Frame for Waveshare 7.3inch ACeP 7-Color E-Paper E-Ink Display Module, 800×480 Pixels

n ned14

VIEW IN BROWSER

updated 1. 7. 2024 | published 1. 7. 2024

Summary

Attractive, durable frame for a very delicate panel

Hobby & Makers > Other Ideas

Tags: thingiverse waveshare waveshare7inch eink epaper

I recently saw mention of an affordable seven colour e-paper display on HackerNews and I wanted to try one out as I have a need for a dashboard showing historical sensor readings, and each line would have its own colour. I decided on the 7.3 inch model from Waveshare as being about as much as I'd be happy to punt. When it arrived, it worked well but was very thin and not at all durable if left loose, so I needed some sort of durable holder.

I was surprised to discover none on any of the 3d printing sites had a frame for this specific size of display with its ribbon at the bottom, so I decided to make my first ever 100% me alone 3d design which is this!

Undoubtedly somebody more experienced could do better than this - the rear supports do not have a smooth curve, and some of the seams are slightly prominent in places. I think these mostly from my settings in PrusaSlicer than the STL however. I attach the MeshMixer files so you can fiddle with the design yourself.

The photos are of a black shiny PETG print which I thought would look nice, the "piano shine" you get off PETG I've never seen off any PLA I've tried the shiniest of which has a duller shine. I should have used a hotter temperature, I used 220 C and undoubtedly 230 C would have been better. Still, it's "good enough" for my purposes and it'll work just fine in PLA too, an earlier prototype was in PLA and it came out great.

As you'll see in the photos, I had an ulterior motive of fitting in a Waveshare PoE hat for the Raspberry Pi Zero in its case. The space is exactly right to hold one of those in place, and it provides extra ballast to keep the frame in place. Even if you don't fit your device into the frame, I made the base 5mm thick to provide a bit of weight to make the frame stable.

In between the two middle supports there are pins to mount the cable connector. They are also spaced exactly apart to hold the cable connector horizontally. On my PLA print the pins came in a touch high, the PETG print the pins came in a touch low. Can't win I guess.

Something I did consider but haven't tried is turning on "fuzzy skin" in the slicer. I think that could produce a very nice looking frame indeed.

Print Settings

Printer Brand: Anycubic

Printer: Kobra Go

Rafts: No

Supports: Yes

Resolution: 0.2mm

Infill: 20% Gyroid

Filament: YOYI PETG Black

Notes:

For the PETG I used, min nozzle 230 C, bed 100 C. The PLA prototype was 190 C nozzle and bed 60 C.

You will likely need supports for the top part of the outer frame, otherwise it sags (you can see it sagged a bit in my photo). It's not end of the world if it does sag slightly, a sharp knife can fix it.

Post-Printing

Cleanup

A rub down with a rough cloth gets rid of the PETG "hair". A few surface imperfections can be removed with a sharp knife. My earlier prototype was printed in PLA and apart from where the seams knit together producing a mild protrusion, it needed almost no cleanup after apart from removing the very few supports needed.

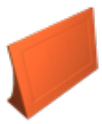
How I Designed This

Meshmixer

My first ever 100% me alone 3D print design! Done in Meshmixer and sliced in PrusaSlicer.

Category: 3D Printing

Model files



frame2.stl

Other files

framemix.zip

Find source .stl files on Thingiverse.com

License

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

Attribution-ShareAlike

✖ | Sharing without ATTRIBUTION
✔ | Remix Culture allowed
✔ | Commercial Use
✔ | Free Cultural Works
✔ | Meets Open Definition