



Printing Guide:

Meek

Please read the following pages carefully to get a successful first print on any printer.

Bambu 3MF file + recommended settings included!

- Works for any printer in [Orca Slicer](#)!

Troubleshooting is at the end of this file.

For private use only!

Commercial license, more designs & content at:

→ [Factorian Designs](#)

Get my favorite equipment & filament here:

→ [Favorite Equipment](#)

Settings - Vasemode

The vase is designed to be printed in Vasemode / Spiralmode. You can print it normally, but Vasemode will be cleaner and faster.

- Activate Vasemode / Spiral vase mode

I recommend using a 0,6 mm nozzle or bigger.

Settings:

- 5-6 bottom layers
- bottom, solid infill and top pattern: „concentric“

0,4 mm nozzle:

- 0,7-0,8 mm linewidth & 0,16-0,2 mm layer height

0,6 mm nozzle:

- 0,9-1,0 mm linewidth & 0,18-0,2 mm layer height

0,8 mm nozzle:

- 1,1-1,2 mm linewidth % 0,2-0,22 mm layer height

Go slowly when using those wide lines! Overhangs are also better this way.

→ Look at the “Trouble shooting” section for watertight settings.

Settings - Normal mode

If you hate Vasemode, here are some recommendations for the normal print mode:

- Use Arachne Wall generator
(standard in Cura but not Bambu Studio)
- Disable z-hop, this setting is not needed for this part
- 5 bottom & top layers
- Bottom, solid infill and top pattern: „concentric“
- 0,2 mm layer height
- Wall printing order “outer/inner/outer”

→ Look at the “Trouble shooting” section for watertight settings.

If you are using matte PLA, use the same speed for the walls and overhangs to get an even matte surface finish.

Size Guide:

vase in pictures are 180mm and 230mm tall
print at 100% size = 70x70x180 mm

Printing at **smaller sizes**:

Be sure to use vasmode, normal mode won't work properly for very small sizes

Printing at **bigger sizes**:

Nothing to do here. Go as large as you want.

Tipp:

A little gravel, sand or stones should be added on the inside to keep your vase from falling over

For private use only!

Commercial license, more designs & content at:

→ [Factorian Designs](#)

Get my favorite equipment & filament here:

→ [Favorite Equipment](#)

Used Colors / Filament

I highly recommend using matte & pastel colors.
Here is what Filament I use for my pictures:

- [Matte Charcoal Black](#) -
- [Matte Ivory White](#) -

*Affiliate links included. I may earn a small commission, at no extra cost to you.

Thanks for the support!

For private use only!

Commercial license, more designs & content at:

→ [Factorian Designs](#)

Get my favorite equipment & filament here:

→ [Favorite Equipment](#)

⚠ Troubleshooting: ⚠

My print is not watertight:

- Increase your printing temperature slightly
(5-10 °C should be enough)
- Go slower! *(100 mm/s outer & inner walls is safe)*
- Make sure your filament is dry
(wet filament also decreases print quality)
- Make sure you are not under extruding
(if you do, maybe increase your flowrate by 1-3%)
- Reduce your acceleration values
(half of your standard is safe)
- Using the concentric pattern helps
(for top, solid-Infill and bottom surfaces)

The above values are just references, modify as you see fit for your machine.

I don't want to reprint it to make it watertight ...

Just add a thin sealing coat of:

- Clear Acrylic Spray
- Polyurethane (PU) Sealer
- Epoxy Resin (Best Overall Seal)

For private use only!

Commercial license, more designs & content at:

→ [Factorian Designs](https://factoriandesigns.com)