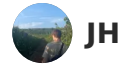




Bottle Cage



[VIEW IN BROWSER](#)

updated 22. 1. 2025 | published 22. 1. 2025

Summary

Clamps bottles with diameters from 60mm-90mm and the insert in the bottom secures against failures due to layer adhesion

[Sports & Outdoor](#) > [Outdoor Sports](#)

Tags: [bottle](#) [can](#) [holder](#) [mount](#) [sport](#) [benchmark](#)
[halterung](#) [bicycle](#) [racing](#) [bike](#) [cage](#) [bottleholder](#)
[fahrrad](#) [stresstest](#) [halter](#) [bottlecage](#) [flaschen](#) [dosen](#)
[getraenkehalter](#) [getraenke](#)

Update: Just uploaded "V3" Versions (slightly better looking, slightly better function)

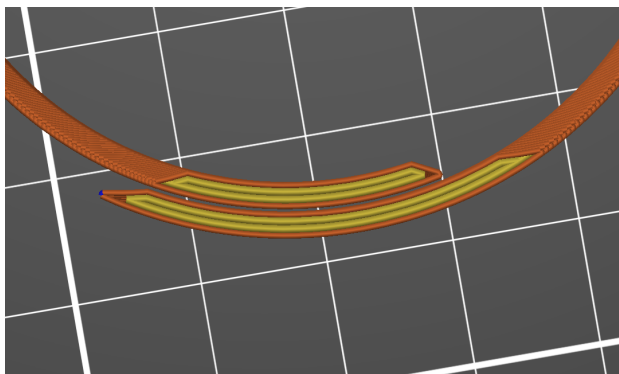
Backstory

I was unsatisfied with most of the printable bottle cages because they usually only fit one specific bottle/can size and do not "grip" smaller bottles. Also most cages are printed in orientations that either lose strength with the clamping section or solely rely on layer adhesion at the bottom section to keep the bottle from sliding out. And I don't trust layer adhesion that much. Because of this I designed a bottle cage myself that has two parts: The cage itself **and** a separate insert that is printed in

another orientation to give maximum security that your bottle will not blow out the bottom of your cage when you are in some bumpy terrain. (spoiler: If you trust your layer adhesion a lot, you will find a file for each cage that prints in just one piece). Also since we want to use our bottle cages outdoors we should print them with something weather resistant like ASA, PCTG, PC, PETG, etc. Fortunately these Materials give quite some flex which we can actually make use of when printing something like a bottle cage!

One 4 All Bottle Cage

This bottle cage is as easy to print as it is kind of an 3D-Printer stresstest at the same time. The overhangs are not super steep but quite challenging due to being rather thin. Also the gap between the overlapping section of the cage is just 0.2mm (as seen in picture below).



This on one hand is very beneficial since a little bit of material is going to stick/fuse together and therefore makes the printing of the whole upper section a little bit more stable - at the same time it is just wide enough of a gap so they are pretty easy to separate without any use of tools and therefore giving us the clamp-style flex to

secure our drinks in place!

I have been testing one out of PETG for a couple of weeks now in temperatures ranging from +15°C to -10°C with different bottles and cans (0,3L & 0,5L glass bottles, cans, 0,5L pet bottles and even up to 1L pet bottles) and it works great!

You can attach the cage to almost any bicycle since it has **variable mounting points ranging from 60-70 mm** distance of the screws. (The Performance V2 even fits 60-80mm). Anyways, I came up with:

Two different iterations: Everyday use (left) and performance (right) - (V2 in pictures).



Everydays Use: This cage is a little bit sturdier. It should fit almost **any type of frames and screws**. This one uses about 35g of filament (PETG) and it might also be suitable for Mountainbikes as well.

Performance: This is for the sporty ones of you! This cage is just 25g of filament and therefore lighter than most bottle cages you can buy with a small disadvantage in rigity (still

works great - tested for couple of weeks). **BUT** for this design you should **use screws with a flat head** (2-3mm) due to the thinner mounting section.

Printing Advice

Prints great with default settings in Prusaslicer! However, key to having a successful beautiful print is the position of the seams. Usually your slicer would not put seams at positions of overhangs - but check it before printing and if it somehow did put seams at the overhangs, you might want to block these (e.g. with prusaslicers seam-painting tool). I also would recommend to use solid infill at the mounting sections - therefore prusaslicer height modifier will come in handy!

Recommended: **0.2mm layerheight!**

Print **without supports!**

Installation

Once printed there are two things you need to do to properly install the bottle cage:

First we go over the insert that gives us maximum protection against failing layer adhesion at the bottom lip. This insert prevents the bottle from sliding out downwards. You will find this L-shaped object on the print bed. It has quite a tight fit to the cage and is printed sideways so we have continuous layers supporting the bottom of the bottle. Now let us look at the installation:



Once printed, do not slide the insert in completely yet! There is a small edge at one end that we first need to put into the matching negative of the cage. See the picture on the left! Only after we put this one in place we can complete sliding the security support in the

cage. You will not be able to properly bring it into place if you do not do it like shown on the left picture. Once in place it should look like on the right.



Now there is only one last thing to do: The Cage has a “main-flap” that goes around 270° to support the bottle from all sides. The other flap goes around just 180°. This second one is our so called “support-flap”! After printing the support-flap initially is on the inside and the main-flap on the outside. We want to change this! So you have to bend both flaps to the outsides just so much so you can change which one is on the inside and which one is on the

outside. See the pictures on the left for better understanding: Left is just off the printbed, right is how it should look like after positioning the flaps.

If you are wondering “why not directly design it like this then?!” - If you are driving around, not every street or road is perfect. You will always have little bumps and vibrations. The very close distance between both flaps would allow them to rattle against each other. But now that we have changed which one is on the inside and which one is on the outside, they have a thoroughly slight tension against each other so there is no rattling noise while riding on bumpy roads.

I hope you like the design! Happy printing.

Model files



Updated files V3

4 files



everyday-v3.stl

☐ About 35g of filament, 4h print time



everyday-v3-onebody.stl

☐ one-part-print (no additional security for bottom!)



performance-v3.stl

☐ sporty - about 25g of filament if sliced with 15% infill



performance-v3-onebody.stl

☐ one-part-print (no additional security for bottom!)



Older Versions V2

4 files



old-everyday-v2.stl

☐ great for everyday use - 35g



old-everyday-v2-onebody.stl

☐ one-part-print (no additional security for bottom!)



old-performance-v2.stl

☐ sporty - about 25g of filament if sliced with 15% infill



old-performance-v2-onebody.stl

☐ one-part-print (no additional security for bottom!)

License ©

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



Attribution-NonCommercial

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