GSAPP FAB LAB 3D PRINTING
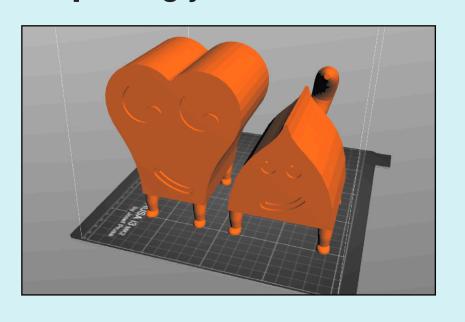
model prep, slicing, and operating a printer

Step-by-step overview:

1. **Export** an *.stl file from Rhino/wherever.

2. **Import** file into MakerBot/Prusa software

3. **Select** print settings: detail, support, and bed adhesion options

4. **Export** file to USB (Makerbot), SD Card (Prusa)

5. **Note** material usage (weight for Makerbot, filament length for Prusa)

6. **Pay** for print. Attach receipt to printer to claim for period of use.

7. If all printers are occupied, get name placed in **queue**.

8. **Prepare** machine: clean beds, wipe Prusa with alcohol, check filament

9. Start print and **WATCH** for early adhesion to bed.
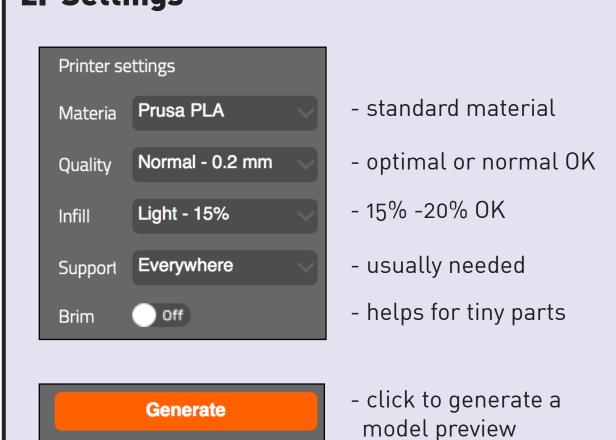

Prusa Control

It's your responsibility to see that your files are set up smartly.
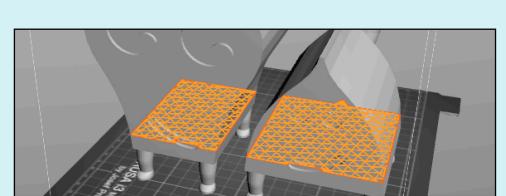
1. Importing your file

After exporting an STL from your modeling program, import it into Prusa Control. If your model appears small, you may need to scale between inches and mm. To convert, scale 2540%
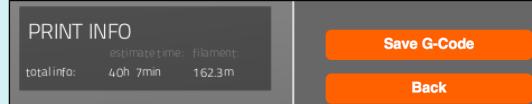
2. Settings

Printer settings

Materia	Prusa PLA
Quality	Normal - 0.2 mm
Infill	Light - 15%
Support	Everywhere
Brim	Off

- standard material
- optimal or normal OK
- 15% -20% OK
- usually needed
- helps for tiny parts

Generate

0%

- click to generate a model preview

3. Exporting

The program provides a preview layer-by-layer of your print. Check to make sure there is material all the way through.

PRINT INFO — total info: 40h 7min, filament: 162.3m — **Save G-Code** / **Back**

Note the filament use. Show the shop crew this number to calculate payment. Export to an SD card!

Prusa MK2/MK3 Printers

Follow these steps to ensure the best chance of success for your prints.

1. Clean and inspect the printer
Remove all material from the previous print, and wipe the print be thoroughly with alcohol or acetone.

2. Check amount of filament
One whole roll of filament is about 250M. If you're not sure if there's enough, ask a crew member.

3. Insert SD card and load file
Browse with the knob to "print from SD" and locate your file, select it to start printing.

4. Stay to watch first layer adhesion
This is extremely important. Watch the first few beads of material the printer lays down. If the material is sticking and LOOKS good, you're OK.

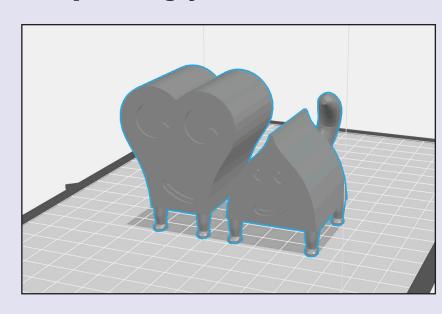### 5. Live-adjust Z if necessary

Adjusting Z:
−0.640 mm

The first bead of plastic should look nicely "painted" on.

If it looks too much like a wire, or is not sticking, click the knob, navigate to "live adjust Z", and dial the number (typically down) until you see the correct bead.
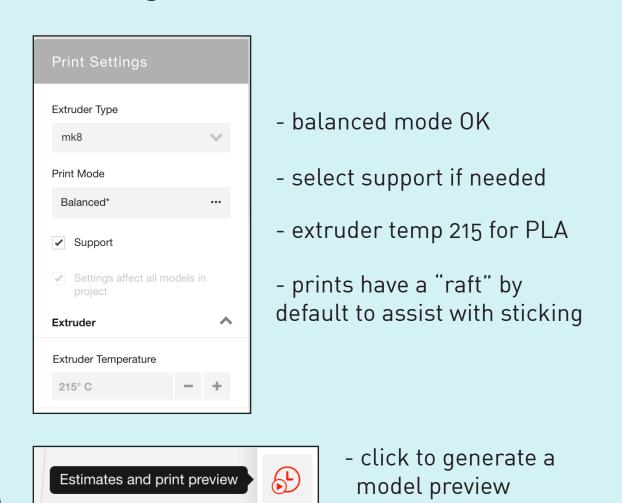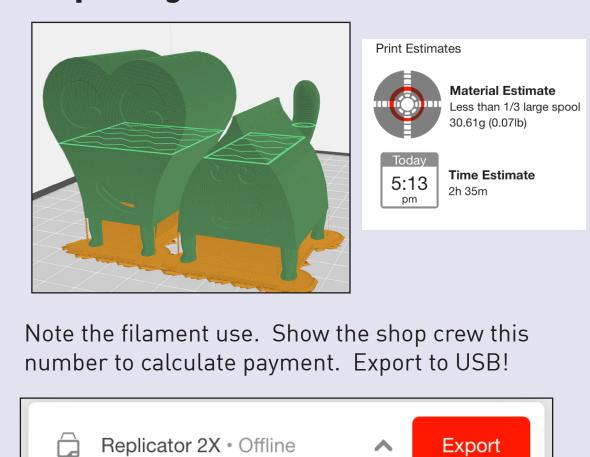

Makerbot Print

1. Importing your file

After exporting an STL from your modeling program, import it into Makerbot Print. If your model appears small, you may need to scale between inches and mm.

2. Settings

Print Settings

Extruder Type
mk8

Print Mode
Balanced*

☑ Support

☑ Settings affect all models in project

Extruder

Extruder Temperature
215° C   − +

- balanced mode OK
- select support if needed
- extruder temp 215 for PLA
- prints have a "raft" by default to assist with sticking

Estimates and print preview

- click to generate a model preview

3. Exporting

Print Estimates

Material Estimate Less than 1/3 large spool 30.61g (0.07lb)

5:13 pm — **Time Estimate** 2h 35m

Note the filament use. Show the shop crew this number to calculate payment. Export to USB!

🔒 Replicator 2X • Offline   **Export**

Makerbot Replicator 2X

Follow these steps to ensure the best chance of success for your prints.

1. Clean and inspect the printer
Remove all material from the previous print, make sure blue tape bases are intact and not ratty.

2. Check amount of filament
One whole roll of filament is about 250M. If you're not sure if there's enough, ask a crew member.

3. Insert USB and browse to file, and start

4. Wait to observe quality of first beads (see Prusa details above)

5. Watch the print for 5-10 minutes before leaving!