

Generic Box Template for personal projects in OpenSCAD



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Summary

When doing electronic projects there always is the search for a housing.

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When doing electronic projects there always is the search for a housing. There are plenty on the market but rarely fit the exact size and usually can not be considered cheap for some molded plastics.

In addition much mechanical work needs to be done to implement all elements needed.

That was the reason why I decided to create an OpenSCAD template to automatically create a basic empty box by defining a hand full of parameters.

The housing consists of the main box and a fitting cover. I decided to handle both in separate OpenSCAD projects since I wanted to center the design since empty, it is symmetrical in X- and Y-axis.

Print instructions

Depending on the size of the box it is helpful to print both parts with a brim to avoid lifting corners of the large area.

Most of my boxes so far I printed with PLA in 0.2mm layer thickness and a 0.4mm nozzle. For the hole diameters of the screw and screw head I added 0.2mm to make it fit through easily.

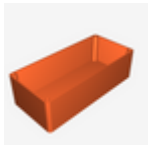
Model files



templatebox.scad



templatecover.stl



templatebox.stl



templatecover.scad

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