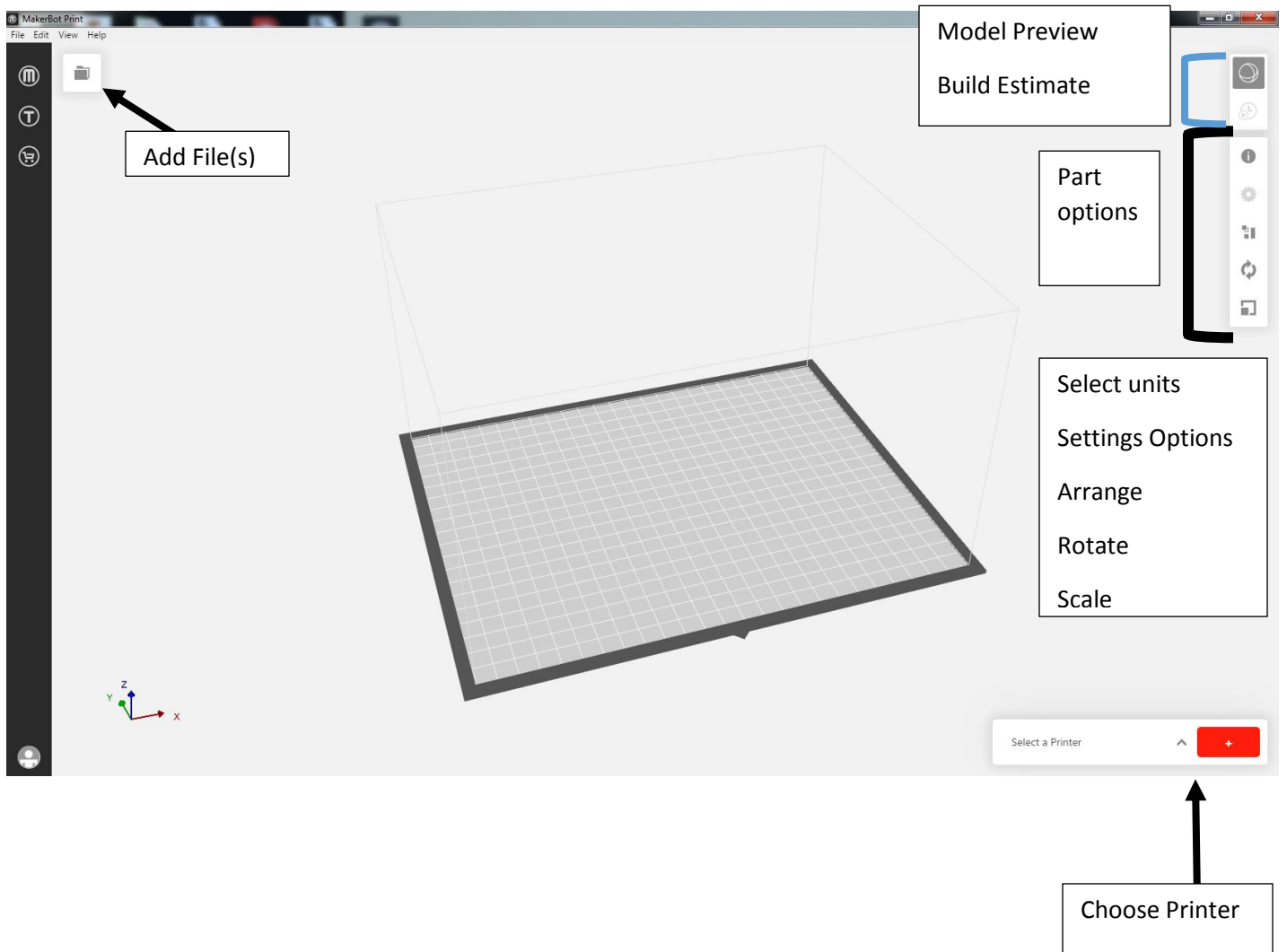


Makerbot Print Software

By: Matthew Jourden

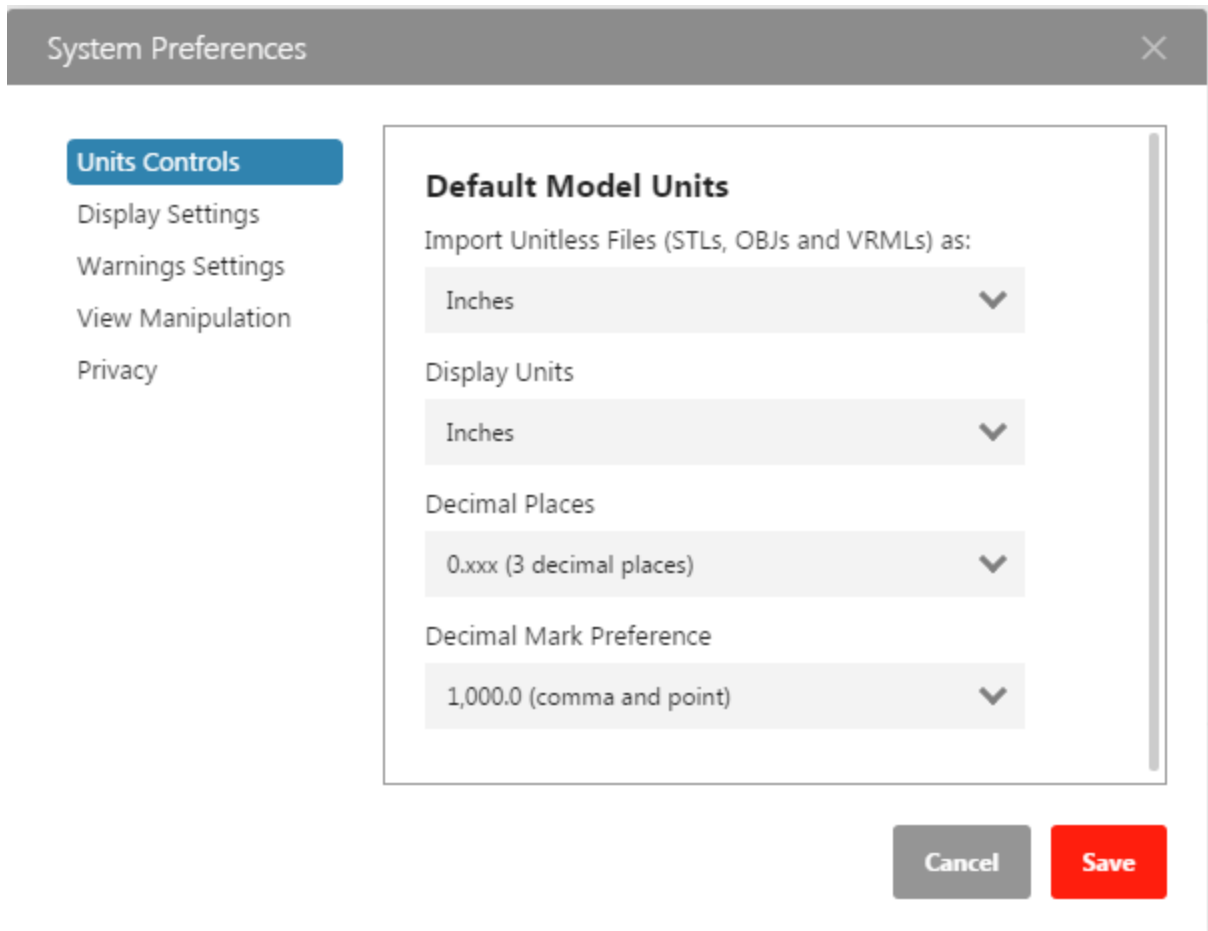
Brighton High School

1. Open Makerbot Print from the Desktop
2. Upon opening you will be required to login. You may make your own or use the following
 - a. Username: mjourden
 - b. Password: bhs
3. Screen



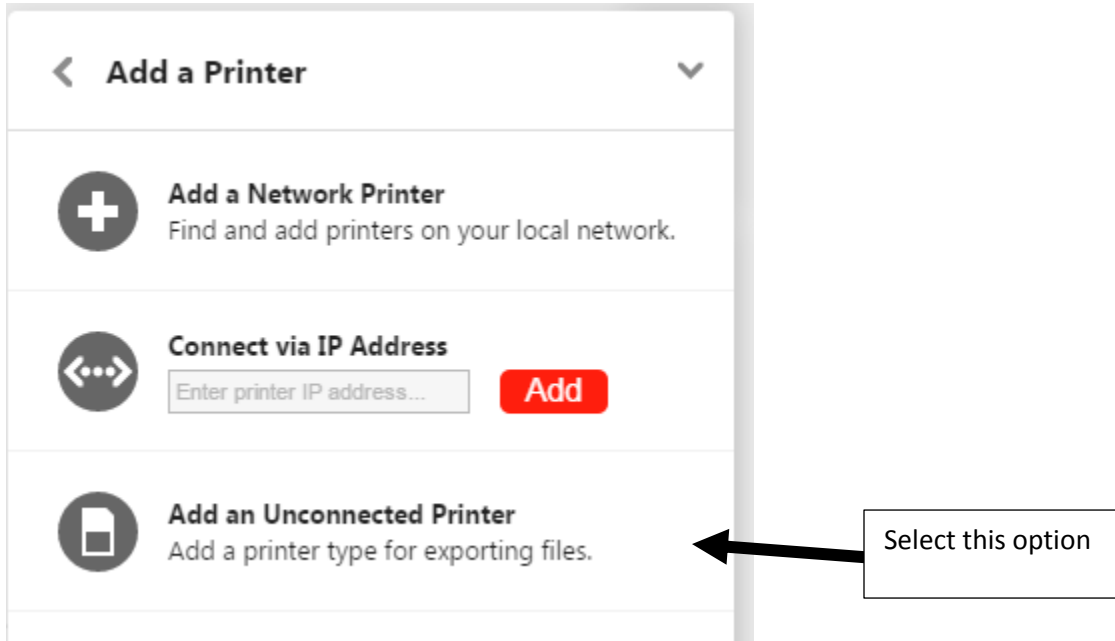
4. Changing Units.

- a. File > System Preferences >
 - i. Import Unitless Files = Inches
 - ii. Display Units = Inches
 - iii. Click Save

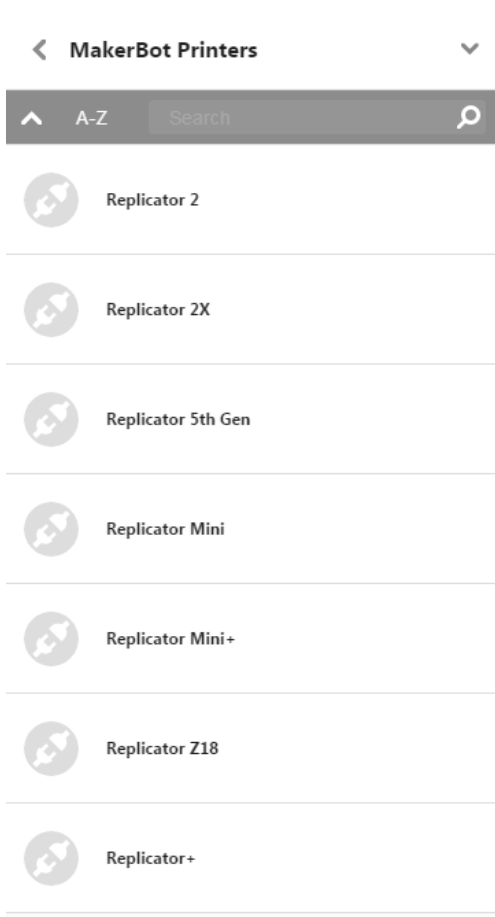


5. Choose a printer

- a. Printer Selection > Select the + sign in the lower right corner of the screen > pop up menu will appear

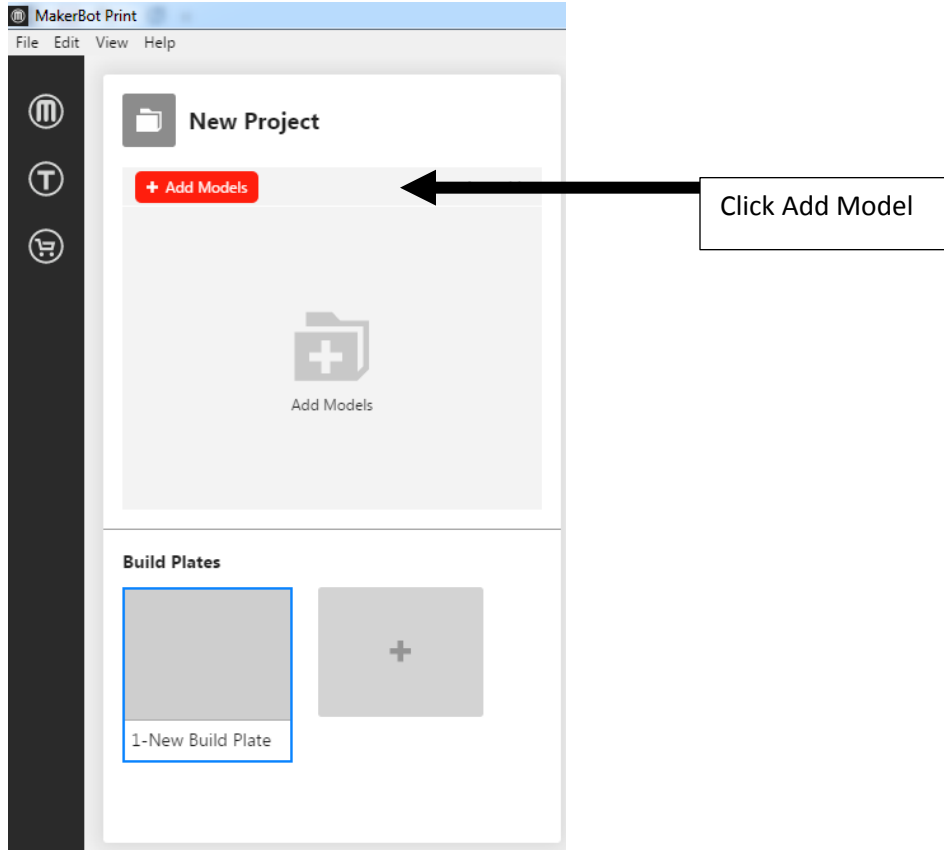


- b. Pop-Menu will appear > Select the desired type of printer
NOTE: BHS ONLY HAS REPLICATOR 2 and REPLICATOR 5th GEN



6. Adding a part

- a. Select the folder in the top left corner of workspace > Click Add Model > Navigate to the .stl and load it on the build plate



NOTE: May add more than one part on the build plate

7. Move, Rotate, Scale part within the build volume and place on plate

NOTE: Make sure the part is on the build plate

8. Settings

5th Gen must use the Smart Extruder+

Replicator 2 will not have this option

Select Settings

Print Settings

Extruder Type

Smart Extruder+

Print Mode

Balanced

Support

Settings affect all models in project

Add a Custom Setting

Default Settings

Click Add a Custom Setting to be able to change various parts of the print (I.E Shells, Infill, etc.)

Advanced Print Settings

Quick Settings

Printer

Extruder

Rafts

Shells

Infill

Floors

Supports + Bridging

Raft + Base

Added:

Done

Number of Shells

Adjust the number of outermost layers of the model while printing.

Increasing the number of shells creates thicker, stronger walls but slower prints.

Raft

Turns rafts on or off.

Rafts help models adhere to the build surface and create help create an even first layer. Printing without rafts can cause models to curl up and/or become dislodged from the build plate during printing.

Infill Density

Checking the box will add the ability to change the default setting for that file

Common Setting Changes

Shells

1. Number of Shells
2. Layer Height

Supports + Bridging

1. Support
2. Support Angle
3. Support to Model Spacing
4. Support Density

Raft

1. Raft On/Off

The screenshot shows the 'Print Settings' panel with the following settings:

- Extruder Type: Smart Extruder+
- Print Mode: Balanced
- Support
- Settings affect all models in project
- Quick Settings (expanded)
- Layer Height: 0.2 mm - Balanced Detail &...
- Number of Shells: 2 Shells - Recommended
- Supports Bridging (expanded)
 - Support Angle: 68°
 - Support Density: 20 %
 - Support to Model Spacing: 0.4 mm
- Rafts (expanded)
 - Raft

At the bottom of the panel is a button labeled 'Add a Custom Setting'.

A large black bracket on the right side of the panel highlights the 'Supports Bridging' and 'Rafts' sections. An arrow points from a text box labeled 'After checking the boxes' to the right side of the panel.