

Nautilus Shell Google Home Mini Holder



KniRider

[VIEW IN BROWSER](#)

updated 12. 4. 2022 | published 11. 1. 2021

Summary

I took the Nautilus Shell Speaker design and made it hold a Google Home Mini.

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

Tags: [mini](#) [home](#) [google](#) [shell](#) [googlehomemini](#) [seashell](#)
[nautilus](#)

I took the Nautilus Shell Speaker design and made it hold a Google Home Mini. My wife actually wanted me to do this so I hopped right to it since she thinks everything else I print is stupid ;)

Took me a long time to do this. Tried with the nautilus shell design that was just a wall hanging and that was a disaster then I found this one i used to remix from and it was a ton better but still went through almost 2 full rolls of filament to get it prototyped. The so-called final version in the pictures has a layer shift due to support failing on me but I said its good enough (so did she). I may print it again sometime just to make it look a little prettier on the back side but it works so no hurry.

The 3mf file I uploaded is my Cura file that has support blockers included (sometimes they show up at the right places, sometimes they dont - no idea why so you may have to move them around). You want to make sure

no support is on the inside of the shell or the entire back part as it doesn't need it (as long as your printer is doing a good job). You want support in the front opening and the bottom to give some support for printing but NOWHERE ELSE. If you get support on the inside you will never get it out. I uploaded a picture of the support blocker placement as Cura on my end wants to keep moving the bottom one around - BAD CURA!.

I included the SPIKES used for displaying that EricSinclair used for the speaker. I had to drill the holes a shade bigger in the shell to get them to fit properly. Maybe it is my printer being a bit off or whatnot but just so you know. I did not want to force them because I didnt want to break the shell. I think I used a 15/64th bit (About 6mm from what Google tells me).

This will take a good bit of filament (almost half a roll) if you use the settings i did. If you opt for more infill or more supports then it will of course take more. Time is a good 30+ hours, again depending on resolution, speed, infill and supports you use. I printed at 80mm/s.

Would love to see makes!

Print instructions Category: Decor Summary

I took the Nautilus Shell Speaker design and made it hold a Google Home Mini. My wife actually wanted me to do this so I hopped right to it since she thinks everything else I print is stupid ;)

Took me a long time to do this. Tried with the nautilus shell design that was just a wall hanging and that was a disaster then I found this one i used to remix from and it was a ton better but still went through almost 2 full rolls of filament to get it prototyped. The so-called final version in the pictures has a layer shift due to support failing on me but I said its good enough (so did she). I may print it again sometime just to make it look a little prettier on the back side but it works so no hurry.

The 3mf file I uploaded is my Cura file that has support blockers included (sometimes they show up at the right places, sometimes they dont - no idea why so you may have to move them around). You want to make sure no support is on the inside of the shell or the entire back part as it doesn't need it (as long as your printer is doing a good job). You want support in the front opening and the bottom to give some support for printing but NOWHERE ELSE. If you get support on the inside you will never get it out. I uploaded a picture of the support blocker placement as Cura on my end wants to keep moving the bottom one around - BAD CURA!.

I included the SPIKES used for displaying that EricSinclair used for the speaker. I had to drill the holes a shade bigger in the shell to get them to fit properly. Maybe it is my printer being a bit off or whatnot but just so you

know. I did not want to force them because I didnt want to break the shell. I think I used a 15/64th bit (About 6mm from what Google tells me).

This will take a good bit of filament (almost half a roll) if you use the settings i did. If you opt for more infill or more supports then it will of course take more. Time is a good 30+ hours, again depending on resolution, speed, infill and supports you use. I printed at 80mm/s.

Would love to see makes!

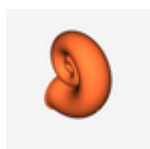
Decided to upload my Cura profile also that I used....REMEMBER - I have an Ender 5 with a BLTOUCH so you may have to edit the starting gcode to eliminate the G29 if you do not have auto bed leveling (If it carries it with the profile, I have no idea if it does) and you will have to adjust for temperature/bed heating, etc.

If you want my gcode - Ender 5 with bltouch - just ask and I will upload. It is 125mb so I wont upload unless someone wants it.

Model files



cura-nautilus_google_mini_holder.3mf



nautilus_google_mini_holder.stl



speaker_spikes_30mm.stl

[Find source .stl files on Thingiverse.com](https://www.thingiverse.com)

License ©

This work is licensed under a
[Creative Commons \(4.0 International License\)](https://creativecommons.org/licenses/by-nc-sa/4.0/)



Attribution—Noncommercial—Share Alike

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