Raise3D Pro2Plus
Validated Settings Verification

Training Documentation for 3D Printing

Table of Contents

Introduction

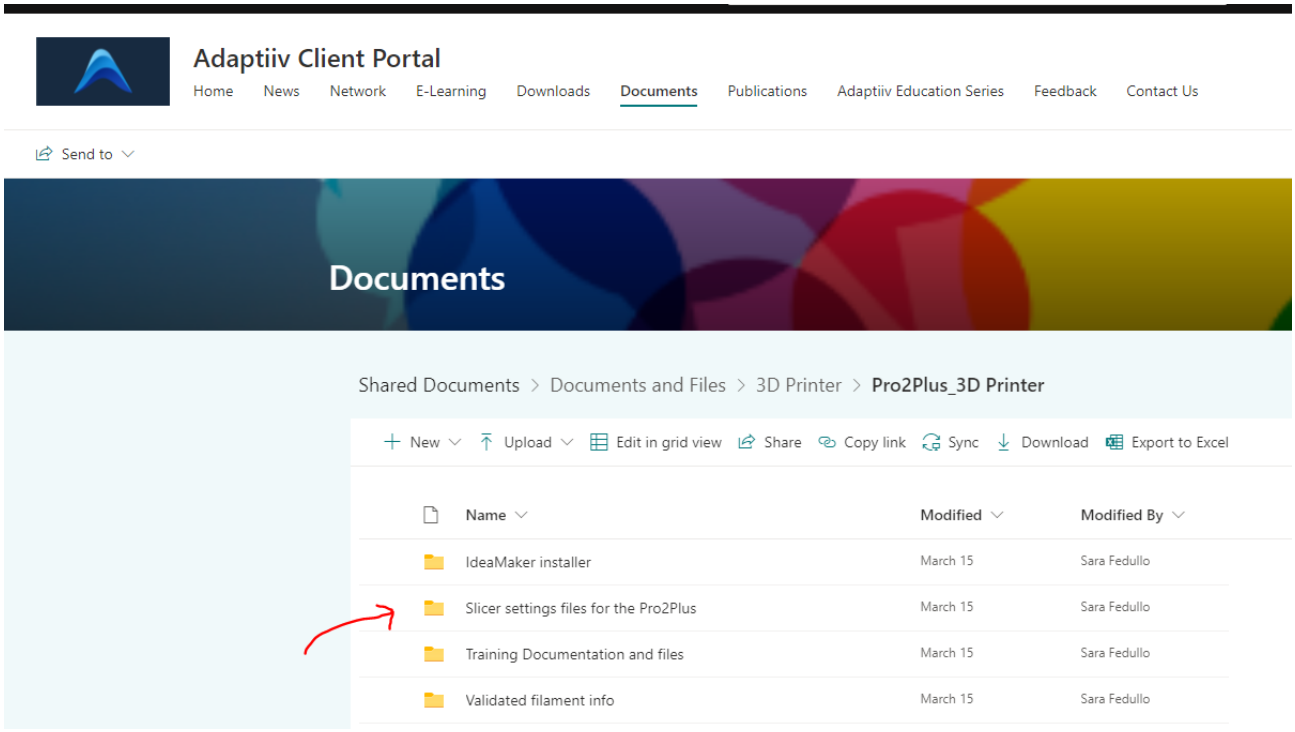
When importing printer settings in the IdeaMaker slicer software for the first time, it is important to check three main settings components: Infill Density, Platform Addition, and Supports.

The following document illustrates the basic steps needed to complete this verification procedure for each of the filament types and applications pursued by Adaptiiv Medical Technologies for the Pro2Plus 3D Printer.

Downloading and Importing 3D Printer Settings in IdeaMaker

1. Download the validated printer settings file from the Adaptiiv Client Portal

    a. Documents → Documents and Files → 3D Printer →Pro2Plus 3D Printer → Slicer settings file for the Pro2Plus



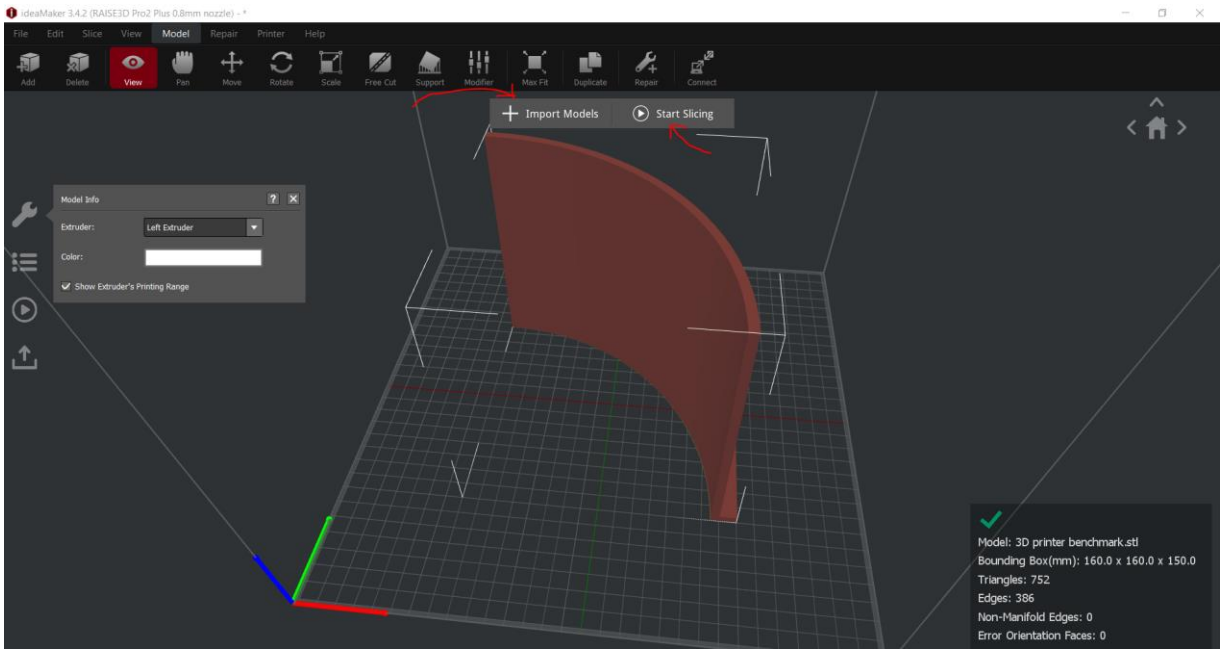
2. Import the pre-downloaded settings in the IdeaMaker 3.4.2 slicer software

    a. Slice → Manage templates →Gear icon next to Printer Type → Import (bottom left of the pop-up Printer settings window) → Select the pre-downloaded settings → OK →Save

    b. Manage Templates →Import (bottom right)→Select the printer settings bin file → OK → Close

3. Import a model in the IdeaMaker slicer software by selecting the corresponding STL file previously created in the Adaptiiv software.

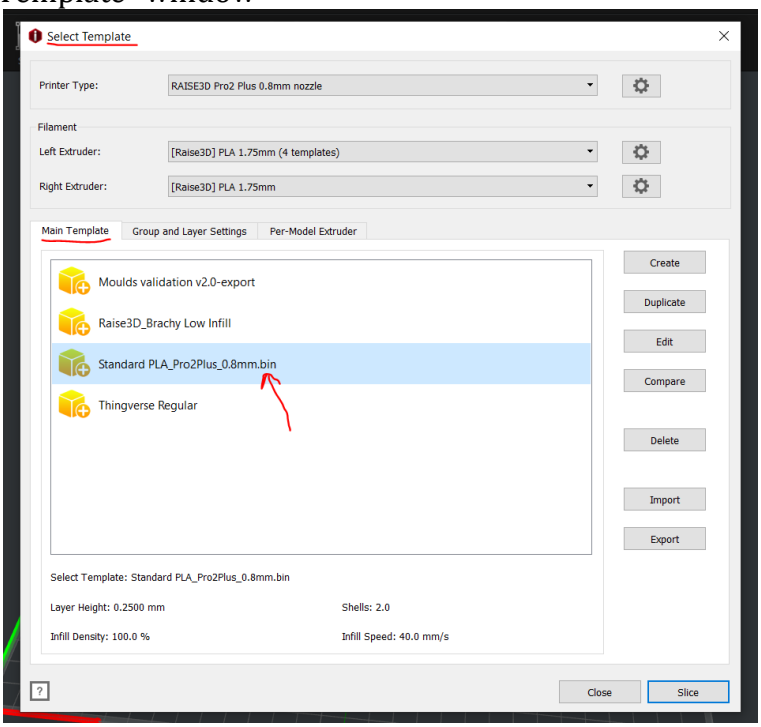
4. Select the “Start Slicing” icon.



5. Follow the filament specific procedures outlined in this document.

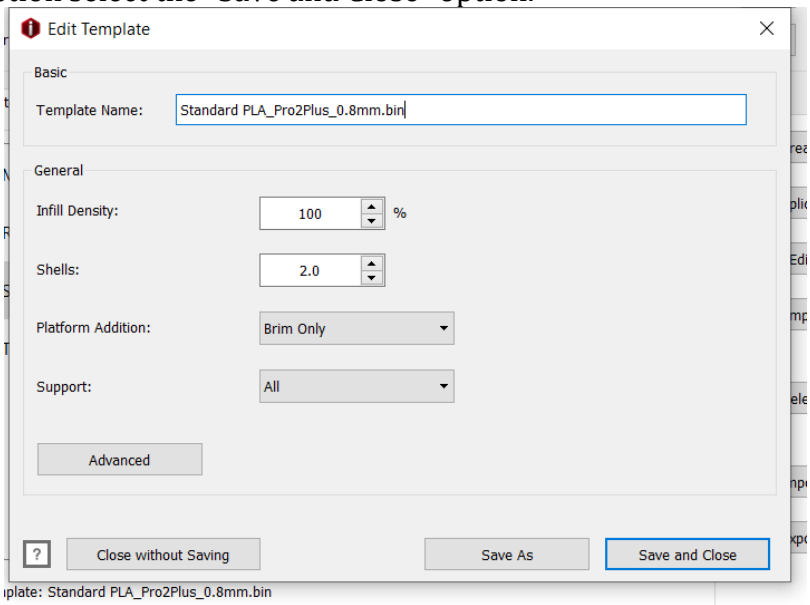
1. Printing with 3D Fuel Standard PLA

1. Click on the Standard PLA settings template found in the “Main Template” tab of the “Select Template” window



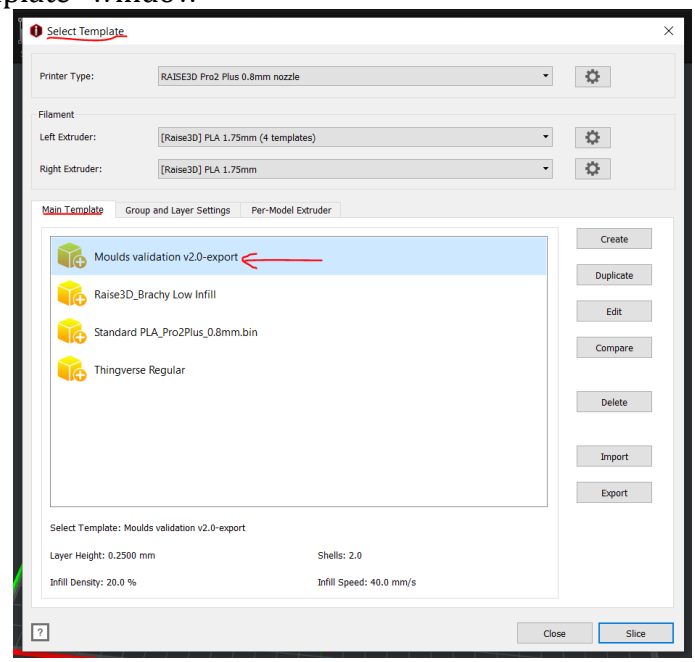
2. Select the “Edit” button found at the top right

3. Verify that the general settings listed match those illustrated in the figure below. At completion select the “Save and Close” option.



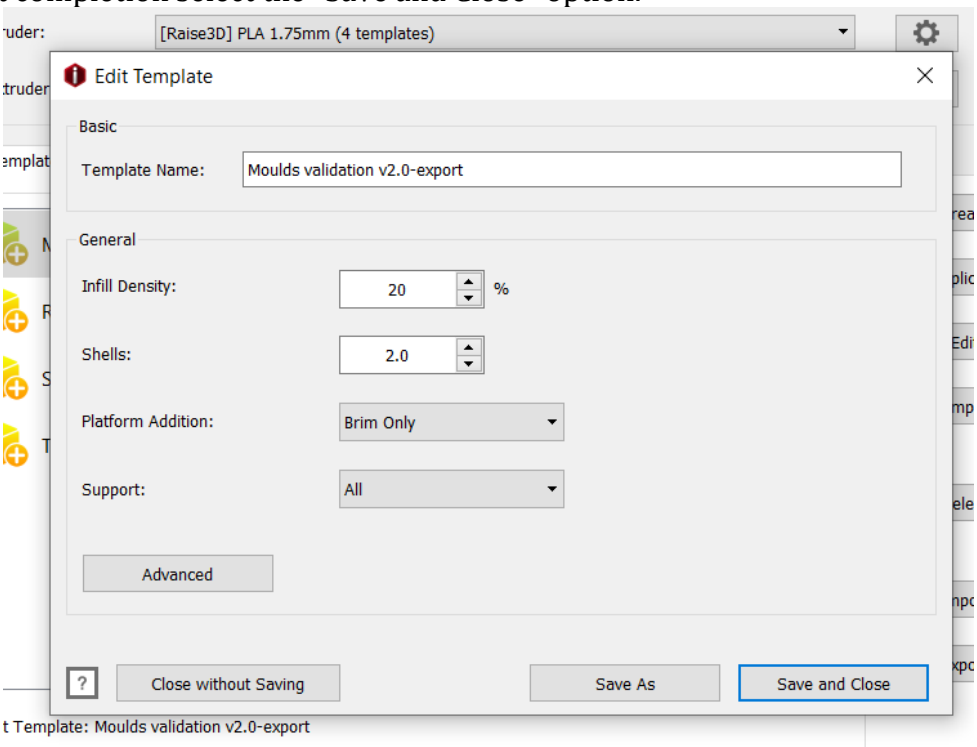
2. Printing with 3D Fuel Standard PLA – Moulds

1. Click on the PLA Moulds settings template found in the “Main Template” tab of the “Select Template” window



2. Select the “Edit” button found at the top right

3. Verify that the general settings listed match those illustrated in the figure below. At completion select the “Save and Close” option.

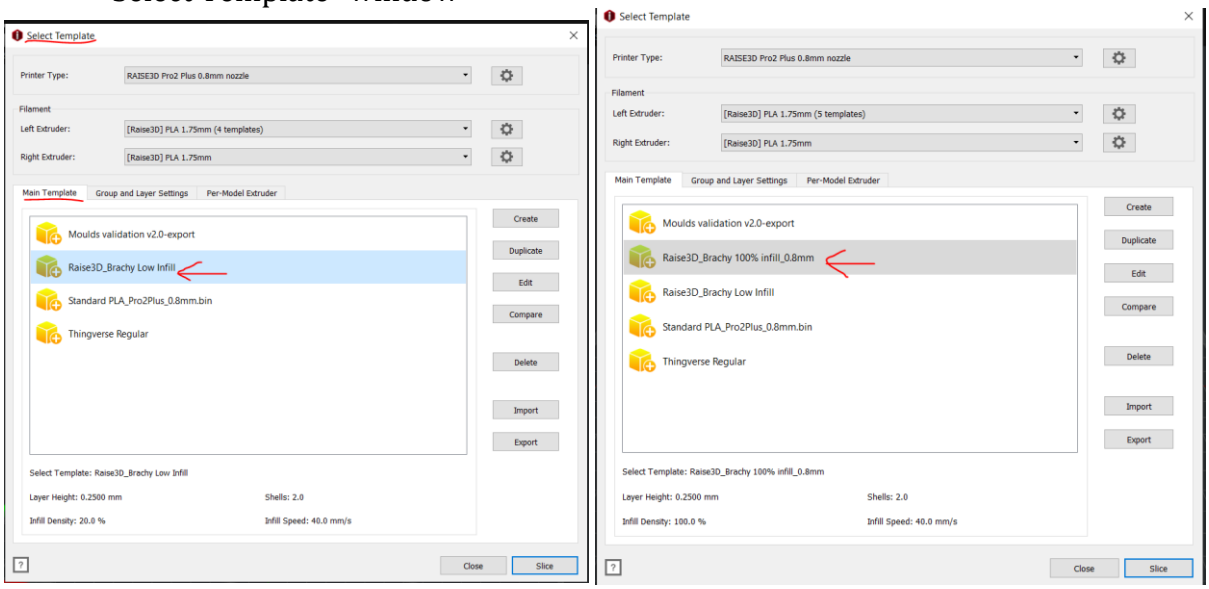


4. Printing with 3D Fuel Standard PLA – Brachytherapy

WARNING!

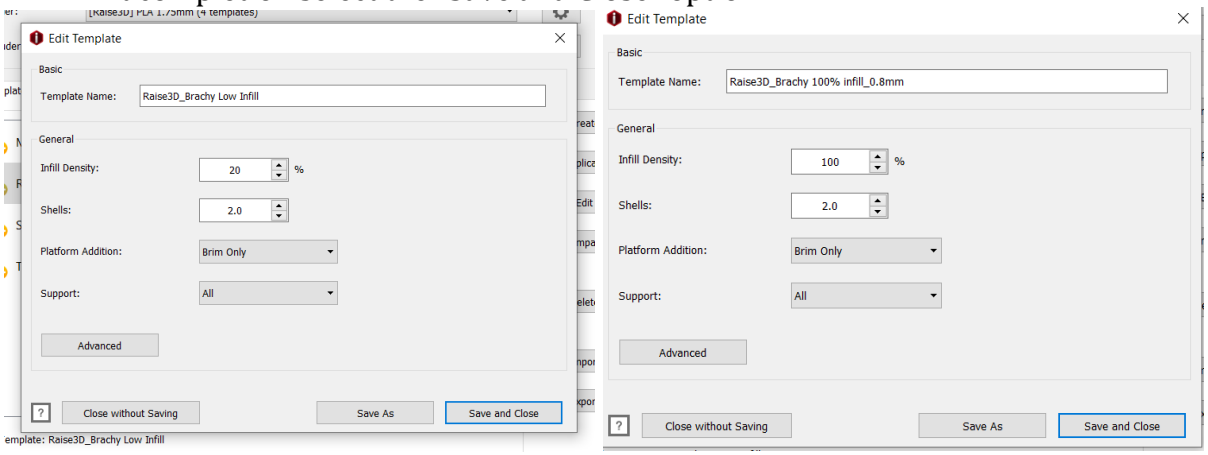
Adaptiiv tested brachytherapy solutions printed at both 100% infill and 20% infill. No spatial fidelity discrepancies were observed in either infill percentage variation, however it was concluded that users are recommended to always print at 100% infill to ensure that the physical density of the brachytherapy applicator is closest to water equivalent.

1. Click on the PLA Brachy settings template found in the "Main Template" tab of the "Select Template" window



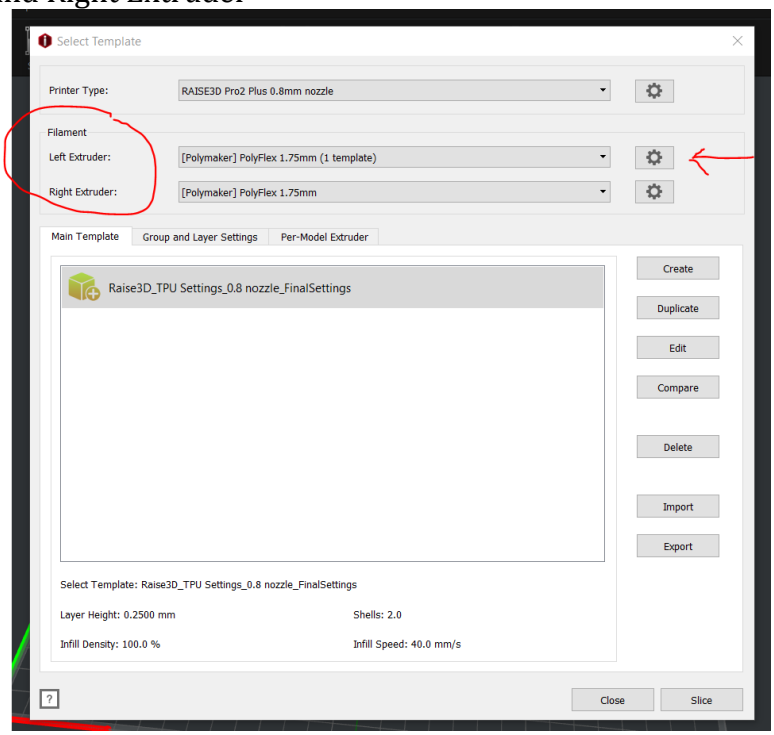
2. Select the "Edit" button found at the top right

3. Verify that the general settings listed match those illustrated in the figure below. At completion select the "Save and Close" option.

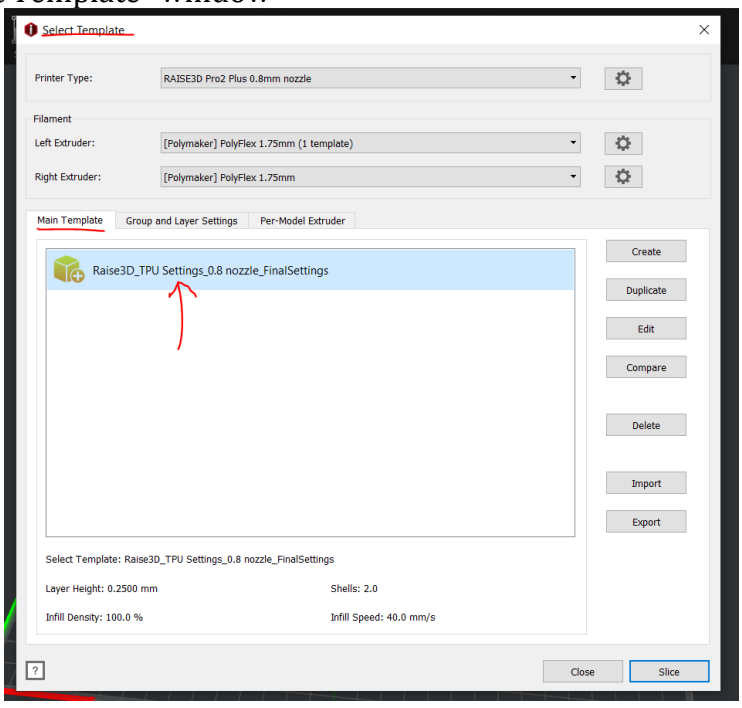


4. Printing with Raise3D 95A Premium TPU

1. Ensure that the Filament template “Polymaker PolyFlex 1.75 mm” is selected for the Left and Right Extruder



2. Click on the Raise3D TPU settings template found in the “Main Template” tab of the “Select Template” window



3. Select the “Edit” button found at the top right

4. Verify that the general settings listed match those illustrated in the figure below. At completion select the “Save and Close” option.