

Raise3D Premium PVA Practice Guide

Summary

Raise3D Premium PVA is a PVA-based support material that can be used for PLA-based, TPU-based, Nylon-based and PETG-based modeling materials.

Recommended Printing Conditions

| | |
|---------------------------------------|--|
| Recommended printing temperature (°C) | 220 - 230 |
| Recommended printing speed (mm/s) | 45 - 90 |
| Build plate surface | BuildTak® with PVA (glue stick) coating |
| Build plate temperature (°C) | Room temperature - 70 |
| Model cooling fan | Not sensitive – use the settings for the modeling material |

Printing Guidelines

1. When a raft is used, always print the raft with Raise3D Premium PVA, for the raft-model distance (“air gap”): 0 mm.
2. Support settings (example in IdeaMaker):
 - Recommended support density: 20% - 30%.
 - Support infill angles: 0° and 90°.
 - Dense support layers: 3 - 5.
 - Dense layer infill: 100%.
3. Ooze shield outlines: set to 1-3.
4. Upper/lower vertical separation layers: set to 0.
5. Maximum overhang angle: 0°.
6. Support-removal procedure:
 - Place the printed model with support in water bath at room temperature
 - In 2-5 min the support should turn into a “gel-like” state. You can take the model from the bath and remove support as much as possible.
 - If this process can't get good result, you better heat the water up to 40°C to accelerate the process and repeat again



- Place the model (with residual support) back to the water bath. then change the water every 2 - 3 hours (repeat 2 -3 times), the support should completely dissolve. Mechanical stirring and ultrasound agitation can be used to further expedite the process.
- If this process can't get good result, you better heat the water up to 40°C to accelerate the process and repeat again

*The material described above is still developmental and meant to be used for experimental testing only. It may not represent the properties and quality of the final product.

