Build Instructions - README.txt

```
This is the DNL DD20.1 3D-printable Glock Frame
In addition to Glock 20 uppers/slides, it is compatible with Glock 21
uppers/slides

Original model developed by FMDA

Bill of Materials:

- DD17.2 spec rails
- Full-size gen 3 locking block (same as DD17.2)
- Gen 5 glock takedown lever spring (many pen springs will work, should be
around 3mm x 10mm)
- Large frame glock lower parts kit (.45/10mm)
- Large caliber magazine of your choice
- G20/21 compatible upper

The files found in the main directory - the documentation and the base files -
are drawn as a blank slate - for a whole collection of remixes (both fun and
possibly functional), refer to the "Remixes" folder.

For help finishing your build, refer to the 'DD17.2 Build Tutorial' pdf found
in this download package, as well as the video you can watch and download here:
https://lbry.tv/@Ivan's_CAD_Streams:c/FMDADD17.2GlockFrameTutorial:e as the
instructions for assembly are identical to the DD17.2 with the exception of the
takedown lever that uses the gen 5 spring.

For general print settings:

This frame has been tested to work well in eSun PLA+. In any material similar
or superior to PLA+, this frame should work as well.
(You can print in Glass-Filled Zytel for a very strong frame. I recommend you
learn how to print Zytel on your machine, then use those settings. Keep the
nozzle hot - 290 - 300C ideally. Run a hot soldering iron over the outside
layers of the finished frame to remelt those layers and you'll be left with a
really tough frame.)

**Note that all STL files are oriented and scaled properly, and changing their
scale or orientation may result in your parts not fitting/working properly.**
**Note on orientation - there's a lot of debate about which orientation to
print pistol frames in. I've found upright works great for this particular
frame when printed in PLA or Zytel. If you print in something that is prone to
warp (like ABS), you might want to print it inverted to keep the bulk of the
frame close to the heat coming off the bed (which will help discourage warp).
**Note that you MUST follow these print settings in order to get the best
```

```
possible results - changing nozzle size or layer height may cause you to have
to hand fit things!**

My print settings for PLA+ were as follows:


Nozzle: 0.4mm
Layer Height: 0.16mm
Temp: 220/60C
Infill: 100% (or 999 walls)
Supports: On/full (tree supports are ok) - use a close support interface gap.
Your interface gap should not be larger than 0.16mm unless you know what you
are doing with your setup.
Cooling fan: 20% normal, 50% on bridges. Off for the first 10 layers. Keep the
fan low to prevent warp.
Bed Adhesion: gluestick or hairspray or both

Post processing: If you print your frame upright, you can clean up the rough
bottom-facing areas of your print with a soldering iron - after removing
supports, take your soldering iron and melt the rough patches on the frame
together to make them smooth.

All other settings can/should be left as default or whatever you know works
best with your printing setup.


This project is published under the Unlicense.
```