HowTo use our 3D Printer

3D Printer Status: **WORKING (see 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!

For example:

```
Do not use metal tools to remove printed objects.
Do not touch the the built plate with your fingers.
```

Both will damage the Kapton Band.

Material

At the moment only use ABS (green, red, white, blue, orange).

Do not use PLA (black) or Ninja Flexible Material … for those materials we have to wait for a firmware upgrade (14.07.2014).

Known Issues

Date	Description	Solution
10.08.2015	again broken fans - ordered replacement	
16.02.2015	Makerware crashes (segfault)	http://www.ichdruck3d.de/makerware-stuerzt-unter-linux-ab-solved/
08.12.2014	Use the left extruder (the right one is unplugged)	

Solved Issues

Date	Description	Solution
31.08.2015	Broken Capton Band	Changed Capton Band
10.08.2015	Broken Temp Sensor	Replaced temp sensor
29.07.2015	Broken Capton Band	Changed Capton Band
03.05.2015	Broken Capton Band	Changed Capton Band
01.12.2014	Broken Gear	Fixed with a new gear holder
21.10.2014	Broken Capton Band	Changed Capton Band
17.09.2014	Broken Capton Band	Changed Capton Band
08.08.2014	Rigth extruder not working (“Heating Failure”).	Reattached thermocable at extruder. Seems to be very fragile…

Date	Description	Solution
08.08.2014	Kapton Band had holes	Replaced it
23.07.2014	Still jumpy X Axis	Got Cables from Support (hafners-buero.com), Replaced them.
16.07.2014	Still jumpy X Axis	Requested new Cables from Support
15.07.2014	Joggy Movement of motors	Strengthend X Axis belt. Resolderd X Axis Connectors
15.07.2014	SD Card Slot does not hold Card	Fixed Slot (push sd card softly in - do not use any tape around an SD Card)
14.07.2014	Broken Fan	Replaced Fan
14.07.2014	Jumpy Stepper Motors	Used WD40 as lubrication
05.06.2014	Greasy Kapton Band	Replaced Band

Updates

Date	Description
08.08.2015	Firmware update to 7.6
14.07.2014	Firmware update to 7.5

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

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 “left”
4. select “Make”

5. Export for “The Replicator 2X”
6. Select Left: “Makerbot ABS”
7. Resolution: Standard
8. Raft: “Left Extruder”*
9. Supports (only if you have overhanging parts in you object): “Left 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: 70mm/s
    2. Speed while traveling: 150mm/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”.