3D Printing
Eric Nutsch
Mechanical Engineer
Extraordinaire!

ericnutsch@gmail.com

Adam Day
Run-of-the-mill IT guy
and Linux Jedi!

aday@twinfallspubliclibrary.org
3D Printers
Additive manufacturing is a process of joining materials to make objects. Example: 3D printing

Subtractive manufacturing is a process of removing materials to make objects. Example: CNC mill/router
Fused Deposition Modeling (FDM)

1. Filament is melted and extruded
2. Model is built by adding layer after layer

Cost effective but lower print quality.
Stereolithography (SLA)

1. Print surface submerged in a resin and then lifted
2. Light sensitive resin layer covers surface
3. Light is projected onto surface.
4. Object is lifted out of resin. Higher print quality but more expensive.
Selective Laser Sintering (SLS)

1. Thin layer of powdered material spread over the print area
2. Laser fuses powder

High temp materials
Metal printing
Hardware
PrintrBot Simple Metal

Model: 1403
Build Volume: 6" x 6" x 6" / 150mm x 150mm x 150mm / 216 cubic inches
Print Resolution: 50 Microns
Print Speed: 80mm/sec max recommended
Filament: 1.75mm PLA
Extruder: Alu Extruder V2 (direct drive) with 1.75mm Ubis Hot End with 0.4mm nozzle
Auto Leveling: Auto-Leveling Probe
Print Bed: Not heated – powder coated 5052 aluminum (0.3mm level tolerance)
3D Printer Axis
Filament holder

Spools of filament can hang from the holder.

Be sure the spool has free rotation around the spool holder.

Remove any tangles or knots and make sure there is enough filament to complete the print.
Print bed & Buildtak

The surface that the print adheres to.

Buildtak is a textured surface that helps with print adhesion. Works well with PLA. Make sure print nozzle is about 0.25mm above surface.

Clean by scraping. Do not use solvents or chemicals.
Stepper motors

The printer controls distances and speeds by sending pulses to the motors.
200 pulses = 1 revolution

Four stepper motors on PrintrBot:
1. Extruder
2. X axis
3. Y axis
4. Z axis
Belt Tensioning

Belts need to be periodically tensioned over time.

Tighten allen bolts to increase belt tension.

A sign of a loose belt will first show in print quality. Check belt tension.
Controller board

Interprets print commands coming from the computer and controls the stepper motors and nozzle/bed temperatures.

Located on underside of Printrbot.
Micro USB cable

USB-A to USB Micro-B cable.

Be sure to use a data USB cable not a charging USB cable.
Z lead screw

Controls the z height
Connected to the z stepper motor
Extruder Assembly

- Extruder Block
- Idler Tension Screw
- Idler
- Hobbed gear
- Fan & Shroud
- Z height probe
- Hotend & Nozzle
Extruder

Hobbed bolt feeds filament
Lever and spring grip filament
Pressing lever releases grip on filament for removal
Tightening nut reduces grip on filament
Cooling Fan & Shroud

- Helps cool print layers
- Shroud directs the air to the nozzle tip
- Helps with print quality
- Permits faster prints and bridging
Hotend & Nozzle

The nozzle is where the filament is melted and forced out of a nozzle. Accepts 1.75 mm filament. 0.4mm nozzle diameter. Red insulating silicone helps maintain nozzle temp. (do not touch).
Z sensor

Inductive sensor that measures the height above the print bed. Printer auto-levels at the beginning of every print by measuring at 3 locations on the print bed. Light at the top indicates the bed has been detected. Hex nuts for mechanical calibration.
Software
Download the getting started package
Download includes Cura 15.04.2 installation for PC/Apple and Cura profile for the PrintrBot Simple.
Extract to a folder on your Desktop.
Start Cura

http://printrbot.com/project/simple-metal/
One time setup: Select Machine

Select your machine

- Ultimaker 2
- Ultimaker 2 extended
- Ultimaker 2go
- Ultimaker Original
- Ultimaker Original+
- Printbot
- Lulzbot TAZ
- Lulzbot Mini
- Others (Ex. RepRap, MakerBot, Wanhao)

PrintBot Selection

- Single Metal
- Metal Plus
- Simple Maker's Kit

Cura is now ready to be used!
Quality

- Layer height (mm): 0.1984
- Shell thickness (mm): 0.8
- Enable retraction: checked

Fill

- Bottom/Top thickness (mm): 0.4
- Fill Density (%): 15

Speed and Temperature

- Print speed (mm/s): 45
- Printing temperature (C): 210

Support

- Support type: None
- Platform adhesion type: None

Filament

- Diameter (mm): 1.75
- Flow (%): 100.0
One time setup: Load Profile

Under the File menu click on Open Profile... point it to the file called: Simple.Plus.GettingStarted.ini
One time setup: Cura Preferences

In preferences change the Print Window option to Pronterface UI.
First Print!!
Calibration Cube
Process Overview

1. Download calibration cube
2. Load calibration cube into Cura
3. Review slice settings
   a. Layer height
   b. Temps
   c. Speeds
4. Make sure printer is detected
5. Start print
Load Model File

You can load the calibration cube STL file a few ways:
- Double clicking on the .stl file
- Clicking the **load model** button
- File > Load model file …

Model file needs to be in .stl format.
Example: calibrationCube.stl
<table>
<thead>
<tr>
<th><strong>Quality</strong></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Layer height (mm)</td>
<td>0.1984</td>
</tr>
<tr>
<td>Shell thickness (mm)</td>
<td>0.8</td>
</tr>
<tr>
<td>Enable retraction</td>
<td>✓</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>Fill</strong></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Bottom/Top thickness (mm)</td>
<td>0.4</td>
</tr>
<tr>
<td>Fill Density (%)</td>
<td>15</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>Speed and Temperature</strong></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Print speed (mm/s)</td>
<td>45</td>
</tr>
<tr>
<td>Printing temperature (C)</td>
<td>210</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>Support</strong></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Support type</td>
<td>None</td>
</tr>
<tr>
<td>Platform adhesion type</td>
<td>None</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>Filament</strong></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Diameter (mm)</td>
<td>1.75</td>
</tr>
<tr>
<td>Flow (%)</td>
<td>100.0</td>
</tr>
</tbody>
</table>
Make sure the printer is powered on and the USB cable is connected.
Pronterface Overview
## Pronterface: Status Bar

### Print with USB

1. Opening serial port, Temperature: 21 Bed: 20
2. Detecting baudrate, Temperature: 21 Bed: 20
3. Operational, Temperature: 21 Bed: 20

### Pronterface Print Button

1. Printing on COM3
Things to watch for

Make sure the temperatures are what you expect.
The printer homes and calibrates.
Make sure the nozzle doesn’t crash into the bed.
Be ready on the power.
Watch the first layer. Check for good bed adhesion.

If the first layer doesn’t stick or lifts off the bed, cancel the print, home the printer and restart.
If the printer doesn’t seem to be responding: close out of all software and power off the printer. Power on the print and start Cura.
Calipers

Loosen nut if calipers will not move freely
Turn calibers on
Close all the way and press the zero button
Measure object
Fan Shroud

Print the fan shroud from PrintrBot and attach it to your printer.

http://printrbot.com/project/simple-metal/
Print Challenge!
Thingiverse is a vast and wonderful place
Finding an object on Thingiverse

http://www.thingiverse.com/thing:66181
A 3D Printer goes...

Infill is the computer generated paths inside of the geometry.

0% is hollow
95% to 100% is solid

Cura only uses Rectilinear infill
Slic3r also has honeycomb infill
Scale

Click on the **scale** icon

Type a number into **Scale X**

1.0 is the original object size

2.0 is 200% of the size

0.5 is 50% of the size

Click the lock to allow stretching of the part
Click on the **rotate** icon
Locate the circle that represents the desired plane of rotation
Click on the circle and rotate to desired angle
On the bed plane it can be used to better fit large or multiple objects.
Multiple Objects

Right click on the object
Click **Multiply Object**
Type in the number of objects
Cura will limit to the number that will fit on the bed
Objects can be deleted by pressing the “**delete**” key
Drag and drop to add other objects
Print Object Orientation

Usually the best object orientation has already been selected by the creator. Usually the largest flat surface is chosen to be against the bed. Other configurations may be desirable for better print quality or more objects per print batch.
Skirt

Shows the bounds of the print.
Helps you verify bed adhesion.
Helpful for priming the nozzle before the object is printed.
Scaffolding

Scaffolding is removable computer generated print material to aid in printing object overhangs. When part of the object’s geometry exceeds the “overhang angle” scaffolding is added below it. When possible rotate your object to avoid needing scaffolding.
View Modes

Click on the **view mode** icon

Click on **Layer**

Drag the slider at the right to see the print at various stages.

This is useful for seeing infill and printer operations before the print starts.
Shell Thickness

Top turtle: 2 shells  
Bottom turtle: 10 shells  
Shell thickness/ nozzle diameter = # of shells  
4mm / 0.4mm = 10 shells

A shell is one nozzle wide perimeter pass  
Increases wall thickness  
Lots of shells is one method of making solid objects.  
Balance with infill for good strength vs weight.
Layer Height

0.4mm Layer Height
15min 45sec Print Time

0.3mm Layer Height
20min 14sec Print Time

0.2mm Layer Height
28min 15sec Print Time

0.1mm Layer Height
55min 44sec Print Time
Print Troubleshooting
<table>
<thead>
<tr>
<th>Quality vs Time</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Taking too Long</strong></td>
</tr>
<tr>
<td>Cura time estimate</td>
</tr>
<tr>
<td>Increase layer height</td>
</tr>
<tr>
<td>Decrease infill percentage</td>
</tr>
<tr>
<td>Reduce scale (if possible)</td>
</tr>
</tbody>
</table>

**Advanced**

Increase travel speed

Decrease travel speed
Object Selection

Think about how the print will be oriented on the bed. Some 3D models won't be easy to place flat on the print surface.

Example:
http://3d.si.edu/explorer?model id=1336
Watching a Print

If a print fails you can cancel it at any time from Pronterface.
- Nozzle clogged
- Ran out of filament
- Did not adhere to bed
Infill & Top Layers

Increased percentage of infill will increase the top of the print quality. From left to right, 15%, 20%, 25%
Rectilinear infill (squares)
Removing a print

Wait for the print to cool to avoid warping. Use a scraper and pry one corner with steady pressure. Do not hammer or pound on bed because it may damage the printer or you.