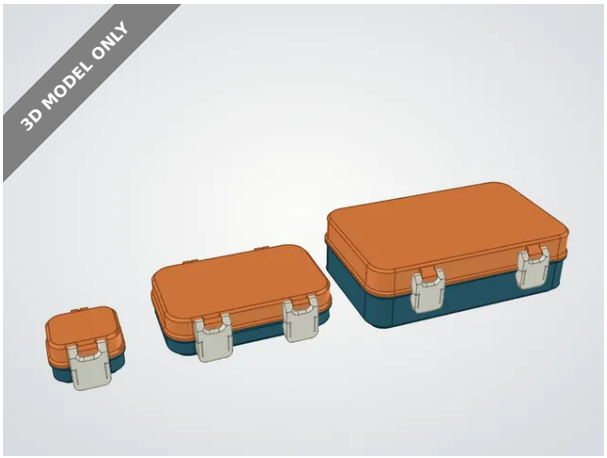

Parametric Storage Box with Snap Latches - FreeCAD

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VIEW IN BROWSER

updated 17. 10. 2026 | published 17. 10. 2026

Summary

Fully parametric hinged box with snap latches, built in FreeCAD. Change 5 values in a spreadsheet to get any size.

Hobby & Makers > Organizers

Tags: parametric box organizer case container storage hinge latch freecad snaplock

Parametric Storage Box – FreeCAD editionInspiration & credit

This design is inspired by the excellent SnapLock V3 box by Hugo on MakerWorld (link above). The snap-latch and hinge concept comes from his work – all credit for the idea goes to him. If you like this box, go check out his designs and support him.

Link here: https://makerworld.com/en/models/2776423-snaplock-v3-single-latch

What is this?

A fully parametric box modeled from scratch in FreeCAD (1.1). Open the .FCStd file, change a few values in a spreadsheet, press recompute,

and you get a new box of any size with the hinges, locks and latches automatically repositioned. The goal was to have a version that anyone can edit with free, open-source software.

Parameters (spreadsheet "Parameters")

Inner_width – inside width (min 47 mm)

Inner_length – inside depth (min 15 mm)

Bottom_height – height of the bottom part (min 20 mm, the latch is 40 mm tall)

Top_height – height of the lid (min 18 mm)

Corner_fillet – corner radius (automatically limited so the hinges always stay on a straight wall)

Walls, floor and ceiling are 2 mm, snap clearance 0.2 mm (also editable). Outer size = inner size + 4 mm.

Smart layout: below 60 mm inner width the model automatically switches to 1 hinge + 1 latch centered; above, it uses 2 hinges + 2 latches.

Assembly & animation

The FreeCAD file also contains an Assembly (FreeCAD Assembly workbench) with the hinge and both latch pivots as joints, plus an opening animation – handy to check clearances before printing.

Included files

parametric_storage_box_FreeCAD.FCStd – the parametric source (default size 150 × 90 mm)

STEP – closed assembly of each size

STL – ready to print, 3 sizes (inner dimensions):

Small 55 × 45 mm, bottom 20 / lid 18 mm – 1 latch

Medium 150 × 90 mm, bottom 20 / lid 18 mm – 2 latches

Large 220 × 130 mm, bottom 40 / lid 25 mm – 2 latches

Each size = 1 bottom + 1 lid + the latch (print ×1 or ×2 as indicated in the file name). The bottom is exported floor-down and the lid is already flipped (top face on the bed). The latch is exported in its assembled orientation – lay it flat in your slicer.

How to make your own size

1. Open parametric_storage_box_FreeCAD.FCStd in FreeCAD 1.1 or newer.

2. Double-click the Parameters spreadsheet and edit column B (MIN / MAX / DEFAULT columns are there as a guide).

3. Press Ctrl+R (recompute).

4. Export BOTTOM, TOP and Latch_R as STL (File → Export).

Printing

PETG works great for the latches, PLA is fine too. Print the latches in a contrasting color for a nice look!

Model files

STEP 3 files

box_large_220x130_assembly.step

box_medium_150x90_assembly.step

box_small_55x45_assembly.step

STL 7 files

box_large_220x130_bottom.stl

box_large_220x130_lid.stl

box_medium_150x90_bottom.stl

box_medium_150x90_lid.stl

box_small_55x45_bottom.stl

box_small_55x45_lid.stl

box__latch.stl

parametric_storage_box_freecad.fcstd

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