

Makerbot Print Quick Reference Guide

Makerbot print software: basics_ 2
Menu bar_ 3
Printer model panel_ 3
Custom print mode settings _5



CENTRE for
INNOVATION and
ENTREPRENEURSHIP



1. MakerBot Print Software Basics

Login to a computer near the 3D Printers, select the MakerBot Print software from the desktop and ask a Creative Technologist for the Username and Password.

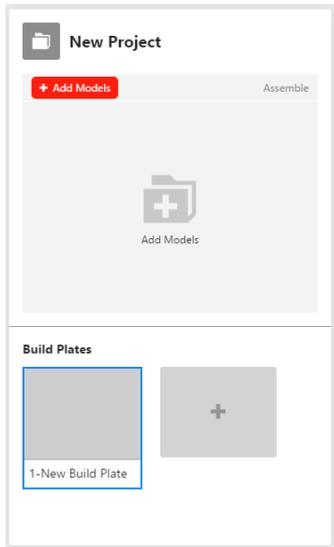


Figure 1: New Project Dialogue Box

i.) Open up a 'New Project' by loading a '.stl' file into the MakerBot Print software shown in Figure 1 above. Clicking

either  or  will open up the 'Insert File' window. Locate your '.stl' file (whether you saved it in a USB Drive or in the computer), this will upload your design onto a Build Plate shown in Figure 2 below.

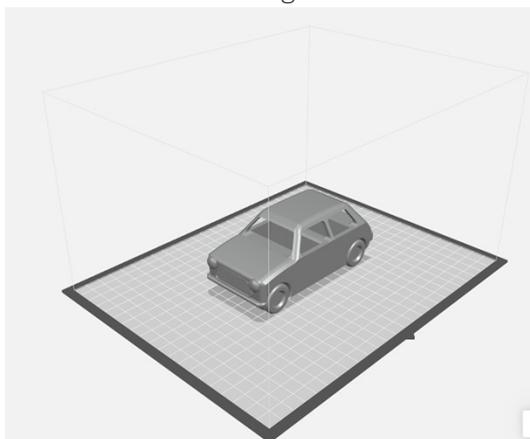


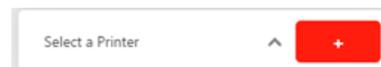
Figure 2: Car on Build Plate

ii.) Tweak the settings using the Print Model Panel shown on the right side of the software. The Print Model Panel (Using MakerBot Print, n.d.) and short descriptions are shown in Figure 3 below.



Figure 3: Print Model Panel and Descriptions

iii.) Select a Printer by clicking the



button.

This will open the Printer Panel shown in Figure 4 below. Make sure both MakerBot Replicator+ 3D Printers are on (Plugged in and switched on). Hovering over one of the Printers in the Printer Panel will allow you to select a printer, click the 'Select' button to use a Printer.

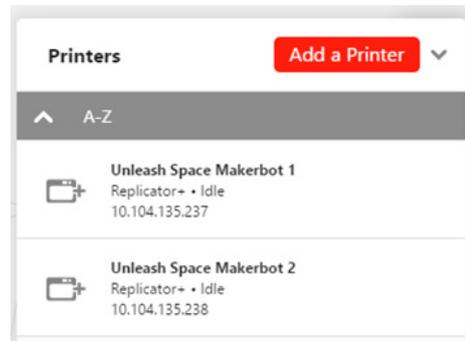


Figure 4: Printer Panel

iv.) Estimate how long the 3D printing will take by clicking on the 'Estimates and Print Preview' button from the Print Model Panel. An example of the Print Preview Dialogue box is shown in Figure 5 below.



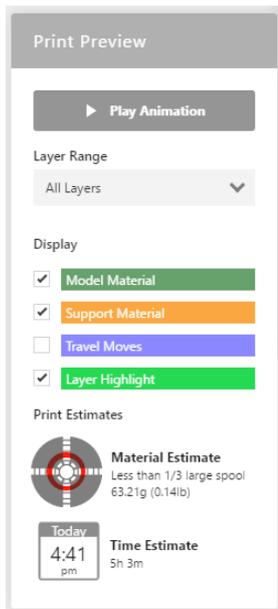


Figure 5: Print Preview Dialogue Box

- v.) Print by clicking on the red 'Print' button shown in Figure 6 below.

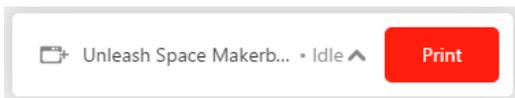


Figure 6: Print Button

2. MakerBot Print Software: Menu Bar

Every software consists of a Menu Bar, this is indifferent for the MakerBot Print software. This section will briefly describe each button. The Menu Bar is shown in Figure 7 below.

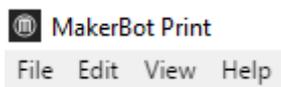


Figure 7: MakerBot Print Menu Bar

- i.) **File** – Allows you to create new projects, open and save, projects, insert and import files, and access system preferences. If you have a multi-file STL assembly, select File > Insert STLs as Assembly.
- ii.) **Edit** - Allows you to undo an action, redo an action, as well as, copy and paste objects.
- iii.) **View** - Allows you to view your model from an array of 2D views, 3D views, and multiple perspectives.

- iv.) **Help** - Enables Help with MakerBot Print and allows you to check for updates.

3. MakerBot Print Software: Printer Model Panel

The Print Model panel was briefly covered in the Section 1

(ii.). This section will cover each of the Print Model panel buttons shown in Figure 8 below.

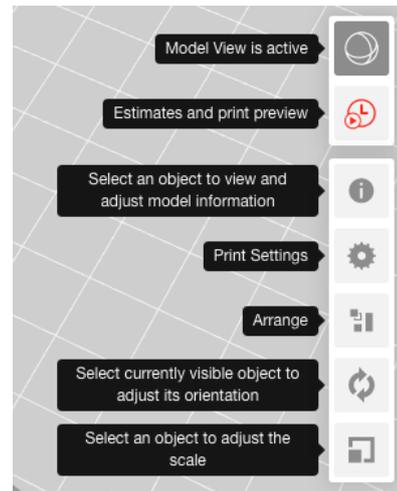


Figure 8: Print Model Panel

- i.) **Model View** (shown in Figure 9 below)



Figure 9: Model View Icon

- a. Prepare and arrange your 3D models on the build plate.
- b. Click and hold the right mouse button and move the mouse to be able to view your model from any angle.
- c. Press and hold the **Shift** key and the **right mouse button** together to move the build plate in space.
- d. Zoom using a track pad or the scroll wheel on your mouse.
- ii.) **Slice Preview** – Slices the 3D model into layers so that you can view the path the Smart Extruder+ tool will make while printing the model. The Print (Slice) Preview Dialogue is shown in Figure 10 below.



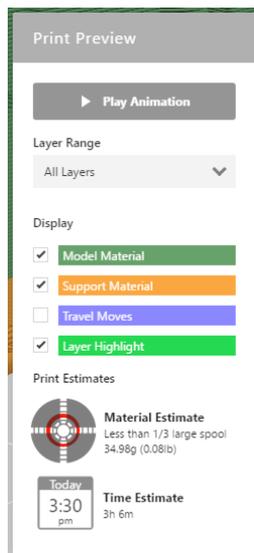


Figure 10: Print (Slice) Preview Dialogue Box

- a. **Play Animation** shows all the layers that make up the sliced model and animates them. Play Animation shows you how your 3D model will be printed, and is a good tool to use to confirm that your file does not have errors
- b. **Layer Range** allows you to view sliced the model up to a certain layer. You can slice the model and view all layers, up to a certain layer, a single layer, a single layer with one layer above and a single layer with one layer below.
- c. **Display** allows you to turn on or off certain information regarding the slice preview.
 - i. Select the **Model Material** checkbox to turn on or off viewing the rendering of your 3D model.
 - ii. Select the **Support Material** checkbox to view the support structures that may be part of your 3D model.
 - iii. Select the **Travel Moves** checkbox to view the travel moves from a project. Travel moves are parts of the extruder path where the Replicator+ extruder moves without extruding any plastic.
 - iv. Select the **Layer Highlight** checkbox to highlight the selected layer.
- iii.) **Model Info** - Allows you to change the units that your 3D model is measured in. You can cycle between mm, cm, m, in, and ft. Please note that the software will assume your model was designed in millimetres. The Model Info icon is shown in Figure 11 below.



Figure 11: Model Info Icon

iv.) **Print Settings** – shown in Figure 12 below.

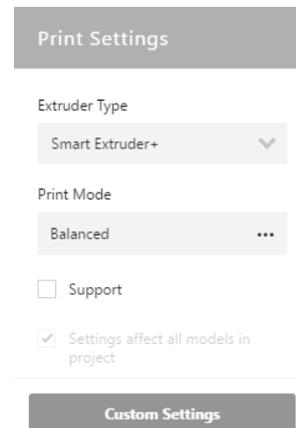


Figure 12: Print Settings Dialogue Box

- a. Select **Extruder Type** to choose which type of extruder you are using. Unleash Space has set the 'Extruder Type' to 'Smart Extruder+' only.
- b. Select the **Supports** checkbox to have your model print with support structures. MakerBot Print will automatically generate supports for any overhanging sections of your object. Supports will be easily removable once you remove your finished object from the build plate.
- v.) **Arrange** - If there are multiple objects on the build plate, select the **Arrange Build Plate** to move the objects to the most optimal placement on the build plate. The **Arrange Dialogue Box** is shown in Figure 13 below.

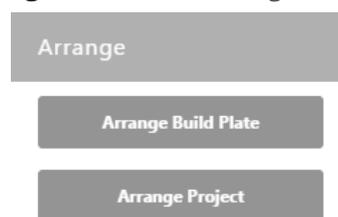


Figure 13: Arrange Dialogue Box

- a. If there are multiple objects on the build plate and all of the objects can't fit on the original build plate, select **Arrange project** to arrange the objects to the most optimal placement on the build plate. Objects that can't be arranged on the original build plate will be moved to another build plate and arranged optimally as well.



- b. Use the right-click menu to copy, paste, hide, or delete your model from the build plate.
- vi.) **Orientation**



Figure 14: Orientation Logo

- a. Select **Place Face on Build Plates** to lay your model flat on the build plate.
 - b. Rotate your model by 90 degrees or by a specific number of degrees along the x-, y-, or z-axis.
- vii.) **Scale**



Figure 15: Scaling Logo

- a. Select **Uniform Scaling** to adjust the scale of your model equally along the x-, y-, or z-axis. Enter the percentage of how much you want your model scaled.
- b. De-select **Uniform Scaling** to scale the model by a specific amount along the x-, y-, or z-axis.

4. MakerBot Print Software: Custom Print Modes Settings

Under the 'Print Settings' in the Printer Model Panel can the Custom Settings be found. These settings are to be used under the supervision of a Creative Technologist.

i.) Default Print Modes.

Print Modes are customized sets of recommended print settings.

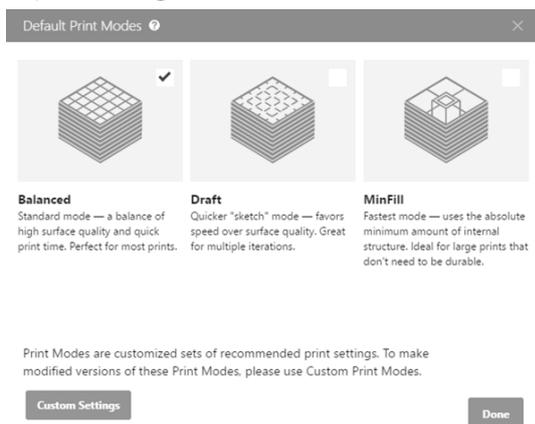


Figure 16: Default Print Modes Dialogue Box

- a. **Balanced:** Standard Mode - a balance of high surface quality and quick print time.
 - b. **Draft:** Quicker "sketch" mode - favors speed over surface quality. Great for multiple iterations.
 - c. **MinFill:** Fastest mode - uses the absolute minimum amount of internal structure. Ideal for large prints that don't need to be durable.
- ii.) **Quick Settings**

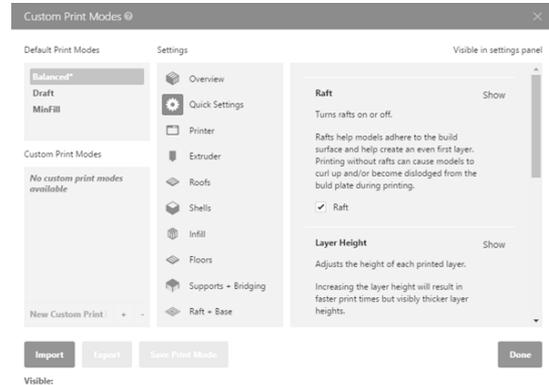


Figure 17: Quick Settings

- a. **Rafts:** Rafts help models adhere to the build surface and 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. Unleash Space has set the default to having rafts on every print.
 - b. **Layer Height:** Adjusts the height of each printed layer. Increasing the layer height will result in faster print times but visibly thicker layer heights.
 - c. **Infill Density:** Adjusts the density of the infill on the inside of the print. If the top of the model is collapsing during a print or the model feels too brittle, increasing the infill density may help. Increasing the density will increase print time.
 - d. **Number of Shells:** The number of outermost layers of the printer model. Increasing the number of shells creates thicker stronger walls, but slower prints. Unleash Space has set the default to the recommended 2 shells.
- iii.) **Printer**

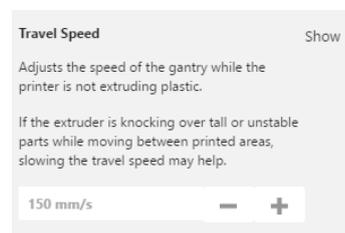


Figure 18: Travel Speed Setting under Printer Custom Print Mode



- a. **Travel Speed:** Adjusts the speed of the gantry while the printer is not extruding.

If the extruder is knocking over tall or unstable prints while moving between printed areas, slowing the travel speed may help. Slowing the travel speed will increase print time.

Minimum: 10 mm/s
Maximum: 175 mm/s

iv.) **Extruder Settings**

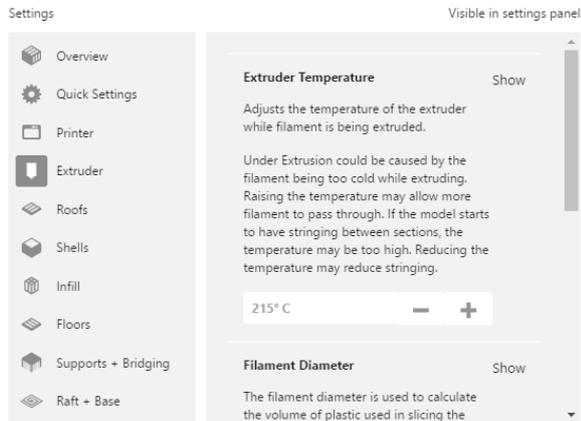


Figure 19: Extruder Settings under Custom Print Modes

- a. **Extruder Temperature:** Adjusts the temperature of the extruder while filament is being extruded.

Under Extrusion could be caused by the filament being too cold while extruding. Raising the temperature may allow more filament to pass through. If the model starts to have stringing between sections, the temperature may be too high. Reducing the temperature may reduce stringing.

Minimum: 100°C
Maximum: 255°C
Unleash Space Settings: 215°C

