# Institute of 3D Printing

Vase Mode

I built this single-wall model to test different acceleration rates on a Craftbot Flow IDEX and used Simplify3D "vase mode" to slice it . . . first time to use vase mode. I expected it to print in a continuous corkscrew. Instead it printed the circumference in one direction, then reversed and printed the circumference in the opposite direction. I marked the reversal point in red. Since the previous layer didn't have time to cool before the extruder reversed, the first segment overheated and is very rough.
Is this the way it's supposed to work?

4sfed ⚪
(@4sfed)
Frequent Member
⭐⭐ | 2 Posts
❓1 ☑1 💬0
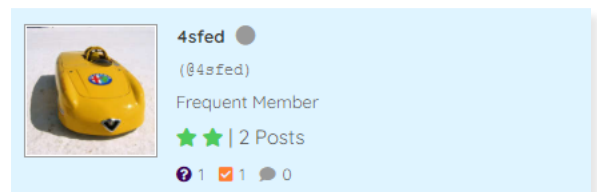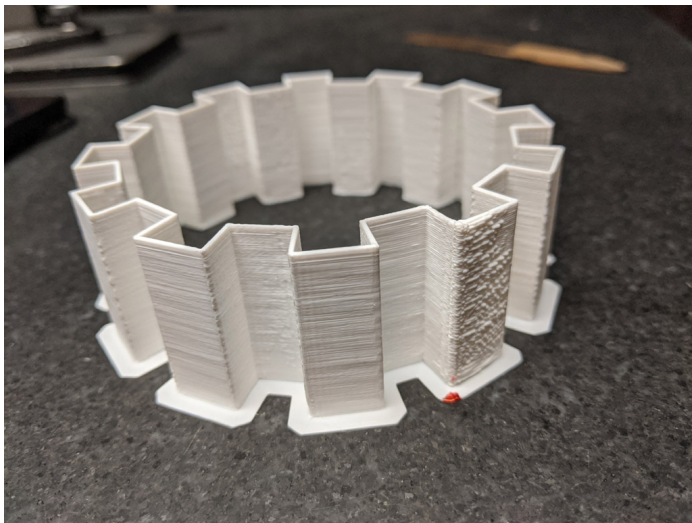
Hi 4sfed,

Welcome to the community! :)

I believe the problem is that you made it a single wall model, so the slicer probably printed the whole thing using single extrusions. Vase mode requires a solid model (think solid cylinder rather than bucket), which the slicer then prints as a single wall. To use the STL you've made already, go to the "advanced" tab, and under "slicing behaviour" tick "merge all outlines into a single solid model". That will make it treat your STL as if it wasn't hollow in the middle so vase mode should work correctly.

When making STL files of vases in future just make them solid and you wouldn't have to take that extra step.
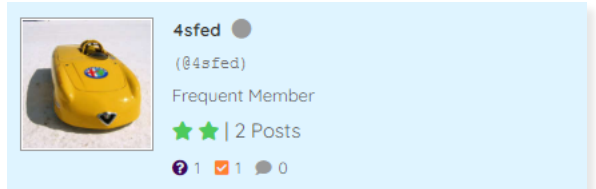
Let me know if that fixes it :)

Regards,
Luke

Luke 🟢
(@1mf5000)
Expert `Moderator`
288 Posts
❓0 ☑194 💬42
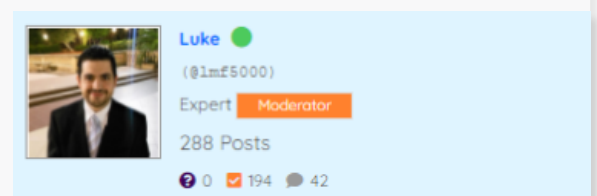
Thanks for the explanation. I rebuilt the model as a solid and sliced it again, using vase mode with no top or bottom layers ... it worked perfectly.

-jim

4sfed
(@4sfed)
Frequent Member
⭐ ⭐ | 2 Posts
❓ 1 ☑ 1 💬 0

Taken directly from our members only community.