All components should be printed with FDM printers.

Consider experimenting with lower wall line count, infill, and top & bottom layers for the main body to improve print time and reduce material cost. For safety reasons, always pressure test with water.

Main Body:

Orientation: Adapters threaded holes facing down.

Material: ABS

Nozzle: 0.4mm

Line Width: 0.4mm for all types of lines.

Layer Height: 0.14mm

Infill: >35%; Triangle pattern.

Wall Line Count: >7

Top & Bottom Layers Count: >18

Speed: 30-40 mm/s for all line types recommended.

Z Seam Alignment: Custom. Use any of the four corners.

Support: Support overhang angle > 85 degrees at 15% density. There should be no support inside the body, but only inside the threaded holes.

Top & Bottom Material Flow: 98.5%

Part Cooling Fan: 2%

Build Plate Adhesion: No brim, raft, or glue required with a clean glass build plate set at 90 degrees and an enclosed chamber.

Adapters:

Orientation: Right side up.

Material: Polycarbonate

Nozzle: 0.4mm

Line Width: 0.4mm for all types of lines.

Layer Height: 0.13mm

Wall Line Count: 10

Speed: 30 mm/s

Z Seam Alignment: Random

Part Cooling Fan: 0%

Support: Off

Build Plate Temperature: 115 degrees C

Build Plate Adhesion: Brim or glue may be required.

Filter Ring:

Orientation: Flat circular surface on build plate.

Material: PLA, ABS or other.

Nozzle: 0.25 or 0.4mm. Cura 5 is recommended for printing with a 0.4mm nozzle.

Layer Height: 0.13mm

Support: Off