Notre Dame Design Deck
3D Printing Tutorial

N3D Safety Code

All posted safety guidelines must be followed at all times.

Introduction

This tutorial walks through the printing of an already created 3D Model using a MakerBot 3D Printer.

Installing MakerWare

- Go to www.makerbot.com/makerware/
- Select your operating system. MakerWare supports Windows 7, 8, 8.1, XP, Mac OS X 10.6+, and Ubuntu Linux. For Windows, you have the option of a 32 or a 64 bit version.
- Follow the directions to download and install the software.
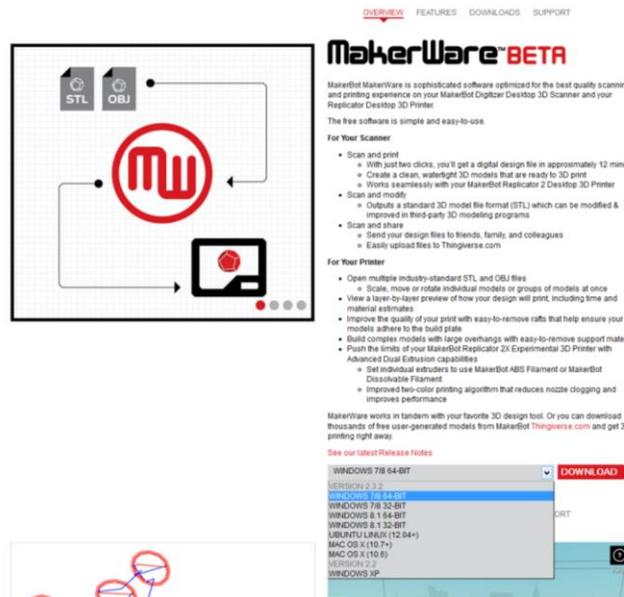

Open MakerWare
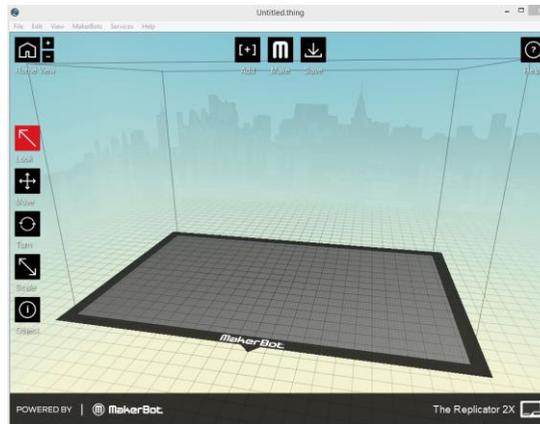
Importing Model

- MakerWare accepts .stl (Binary or Ascii), .obj, and .thing files. Most 3D Modeling programs can export in one of these formats.
- There are several methods to open a model. Choose one of the following:
  - Select the 'Add' button at the center of the top of the screen.
  - Go to File -> Add
  - Go to File -> Open

- The difference between Add and Open is that Add allows for opening multiple models at the same time, while Open replaces whatever you currently have displayed.
- Next, select your desired model. MakerWare opens a preview view of the print bed, along with your model.
- Unless specified by the file, MakerWare assumes dimensions are in millimeters.
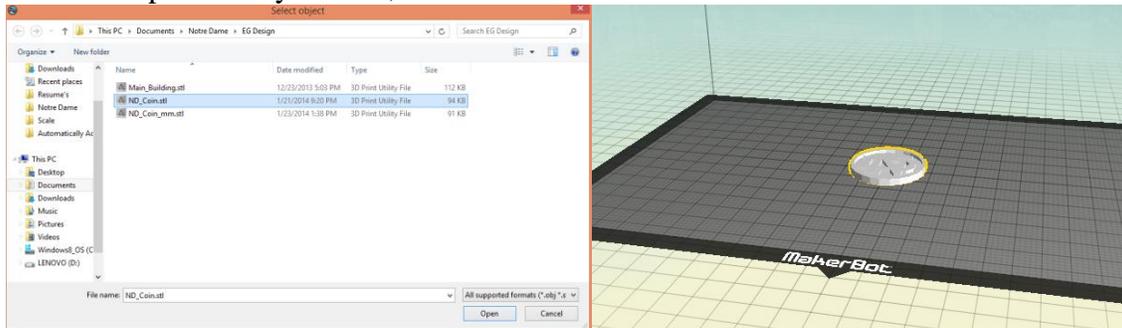
- If you wish to adjust your model, you can use the commands on the left side of the screen to do so. These include:
  - Move
  - Turn
  - Scale
- To use these, click on your desired command. It will turn red. You can then either click and drag your mouse to modify the model by hand, or you can click on the command again, and a popup menu will appear in which you can type in exact values.
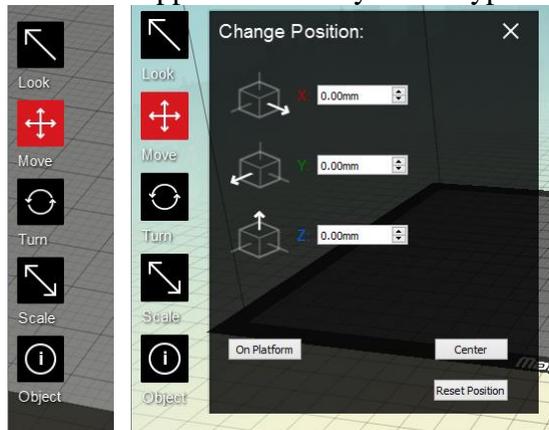
- The location of where the model is printed generally does not matter. However, orientation and scale are particularly important. The proper orientation means that less (or

no) support material is required for the print. The scale is also critical, as details which are too small will not be printed correctly.

- You can also modify the camera with which you view the model.
  - Right click and drag to rotate
  - Rotate the scroll wheel to zoom

Printing the Model

- All of the previous tasks can be accomplished without connecting the 3D printer. Now we need to do so.
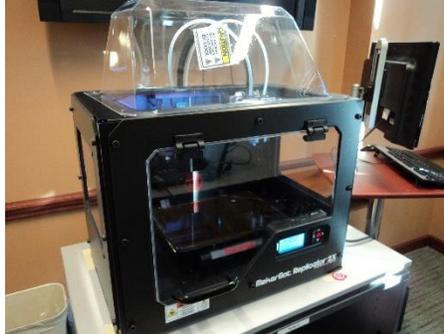
- First, plug the printer into an outlet. This does not automatically power the machine on. You need to flip the switch on the back of the printer (Next to where the cables connect)


- The machine will power on. Next, connect the USB cable to your computer. At this point, MakerWare should automatically detect which MakerBot is connected. However, you can verify this by going to the MakerBots menu at the top of the screen, and going to the 'Type of MakerBot' submenu. If you aren't sure which MakerBot you're using, it should be printed on the front of the machine. All of the MakerBots essentially function the same, the key difference being whether there is a single or a dual extruder.
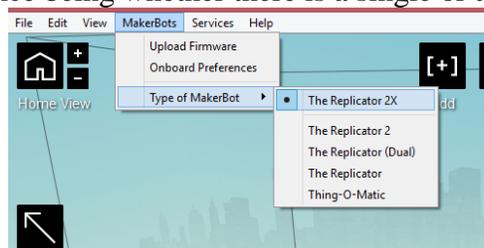

- If using a dual extruding MakerBot, you can click on the 'Object' menu on the left and select which extruder is used to print the model. Note: 'Right' vs. 'Left' refers to the extruders' respective positions when viewed from the front.
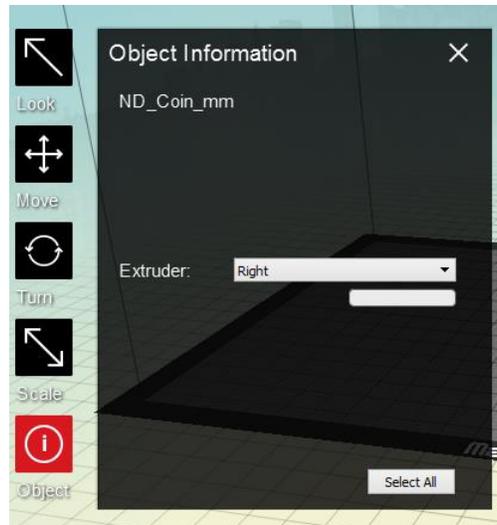
- Next, click the 'Make' button at the center of the top of the screen. A popup appears.
- There a lot of options here that we can tweak to get our print exactly the way we want. Typically the default options are fine however.
    - Again, you have the option to select which MakerBot you are using.
    - You next can select which material you are printing with.
        - If using the MakerBot Replicator 2 (With a single extruder), this will be PLA.
        - If using the MakerBot Replicator 2X (With dual extruders), it will be either PLA or ABS. You can select a material for both extruders.
    - Raft: If this is turned on, a thin layer of plastic is printed under the model. The advantage is that your print is less likely to slip from the print bed and fail. The disadvantage is that additional time and resources are required. Additionally it can sometimes be difficult to remove the raft from the model.
    - Supports: Because the 3D printer starts from the bottom and prints upwards, there can be problems with the process if there are overhangs in the model (Portions of the model which do not have anything beneath them). The solution is for the printer to insert support material. The support material can be extremely difficult to remove. As a result, it is better to design your model with this in mind, or to rotate your model in such a way that minimal or no supports are necessary. However, if required, you can turn on supports here.
    - Resolution: This determines how quickly your model prints, and with how much detail. Standard resolution is typically preferred.
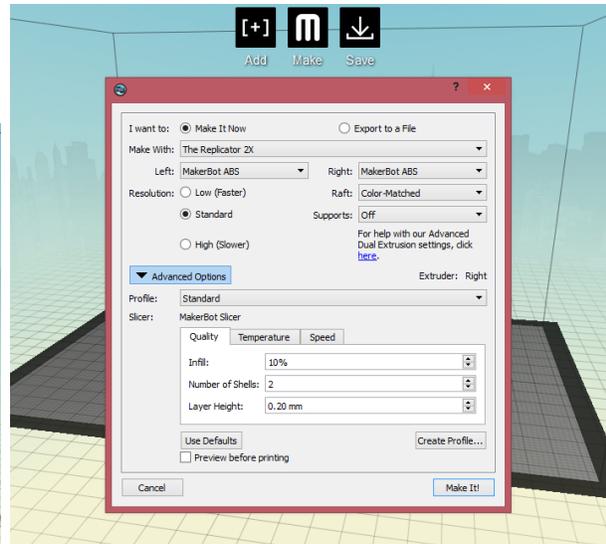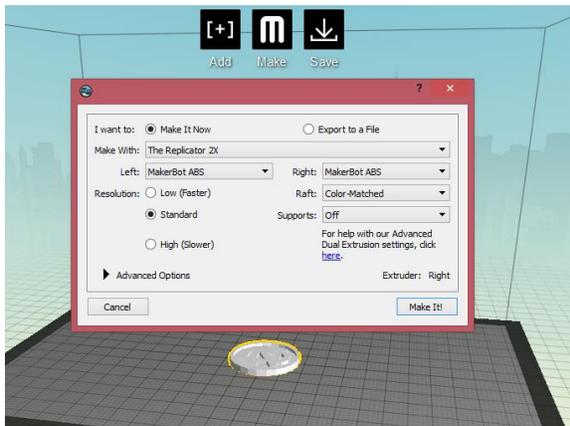
- There is also an Advanced Options menu. Here, you can manually tweak settings such as the temperature of the extrusion, the speed, and the amount of infill (The density of the print, where 100% is solid and 0% is hollow). These options affect the speed and quality of the print, and are automatically calibrated when selecting Low, Standard, or High Quality. They typically do not need to be modified.
- Once you have these options how you wish, you can press 'Make It!' to begin the print.
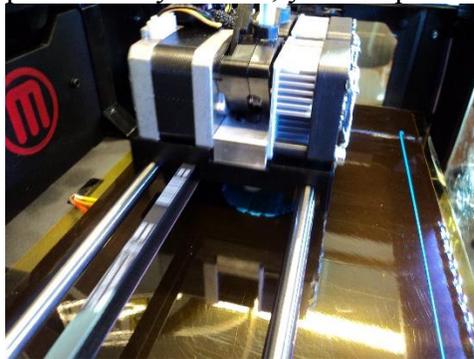
- A progress bar will appear in the lower right-hand corner. You can click on the X to cancel the job. Another method to do this is to press the left control pad button on the actual MakerBot, and then select 'Cancel Build.'
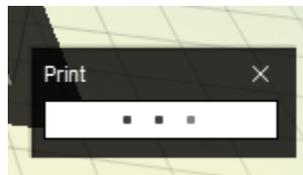
- The printing process itself is slow. The ND Coin created in the SketchUp tutorial requires 20 minutes to complete on Standard resolution.
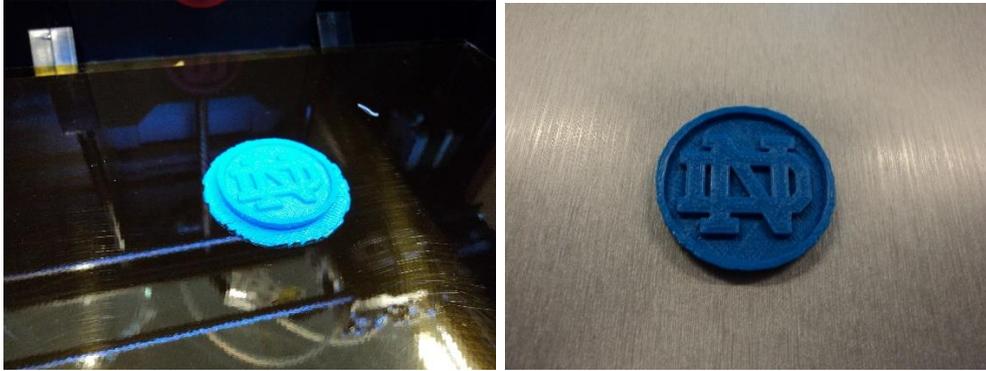
Tips on Using the MakerBot

- All interaction with the MakerBot is done using the control pad on the front of the machine in conjunction with the display next to that. The control pad consists of 5 buttons: 4 directional buttons, and a select button.

- Navigation through the menus is done in a similar way to an iPod. The up and down arrows scroll through the options, the center button selects whatever the cursor is on, and the left button returns you to the previous menu.
- Using the menus, you can accomplish tasks such as changing the filament (If you wish to print with a different color of plastic), calibrating the machine, and more. After selecting one of these tasks, the display of the MakerBot will guide you through the steps required to complete it.

Troubleshooting

If you are having difficulty with a print, there are several things that you can attempt to resolve the issue:

- Print using a raft if you are not
- Change the temperature or speed of the print
- Modify the design, orientation, or scale of your model to make the print simpler to complete.
- Some 3d models contain errors (such as intersecting polygons) that cause the print to malfunction. You can use the Autodesk 3D Print Utility, a free third party application which prepares a model for printing to automatically repair your model (http://apps.123dapp.com/3dprint/install.html)

If none of these steps improve the print, then it is likely that the printer needs to be recalibrated. You do this, you need to level the print bed. This involves ensuring that the print bad using a guided process on the MakerBot, which can be found under Utilities -> Level Build Plate

Deliverable

Design your own object roughly the same size as the ND Coin created in the SketchUp tutorial (Maximum: 1.35" x 1.35" x 0.125"). The object should show some proficiency with SketchUp or other 3D modeling software. Email the instructor (ksonnen@nd.edu) your model as an STL file, or an image of your 3D printed object, with the subject line "EG Design 3D Model Submission".