HowTo use the new 3D Printer "Formlabs Form 2"

3D Printer Status: **WORKING (no Issues)** if you have troubles please write a request (following the report guideline) to simon.haller@uibk.ac.at.

Handling

HANDLE THE PRINTER WITH CARE !!!

- The Form 2 is a precision tool. When used properly, it will help you create beautiful prints. Like any professional equipment, you should treat the printer, materials, and accessories with respect and care to ensure a safe working environment and a long-lasting machine.

Material

Following Resins are available.

- **(in use) Clear Resin 1L** (GPCL02)
  Clear Resin polishes to near optical transparency, making it ideal for working with light or showcasing internal features. Supports print resolutions: 100, 50, and 25 microns.
- (**in stock) Black Resin 1L** (GPBK02)
  Black Resin is formulated for models with very small features and intricate details. Supports print resolutions: 100, 50 and 25 microns.

Formlabs currently sells several resins that are compatible with the Form 2. Their general-purpose hard plastic resin is available in diferent colors: clear, white, grey, black. In addition, the company sells a resin specialized for casting, a flexible resin, a tough resin, and a resin for intraoral surgical guides.

Techn & Design Specs

- Technology: Stereolithography (SLA)
- Self-heating Resin Tank
- Build Volume: 145 × 145 × 175 mm
- Layer Thickness: 25, 50, 100 microns

Known Issues

Date	Description	Solution
19.01.2017	There are no known issues at the moment.	

Solved Issues

Date	Description	Solution
19.01.2017	There are no issues to solve at the moment.	

Updates

Date	Description
03.01.2017	Firmware update rc-1.11.8-64

Creating a 3D Object

Choose a CAD or 3D Software which can handle stl files (examples: MeshLab, Blender, Bricscad, QCad, FreeCAD, VariCad, Open CASCADE, Cycas CityEngine, BRL-Cad, Draftsight, LibreCAD, ...)

Create your 3D model and save it as STL file.

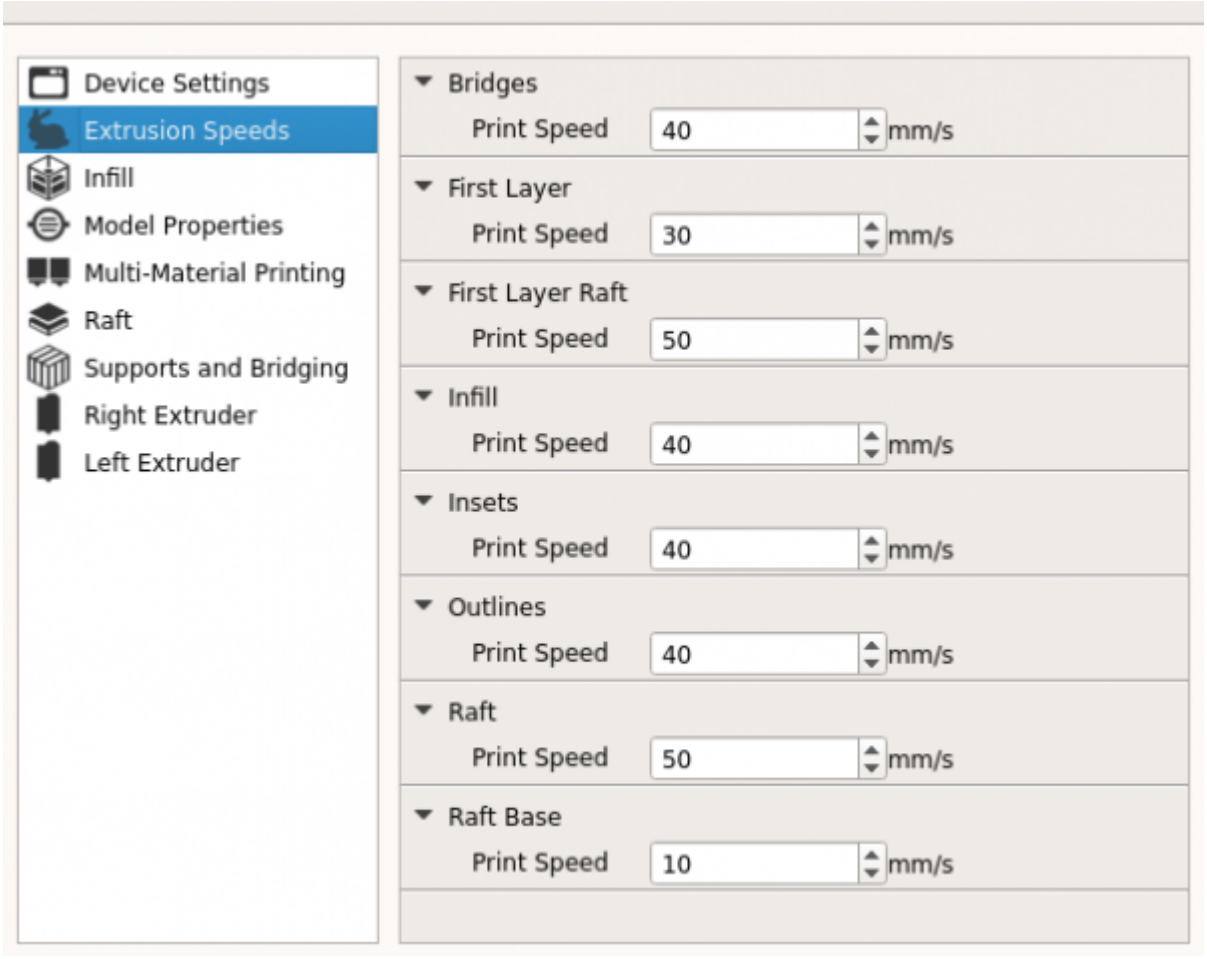
You can also have a look at the makerbot `thingiverse` library - where you can find free (gpl licensed) 3D models ready for printing: http://www.thingiverse.com/

If you want to share your CAD models with the group (or if it is more often used) add your STL file(s) to following repository:

```
ssh://iis.uibk.ac.at/projects/git/CAD-models
```

Makerware and 3D Print Files

Use the following speed settings:



To create 3D print files you have to use the original Replicator 2x software: makerware (Download: Makerware)

1. start Makerware
2. **SELECT CORRECT 3D Printer**
3. MakerBots → Type of Makerbot → The Replicator 2x

Import your STL file in Makerware and use the following settings as default for print-file export (object without quotes refers to the imported stl file; “Object” refers to the menu point):

1. move object to platform (select object → selct “Move” → “On Platform”)
2. place object a bit to the front
3. select object → “Object” → select Extruder “right”
4. select “Make”
5. Export for “The Replicator 2X”
6. Select Right: “Makerbot ABS”
7. Resolution: Standard
8. Raft: “Right Extruder”*
9. Supports (only if you have overhanging parts in you object): “Right Extruder” or “Color matched”*
10. Quality
    1. Infill: betweeen 10% - 40% depending on how stable your object should be
    2. Number of Shells: 2
    3. Layer Height: 0.15mm
11. Temperatur:
    1. Extruders: 233
    2. select “Heat Build Plate”
    3. Build Plate: 114
12. Speed:
    1. Speed while extruding: 40mm/s
    2. Speed while traveling: 70mm/s

Save the x3g file on a FAT formated SD-Card. The SD card used with your MakerBot Replicator x2 must be formatted FAT16 with a maximum capacity of 2GB. Put the SD-Card into the printer and select your file to print.

If (and only if) you want do dig REALLY DEEP and have full control, consider defining you own slicer options as documented here: http://www.makerbot.com/support/makerware/documentation/slicer/ BE AWARE that already the default options differ a lot from what you are used to. So take your time and adjust every single setting to your needs.

* When printing large objects, Makerware may decide to use the two extruders altough only one was chosen, this to let the material/extruder cool down. You can check this by looking at the preview, in the right top corner the material use will be displayed, if both extruders are used “right material use” AND “left material use” will appear and the object will be shown in two different colors. If you do not want Makerware to do so, choose “color matched” instead of “left extruder”.

Instruction for old MakerBot