

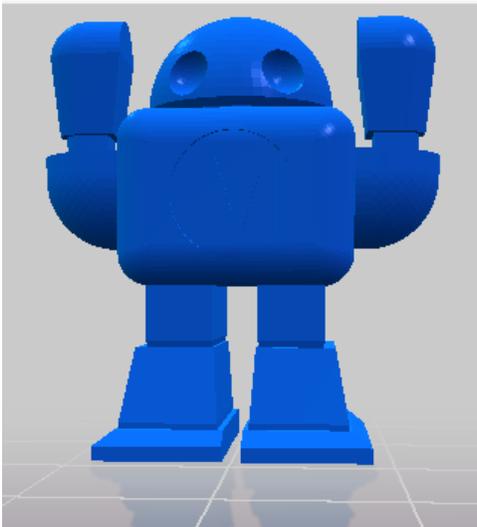
Follow these steps for 3D printing with Formlabs Form1+ stereolithography printer!

(this is intended as a guide for those trained on the Form1+ printer, not as a thorough how-to. if you are unsure what to do, please get help!)

1. Get your model - Doesn't matter where, as long as it's 3-D!

You can:

- Download a model from [Thingiverse](#) or other online sources
- Create a model from scratch in any number of programs (examples: SketchUp, PhotoshopCC, AutoCad)
- Create a model from a real-life object using a laser scanner or photogrammetry

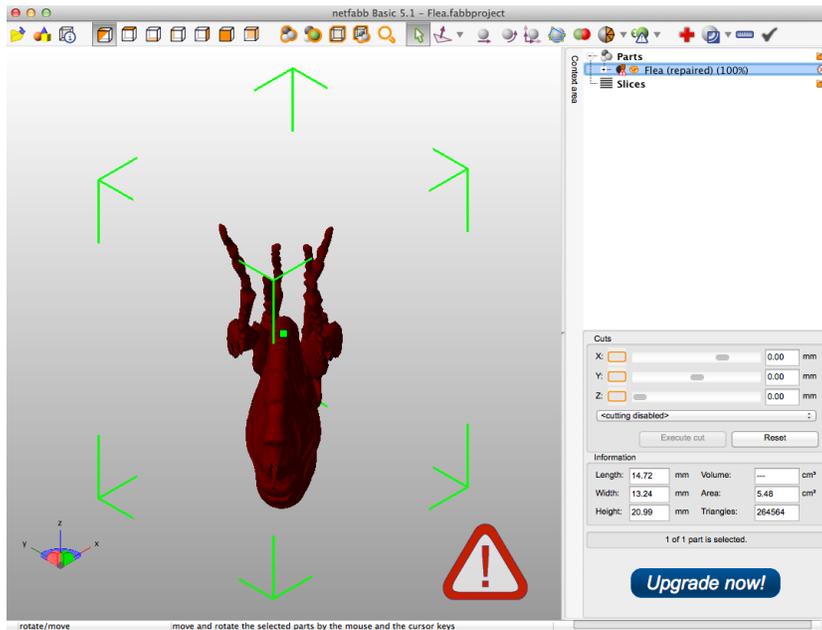


2. Make sure your model is printable!

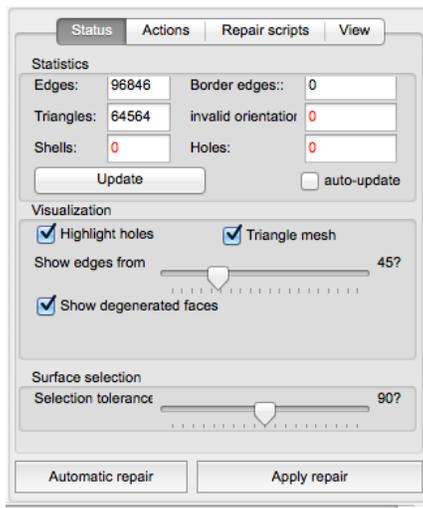
- The recommended program for the Form1+ is netfabb, and it can be downloaded [here](#).

The Form1+ is extremely sensitive to overhangs and holes because of the method it uses for printing. If you are not sure your model is okay, stop here and get help.

2.1. Models will appear in netfabb as 'Parts'

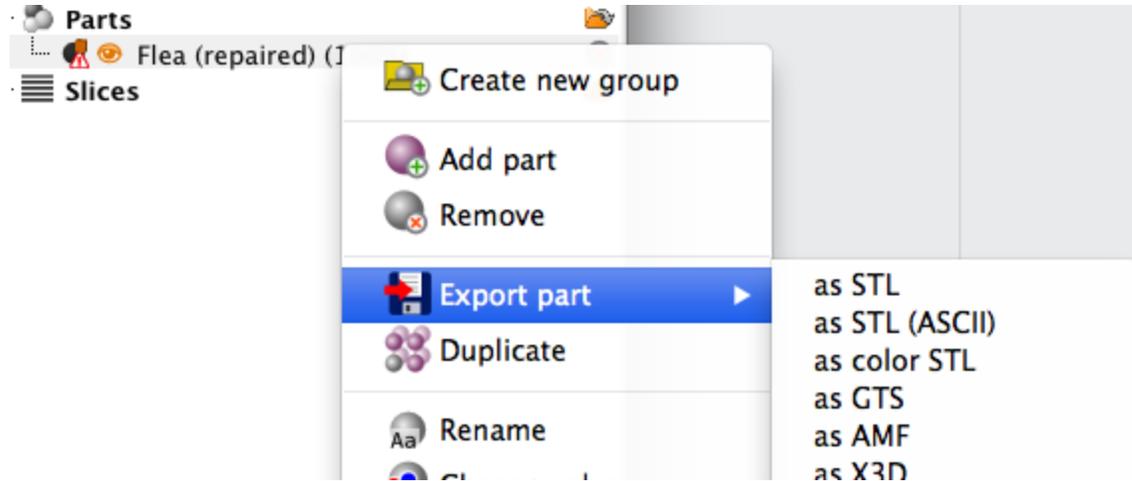


2.2. Clicking the red cross 'REPAIR' button will check a model. The program checks for holes and other errors, and then allows you to perform an auto-repair function.



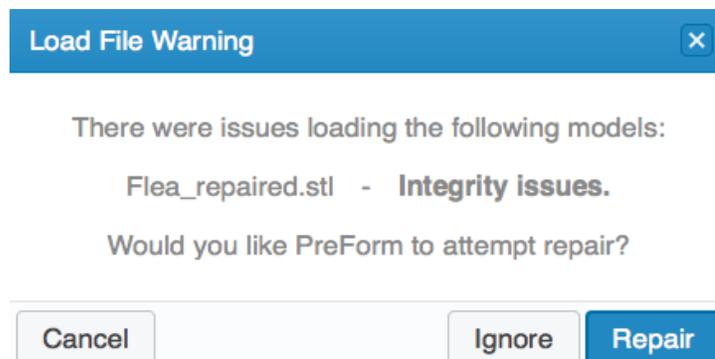
... Flea (repaired) (100%)

- 2.3. When repairs are as complete as possible, export part as a .stl file. If netfabb is unable to complete repairs, they can be completed manually in the next step.



3. Load the .stl file created in the last step into the Form1 interface PreForm.

- 3.1. PreForm can be downloaded for free [here](#).
- 3.2. When opened, PreForm will ask for resolution - use the default unless advised otherwise.
- 3.3. If there are integrity issues, try to repair. If not possible, return to netfabb & check: size, faces, holes



- 3.4. If repairs are successful, adjust sizing & position & orientation

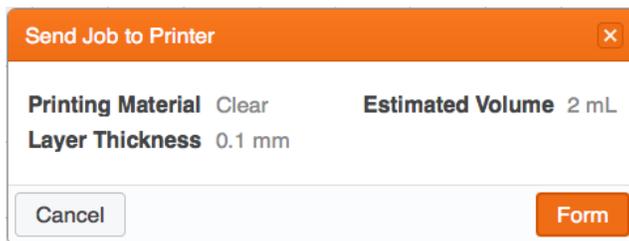
3.5. Basic rules for print preparation:

- PreForm can auto-layout but it is better to choose manually
- Vary print location, keep a record for future printing
 - Do not put tall models on hinge side!
- Print at angles (PreForm will select orientation for you if you want)
- Generate supports (the defaults are generally acceptable)
- Areas will turn red if there's not enough support
 - Adjust support density to increase the areas supported
 - Use 'edit selected' to add supports manually.

4. When ready to print, connect to printer with USB cable.

4.1. Fill printer reservoir to line (overfill is better than underfill)

4.2. Send model to printer by pressing the last button on the line-up



- Printer will double check that build platform is properly in place and the reservoir is filled, confirm.
- Push button on front of printer when all set, print will start.

5. Save job in addition to saving file (saves as a .form) and disconnect computer from printer

6. When print is finished (while wearing gloves!!):

- Remove build plate from printer
- Set build plate upside down on
- Use scraper under edge and gently peel off
- Put in rubbing alcohol bath for ~10 minutes, then agitate for ~5 minutes
- Remove from bath and place on drying sheets, will become more opaque & harden as it dries