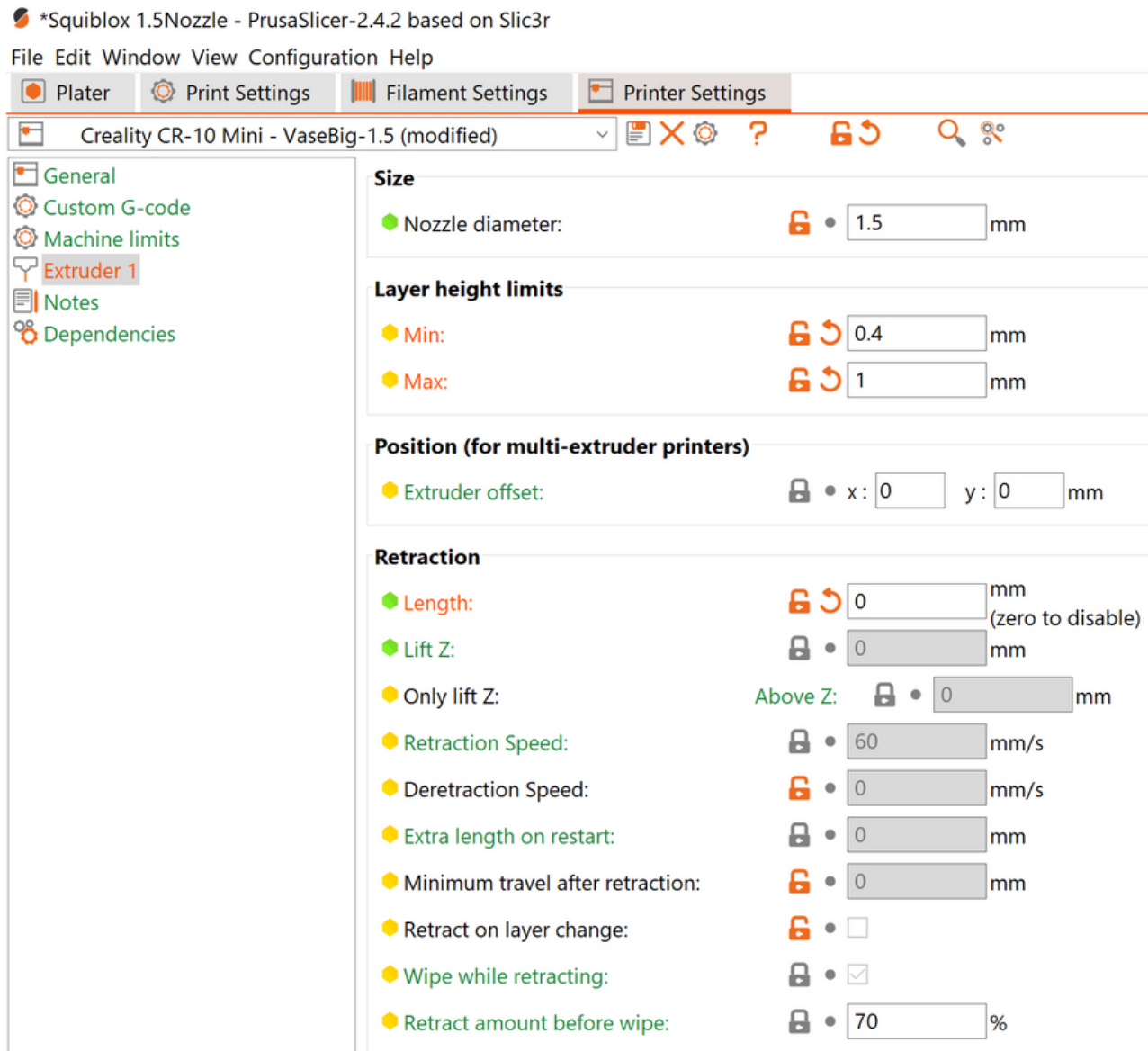

9. Change the vales of **Extrusion width, First layer, Perimeters and External perimeters** all to 2mm

10. Go to the **Filament settings** and then click **Advanced** in the left hand column.

11. In **Filament Properties** click the drop down menu to change the filament type to **PLA**

12. Go to **Printer Settings** and then click **Extruder** in the left hand column.

13. Set the size **Nozzle Diameter** to **1.5mm**

14. Set the layer layer heights limits to **Min: 0.4mm and Max: 1mm**

15. In the **Retraction** section set the **Length** to **0**.

**16.** In the **Plater** view import the object with this symbol on the top dock of tools

17. Use this symbol to highlight surfaces for the object to sit on. **Click the side with the chamfer edge.**

18. Click the **slicer** button in the bottom right of the screen to see how the slicer program will print the object.

19. Click the **export** button in the bottom right of the screen to save the g-code and upload to the 3D printer.
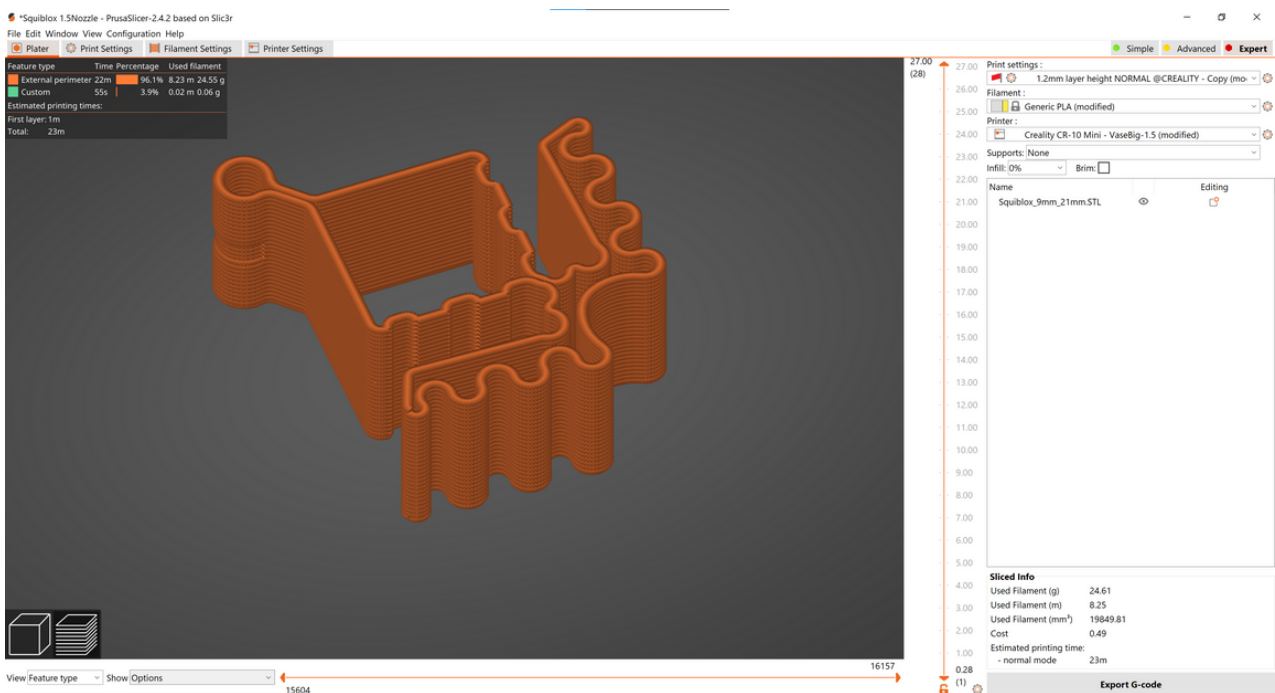
Step 4. Start printing

Start the print and check carefully for a perfect print.