



3D Printing at UNB Saint John

GENERAL

We have three 3D printers available for use in the HWK Commons: the MakerBot Replicator 2 and 2 MakerBot Replicators 5th Generation. 3D printers heat a type of plastic filament (we use one called PLA) into liquid form, then lay it down in thin layers that build up into whatever shape you want to print.

You can either make your own design or download one from the internet. You can also modify a design that you download (though you should check what copyright claims are on it). The 2nd generation printer accepts SD cards and the 5th generation printers use USB or network printing.

Printing is self-serve and on a first-come basis. If your print job is left in the printer, it may be removed and kept at the Student Technology Desk to allow others to use the machine. Please note, however, that ISS and Library staff are not responsible for unattended jobs.

DESIGNING

If you want a place to start from or think the thing you want to print may already exist, check out [[Thingiverse](#)]. You can use the search system here to look through thousands of 3D printer designs and download them.

The simplest site to design on is [[Tinkercad](#)]. You can make an account and work through the lessons it provides to learn the interface, watch the tutorial video on the homepage, or just dive right into the designing process.

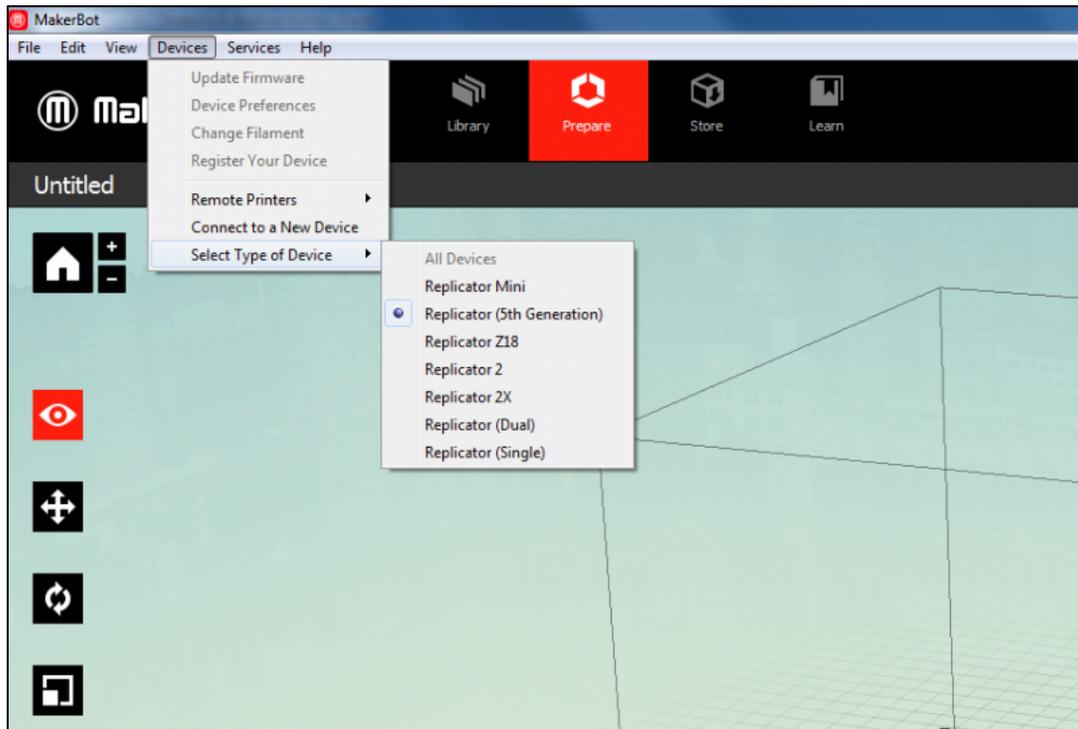
If you know how to use SolidWorks, that is also a good option for making designs.

STEP-BY-STEP PRINTING

MakerBot Print (previously MakerBot Desktop) is required to convert the files to a MakerBot compatible format and also to preview or make building changes to your design. This is free software you can download from the [manufacturer's website](#).

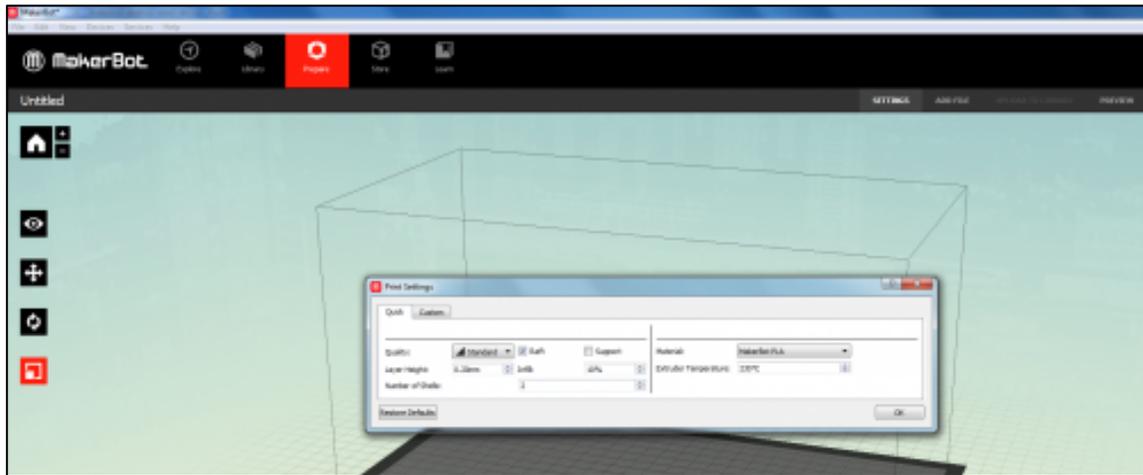
The follow instructions are for the version of MakerBot Desktop available for use in the HWK Commons. The wording may vary slightly if you have downloaded Makerbot Print on your personal computer.

- Open your finished design file in MakerBot Desktop, either by double clicking the file itself or by opening MakerBot Desktop from the Start menu and then clicking **File > Open**.
- Make sure you select the printer you're going to be using by clicking **Devices > Select Type of Device** from the top menu.



- From here you can view how the object should look when printed and adjust the print settings. Most of these can be left in the presets, which are set in **Low**, **Standard**, and **High**. The higher the quality, the longer it will take to print.
- The controls along the left side allow you to change your view of the print, move objects around, **rotate** objects, and **scale** objects. Rotating can help you to place your design on the side where it will have the most support. Be careful when scaling--a smaller or larger size may make the design less stable and require it to be modified.
- 3D prints work best if they have a solid base to work from. Designs that have precarious parts (imagine a model dragon's wings, for example) or not much of a flat bottom have a high chance of failing to print properly. The solution to this is **rafts and supports**. A raft is a thin layer of plastic on the base of the object, and a support is a thin layer that connects everything together (these would support our example dragon's wings). Both of them will snap off relatively easily after printing. You can adjust whether or not your design has these in **Settings**.

- Because of the layering nature of the printing process, the interior of prints are often close to hollow so as to use less material and take less time. You can select how much “**Infill**” there is and what pattern it follows in settings as well.



- When you're happy with your settings, click **Preview**. In the top right, it will tell you how much material the print will use and how long it will take to print. You can use the scroll bar on the left to simulate the printing process layer by layer and more easily see where your design could fail to print. Make sure to check out every angle, especially if the build has failed once and you are trying to troubleshoot the problem.

If using the Replicator 2:

- Take the **SD card** from the front right part of the printer and plug it into the computer.
- Export your file from MakerBot Desktop to the SD card. It should be in **.x3g format**.
- Put the SD card back in the printer. Select **Build from SD** using the arrow pad, then select your file. It should begin printing. If you don't see your file as an option, check the file extension.
- To cancel, hit the **center of the arrow pad** to bring up the menu and navigate to the **Cancel** option.

If using the 5th gen:

- Click **Devices** on the top menu of MakerBot Desktop and select **UNB Saint John MakerBot Replicator**.
- It should appear at the bottom bar of the screen and say **Ready to Print**. You can click **Monitor** to see a camera view of the print surface.
- Click **Print** at the top right to send the job to the printer.
- If you don't see it as an option, or you are on a personal computer, you can export the file from MakerBot Desktop onto a **USB drive** and plug that into the printer itself.