3D printing 101

(This document last updated 11/19/2019)

This document accompanies a live training class at Ace Monster Toys. You must attend the class before using the 3D printers. There are two sections to this document:

- The general section has information common to all 3D printers
- The printer-specific sections give necessary detail for each printer

General

There are 3 steps to making something on the 3D printer:

- Get an STL file
- Slice it
- Print it

Get an STL file

A file in STL (Stereo Lithography) format is the input to the 3D printing process. Export from your 3D modeling program to STL, or download a 3D printable STL file from the internet.

3D modeling is beyond the scope of this class. Check the CAD-CAM group on slack; come to a 3D printing hangout; or look into our Fusion 360 workshops. We have some related articles on our blog as well.

Thingiverse.com is the most popular site for sharing 3D printable designs, and is a great way to get started. Look for proven designs that have actually been printed at least once, and download the STL file. Start small! Big models take much longer to print!

Slice

A specialized software package reads the STL file and creates a *tool-path* that drives the 3D printer. We call this *slicing*, since the solid model will be sliced into layers as a first step.

There are a number of slicing programs available, and they all have lots of settings you can adjust. We make specific recommendations in the per-printer sections below. Stick to the recommended defaults for your selected material and printer unless you have good reason to deviate.

There are a few slicing settings that you probably *will* want to adjust:

Material We recommend PLA unless you have a very good reason to use something else. This general purpose plastic is the default material for the slicing software, and is by far the easiest to work with. We have a bit of PET as well. It

requires higher temperature settings, detailed in the printer-specific sections below. We do not recommend ABS due to the toxic smell and warping issues. If you'd like to use another material, please contact the steward first. Some materials cause extra wear-and-tear that we might ask you to pay for.

Quality The *layer height* is the principal setting that affects print quality. Smaller layers give higher quality but take longer to print. Most slicers have quality 'presets' that set layer height and all other quality-related settings to sensible values.

Support If you have overhanging features in your model, you may instruct the slicer to add support material to be broken away after printing. This sometimes works great; sometimes not. Consider redesigning your model to eliminate the overhangs (perhaps by assembling two pieces instead).

Raft or Brim If you have a large part that is supported by just a few thin points, it may come loose from the bed before the job is complete. If so, the 'raft' or 'brim' features will add a breakaway layer to help bed adhesion. If you have a reasonably large flat surface on the bottom of your model, leave raft and brim turned off.

Infill The printer prints a solid skin around a semi-solid latticework interior. The fill percentage sets the density of this latticework, with 100% being solid plastic. Use the following settings as a guide:

15% - light duty
25% - medium
40% - heavy duty

40% is much denser and stronger than you might think; there's seldom any reason to go higher. You'll just be wasting time and plastic.

Print
Printer setup details are in the specific sections below.

All our printers now use an SD card for standalone printing. Please use this method as it is more reliable than printing via USB and frees up the 3d print computer. Put the slicer output file on the SD card for the printer you intend to use. Don't forget to *eject* the SD card before removing it from the computer.

Please leave the SD cards in their respective printers and don't borrow them for other purposes.

The control panels on our printers all work the same. Turn the knob to choose an option, then click the *knob* to select. There is a button below the knob (labeled with an X on the Prusas). Do *not click this button.* It resets the hardware.

When loading a new spool of filament, it's a good idea to unroll 6-8 feet of it first, to make sure there are no 'overruns' that might tighten up and form snags during the print. Then

carefully roll it back up before loading the spool. Check, every so often, to be sure the filament is feeding freely while the job is underway.

Materials
We have material for member use in the black chest of drawers. Check the label to see what the material type is (PLA or PET). You may also bring your own material. If you're planning a large project we'd appreciate this. Check with the steward for recommended suppliers. Our printers use 1.75mm filament.

No unattended printing!
You may not leave a 3D print job running unattended! (Yes, a 5 minute bathroom break is OK). This has to do with our insurance. We know it's an onerous requirement, since print jobs often take a long time. Sorry about that! Bring your book/homework/screenplay/life story to work on, or write up something cool for the AMT Wiki while you wait!

General Troubleshooting
The Simplify3D website has great photos of typical 3D printing problems.

> http://www.simplify3d.com/support/print-quality-troubleshooting/

Their recommendations will have to be translated to the slicer you're using.

Prusa i3 MK2/MK2.5

We have two nearly identical *Prusa i3* printers the *Mk2* and the *Mk2.5*. These are the newest and best printers in our fleet. They are nearly identical in operation.

Heated bed with PEI print surface
These printers feature a heated bed. It is used even for PLA to improve adhesion and minimize warping as parts cool. Keep the bed set at the slicer's default value (60 C) for PLA. The beds have a special PEI surface that does not require blue tape.

Before each print, use the isopropyl alcohol or denatured alcohol to clean finger grease off the bed. You can do so even if the bed is warm.

> ### Please *DO NOT* put blue tape on the Prusa beds!

You should not need to use a glue-stick, or any other "magic sauce" to get good bed adhesion with PLA. If your print job is not sticking, something else is wrong! (see Live-Z below).

The **MK2.5** has a removable sheet steel piece coated with PEI.

> ### Please *DO NOT* print without this sheet in place!

Automatic bed leveling
The I3 has no provision for manually adjusting the bed. Instead, the firmware probes the bed with an inductive sensor at the beginning of the job and adjusts the nozzle position to match. It works like a dream, but you must

Use a slicer configured for the printer!

...so that the required commands are properly embedded in the g-code.

If you fail to do this, the probing sequence will not happen, resulting in a bad print (best case), or a print-head crash that will at least damage the PEI surface and might even damage the nozzle or printer mechanism.

Slicing

Prusa has created a new slicer called *PrusaSlicer*. It's a rewritten and updated version of the open-source *slic3r* program,. Most importantly, they have created nicely-tuned profiles for their printers. Select the *quality profile*; *filament type*, and *printer* on the *Plater* tab.

Be sure to choose the printer profile that matches the printer you're using.

> *Original Prusa i3 MK2.5*
> *Original Prusa i3 MK2S*   *(ours is not the 'S' but use this profile anyway.)*

This program is installed on our 3d print computer, or you may download it to your own computer.

> http://www.prusaprinters.org/introducing-slic3r-prusa-edition/

Printing

Load/unload filament
First, select *Settings/Move Axis/Move Z* and turn the knob until the nozzle is a few inches above the bed.

Find the *Preheat* item for the filament you're using and click it. Wait for the nozzle to come up to temperature.

Click the *Load/Unload filament* item. It will tell you to preheat if you forgot. Then it will prompt you through the rest.

Start the job
Select *print from SD* and choose your g-code file The printer will warm up first; then probe the bed in nine points and start printing your job.

While the print is happening

The panel will show percent-finished and elapsed time. Use this to estimate overall time.

You may adjust the speed by turning the knob while the main status screen is displayed. By default it's 100% of the speed selected in the slicer. For simple parts you may be able to push the speed a bit faster; for parts with a lot of short head movements, you may want to slow it down to minimize shaking.

There are a few other things you can tweak *while the job is running*. This may enable you to rescue a job without having to reprint the entire thing.

If the first layer is not sticking well, try the *Live Adjust Z* item to tweak the first layer thickness. The manual has excellent photographs of what to look for in section 6.3.9.

The *Tune* menu allows you to adjust speed, temperature, extrusion flow, etc. on the fly. You can change filament on the fly too; Nice if you want to use up that last bit of filament on the spool.

Removing your model

On the MK2 The bed is **not** removable. Carefully use a polished spatula to loosen the model from the bed. Try not to dig into the PEI surface... we want it to last as long as possible.

On the MK2.5 The bed top layer is a spring-steel, PEI covered sheet that's held on with magnets. It may be removed and gently bent to help remove stubbornly stuck parts. ***Do not over-bend! It's easy to permanently damage this sheet!*** Don't expect all parts to simply 'pop off' without using your fingers; small parts, in particular, often require fingernails or the polished spatula.

Manual

The *Prusa* manuals are excellent. We have a hard-copy in the filament drawer. You can also find it online at: http://www.prusa3d.com/drivers/.

Multi color printing!

Well... sort of. The Prusa website has a utility that will insert a pause to change color at a particular layer. You must prepare your job properly first; see the manual. It's fun!

Troubleshooting

If the first layer doesn't stick properly
      stop the job
      clean the bed
      Select 'Calibration/First Layer Cal.' (This replaces the V2calibration.gcode file)
      and tune the first layer with the "Live Adjust Z" feature as described in the manual
      section 6.3.9.

If the machine is badly out of alignment, contact the steward. You may also perform the

calibration steps 6.3.5 and 6.3.6 in the manual. Please let the steward know if you found this necessary.

Type A Machines Series 1

We heavily modified the Series 1 in 2016 and then again in 2017 to make it more reliable and easier to use. It now features an upgraded hot-end; improved cooling fans; an LCD control panel with SD card reader, and automatic bed leveling. These upgrades make the operation very much like the Prusas, but there are a few differences.

Slicing
We have added a profile for our Type-A to the Prusa-Edition Slic3r for use with the Type-A. Use Slic3r as described for the Prusa, except:

> for printer profile select "AMT Type A"
> for filament select "Generic PLA"

The printer profile is available on the wiki if you'd like to install on your own machine.

Printing
The bed must be covered with blue tape just like the Replicator 2. This is a little more difficult to do since the bed cannot be removed.

Changing filament (READ THIS!)
There is an LCD menu item for changing filament, but believe it or not, *it disappears when the nozzle is cold!* You have to preheat first. Click Prepare/Preheat [ABS or PLA]. When the nozzle reaches temperature you'll be able to select Prepare/Change Filament. (They did not consult *me* on this stupid user-interface… someday I'll fix the code in our fork).

The LCD will then walk you through the process. Depress the lever on the extruder and make sure the end of the filament is inserted into the aluminum block just below the pinch roller. When prompted to 'click the button' depress the selector knob; **not** the tiny reset button below it!

Auto leveling!
The Type-A now supports auto leveling much like the Prusa, but the LCD menu is slightly different. To fine-tune the nozzle offset, look for Tune/Z Offset: while the first layer is printing. The value may also be adjusted before the job starts via Control/Motion/Z Offset.

Unlike the Prusa, this setting will not persist beyond a power-cycle. It may be saved by clicking Control/Store Settings, but please discuss this with the steward first. This value seldom needs to be changed.