



Orbital. Structural printing. Print-In-Place lamp



Kagarov

[VIEW IN BROWSER](#)

updated 5. 2. 2023 | published 5. 2. 2023

Summary

Usually when printing objects they talk about walls and filling, but I made a structure without walls, without...

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

Tags: [ikea](#) [lamp](#) [printinplace](#) [lampshade](#) [ikealamp](#)
[illumination](#) [bridges](#) [ikeahemma](#)

Usually when printing objects they talk about walls and filling, but I made a structure without walls, without filling, and also without a bottom and a cover. There are only bridges that connect and lean on each other.

The peculiarity of this project is that it is possible only in 3D printing, it is difficult or impossible to repeat it in traditional technologies, such as injection molding or milling.

The lamp looks very complex, but it's easy to print with the usual settings for printing bridges. This printing can be stopped at the end of any layer to obtain a lampshade of the desired height (maximum - 249 mm).

Perhaps this project was influenced by Tatlin's impressive tower, Shukhov's hyperboloid, and fantastic orbital structures.

I want to use the developed method (for now I called it Structural Printing) in other projects as well.

Shade fits IKEA HEMMA lamp and most other pendants with a socket of about 40 mm.

Good ventilation (or should you turn off the fan) and not too high speed are needed for printing. Even if several threads break or do not weld, it's not scary, the design "self-heals" during printing and in the end it turns out to be quite strong (and also a little springy). No post processing required.

Do not use bulbs that get very hot. I recommend LED, for example.

Check out my project that led me to this <https://www.prusaprinters.org/prints/40606-liquid-cuprum-print-in-place-lampshade>

And also another funny fluffy lamp - Polar Sheep <https://www.prusaprinters.org/cs/prints/40124-polar-sheep-lamp>

Print instructions

Full-height printing time is approximately 8 hours.

I was slicing in PrusaSlicer, my old Cura couldn't do the slicing.

Since the size of the parts must match the height and width of the layer, changing the scale when printing can lead to an unpredictable result (but you can increase the Z height by a multiple of the layer height, in increments of 0.3 mm per layer, for example, select 0.6). I advise you to print 0.3 layer height and 0.4 layer width, with 100% airflow. The height of the first layer must be adjusted like all the others, otherwise the distribution of layers may get lost.

My print speed was 60 mm/s.

A colleague of mine was printing PETG: layer height 0.3, width 0.4, speed 60, no fan, flow 120%, temperature 220.

No support is needed - unlike other projects, the threads should hang freely here. No post processing is required.

I printed with black and transparent PLA, but this is also possible with other materials with adjustments.

Model files



orbitalstructure.3mf



orbitalstructure.stl

License ©

This work is licensed under a
[Creative Commons \(4.0 International License\)](#)



Attribution—Noncommercial—No Derivatives

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