MakerBot: 3D Printing

- Makerbot: 3D Printing
- Makerbot: Changing the Filament

Makerbot: 3D Printing

Needed Materials :

- A file of what you would like to print. (Preferably an STL file.)
- A USB storage device. (Preferably formatted to FAT32)
- A lot of time.
- Access to our Makerbot Cloud Print service (see a SCiL staff member)

Get the Model:

Specifically, to begin the process, you should attempt to get an STL file. Other forms of 3D model files may work, but an STL works best. You should also try to make sure the model itself is set to the correct sizes. The easiest way to do this or transfer from OBJ or another file type to STL is to open your ile in a 3D modeling software, such as Blender or Maya, confirm its size and details, then to export it as an STL from there. Once you have the correct file type and you have opened Makerbot Cloud Print and determined it's functional, you are free to proceed. Do not worry about model size yet, as Makerbot has a scaling feature, but you do need to be sure proportions and details are correct.

File Preparation:

This section below is now outdated and must be rewritten! We now use Makerbot Cloud Print and below uses the Makerbot software that is deprecated.

To prepare the file in Makerbot, once you are signed in, simply click the icon that looks like a folder in the upper left corner. Then hit the plus icon and import your STL or other model file. Once the model is loaded, click on the icon on the left that appears to be a box inside a box. This is the scaling option, and allows you to make your object the correct size. Then, going through the options will allow you to orient the print, arrange the print plate, and prepare the print. Once you have everything placed then choose the appropriate options in the settings, such as if it should have supports or not.

Once you have all the seeing in order, click on the icon that looks like a clock with a play icon to prepare the print. After that, export the makerbot file with the export button and save your work.To do so, you may need to select the kind of printer to be used if you haven't already. The SCIL labs model of printer is a Replicator+, and you'll need to select it as an offline or manual option. Finally, transfer the makerbot file, not the print file, to your USB storage device, and you are ready to use the 3D printer.

Printer Operation:

Follow these instructions as closely as possible, and do not mess with other settings. Although improving the extruder speed or messing with the fan may sound like a way to speed up the process, it will more than likely result in your print turning into a blob or warping horribly. If you follow these instructions and have issues, or wish to alter something such as the color of your print, please find someone trained with the printer to help you.

1. Clear the plate.
    1. There may be debris or plastic from previous prints left on the printing plate. It is recommended before printing to make sure this plate is clear.
2. Turn on the printer.
    1. The power button is located on the back left of the printer. If there are no lights on at all, find and press this button. If there are lights on but the screen is not on, just click the knob to bring it out of sleep mode. Be patient, the printer may be slow starting up or coming out of sleep mode. Once the screen is on, proceed.
3. Plug in the USB Storage.
    1. The slot is located to the right of the selection knob. Make sure the drive is plugged in and wait a moment before proceeding to give the printer a moment to register it was plugged in.
4. Select Print.
    1. It should be the upper-left most icon.
5. Select from USB storage.
    1. It should be the first of the two options.
    2. Be patient, it takes the printer a moment here to retrieve the files from your storage. Proceed when the screen changes.
6. Select your print.
    1. Navigate to your print through your files and select the file you would like to print. This should give you a preview of your print and a time estimate.
7. Select print.
    1. Select the print option and the machine will automatically calibrate itself and begin printing.
8. Wait.

1. It is not uncommon for prints to take longer than an hour to print. Please be patient or find something else to occupy your time.

Clean-up:

Once you have printed your file, you may need to clean it up some. The “Raft” and supports on your print can be removed by hand, though tools are available. You may use the maker space in order to fully clean up your project. I would recommend using files as needed and many prefer to use the exacto knives and other tools present as well. You may need to find an SCIL member or have access to the SCIL key in order to access many of the tools. Be careful when cleaning up your model, the plastic can be brittle and break easily if you handle your model roughly, especially if it has thinner parts. Once you are finished, please remember to clean up the area before you leave. Thank you.

Special Notes/Advice:

- Currently there is an issue with larger prints warping. It is recommended to keep your prints width and depth to approximately 5 cm or lower. Vertical or height has no effect on the warping and you are free to print tall as long as it is within the height limit.
- The variance of the plastic is about 1 mm. Keep this in mind when making your print, as this means any part may shift up to a mm or the model overall may shrink a mm once you have finished printing.
- For those looking to reduce the amount of time on a print, or to reduce the amount of support bridging on a print, you can safely lower support density to 5% in the settings, and raise the support angle to 80%, to do so.
- Any part that hangs about half a cm or more from the main print likely requires support.
- It is recommended when preparing the print to set the widest and flattest part of the print on the bottom. The printer tends to dislike “legs” and things standing on them.
- If your print will take more than 4 hours, it is recommended to have it print overnight as a courtesy to others. If a print will take more than 16 hours, it is then recommended to print it over the weekend instead.
- Remember, the plastic shrinks when cooled. This size difference can range from .2 mm to .8 mm, so keep that in mind when designing parts.

Troubleshooting:

-If your USB storage device cannot be recognized, then it may be formatted in a way the printer cannot read. Try getting a new drive or reformatting yours. FAT32 seems to be the smoothest

functioning storage format.

Makerbot: Changing the Filament

When the Filament has run out, or if you would prefer to print in a different color, you need to change the Filament reel. Here is how.

1. Remove the Old Reel:

The first step to removing the old reel is to make sure the 3D printer is turned on.

Then, using the release button, lift the section containing the reel. It should be located at the back of it, and is easiest and most safely reached from above the machine, not through the printing area. You may need to push and hold the button or small lever next to the handle in order to lift it properly. Make sure when it's lifted fully it locks in place and will not fall back down before releasing it.

After that, using the menu on the machine, choose the options for the filament and choose the option to unload the filament.

Once the machine has finished disconnecting the filament from the extruder, slowly rotate the reel clockwise to remove the filament still in the machine and wind up the reel.

The peg the reel is on should have two small nubs keeping the reel on it. Pinch these nubs and slide the Reel off the peg.

Store the remaining filament in a bag in an appropriate location.

Adding the New Filament:

First, now that the peg is clear and while the filament section is still in the open position, select or acquire a new reel of filament.

Locate the free end of the filament, and then put the new reel onto the peg in such a way that the reel rotating counter clockwise will feed it into the extruder. There should be a small click as the nubs from earlier move in place to keep the reel attached.

Now, make sure the hose that the filament will travel through is plugged into the top of the extruder.

Then, slowly feed the free end of the filament into the filament slot until you feel some level of resistance. This slot on our extruder is outlined in red or has a red 'mouth' for the intake. You should be able to see the filament going through the tube and into the extruder before feeling the resistance that stops it.

Next, using the printers menu options, go into the filament options and select load filament. If the device does not automatically begin feeding filament into the extruder, then gently feed the filament into the mouth or intake until it does so.

When the filament begins to come out of the extruder, you may stop the loading procedure.

Then carefully lower the filament section back into it's normal, closed position.

Clean the printing plate and you are ready to print again.

Troubleshooting:

Below are troubleshooting tips in no particular order.

If you are having trouble feeding the new filament, check the end of it that would be going into the extruder. It may have melted, deformed, or simply just be slightly off. Trimming the end and possibly cutting it or trimming it at an angle so that is has a point feeding into the extruder has been known to fix the issue and help feed the new filament into the extruder.