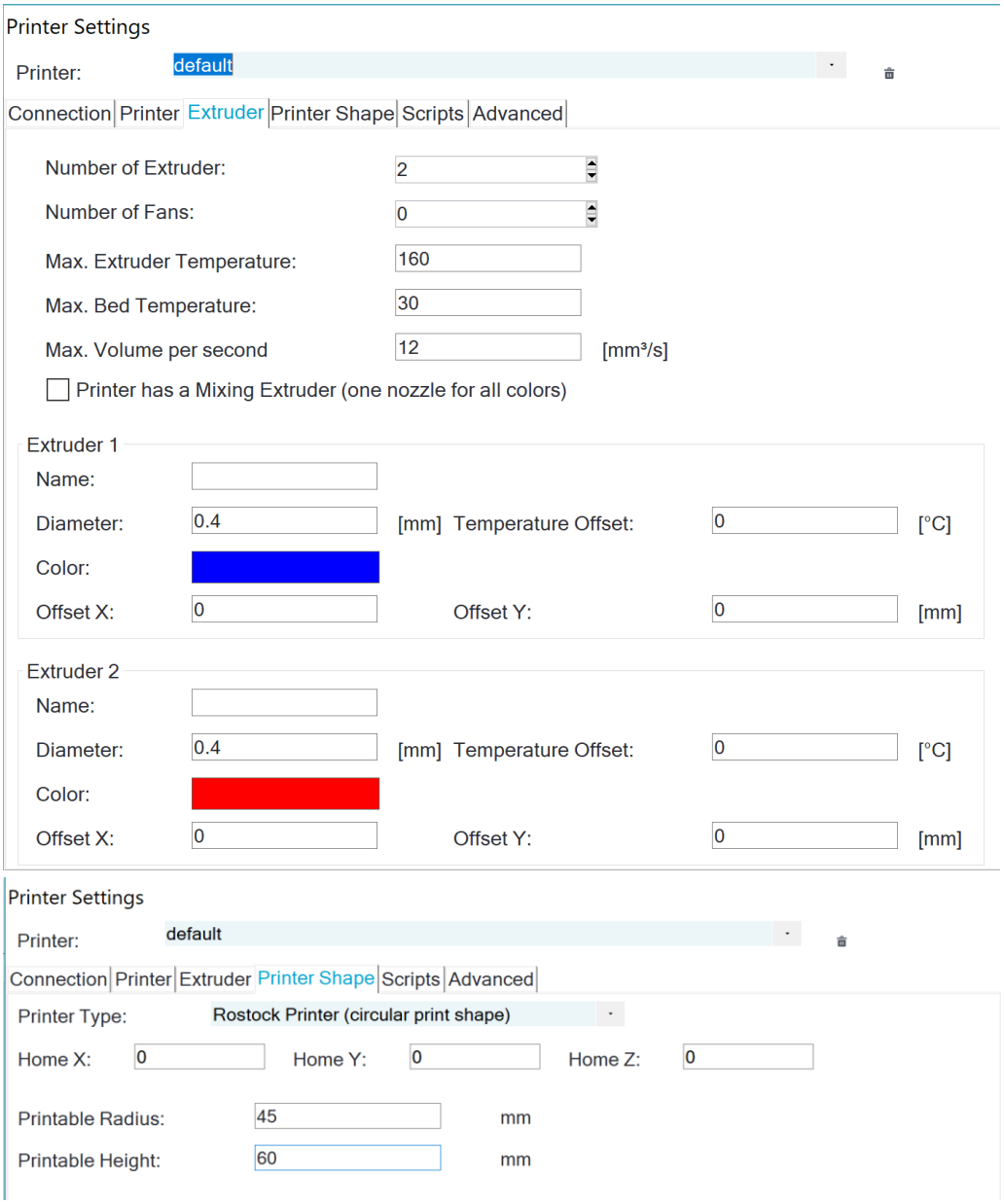
Allevi Guide to Repetier-Host

Setting up Repetier-host

1. Install Repetier-Host.
   a. Close window if Repetier-Host Server opens in your browser and continue.

2. Open Repetier-Host. Go to ‘Config’ > ‘Printer Settings’ and assign the following values in the appropriate sections.

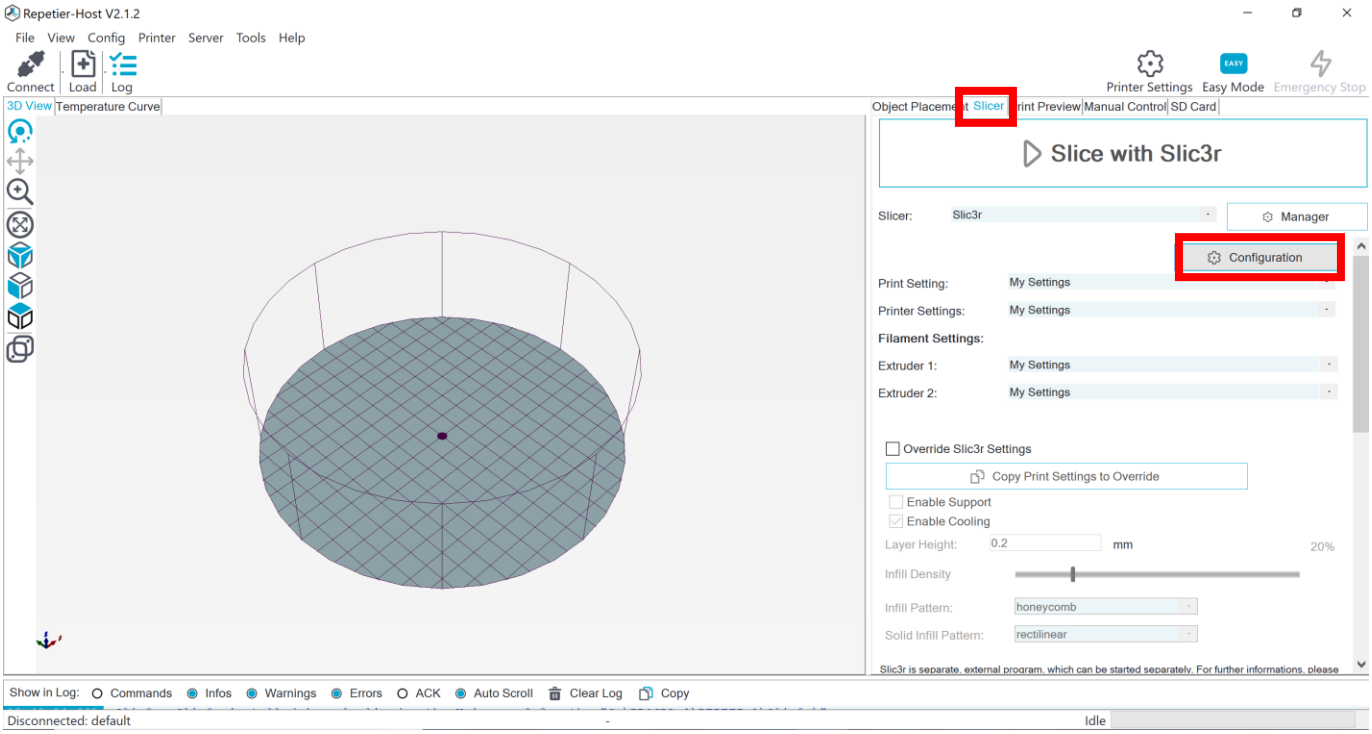


Note: For ‘Number of Extruder’, enter the number of extruders on your printer.

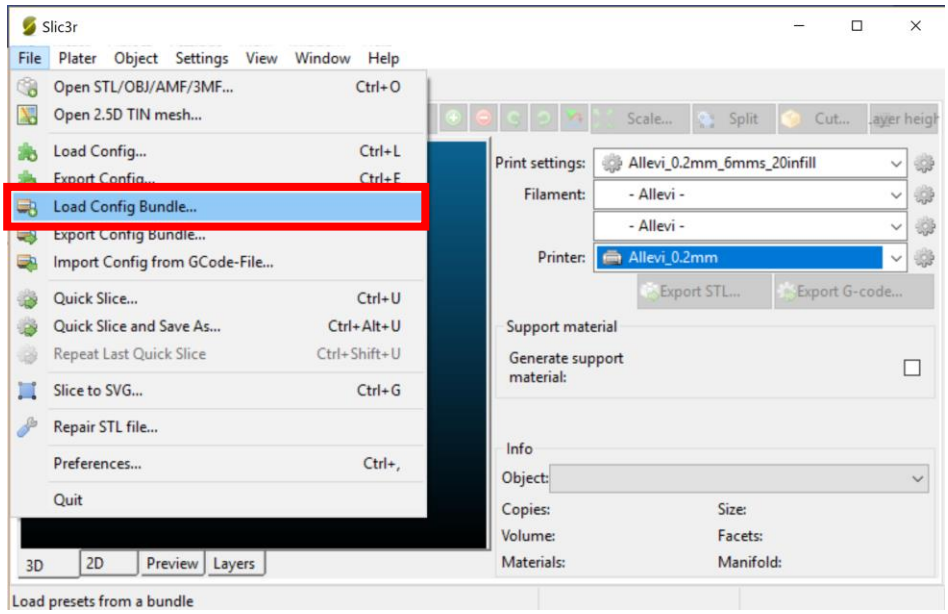
3. Go to ‘Config’ > ‘Units of imported Objects’ and select millimeters.

4. Download the Slic3r Config Bundle file.

5. In Repetier Host, go to the Slicer tab and click “Configuration.”



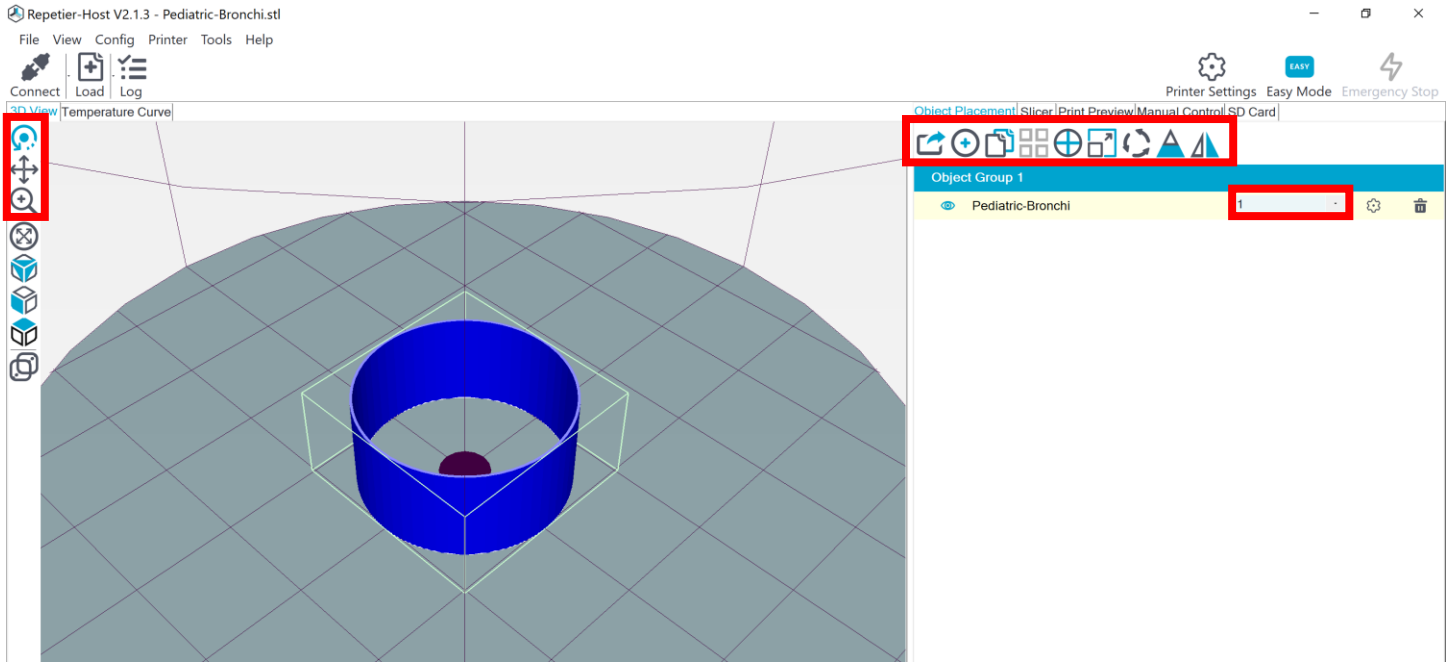
6. Go to “File” > “Load Config Bundle…” and select the Slic3r Config Bundle file you just downloaded.



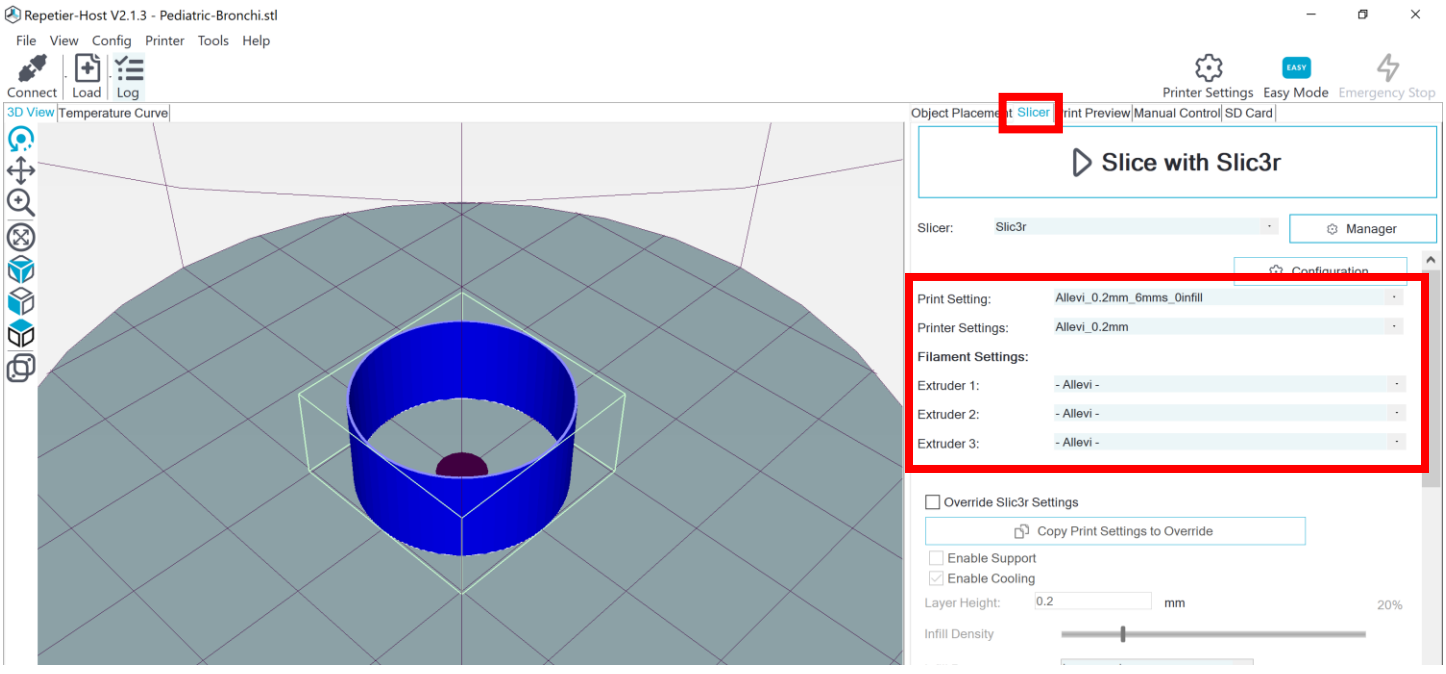
Slicing your STL

You can design STL files on any CAD software, such as SolidWorks or Autodesk Fusion 360. Alternatively, you can find a host of STL files in the “Getting Started” folder. To begin we will use a predesigned STL file.

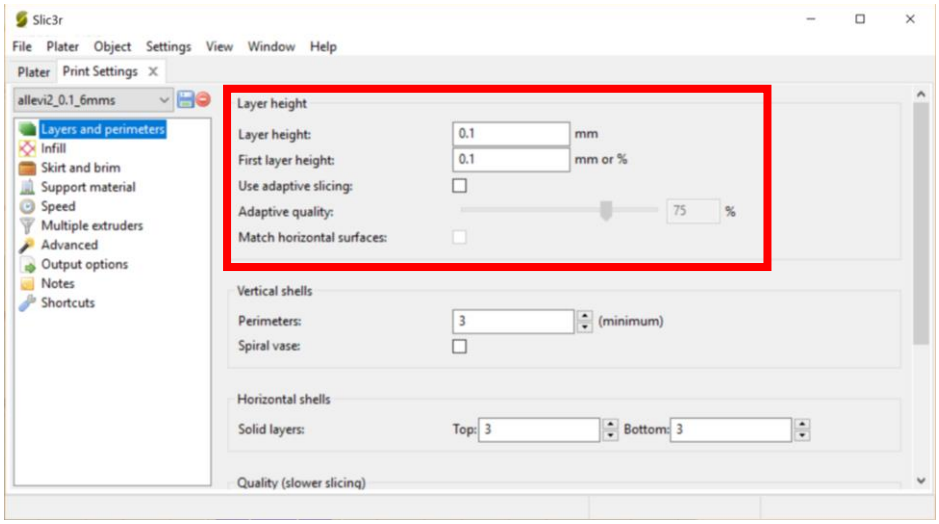
1. Load the Pediatric-Bronchi.STL file into Repetier-Host.
   a. Move the object on the bedplate using the arrows highlighted on the left.
   b. Save, copy, or scale object with the items on the right.
   c. Assign the STL to an extruder by using with the extruder drop down menu.



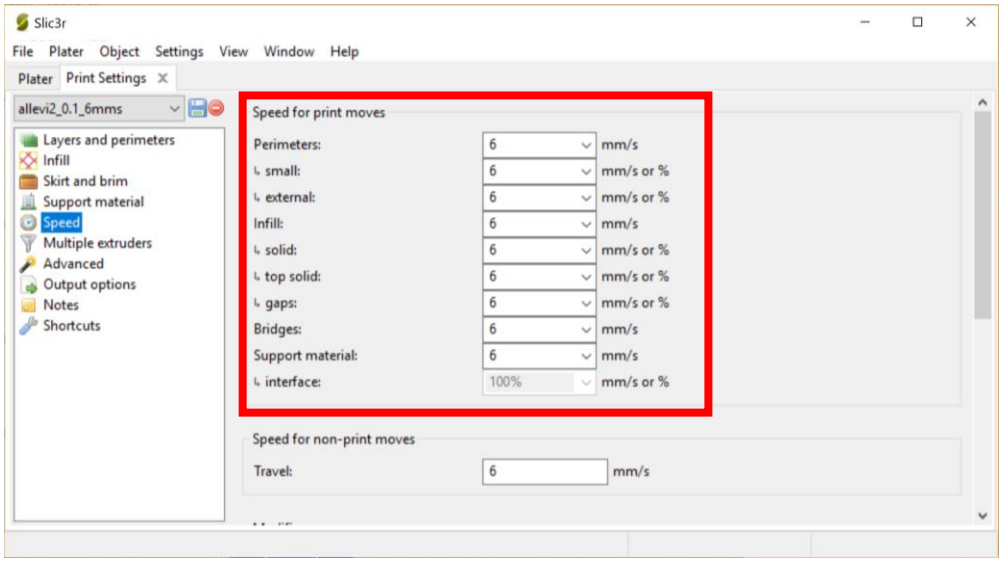
2. Enter your print settings.
   a. Select the “Slicer” tab to assign preconfigured Slic3r settings to slice the file.
   b. Speed and layer height are defined.



3. Select the configuration tab to adjust settings important for bioprinting. Print, filament, and printer settings are accessible under ‘Settings.’
   a. Layer Height – Determines how thick or thin your layers are

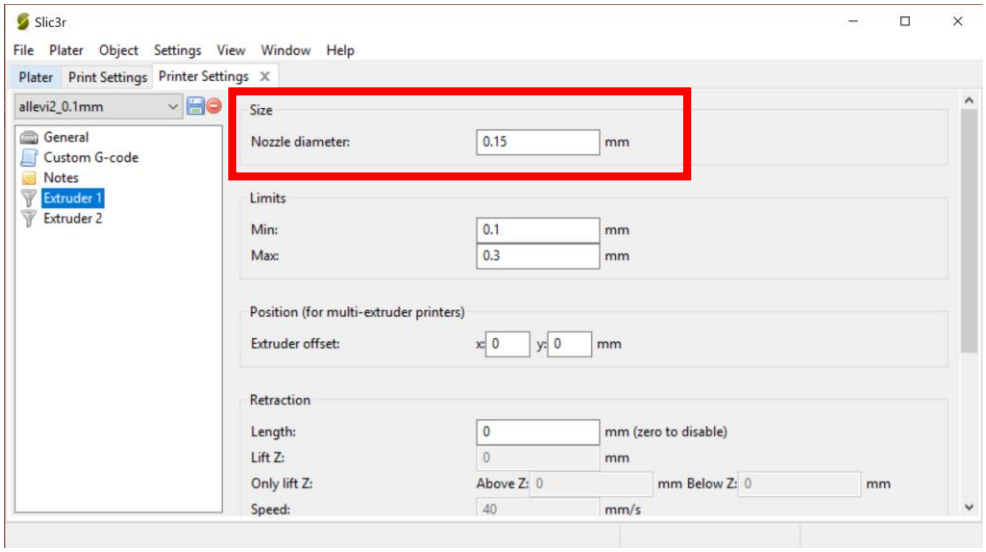


   b. Speed – Determines how fast the extruder will move during the print. Remember to change all the speeds in the column to the desired number



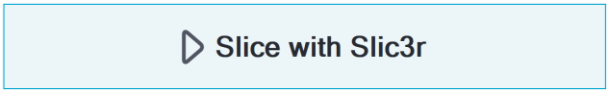
Recommendation: 2 mm/s is good for a small accurate design (example: lattice). 8-16 mm/s is good for a large design (example: ear). **Speed is a print parameter that is very dependent on material and pressure.**

c. Nozzle Diameter – Enter the number that accurately depicts your nozzle tip inner diameter



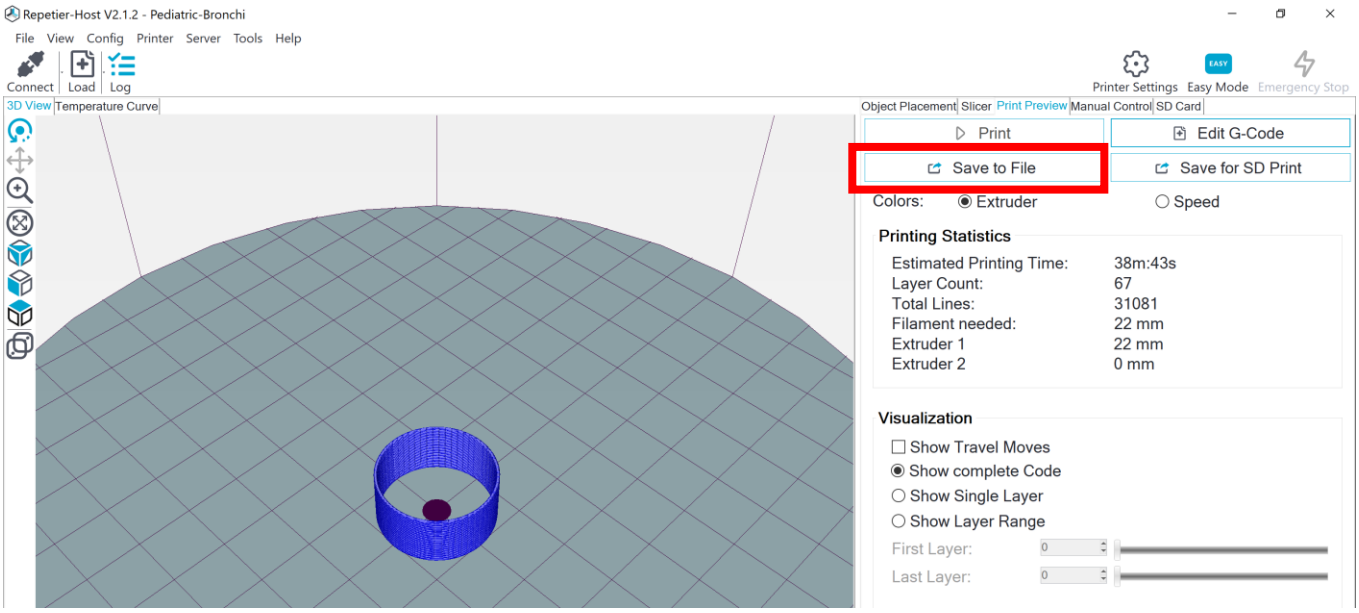
NOTE: Your layer height must be less than your nozzle diameter to correctly slice
NOTE: Once the changes have been made, remember to save and label your settings.

4. Once your settings have been saved, you are now ready to slice the file.



5. You can view the printer movement by viewing the g-code created by your slicer file.
Note: If no g-code is generated, check that your nozzle diameter is thin enough.

a. Save your g-code file.



How does your gcode look? You can compare with our presliced file!