

This model was tested in Cura 2.3.1. To avoid printing problems, we recommend the following settings:

### **Quality**

Layer Height: 0.1 mm  
Initial Layer Height: 0.3 mm  
Line Width: 0.4 mm  
Wall Line Width: 0.4 mm  
Outer Wall Line Width: 0.4 mm  
Inner Wall(s) Line Width: 0.4 mm  
Top/Bottom Line Width: 0.4 mm  
Infill Line Width: 0.4 mm  
Skirt/Brim Line Width: 0.4 mm  
Support Line Width: 0.4 mm

### **Shell**

Wall Thickness: 0.8 mm  
Wall Line Count: 2  
Top/Bottom Thickness: 0.8 mm  
Top Thickness: 0.8 mm  
Top Layers: 8  
Bottom Thickness: 0.8 mm  
Bottom Layers: 8  
Top/Bottom Pattern: Lines  
Outer Wall Inset: 0 mm  
Compensate Wall Overlaps: Check  
Compensate Outer Wall Overlaps: Check  
Compensate Inner Wall Overlaps: Check  
Horizontal Expansion: 0 mm  
Z Seam Alignment: Shortest

### **Infill**

Infill Density: 25%  
Infill Line Distance: 3.2 mm  
Infill Pattern: Grid  
Infill Overlap Percentage: 10%  
Infill Overlap: 0.04 mm  
Skin Overlap Percentage: 5%  
Skin Overlap: 0.02 mm  
Infill Wipe Distance: 0.1 mm  
Infill Layer Thickness: 0.1 mm  
Gradual Infill Steps: 0  
Infill Before Walls: Check

## **Material**

Enable Retraction: Check  
Retraction Extra Prime Amount: 0 mm<sup>3</sup>  
Retraction Minimum Travel: 0.8 mm  
Maximum Retraction Count: 90  
Minimum Extrusion Distance Window: 6.5 mm  
Standby Temperature: 175°C  
Nozzle Switch Retraction Distance: 16 mm  
Nozzle Switch Retraction Speed: 20 mm/s  
Nozzle Switch Retract Speed: 20 mm/s  
Nozzle Switch Prime Speed: 20 mm/s

## **Speed**

Print Speed: 60 mm/s  
Infill Speed: 80 mm/s  
Wall Speed: 30 mm/s  
Outer Wall Speed: 30 mm/s  
Inner Wall Speed: 60 mm/s  
Top/Bottom Speed 15 mm/s  
Support Speed: 60 mm/s  
Support Infill Speed: 60 mm/s  
Travel Speed: 120 mm/s  
Initial Layer Speed: 15 mm/s  
Initial Layer Print Speed: 15 mm/s  
Initial Layer Travel Speed: 30 mm/s  
Skirt/Brim Speed 15 mm/s  
Maximum Z Speed: 0 mm/s

## **Travel**

Combing Mode: All  
Avoid Printed Parts when Traveling: Check  
Travel Avoid Distance: 0.625 mm

## **Cooling**

Enable Print Cooling: Check  
Fan Speed: 100%  
Regular Fan Speed: 100%  
Maximum Fan Speed: 100%  
Regular/Maximum Fan Speed Threshold: 10 s  
Regular Fan Speed at Height: 0.3 mm  
Regular Fan Speed at Layer: 2  
Minimum Layer Time: 5 s  
Minimum Speed: 10 mm/s

## **Support**

Enable Support: Check  
Support Placement: Everywhere  
Support Overhang Angle: 50°  
Support Pattern: Zig Zag  
Connect Support ZigZags: Check  
Support Density: 30 %  
Support Line Distance: 1.3333 mm  
Support Z Distance: 0.1 mm  
Support Top Distance: 0.1 mm  
Support Bottom Distance: 0.1 mm  
Support X/Y Distance: 0.7 mm  
Support Distance Priority: Z overrides X/Y  
Minimum Support X/Y Distance: 0.2 mm  
Support Stair Step Height: 0.3 mm  
Support Join Distance: 2.0 mm  
Support Horizontal Expansion: 0.2 mm  
Use Towers: Check  
Tower Diameter: 3.0 mm  
Minimum Diameter: 3.0 mm  
Tower Roof Angle: 65°

## **Build Plate Adhesion**

Build Plate Adhesion Type: Brim  
Skirt/Brim Minimum Length: 250 mm  
Brim Width: 8.0 mm  
Brim Line Count: 20  
Brim Only on Outside: Check

## **Mesh Fixes**

Union Overlapping Volumes: Check

## **Special Modes**

Print Sequence: All at Once  
Surface Mode: Normal

## **Experimental**

Extra Skin Wall Count: 0

**Disclaimer:** This model will look outstanding if printed on SLA/SLS 3D printer. The accuracy of the model printed on FFF printer can vary from the result shown in the pictures.