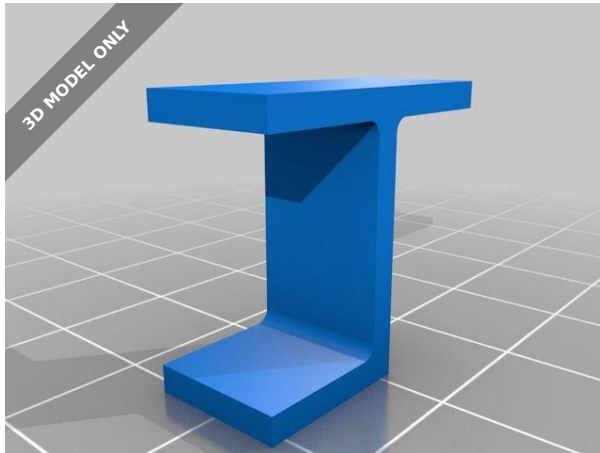




Support test

F FaultyData

[VIEW IN BROWSER](#)

updated 7. 5. 2022 | published 7. 5. 2022

Summary

I made this after remixing the all in one 3D printer test (/thing:3718940) because I realized I didn't want to do a...

[3D Printers](#) > [Test Models](#)

Tags: [support](#) [calibration](#) [supports](#) [testprint](#) [calibrationtest](#)
[printer calibration](#)

I made this after remixing the all in one 3D printer test (/thing:3718940) because I realized I didn't want to do a full 8 hour print just to see if I was making improvements on my support settings.

Print Settings

Printer Brand:

Creality

Printer:

CR-10

Rafts:

No

Supports:

Yes

Resolution:

0.2

Infill:

100%

Filament: Sunlu PLA Plus Orange

Notes:

This is a small model at 10 x 22 x 20 and doesn't take too long to print. It will test supports attached to the build plate as well as supports that are sitting on top of your model.

In Cura, I used a 15% support density in zigzag pattern. I also enabled support interface and used 1mm floor/roof thickness, grid pattern at 30% density. For this print at 0.2mm resolution, the bottom Z distance is 0.4 mm while the top Z distance is 0.2. The floor breaks away rather easily from the model and leaves hardly any marks; The roof is a bit harder to remove, but leaves pretty good surface quality on the underside of the supported overhang. I'll still probably drop the interface density down to 20 or 15% and see if it makes any difference to the ease of removal, or reduces overhang quality. Category: 3D Printing Tests

Model files



support_test.stl

[Find source .stl files on Thingiverse.com](#)

License ©

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



Attribution—Noncommercial—Share Alike

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