Spitfire MkIX

Starter Kit A

Assembly Guide

1. Introduction
2. Calibration
3. YouTube assembly videos
4. Screws & Wires
5. Index of all printed parts with printing advice
6. Mounting system
7. 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

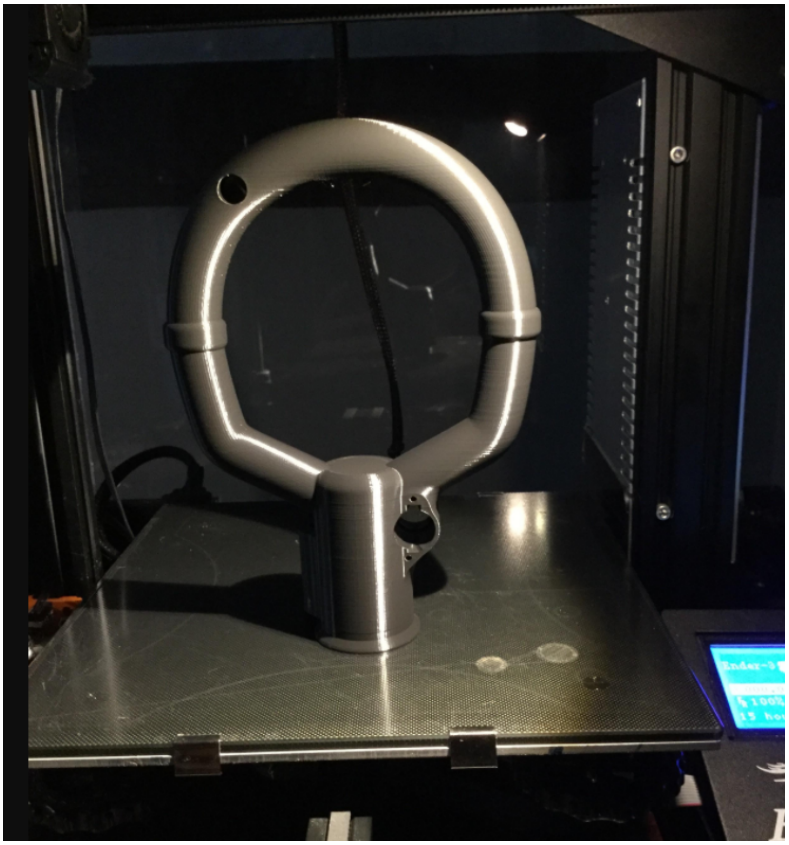
Starter kit A covers the Spitfire MkIX flight stick with its famous spade grip and MkIX two button spade mounted firing system. It also includes the Universal Hub which the flight stick plugs into in order to connect it to your PC. It is called a starter kit because once you have the universal hub you can add on other flight controls like throttle quadrant and trim wheel (coming soon).

I sourced nearly all parts from Amazon & eBay with a few coming from Aliexpress. I was quite nervous about using Aliexpress to begin with but everything arrived exactly as expected albeit with a delay of a few weeks. The beta test team for this project used other suppliers and reported good results using Banggood. Springs were the trickiest to source and availability/cost varies immensely depending on where you are in the world.

For a full list of parts see my document **Spitfire MkIX - Kit A - Bill of Materials.pdf**

Fortunately there is now another sourcing option as someone has set up a business supplying kits for projects like this and are supplying the entire kit of non printed parts for about ⅔ the price you're likely to run to sourcing them separately. See simkitsupplies.com for more details.

There are quite a few printed parts so be prepared that it will take you a couple of days of solid printing. Print the spade grip first - it's very satisfying seeing that come off the print bed!

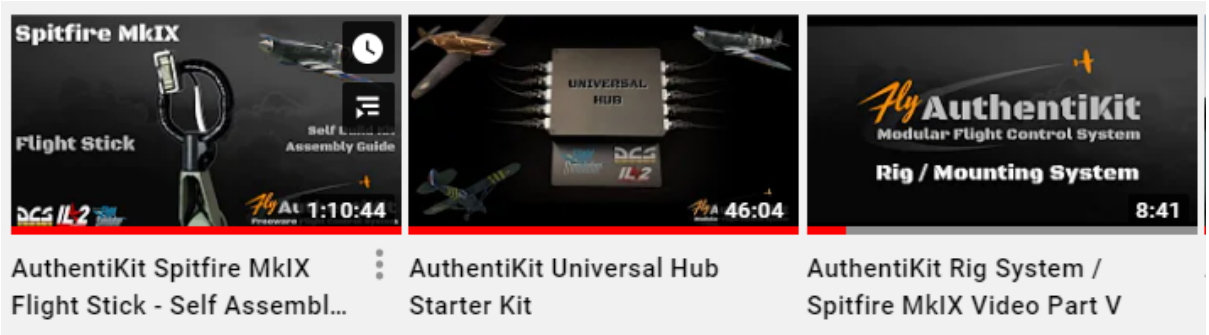


Calibration

Caution - **before you start printing** that spade grip, check the printer settings in section 9 and print the very quick calibration test, then tweak horizontal expansion settings. If you are using multiple colour filaments you are advised to test the calibration on every colour you use - I have found eSun PLA+ green needed a higher flow rate than eSun PLA+ black.

Spitfire MkIX YouTube Playlist

There is no step by step written assembly guide at this time. The primary reference resource for the build is my YouTube channel - see this playlist **authentikit.org/youtube**



You will find other videos about the stick on this channel. Originally the flight stick assembly was covered by 5 videos followed by a version 2 revision video. After a consolidation exercise in July 2021 there is now a definitive single assembly video for the flight stick. The Universal Hub which is required to use the flight stick is covered by a new separate video as it is common to all flight controls. Finally, the rig/mounting system video which was part 5 of the original series is still applicable.

Note that the Universal Hub design in Starter Kit A does not include the cable clips referred to in the Universal Hub video below. If you want that version of the Universal Hub there is a separate download on the AuthentiKit website for the Universal Hub Starter Kit. The two versions of the hub will be harmonised in the near future.

3¾ Hours

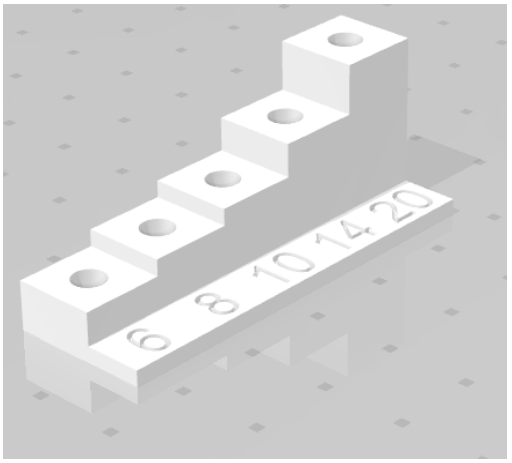
By way of expectation setting, this is how long it took me to assemble the flight stick last time I did it. I would add 50%-100% for your first time. **If this is your first AuthentiKit build then I would actually recommend building the trim wheel and the quadrant in Spitfire MkIX Kit B first. It is an easier first project providing the opportunity to learn the techniques you need for this more complex project.**

Screws

This project uses over 60 screws, most of them countersunk M4 machine screws. **It is important to use the right length screw in the right place**. You'll get away with the wrong screw sometimes but at other times a screw that is slightly too long will break a component. The screws that go into the base of the MagHall 6003 are a good example. To help you use the right screw see this separate document.

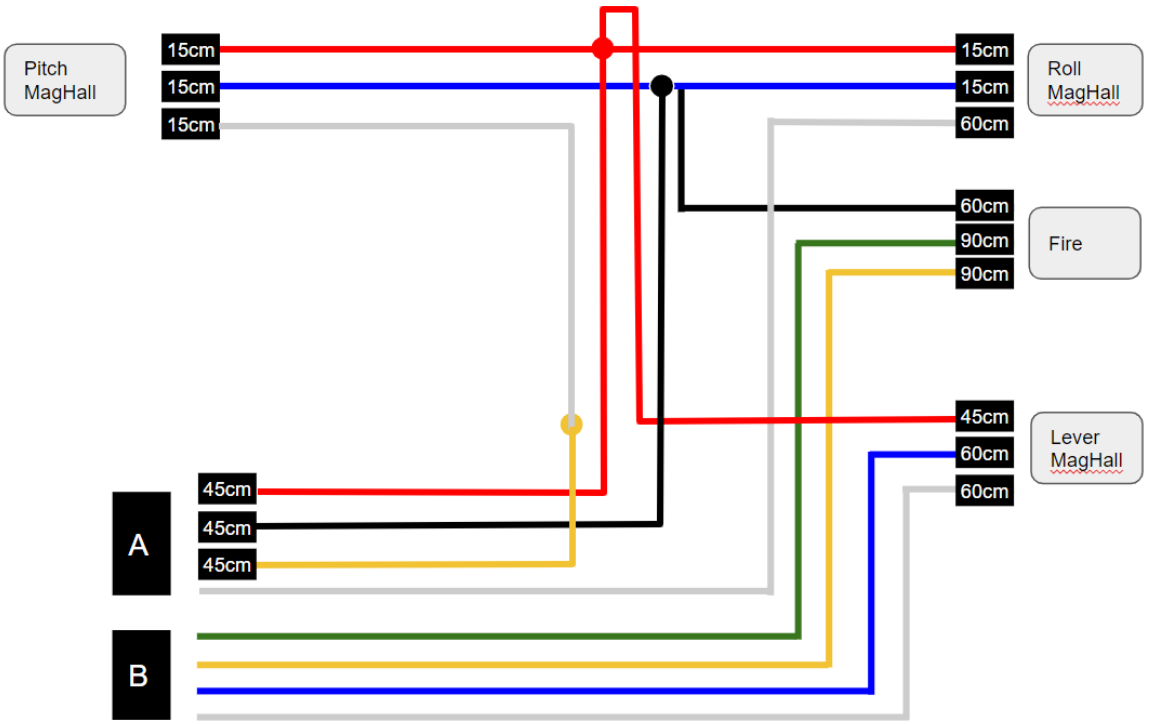
Spitfire MkIX - Kit A - Screws Inventory.pdf

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. It is designed just for M4 screws of length 6,8,10,14 and 20mm.



Wires

Using the right wire length at the start is crucial or you'll find at some point that you don't have enough and need to make a messy joint. Also you need an extra length of wire at the start to pull through body parts which is later snipped off so don't rely on looking at how far the wire has to go as a rule for how long it should be. Use this diagram instead. There is a full size version in the PDF that's part of this project. **Spitfire MkIX - Kit A - Wiring.pdf**



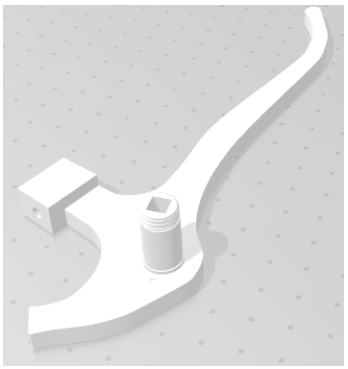
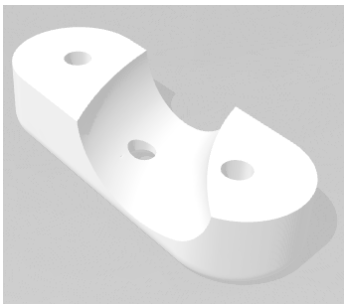
If that's a bit confusing, just follow the video instead and when it refers to a piece of wire just match it with the right colour from this diagram to see how long it needs to be. I know colours are duplicated but the context of what you're working on helps differentiate them.

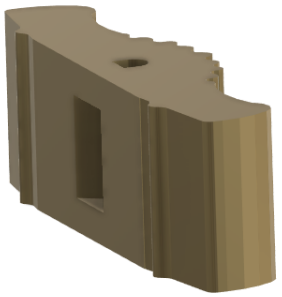
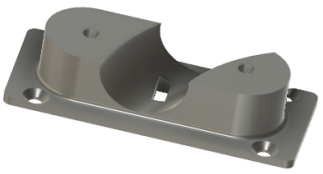
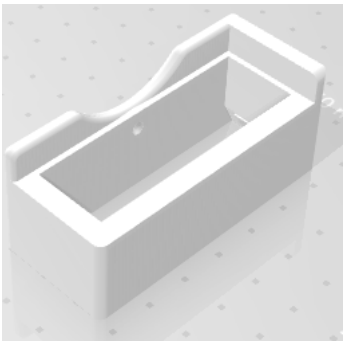
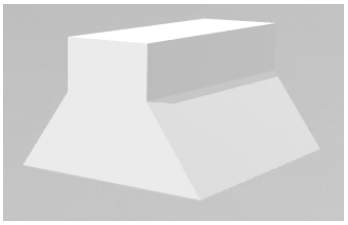
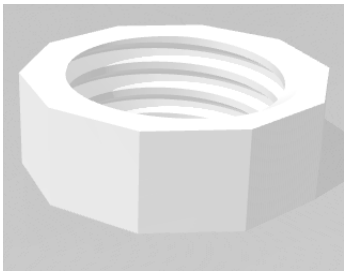
Index of all printed parts with printing advice

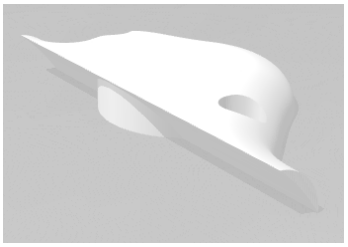
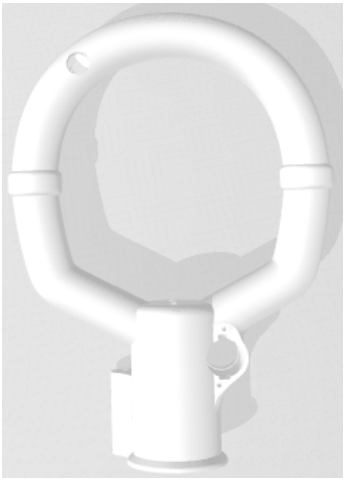
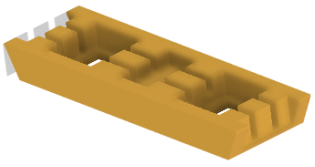
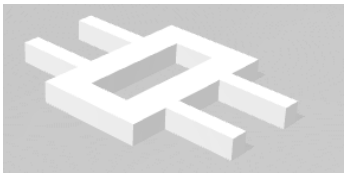
I'd suggest printing most parts at 25% infill and it should be fairly obvious which face should be down on the print bed. Exceptions to this are documented here.

No brim and no support on anything.
There is one exception - the angle bracket.

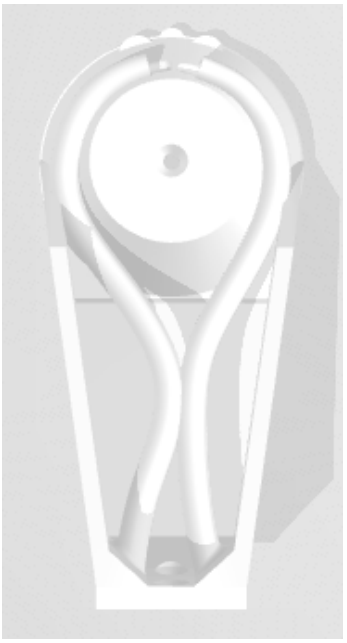
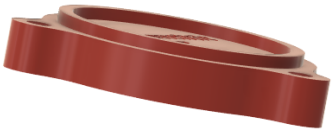
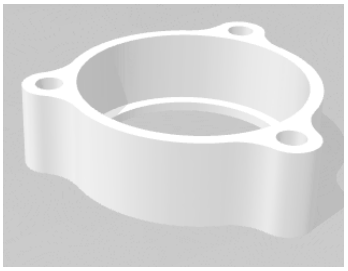
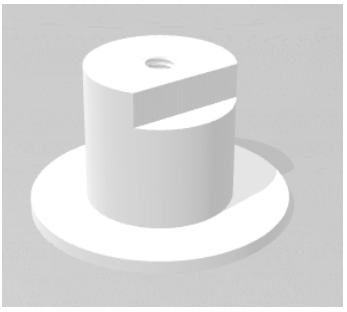
Part 1 - Spade

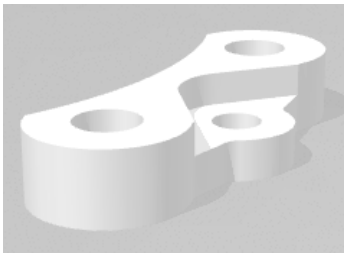
STL	Comment	Infill	Print Colour/orientation
Brake Latch	Not shown in video		Silver 
Brake lever		75% or 33% and selectively strengthen the post to 75%	Silver 
Case base rear			Silver 
Fire button	Make sure it lays flat - check underneath		Silver

			
Fire case base front			Silver 
Fire case			Silver 
Pin plug			Any Colour 
Screw			Any Colour 
Sensor Cover			Black

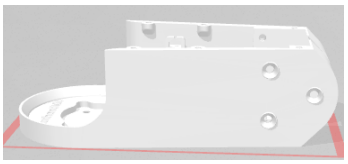
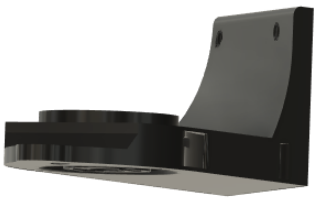
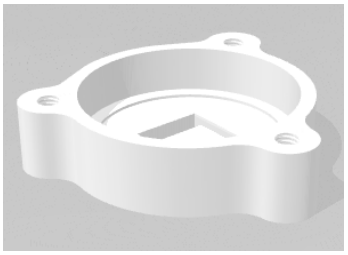
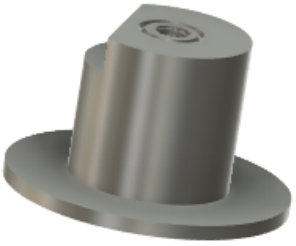
			
Sensor holder			Any Colour 
Spade		33% to be safe	Black 
Tac Case			Any Colour 
Tac Wire Press			Any Colour 

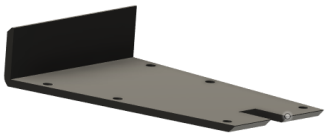
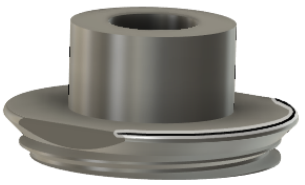
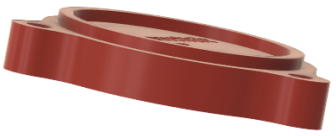
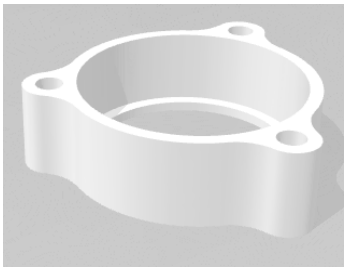
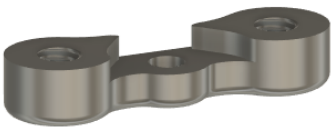
Part 2 - Angle Bracket

STL	Comment	Infill	Print colour/orientation
Angle Bracket	This is the only item that needs a brim or the angle that the bracket stands at means it might tip over.	50% to be safe	Olive Green 
MagHall 6003 Base	An almost identical item is used in the pitch lever this one has a slightly smaller footprint		Any Colour 
MagHall 6003 Cap	As above	50%	Any Colour 
Roll Peg	A very similar item is printed for the pitch lever - the pitch lever one has a circle on the top. This doesn't	50%	Any Colour 

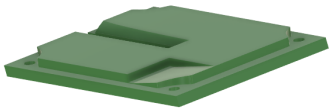
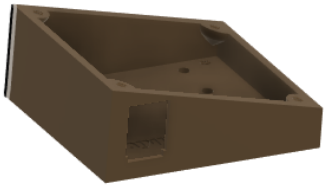
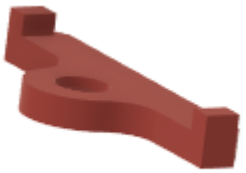
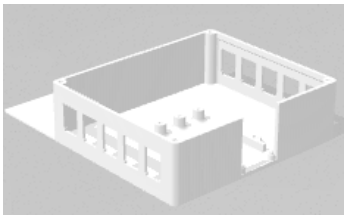
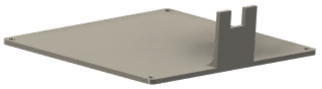
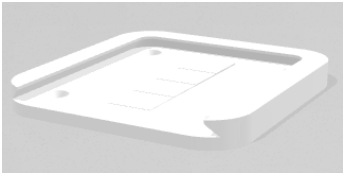
Top Roll Pin Retainer		50%	Any Colour 

Part 3 - Body

STL	Comment	Infill	Print colour/orientation
Body		33%	Olive Green 
Pitch Lever		80%	Olive Green 
Slotted Case			Any Colour 
Pitch Peg	This is like the angle bracket peg but has a circle on top	50%	Any Colour 

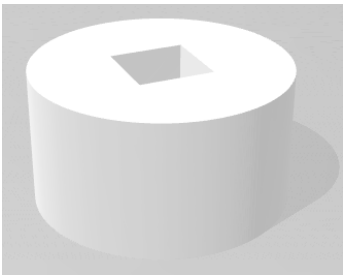
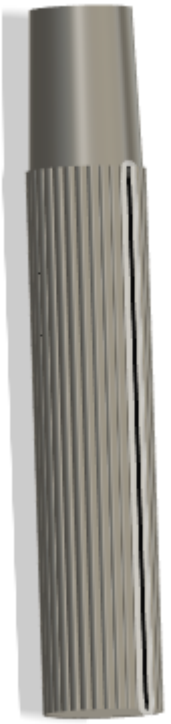
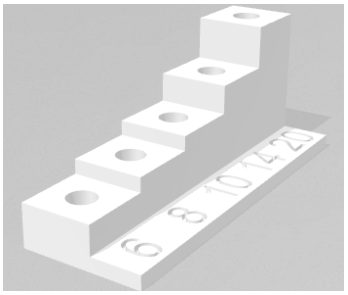
Lid			Olive Green 
New Plug27		50%	Any Colour 
New Base			Any Colour 
New Cap			Any Colour 
Pitch Lug		50%	Any Colour 
Roll Pin Body retain		50%	Any Colour 

Part 4 - Base & Hub

STL	Comment	Infill	Print colour/orientation
Base box lid		33%	Black 
Base Box		33%	Black 
BU clamp			Black 
UniHub Box			Black 
UniHub Lid			Black 
VESA Dovetail		50%	Black 
VESA Wedge			Black

Part 5 - Tools

STL	Comment	Infill	Print orientation
Calibration test	Print this to test your horizontal expansion		
Hex holder short	This fits a standard hex stem screwdriver bit. Use this with PH2 and PH1 bits to make assembly much quicker and easier than trying to do it with the wrong screwdriver.		
M4 6-20	Use this to check you're using the right length of screw		

Plug Spanner	Use this to tighten the pitch lever plug snugly		
Screw wrench			

Mounting System

The flight stick is designed to mount to a VESA bracket on a monitor stand such as this below. The one I have found best is the MDM12DA from allcam.biz which has a 35mm diameter pole. These are not easy to source outside the UK though. An alternative which is very similar and equally robust is the very similar looking stand from Mount-It which has a 48mm diameter pole. That is available in the US, Canada, Australia and probably most other parts of the world.

Amazon have several similar ones that aren't as sturdy but will do as well. You need a 2 arm monitor stand with rigid arm pieces that can be disconnected as you need to take one piece out of one arm to extend the other arm, then mount the stand upside down on a desk like this.

Don’t worry about scratching the desktop because the Universal Hub flap goes between the desk and the clamps both protecting your desk surface and holding the hub in place.

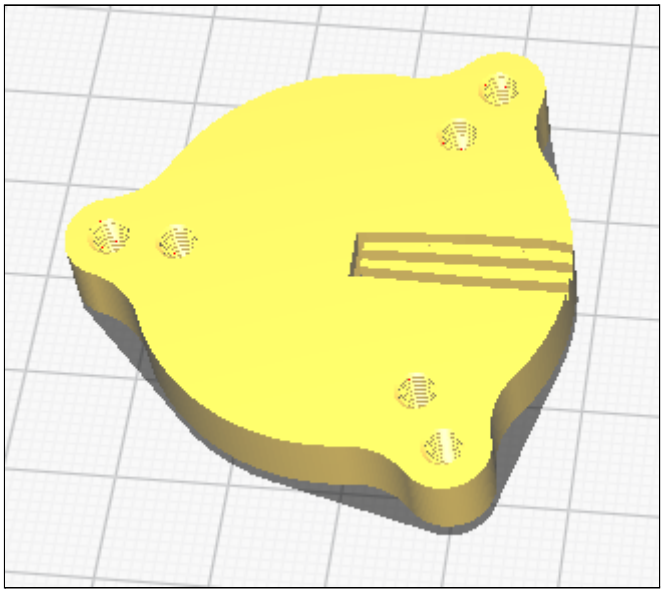
Make a loop with the arms so your left leg can reach the rudder pedals and not catch the arms. Then tilt the VESA plate back 45 degrees and mount the quick release 3d printed VESA wedge.

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.

You will need something that ideally has a print volume of at least 200mm x 200mm x 200mm. If you only have a mini with 180mm x 180mm x 180mm you'll just get away with most things although the spade grip is a few mm taller than 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 angle bracket and main body
- Silver for the fire button and brake lever

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.

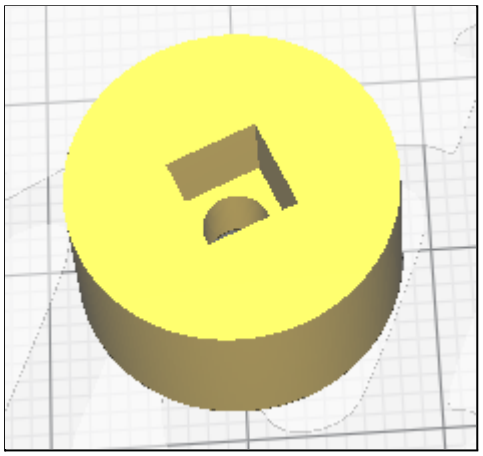
Bed Levelling

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.

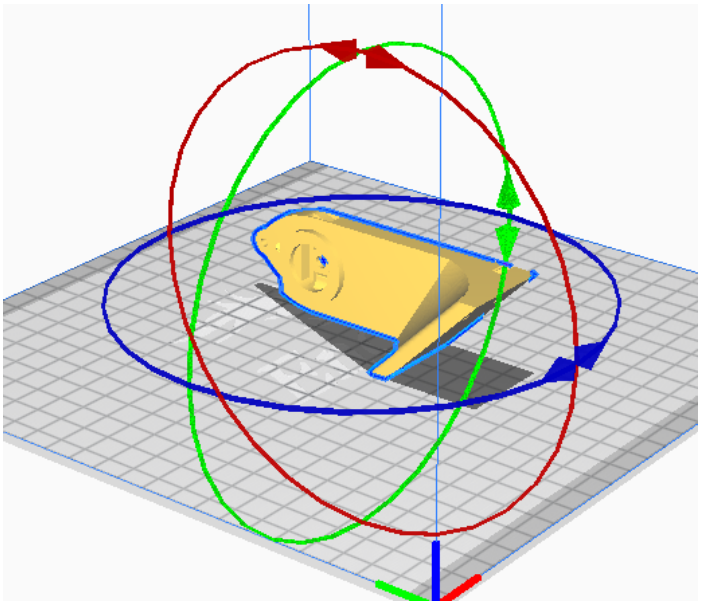


The inner cavity is designed to snugly fit the 5mm^3 cube magnet. The magnet needs to be snugly held but removable by pulling it out with the strength of one of the other cube magnets, but it should not be loose. The outer diameter should fit inside the 60032RS bearing. Again it should be neither loose 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.

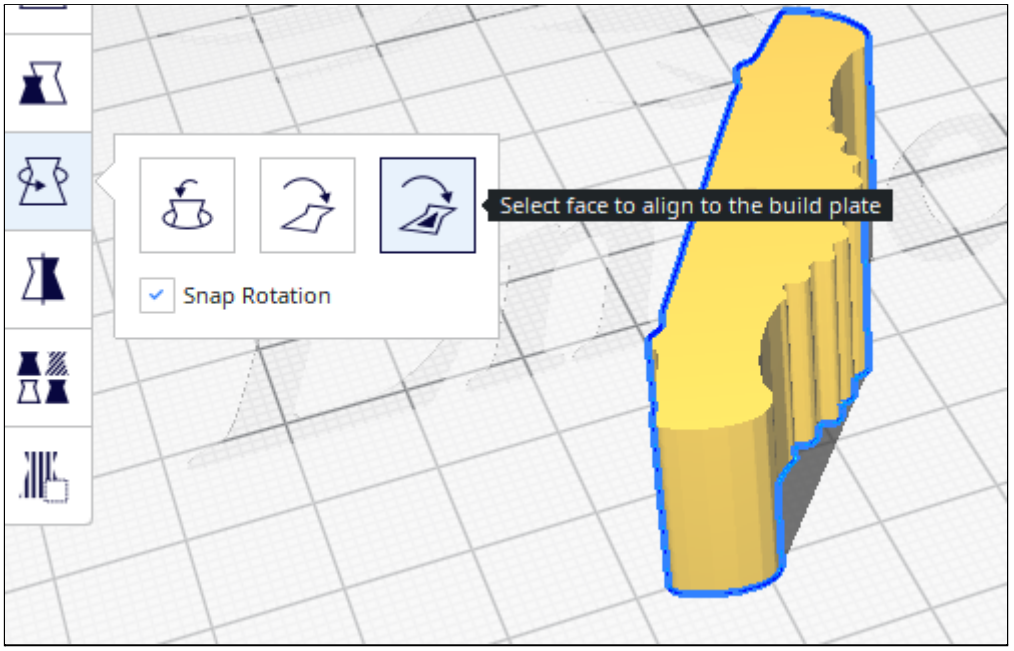
Print Orientation

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.

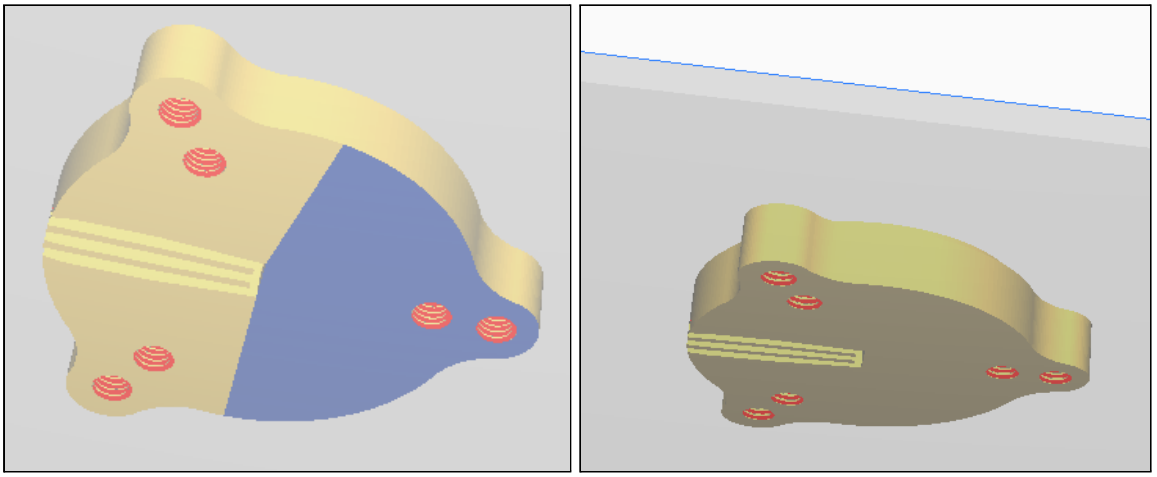


This is a relatively easy example, but there are one or two parts which can't be reliably laid flat unless you use the

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



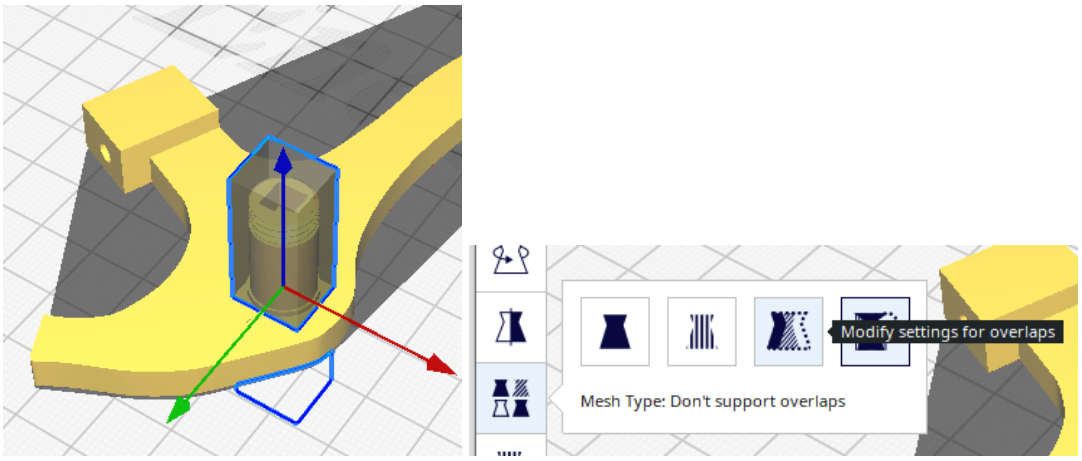
One caveat when you do have to rotate items, especially using the “select face” option is to take a look underneath the bed in Cura and check the item is truly flat on the bed. One or two parts are prone to looking flat when in fact they are not and the print then tends to distort and come off the bed.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

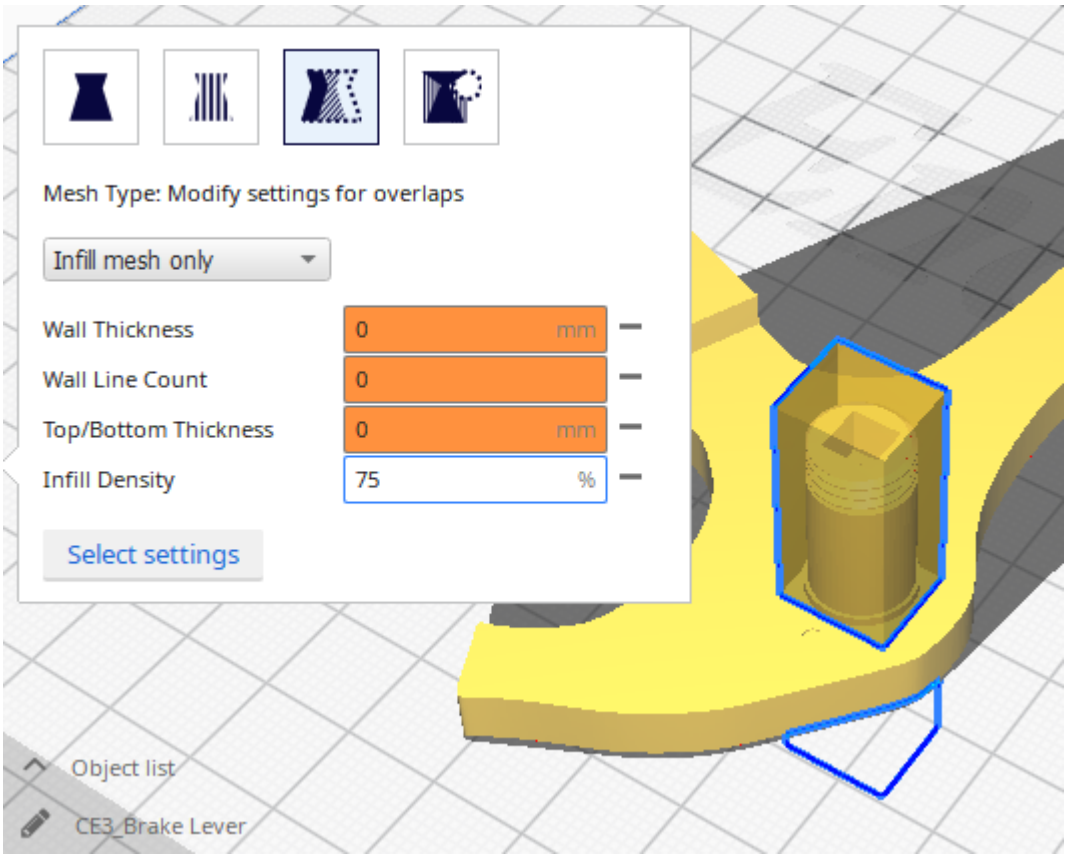


Selective strengthening

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





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