



iPhone 15 Pro Max TPU Case with MagSafe



Grant G

[VIEW IN BROWSER](#)

updated 25. 9. 2023 | published 25. 9. 2023

Summary

This is a relatively easy to print TPU case for the iPhone 15 Pro Max with cutouts for MagSafe magnets.

[Gadgets](#) > [Portable Devices](#)

Tags: [camera](#) [case](#) [pixel](#) [samsung](#) [tpu](#) [phone](#) [apple](#)
[iphone](#) [watch](#) [15](#) [pro](#) [plus](#) [screen](#) [google](#) [flex](#)
[ultra](#) [13](#) [bumper](#) [magsafe](#) [max](#) [14](#) [promax](#)

This is a relatively easy to print phone case for the iPhone 15 Pro Max so you can protect your shiny, new, titanium phone.

The case consists of two pieces: the case and the camera guard. Print the case in TPU or other flexible filament, and the camera piece in any non-flexible filament. I just used PLA. The case features ribbed buttons for tactility and a one piece design that doesn't require any separate button pieces and makes for easy printing.

The inside of the case features an indent for MagSafe magnets that can be bought on amazon, such as [these](#). Haven't gotten mine in yet, will update the case if they don't work.

Make sure you change “PERIMETER GENERATOR” to “CLASSIC” before slicing!

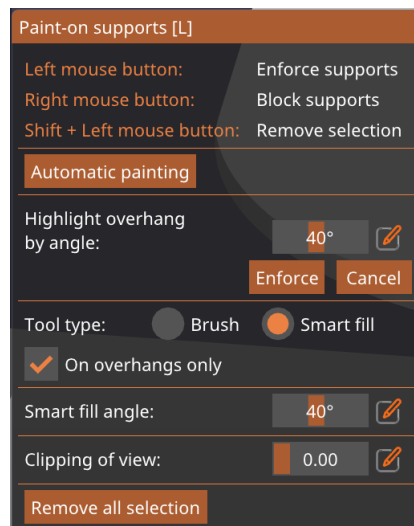
Otherwise the button areas will not print correctly. Use the search function in the top bar if you are not sure where this setting is.

Slicer setup:

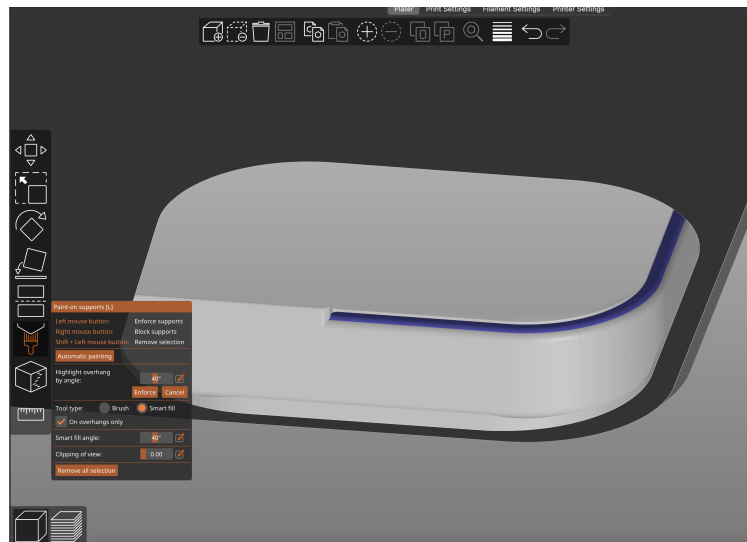
Case

I printed with TPU at a 0.2mm layer height. Make sure the case is on its back. The most important thing will be the support setup, as the front of the case needs support.

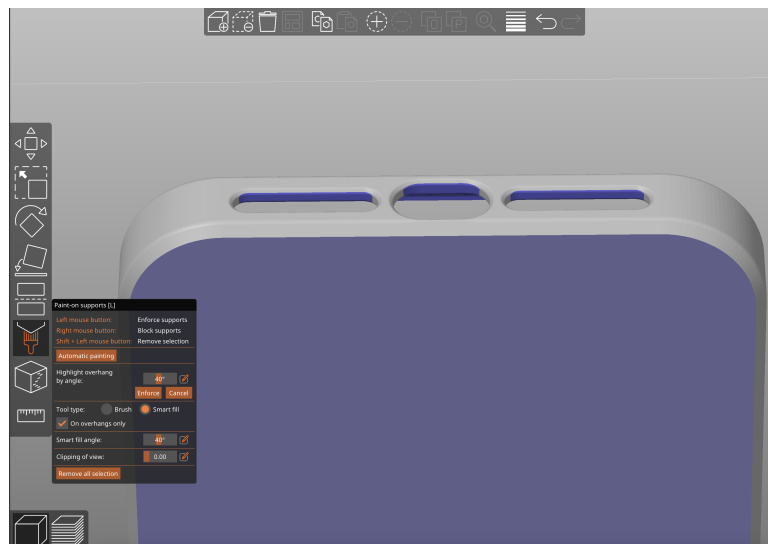
- Select the case and press “Paint on Supports”
- Set the settings for the support painter to match this:



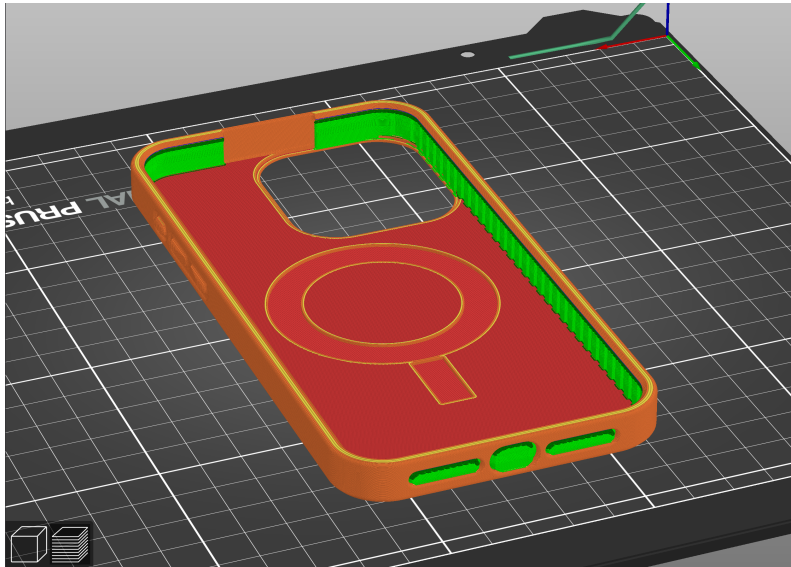
- Change the view so that you are looking up through the bed and through the camera hole like this:



- Select the overhang that is highlighted
- Change the view so that the charge port area is in view, like this:

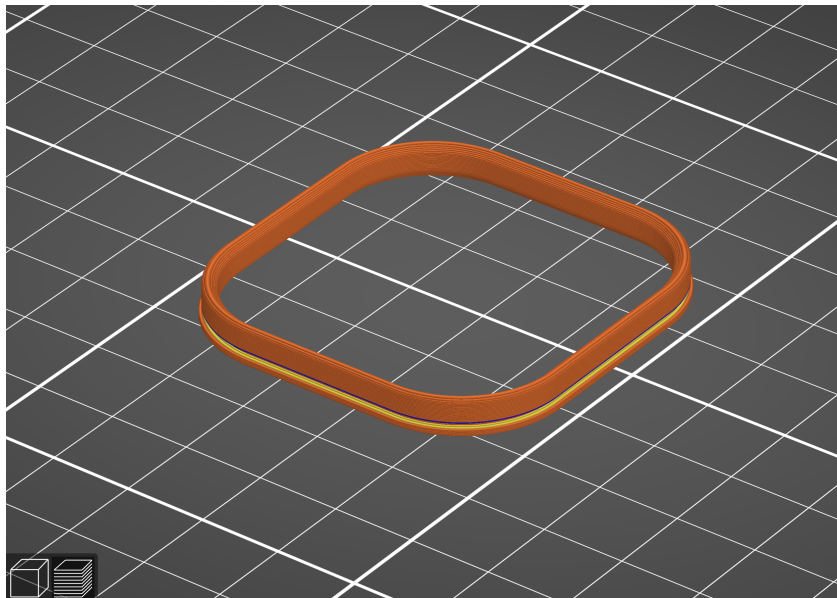


- Select the overhangs for the charge port and speaker grills
- Deselect the “paint on supports” tool
- Change the support setting to “for support enforcers only”
- Should look like this:



Camera Guard

- I printed mine at 0.15mm layer height in PLA. There is an overhang that looks like it should be supported, however the gap is too small and prusaslicer does not want to build actual supports in that gap.
- I was able to print mine just fine without supports, but I am not sure this will work for everyone.
- Leave the orientation the way it is when imported.



If you have any questions, let me know.

If you like the model, please leave a make. I would love to see how others like it!

Happy Printing!

Model files



camera-piece.stl

☐ print in non-flexible filament



case.stl

☐ Print in flexible filament

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