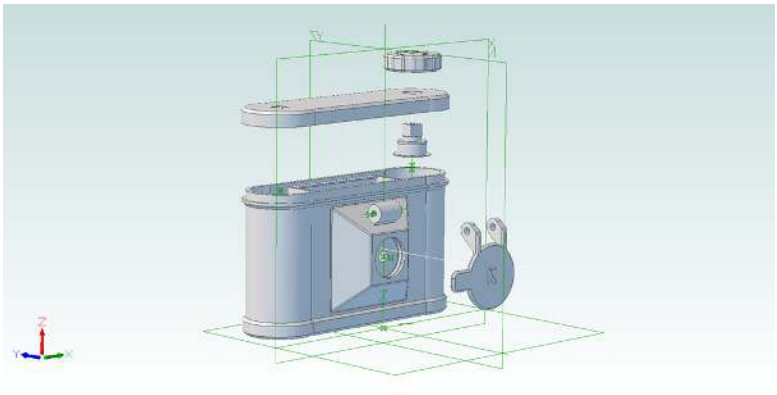
Printing the Flyer 6x6



There are five parts to print, as you can see in the diagram above, which also shows the orientation of the XYZ axes.

- Body
- Top
- Knob
- Winder
- Shutter

License

If you backed my project on Kickstarter, you may print as many cameras as you'd like for yourself or close friends or family for personal use. For anyone else including casual friends, please ask them to go to pinholeprinted.com to get the cameras or files/kits. This way, you help support me in future creations.

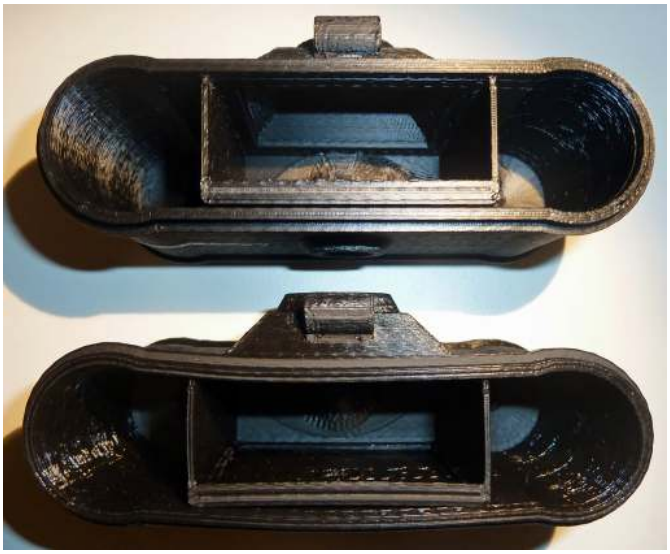
*Reminder: the STL files are **not** open source - please do not publish or post the files, especially not in any public forum. Thank you for respecting my wishes!*

Material & Care

Flyer can be printed in **PLA** or **ABS**. Whichever you use, I recommend printing only in **black**. With any other color, even dark ones, you risk diffuse light penetrating the plastic and fogging or exposing the film.

I also recommend ABS for both durability and temperature stability, but PLA can be used as well. Heat is not good for film or plastic - keep the camera in a camera bag or cover it with a white cloth if you are leaving it out in the sun for a sun track.

If you print your camera in PLA, you must be especially careful not to leave it out in a hot sun or in a hot car - the plastic may soften and collapse. Below is a test photo of two cameras left on a hot dashboard in the Texas sun for one hour - the upper is ABS and the lower is PLA.



Calibration

Calibrating your printer is important for best results. You should be able to download any number of calibration cubes from Thingiverse or Youmagine. Print one and measure it. The calibration process is specific to every printer, so if it is off, you will need to do some research or enlist a 3D printer friend for help.

Calibration is not just ensuring your printer is accurate in the XYZ axes, but also that it is heated to the correct temperature and extruding properly for the filament being used. This is not only specific to every printer, it is also specific to the actual diameter of the filament and filament supplier.

It also helps to print a bridge test or torture test. The more you know about operating your printer for best results, the better your camera will be. Of course, you could just go ahead and try it!

Files and Orientation

There are two files which will need to be unzipped and then sliced for printing.

All parts should be oriented on the print bed in the same relative orientation as the whole for best results. For example, orienting the top in the same direction as the body will ensure that each are off by (roughly) the same amount even if there is a printer calibration error.

Flyer 6x6 A Part 1

- Body STL
- 135mm x 55mm on bed (with brim) x 70mm high
- ~28 meters (28,000mm) of filament
- ~6 hours print time (your printer may be faster or slower)

Flyer 6x6 A Part 2

- Top, Shutter, Knob, Winder (with support) STL
- 127mm x 82mm on bed (no brim) x 15mm high
- ~8 meters (8,000mm) of filament
- ~2 hours print time (your printer may be faster or slower)

Small Bed

If your print bed is less than 6” x 6”, you may still be able to print Flyer by rotating the parts 45 degrees on the Z axis - your print will be diagonal on the bed. If rotated, it is extremely important to rotate all parts in the same direction to minimize the effects of any printer calibration error.

Software

I used Repetier Host 0.90c on one printer and 0.85 on the other to print the Kickstarter Flyer models (http://www.repetier.com). You will use the software that came with your printer or that you are using now, so your setup and process may be different.

The first step is to lay the STL files on the print bed. In Repetier, this occurs when you load the STL. Check that it is laid down correctly - the body should sit on the bed (Part 1), and you should not see the smooth side of the top or shutter with the "F" showing (Part 2) - they should be on the bed. If the smooth side is up, then you need to rotate the parts about the X-axis by 180 degrees to get the correct orientation.

Slicing

Slicing is critical to a good print. The process of slicing converts the 3D stereolithography (STL) model into layer by layer g-code, which tells the printer to move the head, how fast, what temperature the hot end should be, how fast to extrude, etc.

I used different Slicer settings for the body since the tall case was more prone to warping. For the body, I set the brim to 4mm to help anchor the base to the borosilicate glass bed, with the help of hair spray.

For all parts, I used the following general settings:

- Top/bottom layers = 4, concentric finish
- Infill = 20% honeycomb, solid layer every 24 layers
- Brim = 4 for body, 0 for other parts
- for most other settings, you should generally use your printer defaults

Printing

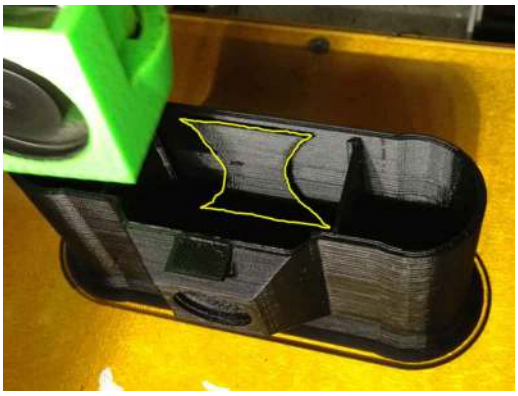
Your printer will likely be different so I cannot offer any specifics. For my two Solidoodle 3's, one has an E3D hot end with a very small melted volume so I run this one at 210C extrusion and heat the bed to 96F. The other has the stock hot end with a large melted volume, so I run this one at 207C and heat the bed to 105F. These temperatures are for ABS plastic and are not calibrated, so the actual temperature may be different.

Post-Processing

Knocking Out the Support Window

There is a support window with scalloped sides in the film plane that needs to be knocked out after printing. Using your finger on the inside and your thumb on the outside, squeeze in the

middle where the film number window is, and move your finger and thumb down until you feel the rigid part. Squeeze hard and run your finger and thumb back and forth - you will feel the support window cracking. Do the same near the top. Now poke out the support window with a screwdriver and remove it. There may be a filament strand at the bottom and sometimes at the top that needs to be pulled out with tweezers or longnose pliers.

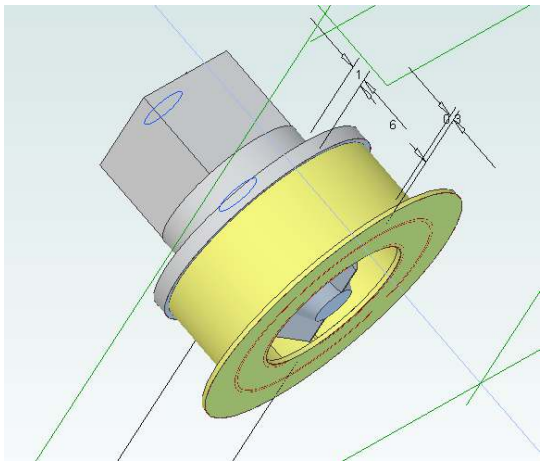


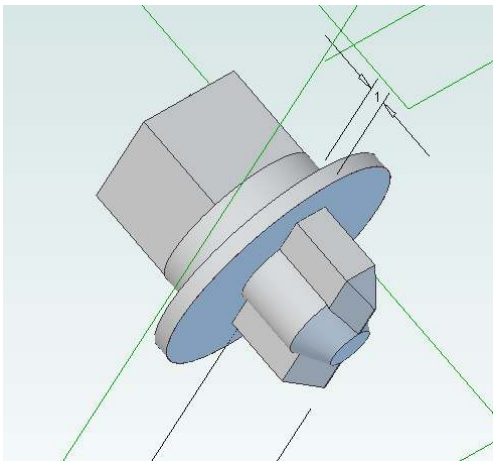
Sanding (optional)

Printing often leaves fine wispy strands when moving the print head. Take a piece of 120 or 150 grit sandpaper and insert it into the film plane, facing the film window, and just rub it back and forth a few times to break any strands. Then reverse the sandpaper and hold the edges forward into the spool cavities and rub it up and down to smooth any rough edges around the corners. After that, wash the body out or you will have a lot of white specks on your first few rolls of film as the particles get on the film and block the light.

Winder

The winder has built-in support which must be removed. This is a trivial task with a pair of longnose pliers - I just place them below the flange and crunch a bit and then pick off the pieces. These pictures should make it clear (yellow is the support material).





Posting

When you successfully print and assemble your camera, please start using it and post your pictures to Flickr or Pinterest and show us what you can do!

- Flickr - http://www.flickr.com/groups/pinholeprinted/

- Pinterest - http://www.pinterest.com/pinholeprinted/

Feedback

If you have issues and need to talk, please use the Google+ forum at https://plus.google.com/u/0/communities/112210063823673129193 or drop me an email.

Troubleshooting

Dimensions/Holes not Round/Binding

If your dimensions are off, your holes may not be round and the knob and winder may bind or the shutter does not close easily. Print a calibration cube and measure the dimensions (the larger the cube, the more accurate your measurements). If they are off, you will need to change your firmware settings. This is generally not required for production models.

Blobbing/Uneven Layers

If your extrusion rate or hot end temperature is not correct, you may experience blobbing or uneven layers.

Warping

This is more likely to occur with the body and top parts, and particularly with ABS, which shrinks as it cools down. The camera body has been designed to allow for a little warp but if the film spool sticks out past the top edge of the body, there has been too much warp. Preventing warp will depend on your printer, but there are generally three factors, all heat-related:

Heated bed

Heated beds have proven to be significantly better with ABS prints. I printed the Kickstarter cameras on a bed heated to 96C with the E3D hot end and to 105C with the stock Solidoodle hot end. The substrate was a sheet of borosilicate glass clamped onto the heated bed, and sprayed with AquaNet Extreme Super Hold prior to each run. The hairspray should be applied lightly to a hot bed just before printing.

Extrusion Temperature

If your extrusion temperature is too high, it can cause blobbing and also more shrinkage as it cools, leading to more warping. On the other hand, if it is too low, the layers will not bond strongly to each other, and the body, in particular may crack on the seams. It's been my experience that if you have one crack, you may

get others later.

Drafts

Cool or cold drafts across your printer bed (not counting the hot end fan if you have one) from running in an unheated garage with changing temperatures can cause some layers to not bond well, developing cracks by the end of the print.