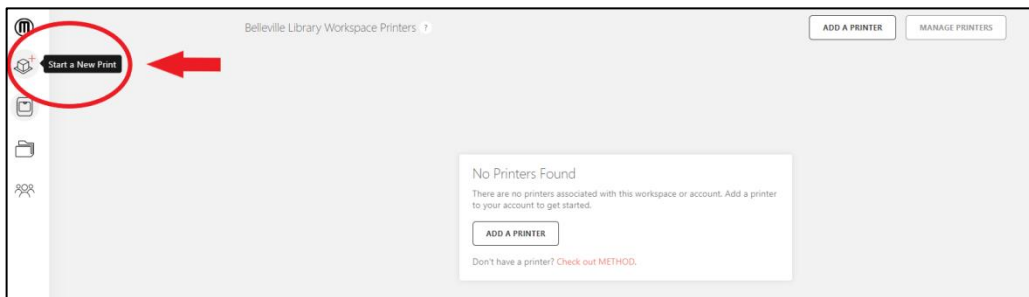




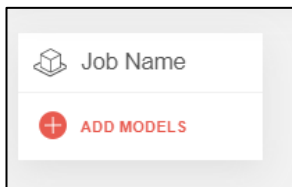
MakerBot CloudPrint Slicing Software

Before sending us your print job, use MakerBot CloudPrint slicing software to prepare your model. This slicing software will convert your .STL or .OBJ file into a .MAKERBOT file that the Library's MakerBot Replicator+ 3D Printer can read.

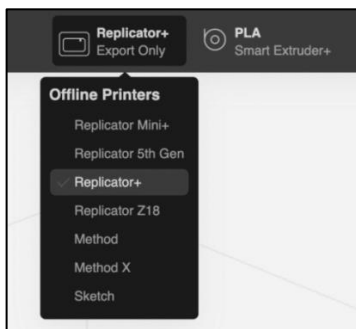
1. Go to www.makerbot.com/3d-printers/apps/makerbot-print/download/, select Get Started for Free, then Sign Up, then follow the prompts to create a free MakerBot account.
2. When you first sign in on MakerBot CloudPrint, you can ignore the No Printers Found message. Get started by selecting the Start a New Print option on the left side menu.



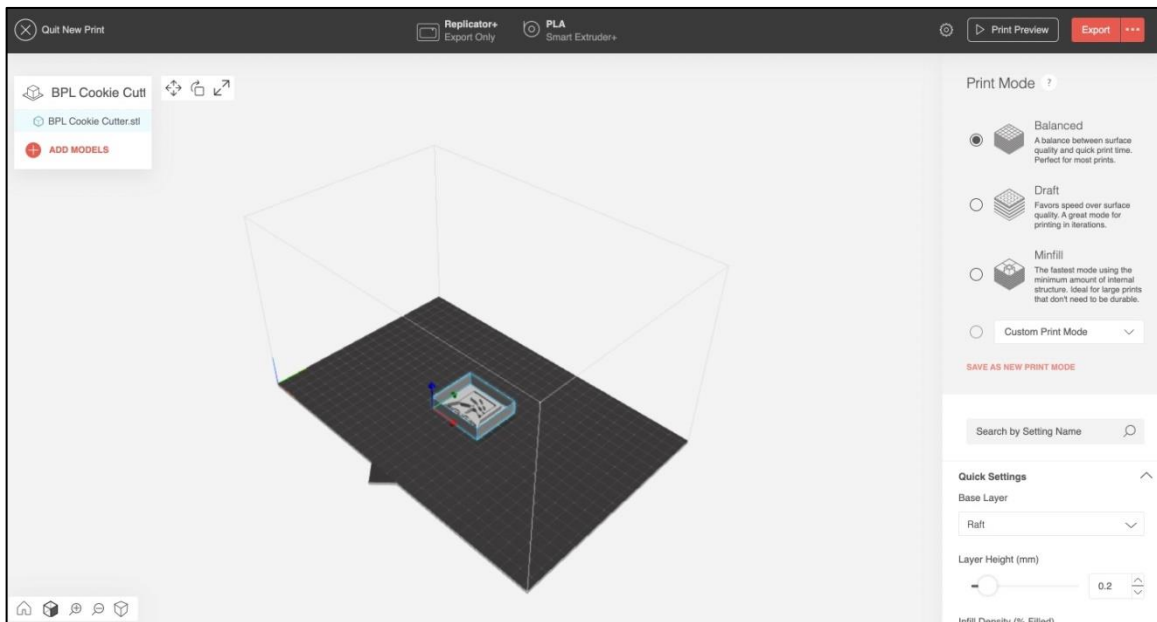
1. Import your object file (usually a .STL or .OBJ) by selecting ADD MODELS, then ADD FILES, then select the file from your computer.



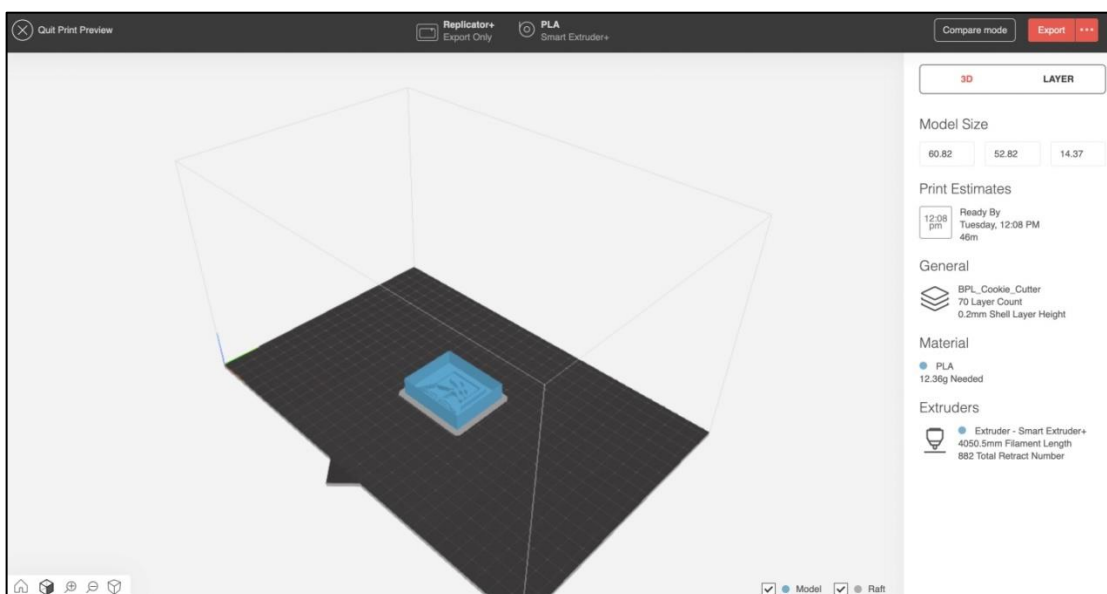
2. On the top menu bar, select MakerBot Replicator+ as the 3D printer and ensure the filament it set to PLA Smart Extruder+.



3. Modify print settings (infill, supports if required, orientation, size, etc.) on the right side menu to your preferences. [Click here](#) to learn more about print settings. For PLA material, MakerBot recommends an Extruder Temperature of 215°C and a Print Speed of 90 mm/s.



4. For a preview of your object, select Print Preview in the top right corner. Print Preview slices the 3D model into layers. **LAYER** view shows the path the extruder tool will make while printing each layer of the object. Toggle to **3D** on the right side menu to see a view of the final product, an estimate of print time and the amount of filament your object requires. If you want to go back and modify the print settings, select Quit Print Preview in the top left corner.



5. When you are happy with your model, select Export and choose the .MAKERBOT file format.

