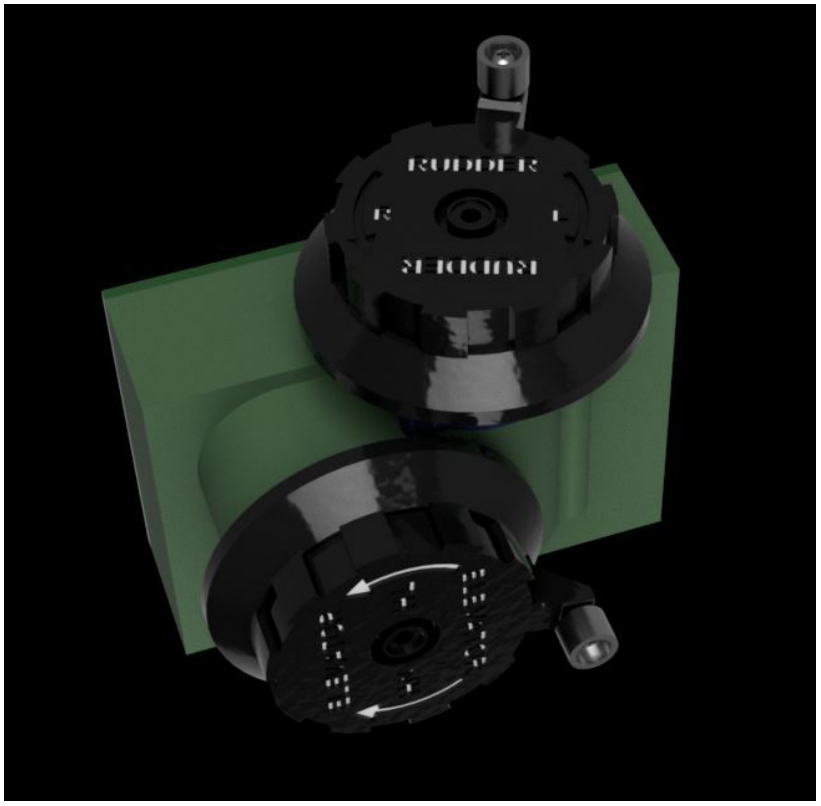
P-40B Tomahawk Elevator and Rudder Trim Wheel Kit

Assembly Guide



1. Introduction
2. Calibration
3. Assembly steps
4. Screws & Wires
5. Index of all printed parts with printing advice
6. Printing advice and recommended slicer settings

For more information about this project and the aims and ambitions for the AuthentiKit system see **authentikit.org**

Introduction

This kit is part of the P-40B Tomahawk range of flight controls by AuthentiKit developed in collaboration with Big Radials. It has been primarily tested with the Big Radials MSFS version of the P-40B Tomahawk. Also part of this range is the P-40B throttle quadrant and P-40B flight stick which at the current time are in development.

In order to use these controls with your flight simulator you will need the following additional kits, also available as free downloads from authentikit.org

- Universal Hub Starter Kit
- Rig Extension Kit A

If you already have the Spitfre MkIX Starter Kit A and Add On Kit B (inc. Rudder Trim) then you already have the above items.

Sourcing

For a full list of parts see my document **P40-B Tomahawk Elevator and Rudder Trim Wheel Kit - Bill of Materials.pdf**

The other sourcing option is simkitsupplies.com - see their website for more details. The price should be less than you'd pay to get the components yourself.

Calibration

Caution - before you start printing check the printer settings at the end of this document and print the very quick calibration test, then tweak horizontal expansion settings. **Do not print large parts until the bearing is a snug fit but not tight.**

P-40B YouTube Assembly Guide

https://youtu.be/FoYRlHgnMdY

Assembly Steps

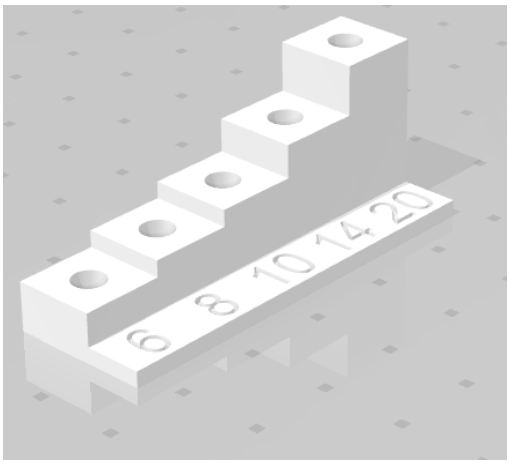
1. Fit the trim wheel knobs to the trim wheels (12mm round head screws)
2. Attach the rudder trim cover to the elevator trim cover (10mm screws)
3. Long Roten6003F peg through bearing
4. Bearing into Roten6003F cap
5. Fit encoder through Roten6003F base (single central hole)
6. Fit cork disc over base
7. Fit base to cap (25mm screws)
8. Fit wires to encoder (Yellow-Black-Green) with ferrules
9. Thread wires through large hole
10. Put encoder to box so 25mm screws are visible and fit encoder to box (8mm max)
11. Test elevator trim friction and adjust - fit the wheel and test
12. Attach cover to box with 3 screws (10mm)
13. Fit encoder to the other Roten6003F base
14. Fit wires to encoder (Yellow-Black-Green) with ferrules
15. Thread wires through slot in box
16. Fit base to box plinth (14mm)
17. Fit cork disc to base
18. Fit short Roten6003F peg through bearing
19. Fit bearing to Roten6003F cap
20. Fit Roten6003F cap to base (25mm)
21. Adjust rudder trim friction
22. Fit elevator trim wires to RJ45 (black - yellow - green) and fit to lower slot
23. Fit rudder trim wires to RJ45 (black - yellow - green) and fit to upper slot
24. Fit back to box (8mm)
25. Fit VESA dovetail to back (8mm)
26. Fit rudder trim wheel to peg (20mm round head)
27. Fit sleeve to elevator trim peg
28. Fit elevator trim wheel to peg (20mm round head)

Screws

This project uses a lot of screws. See the PDF for full details.

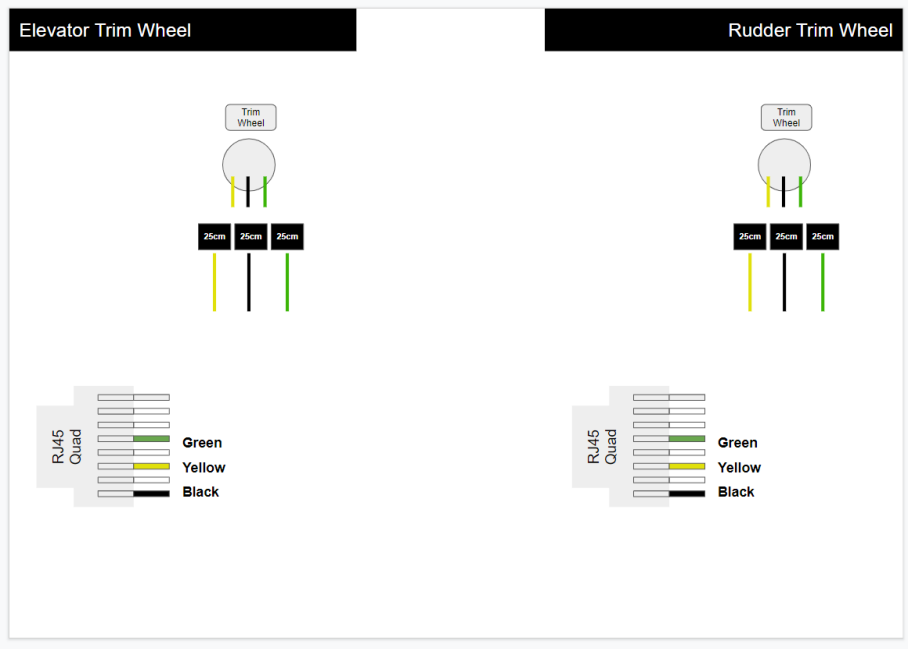
P40-B Tomahawk Elevator and Rudder Trim Wheel Kit - Screws Inventory.pdf

You may find it a little fiddly to tell certain screw lengths apart, so to avoid using a 10mm in the back of the MagHall instead of the 8mm it should be, I have included the following test tool in the STL download for Starter Kit A. It is designed just for M4 screws of length 6,8,10,14 and 20mm.



Wires

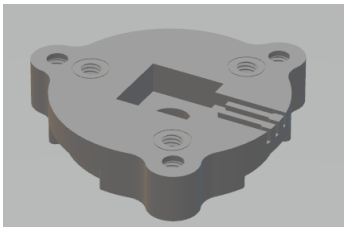
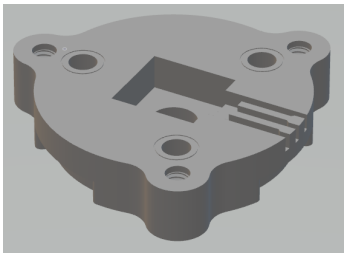
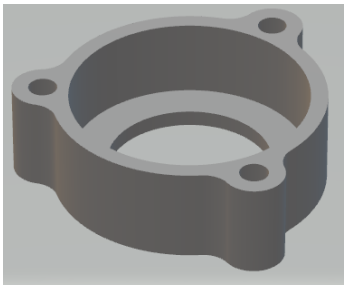
There is a full PDF size version of this diagram in the ZIP for this project. **P40-B Tomahawk Elevator and Rudder Trim Wheel Kit - Wiring.pdf**

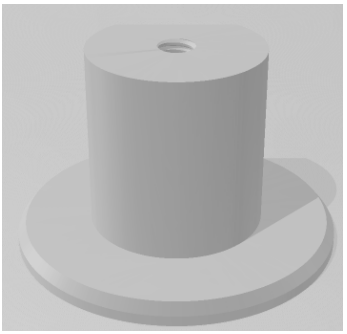
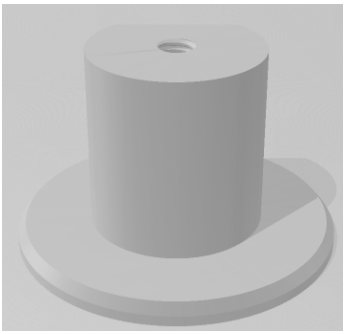
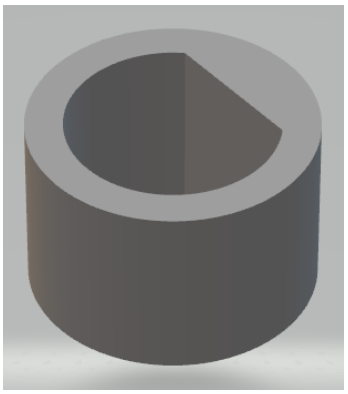
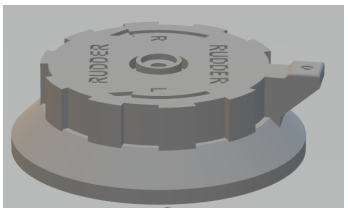
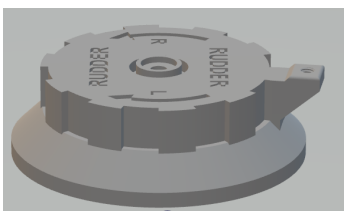


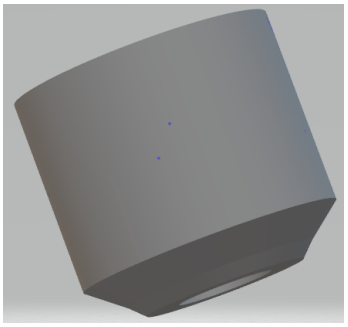
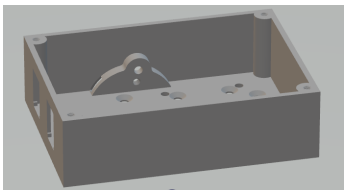
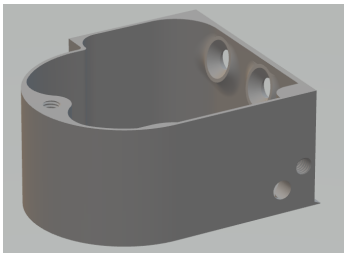
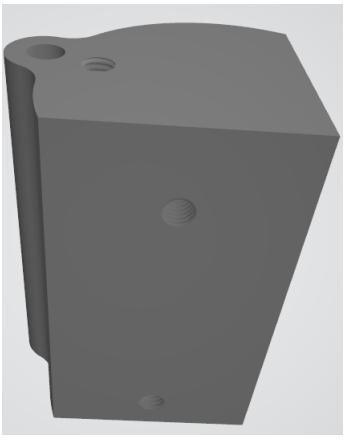
Index of all printed parts with printing advice

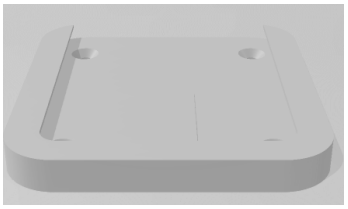
I'd suggest printing most parts at 25% infill and it should be fairly obvious which face should be down on the print bed. Exceptions to this are documented here.

No brim and no support on anything by default.
Exceptions explained below.

STL	Comment	Infill	Print Colour/orientation
Roten Base			 Olive green
Roten Alt Base	With extra fixing holes		 Olive green
Roten Cap	X2 One for elevator trim and one for rudder trim		 Olive Green

Roten Peg Long			 Any colour
Roten Peg			
Sleeve			 Olive Green
Rudder trim wheel			 Black
Elevator trim wheel			 Black

Knob	X2	This needs a brim	
Back plate			 Olive Green
Main Box			 Olive Green
Side Box Elevator			 Olive Green
Side Box Rudder			 Olive Green

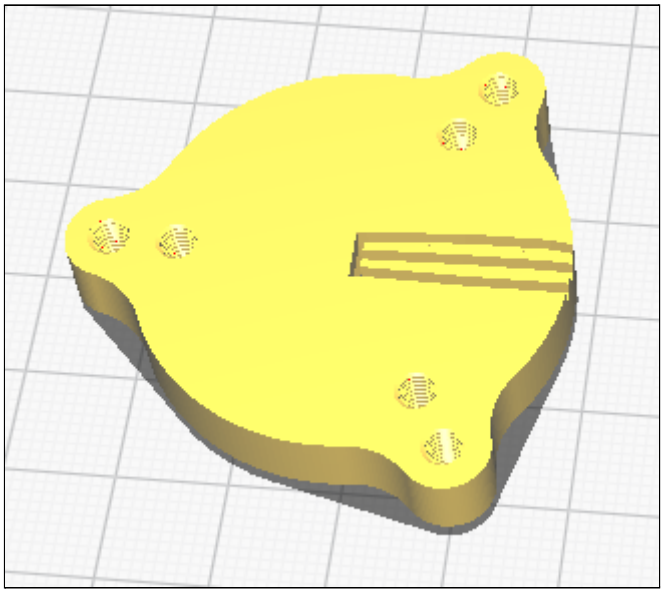
Dovetail	X2 One for elevator trim and one for rudder trim		 Black

Printing Advice & Slicer Settings

I did all the development for this project on the most basic Ender 3. I have since bought an Ender 6SE hoping that it would avoid some of the headaches that come with 3d printing. I like the build quality of the 6SE but sadly at present it isn't printing as well as my Ender 3. If you don't have a printer and this is your impetus to get one my recommendation would be to get an Ender 3 Pro for a budget option or a Prusa if you want to avoid as much hassle as possible. Newer firmware for the 6SE may change my advice and I hope it does as otherwise I really like the 6SE. The next printer I plan to get is a Prusa Mini as I'm hearing a lot of good things about it. There is very good advice on our Discord forum about printers.

You will need something that ideally has a print volume of at least 200mm x 200mm x 200mm although everything so far does fit onto a "mini" with 180mm x 180mm x 180mm.

AuthentiKit parts are designed to require no post printing work. No sanding, reaming out of holes or snipping off brims. This requires very specific slicer settings to ensure that holes are the right size and first layers don't expand too much. Parts are also designed to require **no support**. In **all but one** exception case they use **no brim** and no skirt as many parts have some fine channels and openings in the base and you'll never clean up the brim without damaging these. You'll also find screw holes get bunged up. Below is an example as this item needs to be printed the other way up with these channels on the bed.



The requirement for no supports and also the need for strength in the appropriate dimension means that every part has to be printed at a particular orientation. Orientation is shown in the previous section.

The recommended slicer is **Cura 4.6** or above as this contains some important features such as **horizontal hole expansion**.

Filament Choices

I experimented with a lot of filament. There are some nice matte finish options around but I wasn't too convinced about their strength. In the end I went with eSun PLA+ in the following colours.

- Black for most parts
- Olive green for the panels
- Silver for the fire button, P clip, lugs and posts

eSun is easy to find on Amazon though I had to go to 3dfilaprint.com for the olive green.

You may well want to print in PETG which is known to be stronger. I found PLA+, printed at 215 degrees, to be perfectly strong although some of the beta testers used PETG. If you do go with PETG I'd recommend fine tuning your printer settings first as is tends to come out quite a bit rougher and needs more post print cleaning.

My personal view on the choice of filament is that PLA+ is fine for strength. The issue I have with it is that it will go soft at around 50ºC which happens surprisingly easily if left near a window on a sunny day for a couple hours.

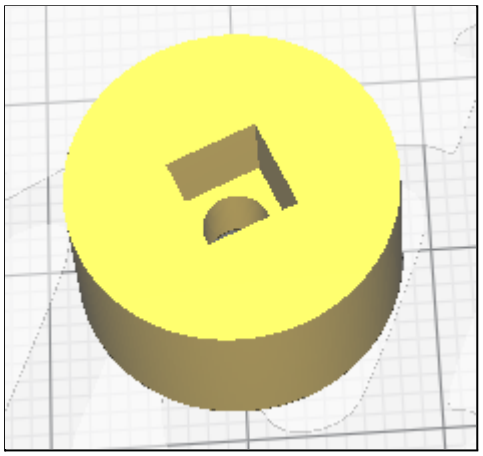
Bed Levelling

You will need to level the bed well to ensure good adhesion, but don't overdo it as you'll end up with too much first layer squishing (horizontal expansion) which stops some parts fitting together.

Incidentally, my personal approach to first layer adhesion is to use a glass bed with microporous coating. I didn't find the Creality beds to be that flat so I went with one from Wisamic. I wash it with detergent poured on neat and rubbed in well before washing off. Then I never touch it with my fingers. Then simply level it properly using the gcode that moves the head for you as otherwise I found I was messing things up moving the print head to different places with my hands. Right now my Ender 3 with this set up is still my workhorse and the new Ender 6SE needs plenty of TLC before I can trust it with complex prints.

Calibration

Before you embark on an 18 hour major part print, please print the calibration STL below and test it! You will probably need to adjust the horizontal expansion settings.

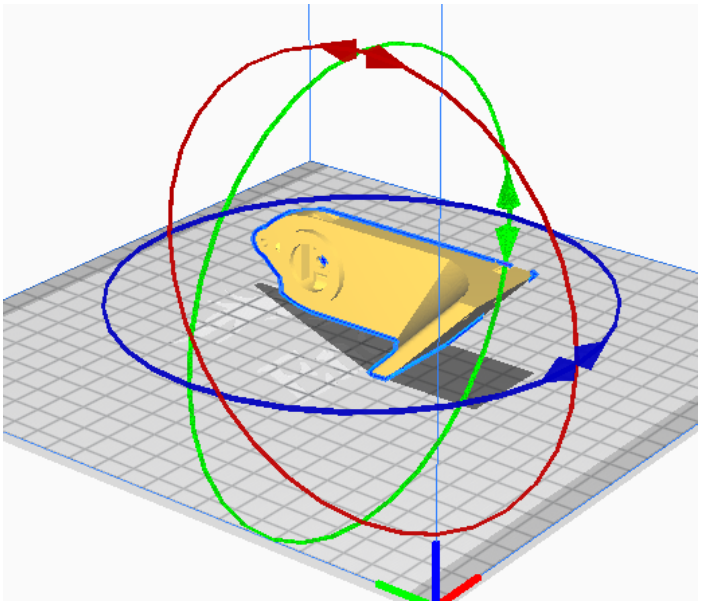


The inner cavity is designed to snugly fit the 5mm^3 cube magnet. The magnet needs to be snugly held but removable by pulling it out with the strength of one of the other cube magnets, but it should not be loose. The outer diameter should fit inside the 60032RS bearing. Again it should be neither loose nor so tight you need to hammer it in.

Start with the horizontal expansion settings below and adjust them to fit your filament and printer.

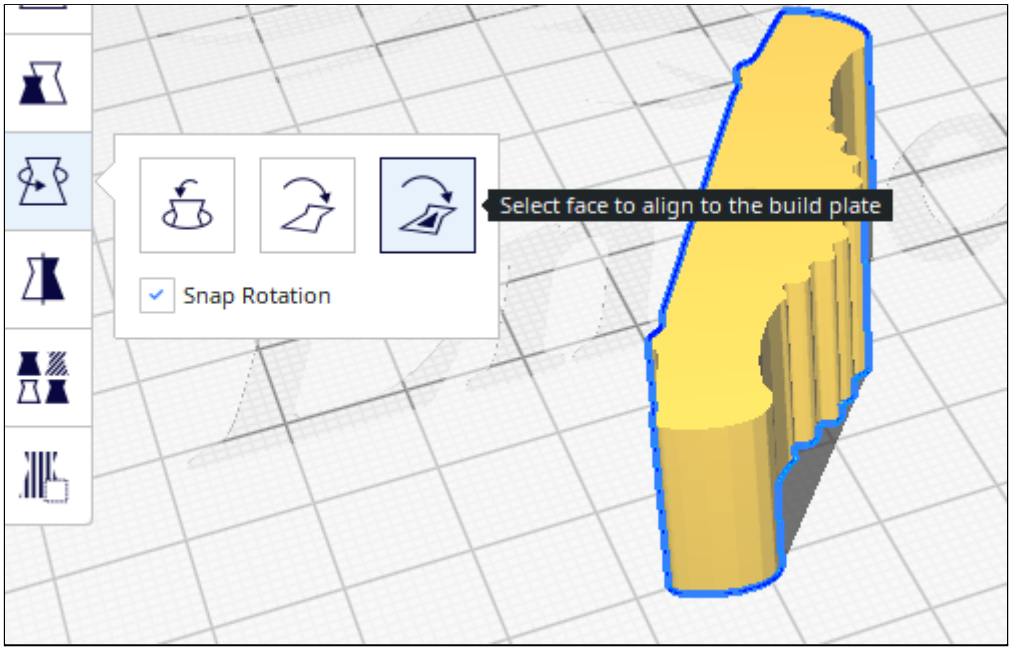
Print Orientation

Many parts require 90 degree or 180 degree rotation in order to lie flat on the bed on the required side. This pitch lever for example needs to be rotated 90 degrees in the green axis to lie it flat on the side it was designed to be printed on.

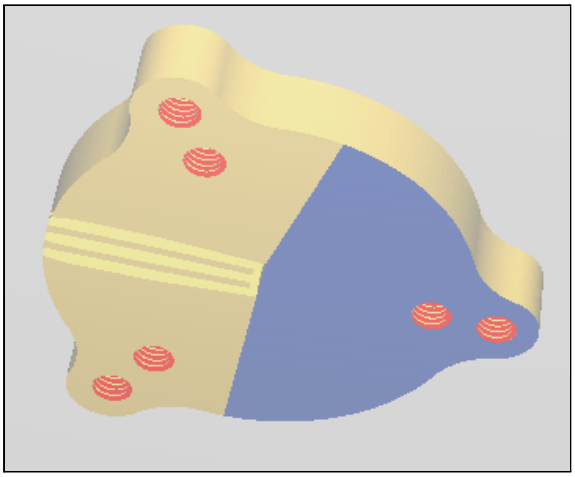


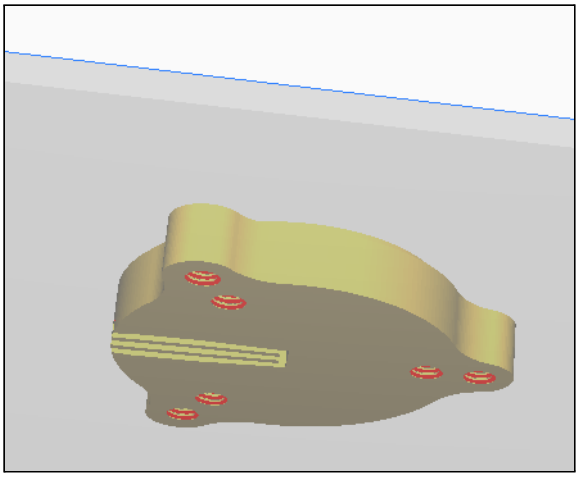
This is a relatively easy example, but there are one or two parts which can't be reliably laid flat unless you use the

Cura tool “Select face to align to the build plate”



One caveat when you do have to rotate items, especially using the “select face” option is to take a look underneath the bed in Cura and check the item is truly flat on the bed. One or two parts are prone to looking flat when in fact they are not and the print then tends to distort and come off the bed. If the print looks like the image on the left from underneath the bed then you may have problems. You want it like the image on the right.

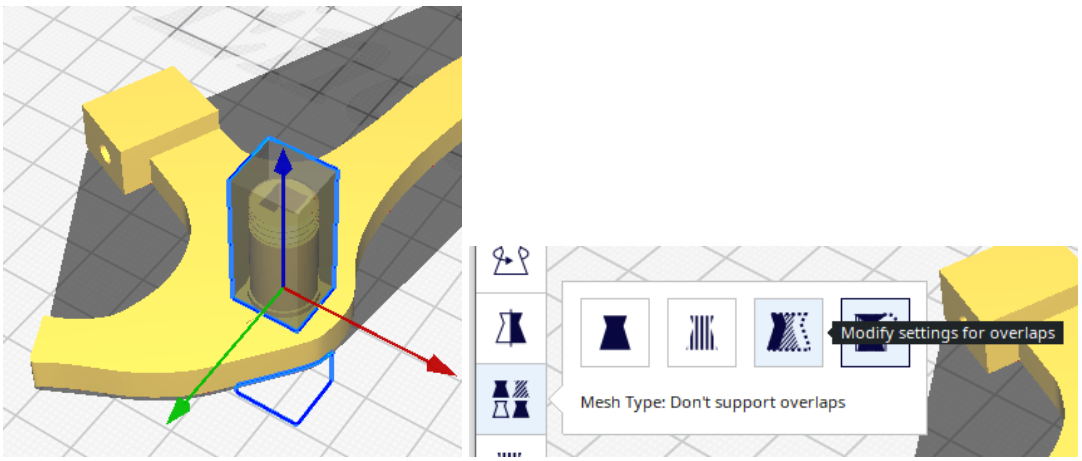


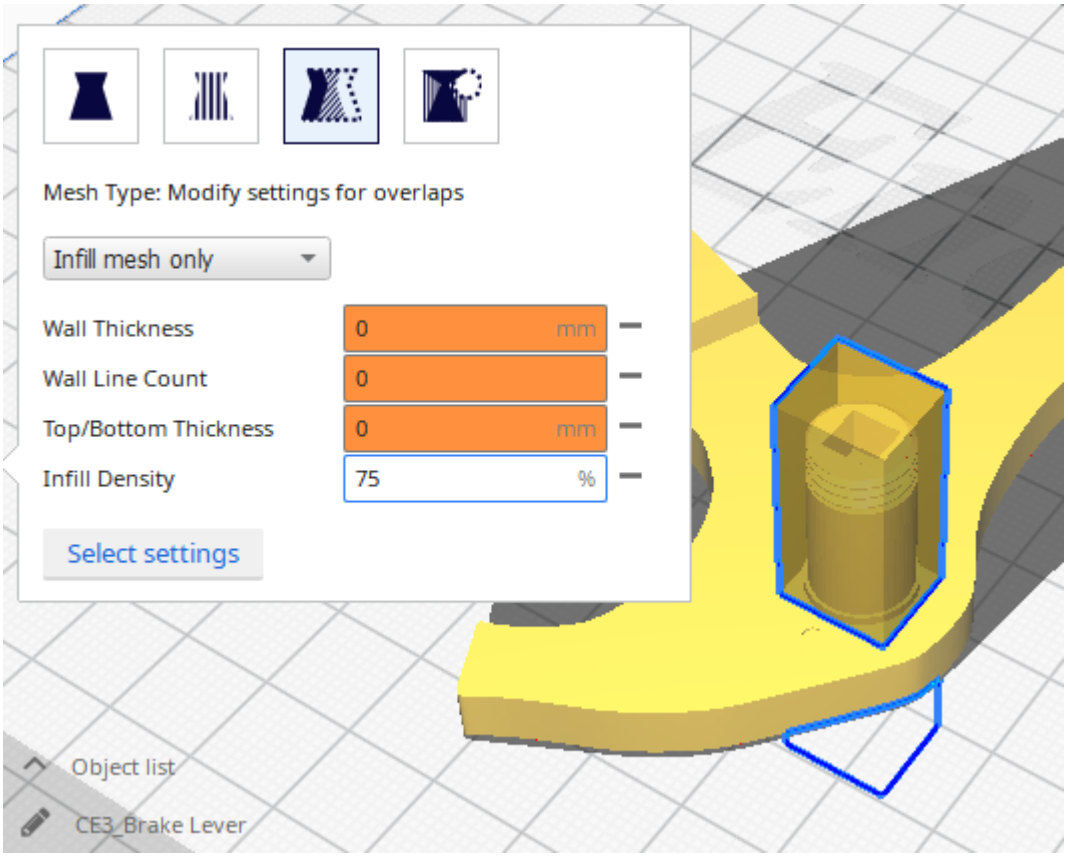


Selective strengthening

Most parts are printed at around 25% infill, with certain items such as the print lever printed a fair bit higher. In some cases you might want to print most of the part at one infill and selectively strengthen areas with greater infill. One example is the brake lever. You can print this at 33% overall, but then print the shaft at 75% plus. Do this using the support blocking tool.

1. Load the lever in
2. Select it
3. Click the **support block tool** then click onto the lever near the shaft
4. Now select the cube that has been placed and use the **scaling tool** to (unlock “uniform scaling”) to make a tall column
5. Now drag the the column so it surrounds the shaft as below
6. Select the mesh type tool (shown below) and pick the option “modify settings for overlaps”
7. Click “Select settings” and choose infill density
8. Now you can say that the infill for the area of the lever inside the cube is 75%
9. Preview the sliced model to see for yourself that the infill is greater where you want it


Modify settings for overlaps
Mesh Type: Don't support overlaps


Mesh Type: Modify settings for overlaps
Infill mesh only
Wall Thickness
0
mm
Wall Line Count
0
Top/Bottom Thickness
0
mm
Infill Density
75
%
Select settings
Object list
CE3_Brake Lever

Key Slicer Settings

Start with the Cura 4.6 defaults which incorporate the latest optimisations for print speeds and retraction settings.

Layer Height	0.12mm (0.24 for 1st layer)	High precision needed to facilitate areas like printed screw threads
Line width	0.4mm	
Outer before inner walls	On	I find I get better dimensional accuracy where I need it
Print thin walls	On	
Fill gaps between walls	Everywhere	There are a lot of slim side walls around 2mmm thick and this setting protects against delamination
Horizontal Expansion	-0.04mm (-0.08 for 1st layer)	Start with this, then print the calibration STL and experiment to match your filament properties. This ensures parts fit together properly and might save you 18 wasted hours !
Horizontal Hole Expansion	0.075mm	A little extra needed for holes
Infill density	Varied	Each part comes with recommended infill density
Slicing orientation	Varied	Each part comes with recommended print orientation
Print temperature	215c plus	Keep it as high as possible subject to stringing to maximise layer adhesion.
Support	None	Support can make parts difficult to fit together. All parts designed to print without.
Build Plate Adhesion	None	Brims can make parts difficult to fit together. Avoid the need with a well levelled bed.
Print speeds	Cura default	Defaults are generally fine subject to the adjustments below.
Small hole max size	8mm	Use this feature to significantly slow down the printing of small holes and features, particularly on the 1st layer

		as there is no brim to aid adhesion.
Small feature speed	50% (same for 1st layer)	Goes with the above