This model was tested in **Cura 3.5.1** and printed on an **Ultimaker 2** in **PLA material**.

Recommendations: For all parts of Locks you need to change "Brim" type to "Skirt" in Build Plate Adhesion section.

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
 Initial Layer Line Width: 100%

Shell
Wall Thickness: 0.8 mm
    Wall Line Count: 2
Outer Wall Wipe Distance: 0.2 mm
Top Surface Skin Layers: 0
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
Bottom Pattern Initial Layer: Lines
Top/Bottom Line Directions: [ ]
Outer Wall Inset: 0 mm
Compensate Wall Overlaps: Check
    Compensate Outer Wall Overlaps: Check
    Compensate Inner Wall Overlaps: Check
Fill Gaps Between Walls: Everywhere
Filter Out Tiny Gaps: Check
Horizontal Expansion: 0 mm
Initial Layer Horizontal Expansion: 0 mm
Z Seam Alignment: Sharpest Corner
Seam Corner Preference: Hide Seam

Ignore Small Z Gaps: Check
Extra Skin Wall Count: 1

Infill
Infill Density: 20%
Infill Line Distance: 4.0 mm
Infill Pattern: Grid
Infill Line Directions: [ ]
Infill X Offset: 0 mm
Infill Y Offset: 0 mm
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: 1
Gradual Infill Steps Height: 1.5 mm
Infill Before Walls: Check
Minimum Infill Area: 0 mm2
Skin Removal Width: 0.8 mm
   Top Skin Removal Width: 0.8 mm
   Bottom Skin Removal Width: 0.8 mm
Skin Expand Distance: 0.8
   Top Skin Expand Distance: 0.8
   Bottom Skin Expand Distance: 0.8
Maximum Skin Angle for Expansion: 90˚
   Minimum Skin Width for Expansion: 0.0

Material
Initial Layer Flow: 100%
Enable Retraction: Check
Retraction Extra Prime Amount: 0 mm3
Retraction Minimum Travel: 0.8 mm
Maximum Retraction Count: 90
Minimum Extrusion Distance Window: 6.5 mm
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: 45 mm/s

Infill Speed: 45 mm/s
Wall Speed: 22.5 mm/s
Outer Wall Speed: 22.5 mm/s
Inner Wall Speed: 45 mm/s
Top/Bottom Speed: 15 mm/s
Support Speed: 45 mm/s
Support Infill Speed: 45 mm/s
Travel Speed: 45 mm/s
Initial Layer Speed: 22.5 mm/s
Initial Layer Print Speed: 22.5 mm/s
Initial Layer Travel Speed: 30 mm/s
Skirt/Brim Speed: 45 mm/s
Maximum Z Speed: 0 mm/s
Number of Slower Layers: 2

Travel
Combing Mode: All
Avoid Printed Parts when Traveling: Check
Travel Avoid Distance: 0.6562 mm
Layer Start X: 0.0 mm
Layer Start Y: 0.0 mm

Cooling
Enable Print Cooling: Check
Fan Speed: 100%
Regular Fan Speed: 100%
Maximum Fan Speed: 100%
Regular/Maximum Fan Speed Threshold: 10 s
Initial Fan Speed: 0%
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
Generate Support: Check
Support Placement: Everywhere
Support Overhang Angle: 60°
Support Pattern: Zig Zag
Connect Support ZigZags: Check
Support Density: 15 %
Support Line Distance: 2.2 mm
Support Z Distance: 0.13 mm

Support Top Distance: 0.13 mm
Support Bottom Distance: 0.13 mm
Support X/Y Distance: 1 mm
Support Distance Priority: Z overrides X/Y
Minimum Support X/Y Distance: 0.25 mm
Support Stair Step Height: 0.3 mm
Support Stair Step Maximum Width: 5.0 mm
Support Join Distance: 2.0 mm
Support Horizontal Expansion: 0.2 mm
Support Infill Layer Thickness: 0.1 mm
Gradual Support Infill Steps: 0
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 (for all parts of locks use "Skirt")
Skirt/Brim Minimum Length: 250 mm
Brim Width: 8.0 mm
Brim Line Count: 15
Brim Only on Outside: Check

Mesh Fixes
Union Overlapping Volumes: Check
Merged Meshes Overlap: 0.15 mm

Special Modes
Print Sequence: All at Once
Surface Mode: Normal

Experimental
Slicing Tolerance: Middle
Maximum Resolution: 0.01 mm
Flow rate compensation max extrusion offset: 0 mm
Flow rate compensation factor: 100%

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.