Prusa XL General Tutorials

The Prusa XL is an advanced 3D printer designed for high-quality, multi-material printing. It is capable of printing with five colors simultaneously using its Multi-Material Unit (MMU) and multiple extruders. The printer features a large build volume of **360×360×360 mm (14.17''×14.17''×14.17'')**, offering a vast area for creative projects.

Supported Materials:

- PLA, PETG, ABS
- Flexibles (TPU, TPE)
- Composite filaments (carbon fiber, fiberglass) – currently, there is no dedicated space for handling these filaments.
- PVA for supports

Supported File Formats:

- **G-code:** The standard format for 3D print files, containing both geometry and print settings.
- **3MF:** A versatile 3D file format used for multi-material printing, retaining color information.
- **STL:** A widely used 3D geometry format that can be processed by the slicer to generate G-code.

Step-by-Step Guide to Slicing with PrusaSlicer for Prusa XL

Important: Hovering over a setting will display a pop-up explanation about each specific setting.

Step 1: Open/Download the Latest Version of PrusaSlicer

Step 2: Setup Prusa XL in PrusaSlicer

1. Open **PrusaSlicer**.
2. Go to **Printer Settings** by clicking on the "Printers" tab in the top-left menu.
3. Click **"Add Printer"** and select **Prusa XL 5T** from the available printer profiles.
4. If Prusa XL is not listed, download the latest version of PrusaSlicer or manually add a custom printer profile by entering the following settings:
   - **Build Volume:** 360 x 360 x 360 mm
   - **Extruder Count:** 5 (for multi-material prints)

- **Max Extruder Temp:** 300°C
- **Max Bed Temp:** 120°C
- **Bed:** Magnetic, heated, textured steel

Important: Make sure to select **Original Prusa XL - 5T Input Shaper** for faster and higher-quality printing.

Input Shaper Benefits:

- Reduces vibrations, ensuring smoother motion.
- Minimizes print artifacts such as ringing and ghosting.
- Improves print quality at higher speeds.
- Enhances overall printer stability and efficiency.
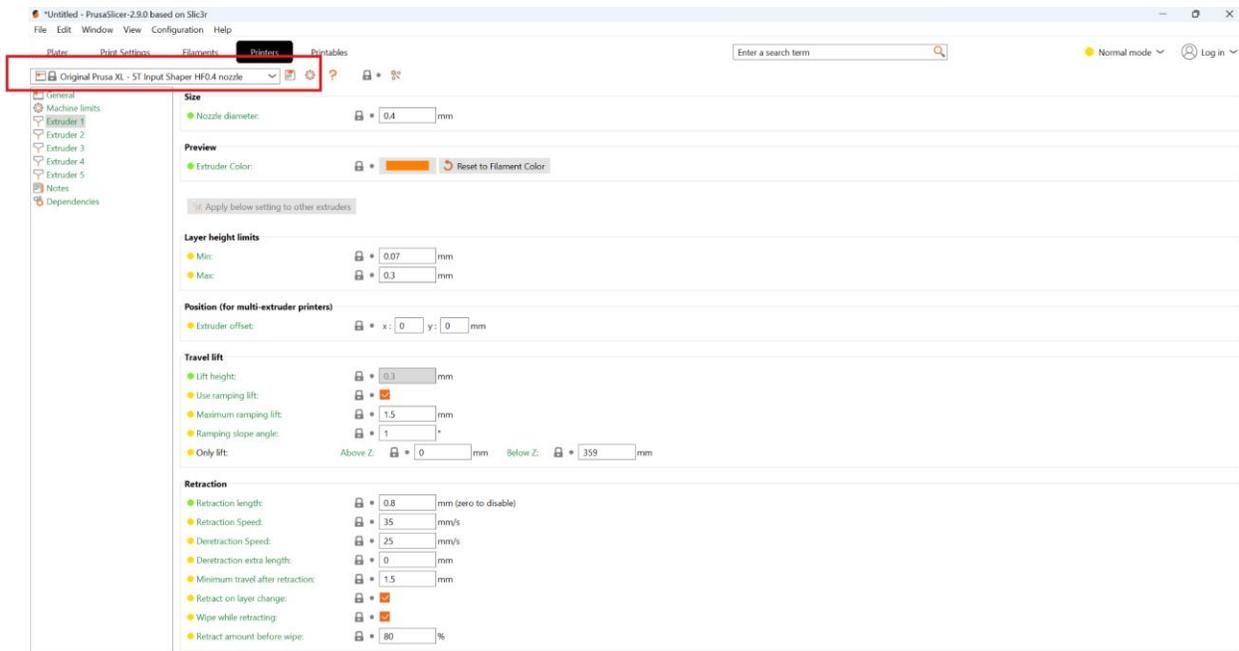
Step 3: Prepare Your Model

1. Open your 3D model (**STL, 3MF**) in **PrusaSlicer** by clicking **Add** in the top-left menu.
2. Rotate, scale, or move the model if necessary using the right panel options.
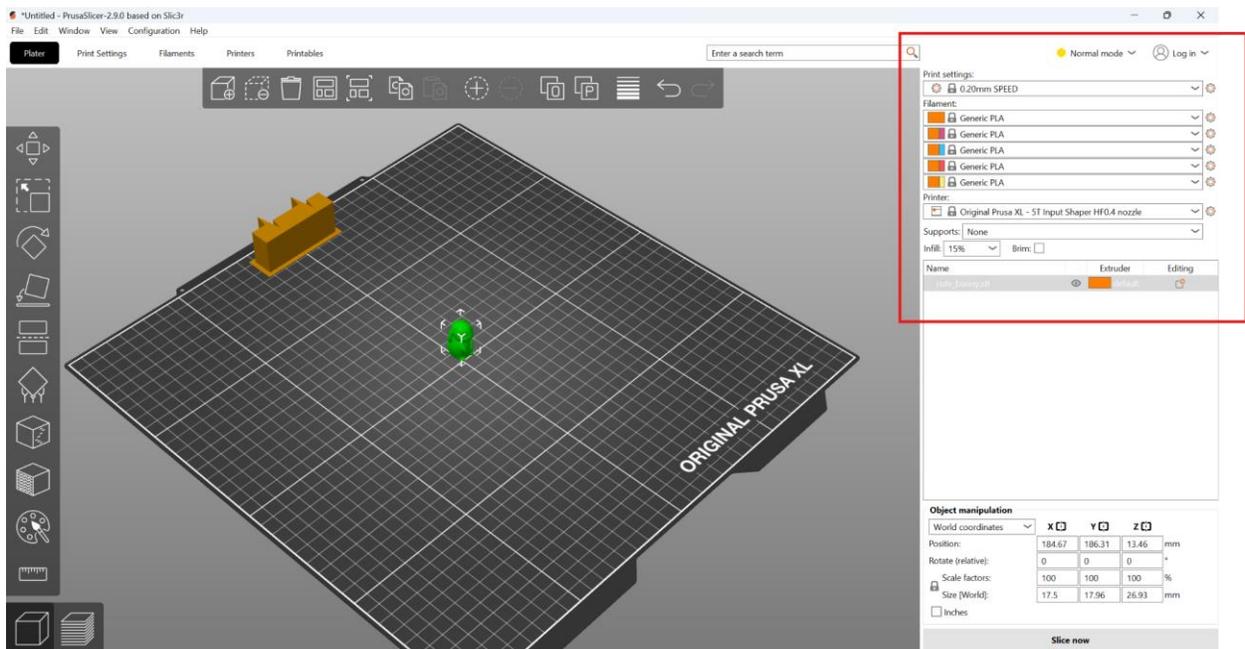
Note: For multi-material printing, ensure that your model has designated color assignments or distinct parts managed by multiple extruders. Check the tutorial on assigning colors to parts or meshes of the model. Link Here.
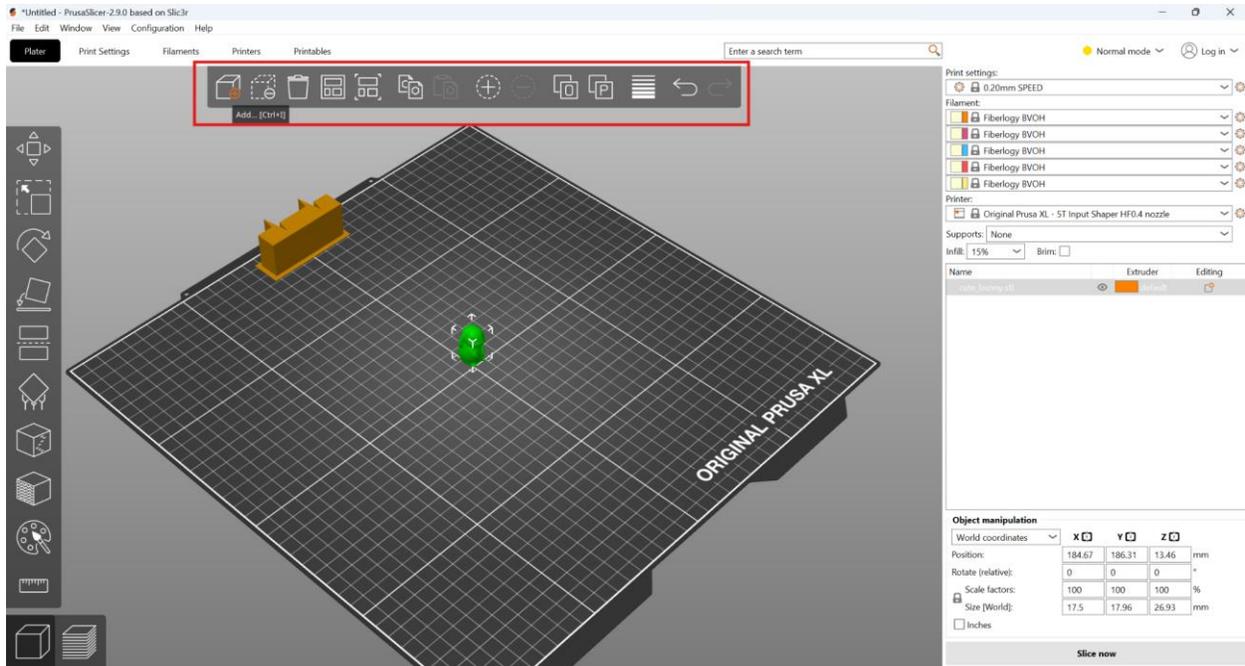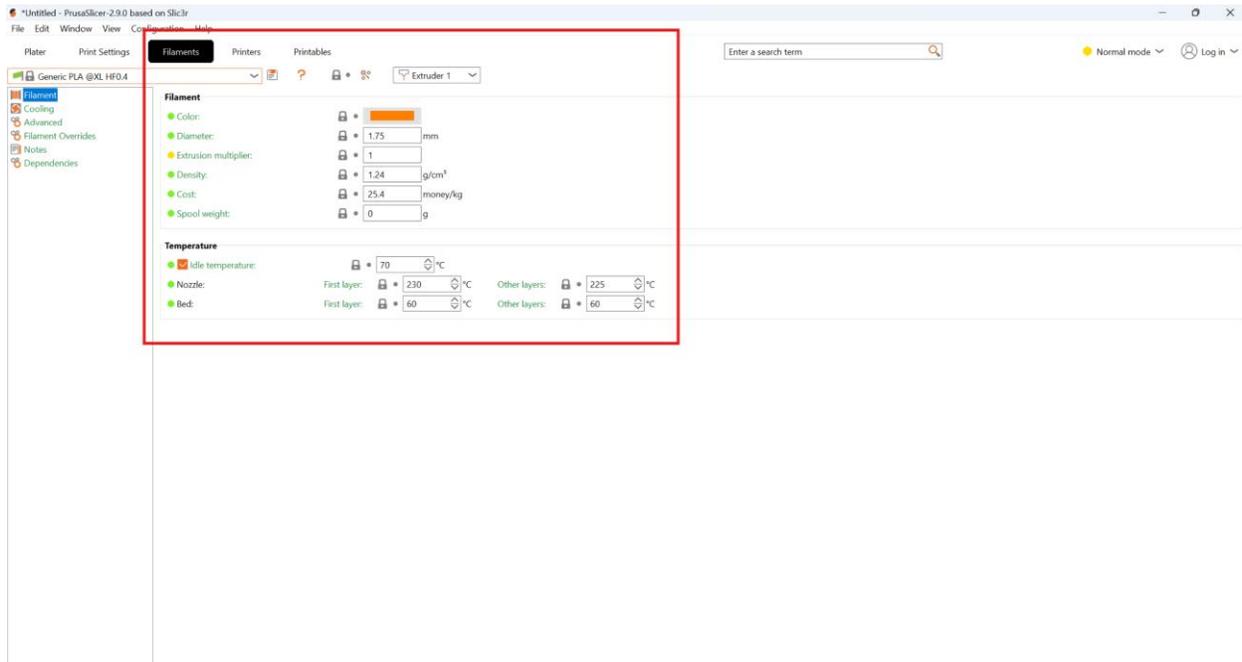
Step 4: Configure Print Settings
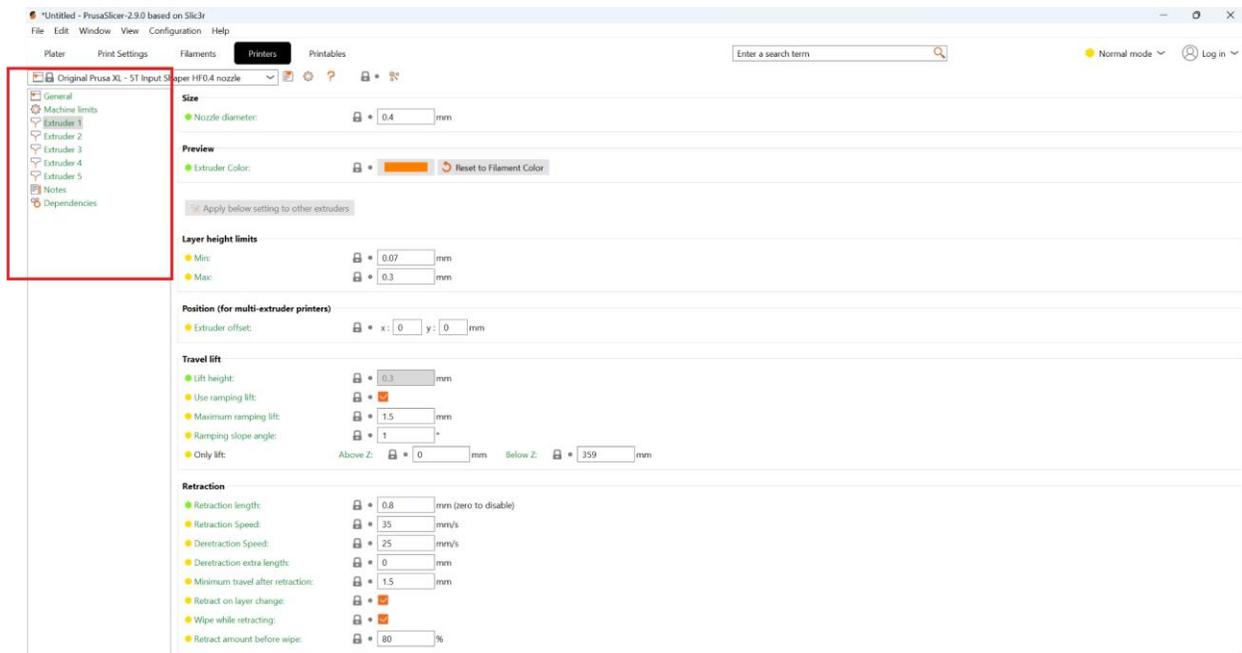
1. Select Material:

- Choose the material type for each extruder (e.g., PLA, PETG).

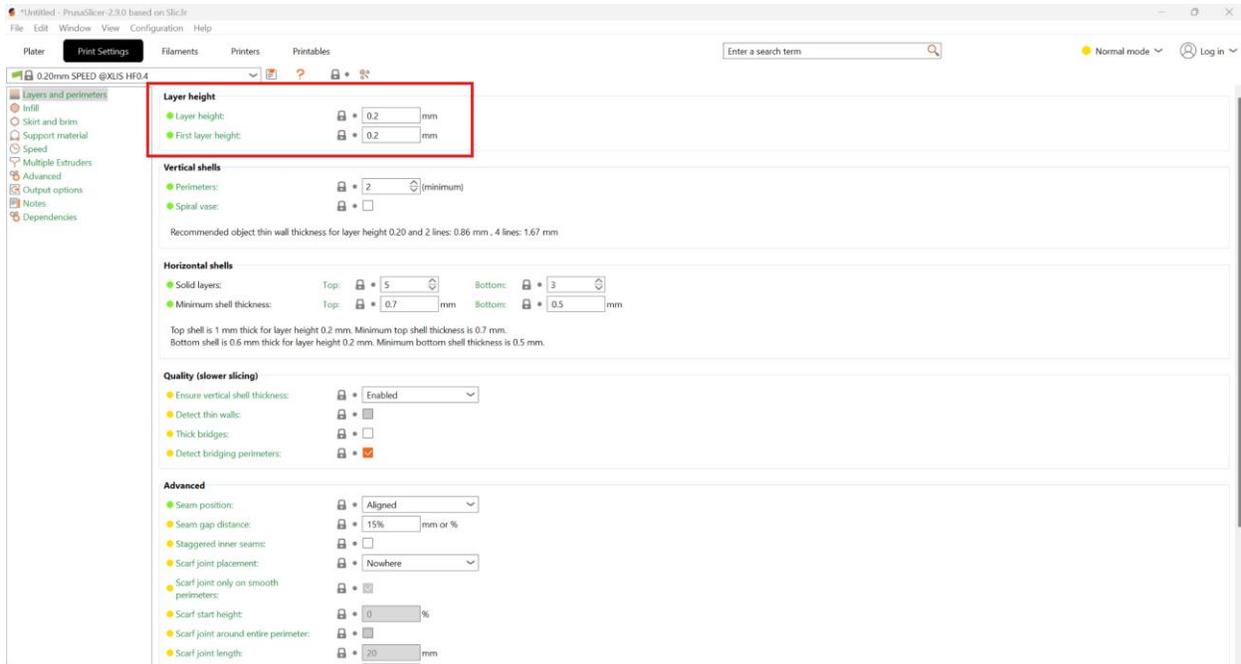- Click on the **Filament Settings** tab to set temperatures, speeds, and material properties.

- If using multi-material, configure each extruder individually under **Extruder Settings**.
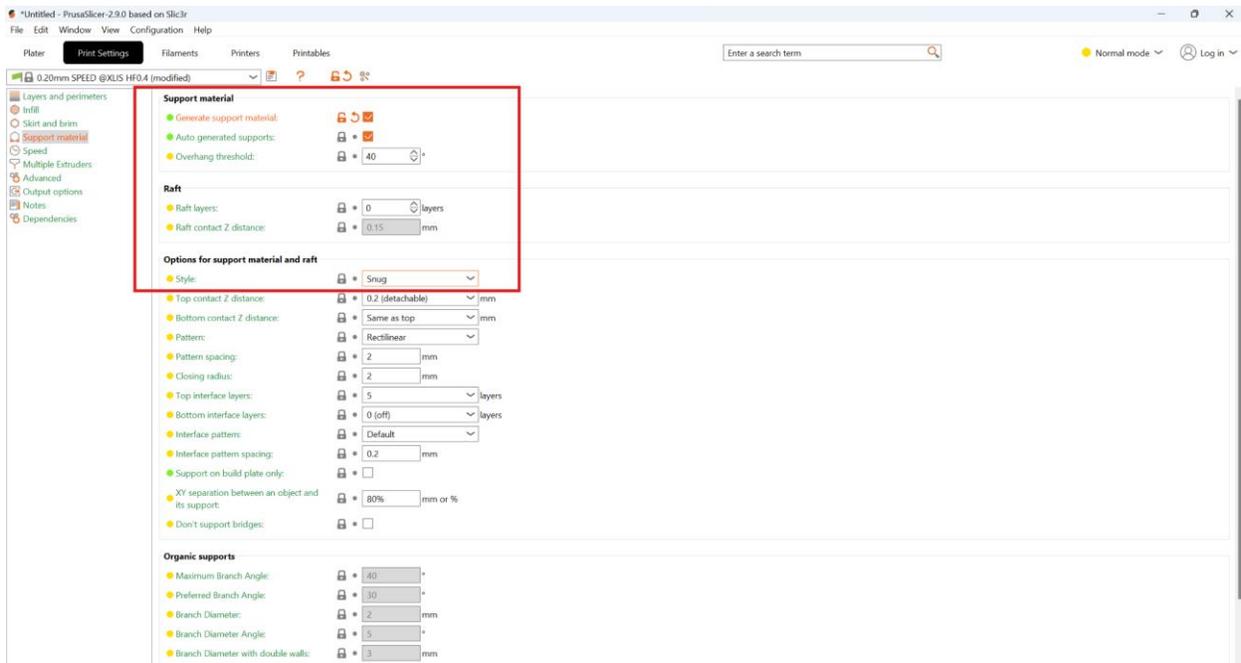

2. Layer Height:

- **0.2 mm** for standard quality.
- **0.1 mm** for high quality.
- **0.3 mm** for fast printing.
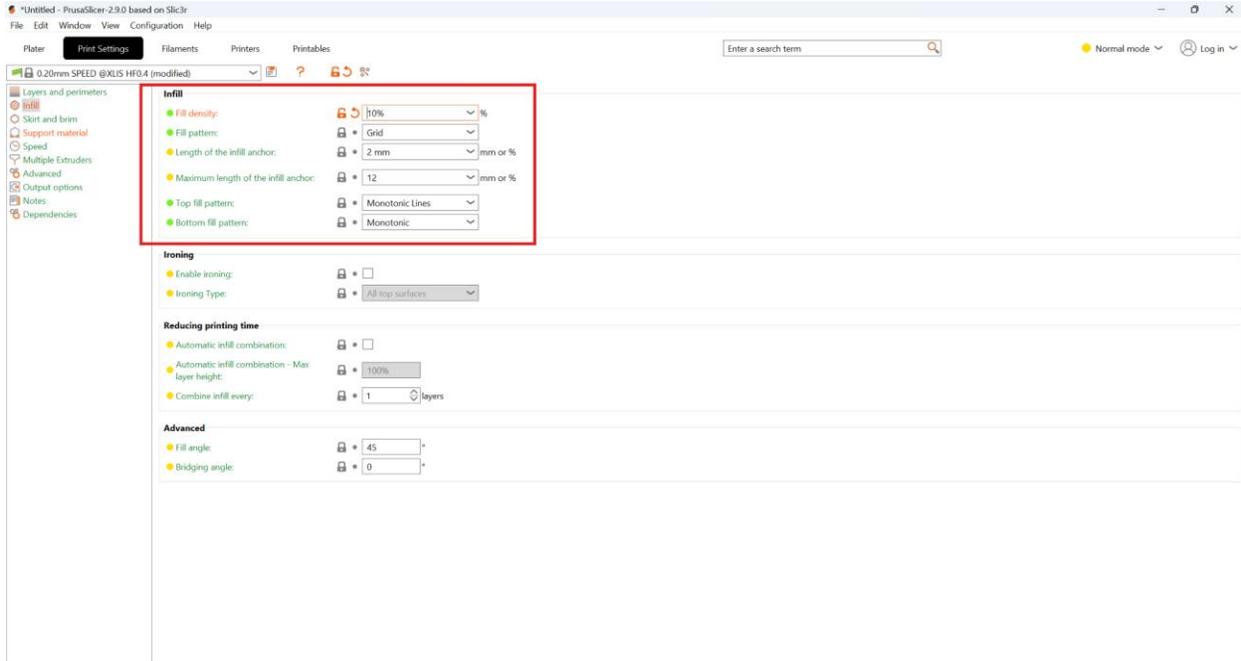
3. Supports:

- Enable support if needed.

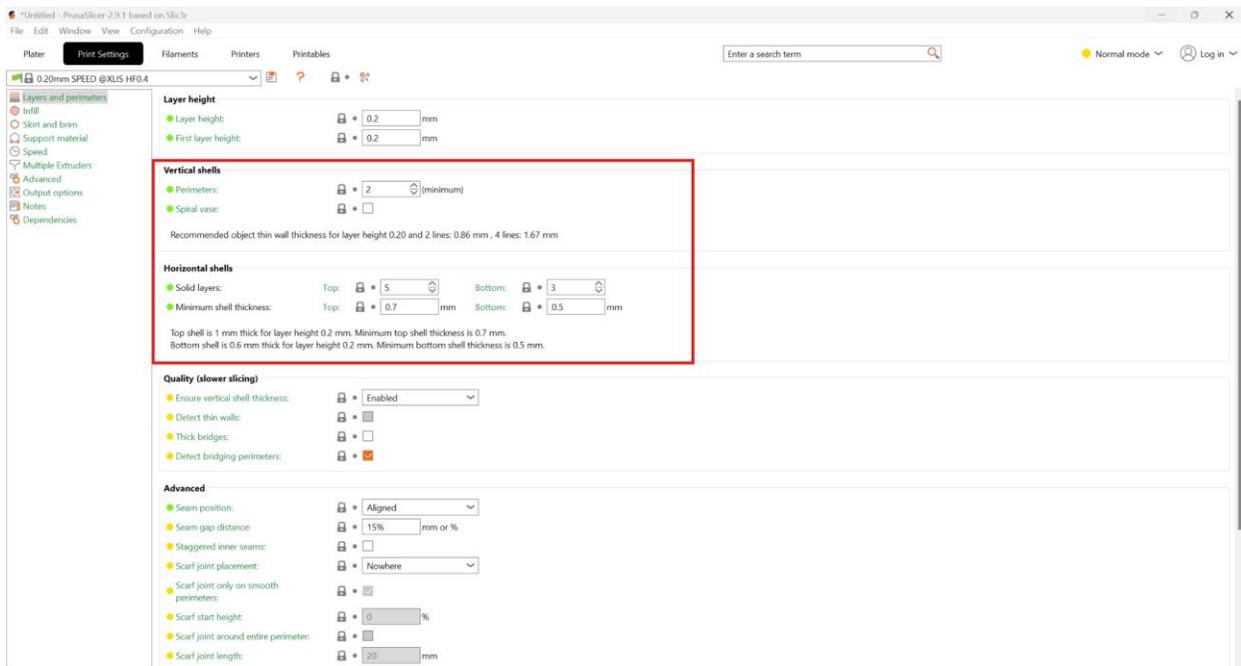**Note:** Your geometry will behave differently depending on the support system used.

4. Infill:

- **5-10%** for draft prints.

- **10-20%** for standard prints.
- **40-50%** for functional prints (most effective with cubic and gyroid patterns).

Note: High-strength prints should have additional shells (3 to 6). Additional shells increase print time.

5. Speed Settings:

- **100 mm/s** (standard printing speed).
- **400 mm/s** (maximum with Input Shaper).

Note: Input Shaper allows speeds up to **400 mm/s** with precision. Slower speeds (**100 mm/s**) can further improve print quality.
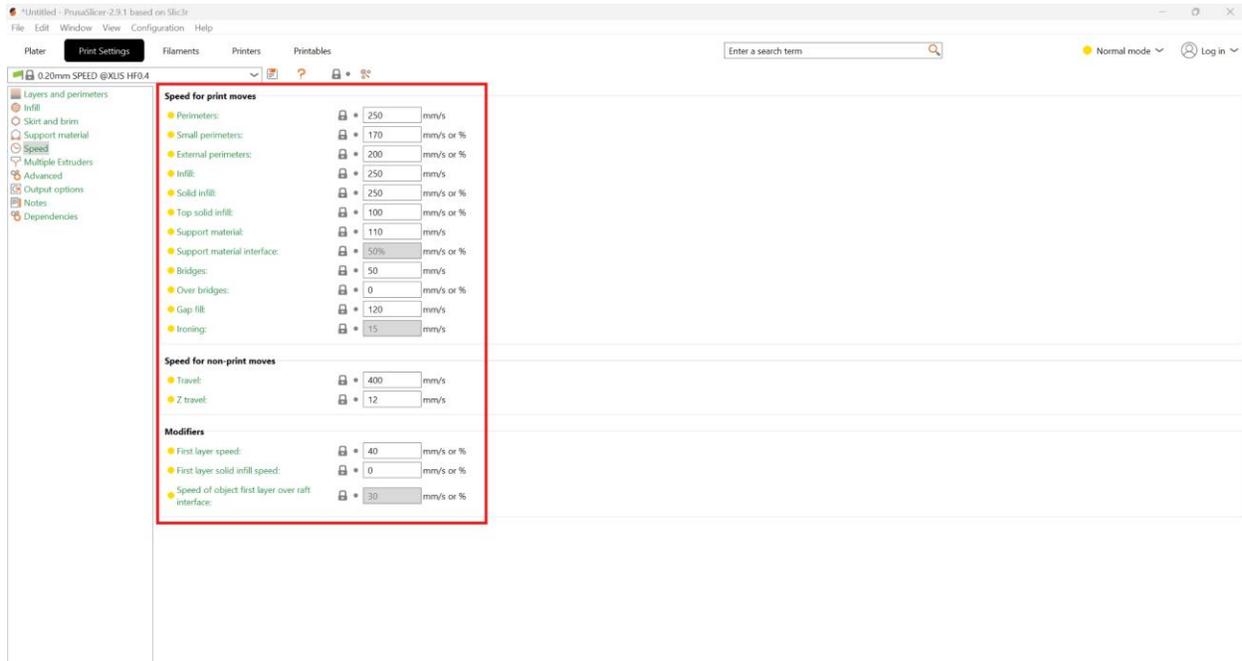
Perimeter Speed:

- Determines how fast the printer moves when printing outer and inner walls.
- **Slower speeds improve surface quality.**

Infill Speed:

- Controls speed for internal structures.
- **Faster speeds reduce print time without affecting surface quality.**

Balancing Speed and Quality for Multicolor Printing:

- Lower perimeter speeds ensure **smooth color transitions** and **clean details**.
- Higher infill speeds reduce print time.
- Use **Input Shaper** to counteract vibrations.
- Experiment with speed settings based on filament type and model complexity.

Step 5: Assign Materials for Multi-Material Prints

**Simple Multicolor Printing (by Surface Coloring):**

1. Under **Plater (Main Workspace)**, select parts to be printed with different materials.
2. Use the **Material Editor** on the left panel to assign each part to an extruder (1–5).
3. **PrusaSlicer** will generate a toolpath for each material.

Note: Multi-material printing is encouraged with the XL, except when objects exceed other printer bed capacities.
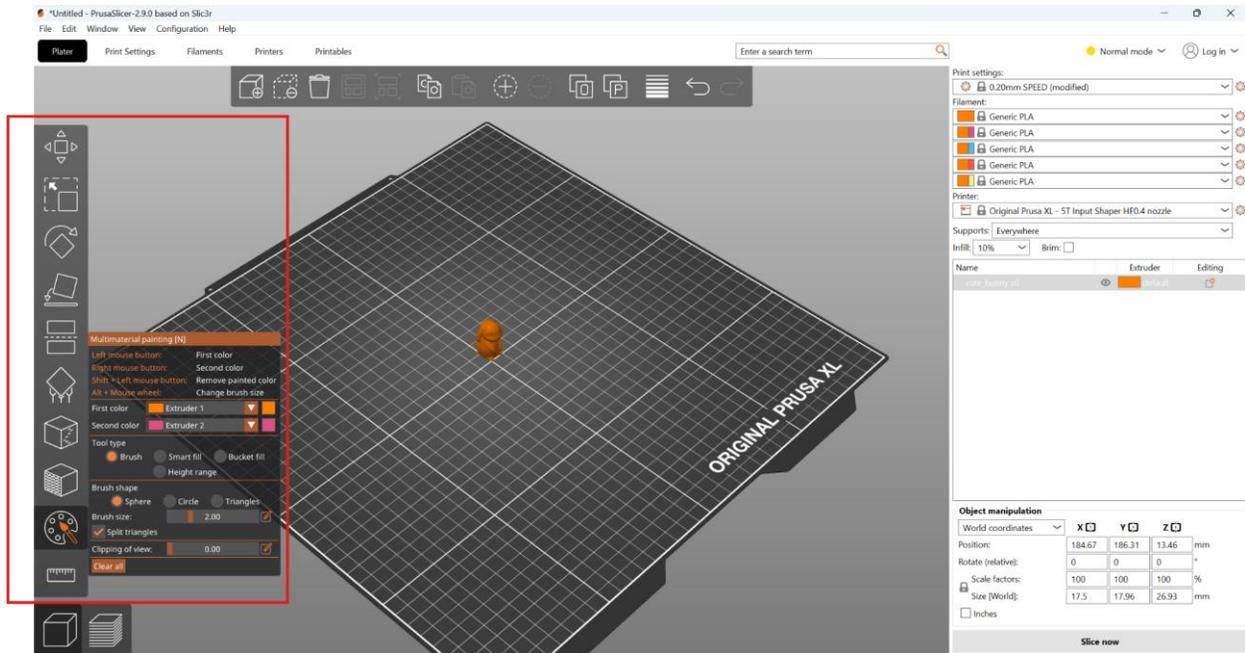

Step 6: Preview the Print

1. Click on the **Preview** tab.
2. Inspect for potential issues:
   - Missing supports or incorrect layer transitions.
   - Incorrect color layers in multi-material prints.
3. Simulate the print process to ensure correct slicing.
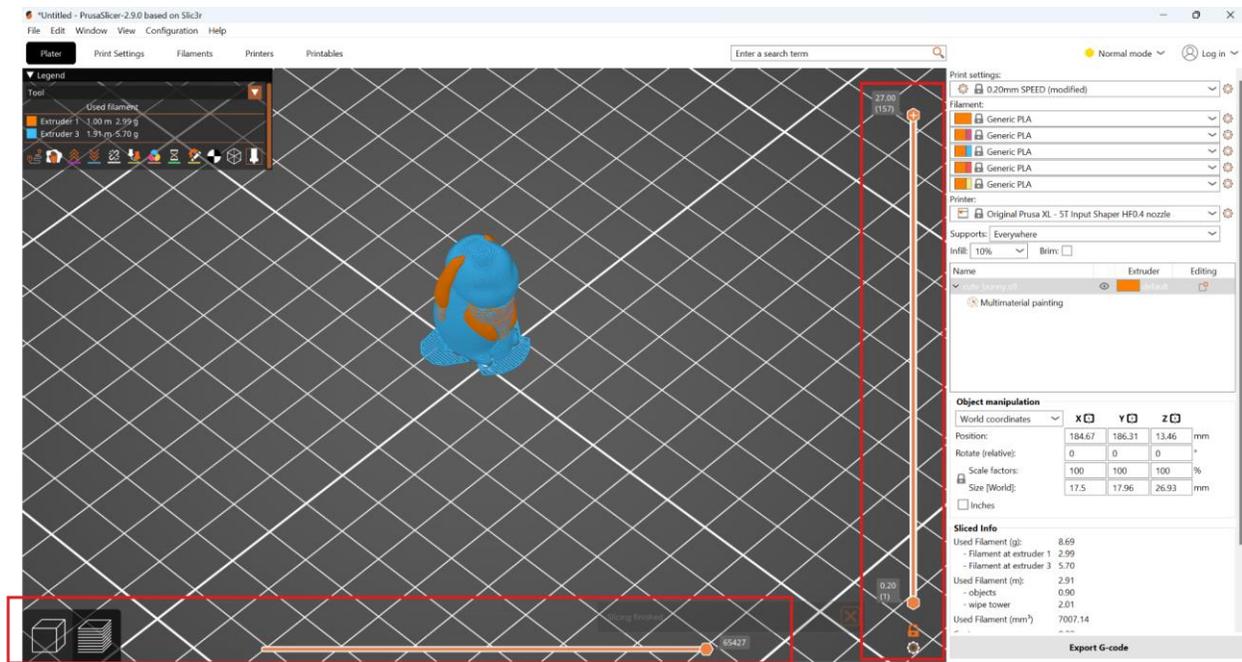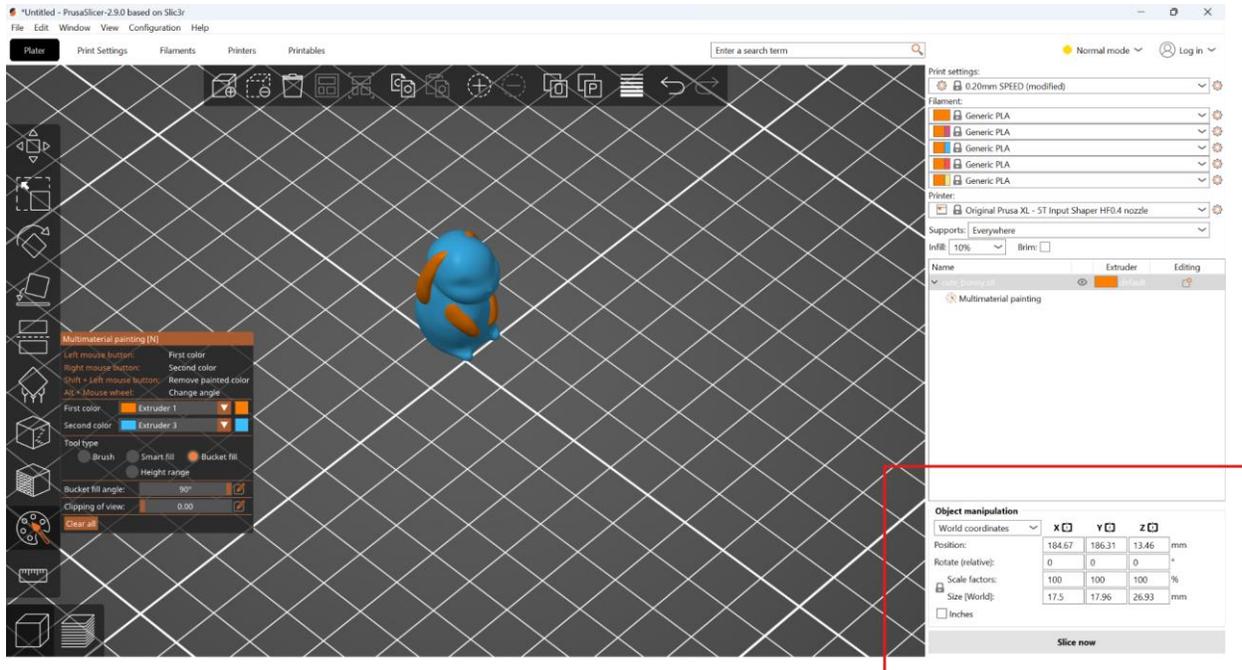
Step 7: Export G-code

1. Click **Export** in the bottom-right corner.
2. Choose **G-code** for Prusa XL and save it to your storage device.
3. **Submit the G-code** to Simply Print and await AT team approval.
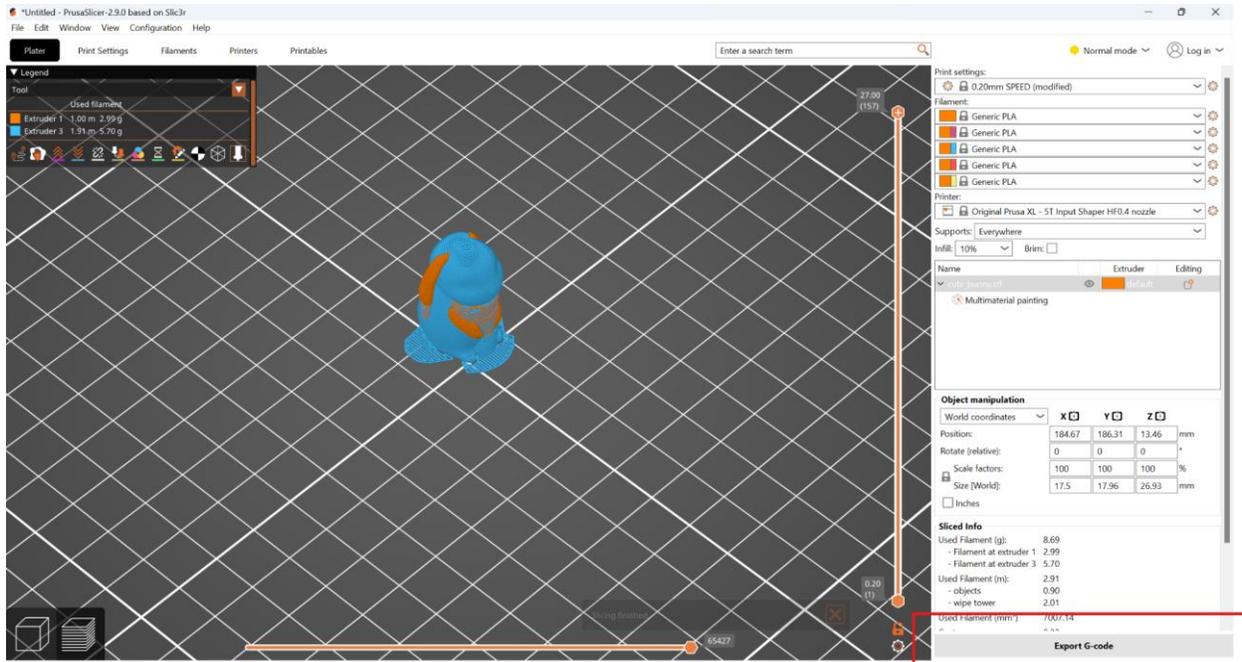
Step 8: Prepare the Printer

Important: Students engaging directly with the machine **must consult AT before proceeding.**

1. Load the selected materials into each extruder.
2. Preheat the bed and extruders.
3. Load the G-code into the **Prusa XL**.

Step 9: Start Printing

1. On the **Prusa XL**, navigate to **Print**.
2. Select the **G-code** file.
3. Confirm settings and start the print.

Step 10: Post-Processing

1. Remove the model carefully from the bed.
2. If using **PVA supports**, dissolve them in water. **Do not dispose of water into the main drain.**
3. Clean excess filament from extruders and ensure bed cleanliness for future prints.