



## Silky Crease Vase - featuring a dual wall vase mode design



Andreas Mass

[VIEW IN BROWSER](#)

updated 3. 5. 2023 | published 3. 5. 2023

### Summary

A silky smooth vase design. Featuring a clever dual wall variant which can still be printed in vase mode.

[Household](#) > [Home Decor](#)

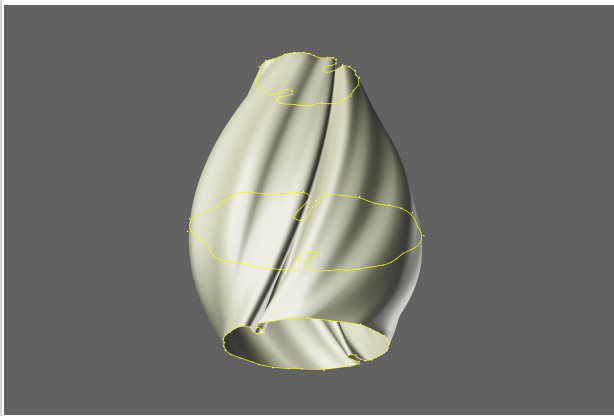
Tags: [decor](#) [fold](#) [home](#) [silky](#) [smooth](#) [vase](#) [vasemode](#)  
[crease](#)

This took a lot longer than intended thanks to a bunch of iterations but I'm glad I finished in time for the contest deadline!

I designed a vase that not only has a silky smooth appearance but that can also be printed in three different ways, yet still without revealing an obvious seam.

The vase is 20cm in height and roughly 15cm wide/10cm deep, but you can scale it according to your needs.





### 01. Traditional Vase Mode

First print option is the traditional vase mode. Use the “\_universal.stl” file for that. It's pretty straight forward here: enable “Spiral vase” mode and set a desired number of bottom layers. Done.

As with all vase mode prints (especially printed with 0.4mm nozzles) I recommend increasing the line width and also the extrusion multiplier by a bit. Setting “Infill/perimeter overlap” to something like 35% also helps to ensure that the walls and bottom layers are fully sealed.

Print example on the left was done using 3D Jake's magic PLA Metallic Emerald and the vase was scaled down to 14cm height.

## 02. Regular Print

The second print option is printing it regularly with your desired number of perimeters. Use the “\_universal.stl” file in this case as well. Just make sure to set the following.

**Top layers: 0**

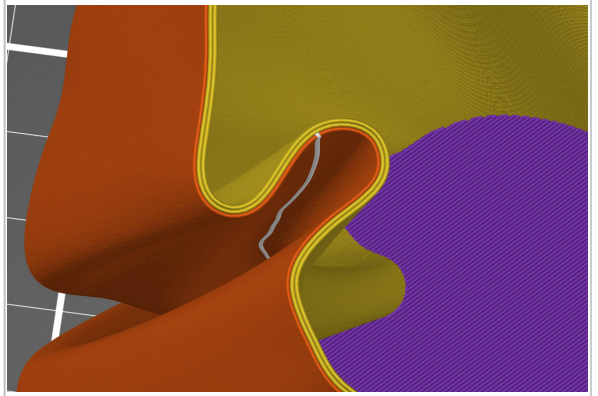
**Infill density: 0%**

**Seam position: Aligned**

**Ensure vertical shell thickness: off**

With the seam position set to aligned the slicer will automatically place the seam within one of the creases as seen on the right. Or if preferred you can also paint on the seam position yourself. Either way, placing the seam inside one of the two creases will nicely hide it when looking at the vase from the front.

Disabling the "Ensure vertical shell thickness" option will prevent the slicer from adding additional material along the inner walls in sloped areas. The print example on the right was printed in Prusament Viva la Bronze PLA, with the vase scaled down to 14cm height and with 3 perimeters.







### 03. Dual Wall Vase Mode

And as a third print option I created a special model

("\_dualwall\_vasemode.stl") which allows you to print the design in vase mode, yet with two walls.

This could be desirable if you want the vase to have a thicker appearance but still have the advantage of consistent, uninterrupted extrusion. Especially interesting if you use a 0.4mm diameter nozzle but find that normal vase mode leads to thin/flimsy looking prints. Having two walls also decreases the chance of ending up with a non-watertight vase.

This design could also be interesting if you would like to experiment with more translucent filaments and see how light shines through the walls.

While I tested this model only with a 0.4mm nozzle and the further described settings are based on that, the design should also work with bigger nozzle diameters. The distance between the two walls is 3mm.

First, **enable "Spiral vase" mode**, set **layer height to 0.2mm**, and the **bottom layers to 5**. You can also print with a different layer height but just make sure that the number of bottom layers will cover at least 1mm of height. That is because the thickness of the bottom in this model is exactly 1mm. Scaling the vase up/down will ofc change the bottom thickness as well so beware.

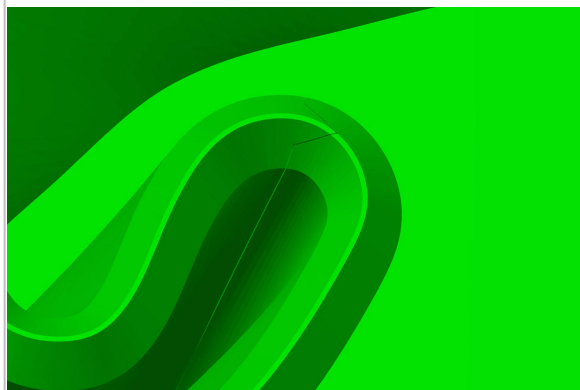
### **VERY, VERY IMPORTANT!**

To print this model you need to set the following under the "Advanced" print settings.

**Slice gap closing radius: 0mm**

This setting is very important because at the default value (0.049mm) the slicer will detect the thin gap that splits the wall (seen on the right) and automatically ignore/close it when you attempt to slice it. Once you set the closing radius to 0mm the slicer will treat the model properly and generate a continuous spiral path.

Additionally I highly recommend to **increase the extrusion multiplier by about 0.05**. So if your usual extrusion multiplier is set to 1.0, set it to 1.05. Doing so will ensure the wall perimeters will nicely overlap within the gap. Btw. I gave the gap this pointy shape on purpose because I've found in my tests that compared to a straight cut this reliably seals the gap and makes the vase watertight. I also increased the "Infill/perimeter overlap" to 35% so the outer wall nicely seals with the bottom layers... just in case.



#### Extrusion width

● Default extrusion width:		0.45	mm or %
● First layer:		0.45	mm or %
● Perimeters:		0.45	mm or %
● External perimeters:		0.45	mm or %
● Infill:		0.45	mm or %
● Solid infill:		0.45	mm or %
● Top solid infill:		0.45	mm or %
● Support material:		0.35	mm or %

#### Overlap

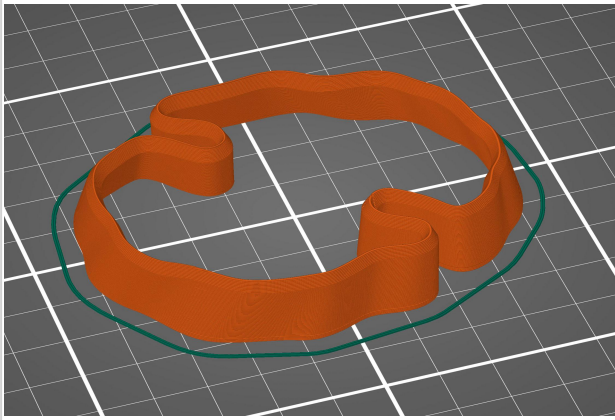
● Infill/perimeters overlap:			35%	mm or %
------------------------------	--	--	-----	---------

#### Flow

● Bridge flow ratio:		0.95
----------------------	--	------

#### Slicing

● Slice gap closing radius:			0	mm
● Slicing Mode:		Regular		
● Slice resolution:		0	mm	
● G-code resolution:		0.0125	mm	
● XY Size Compensation:		0	mm	
● Elephant foot compensation:		0.3	mm	



I highly recommend to test the print first before committing to it fully. For that just move the vase down into the print bed inside the slicer until only the top edge is printed.

Since vase mode does not support top layers I chamfered/sloped the upper edge in a way that the spiraling perimeters eventually meet and overlap each other.



Once you have done a print test, checked that the gap in the wall and upper edge is sealed/bonded nicely you print out the whole thing.

Happy printing and please share your makes if you can. I love seeing my designs printed by the 3D printing community.







**silky\_crease\_vase\_dualwall\_vasemode.stl**

# License

This work is licensed under a  
**Creative Commons (4.0 International License)**



**Attribution—Noncommercial—Share Alike**

---

- ✗ | Sharing without ATTRIBUTION
- ✓ | Remix Culture allowed
- ✗ | Commercial Use
- ✗ | Free Cultural Works
- ✗ | Meets Open Definition