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

- Design guide from formlabs

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.

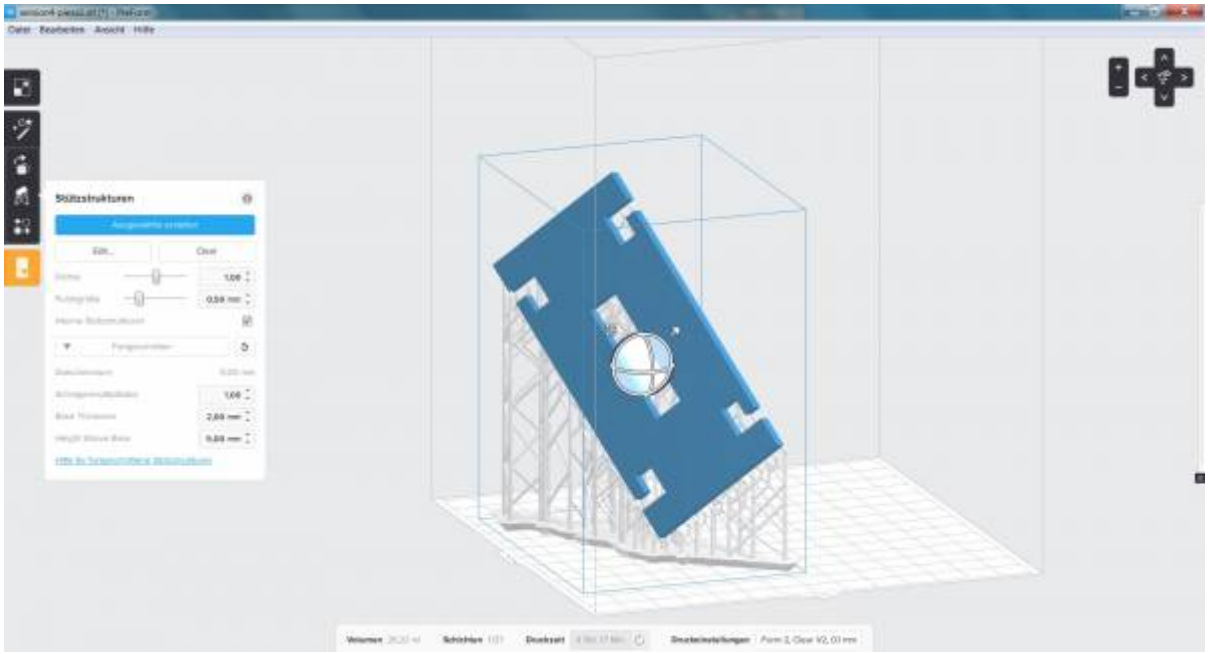
You can also have a look at the makerbot `thingiverse` library - where you can find free (gpl licensed) 3D models ready for printing: http://www.thingiverse.com/

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
```

PreForm Software and 3D Print Files

- PreForm prepares your models for printing on the Formlabs printer.
- The Software will be available on a Laptop stationed besides the Printer.



To print 3D files you have to use the original PreForm software which will will be available on a Laptop stationed besides the Printer (Download: PreForm)

1. Prepare your .STL or .OBJ files for Print.

2. Start the PreForm Software.
3. Open your .STL or .OBJ files. Most parts print better when support at an angle. For automatic set-up, use PreForm tools to “orient all,” “generate all,” and “layout all.” PreForm will do these steps in sequence if you select “One Click Print.”
4. Confirm the print's material and layer height, then click the orange print icon. Select “which printer to upload the file to” from the printer dialogue.
5. You will see your file uploading to the print queue on the printer's touchscreen. Select the file name and confirm the print by pushing the button on the printer.
6. Before printing, the resin tank will fill and heat up to around 35° C. Once the tank is full, your print will automatically start.
7. When your print is finished, wear gloves to remove the build platform. Turn the platform over to prevent dripping while transferring to the finish kit. Make sure to close the printer cover.
8. Attach the build platform to the jig and slide the removal tool under the base of your prints to release the part. Use the removal tabs for easy entry points.
9. Fill two rinse buckets with isopropyl alcohol (IPA). Keep the lids closed. IPA is flammable!
10. Drop your print in the rinse bucket and leave for 10 minutes. Agitate your part and repeat with the second rinse bucket. This two step process will help your IPA last longer.
11. Once dry, use the flush cutters to remove supports from your print. You can sand support marks and polish your parts for a clean finish. You may need to UV post-cure your part if using our Functional Resins.

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