PolySupport™ User Guide

polymaker

Key Features

PolySupport™ is an entirely new support material for desktop FDM/FFF 3D printers. Unlike any existing support material, PolySupport™ generates support structure that can easily and cleanly be removed by hand or any simple tools. It redefines what is possible with desktop 3D printing.
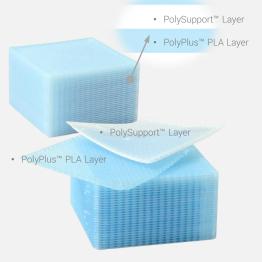So how does PolySupport™ work? It was designed to have 3 main features:

• The cube printed with alternating layers of PolySupport™ (white) and PolyPlus™ PLA (translucent blue)

■ Optimized interlayer adhesion

We have carefully optimized the interlayer adhesion between PolySupport™ and PLA – so the bond is sufficient to support the structure being built, while still weak enough to ensure easy and clean support removal from the finished part.
Here is a simple demonstration of this feature. The photo to the right shows a cube printed with alternating layers of PolySupport™ (white) and PolyPlus™ PLA (translucent blue). As you can see, each layer can be peeled apart easily without affecting the adjacent layer.

• PolySupport™ Layer
• PolyPlus™ PLA Layer
• PolySupport™ Layer
• PolyPlus™ PLA Layer

• Easy to break away by hand

■ Easy to break away

PolySupport™ features good inherent strength and medium stiffness, allowing the support structure to be removed easily and in some cases as a single piece, greatly facilitating the overall removal process of PolySupport™.

■ Excellent for large support structures

Compared to soluble support materials, PolySupport™ is much more simple to use. The support structure can be removed very fast with no need for any liquid solvents, which makes it especially good for parts with large overhangs.

• Large support structures can be removed quickly and easily

polymaker

Printing with PolySupport™

Basic printing conditions

When used on dual- or single-extrusion printers, it is recommended that PolySupport™ be printed under the following conditions:

Nozzle temperature: 225 – 235 ℃

Printing Speed: ~ 50 mm/s or less

Notice:

☐ Make sure that all the printing speeds (including the speeds for shells/outlines, infill, raft, etc.) are no more than 50 mm/s

☐ Make sure that all the support structure (including the raft) is printed with PolySupport™.
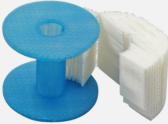
• PolySupport™ can be used on both dual and single-extrusion printers. It can be used either as a dedicated support material (dual-extrusion), or as both the modelling and support material (single-extrusion). In both cases the support structure can be easily removed. So no matter what printer you may have you can rely on PolySupport™ to support your creations.

• PolySupport™ Filament

• Open support： the open support structure of the F-22 Raptor model can be removed directly in a single piece

Classification of support structures

Before we discuss the printing settings for PolySupport™, it is important that we identify the two different types of support structures. Generally speaking, we classify support structures into two categories: open support and closed support.

☐ Open support: support that does not enclose any portion of the model. Open support can usually be removed in a single piece. The support under the F-22 Raptor model (the photo to the left) is an example of open support.

☐ Closed support: support that encloses some portion of the model. In this case the support must be broken apart and separated in order to be removed from the finished part. For example, in the spool model (the photo to the left), the hub is fully surrounded by the support structure, which can't be removed unless it is broken apart.

It should be noted that there is no clear distinction between open support and closed support. Sometimes as the support settings are varied, an open support can turn into a closed support and vice versa.

• Closed support: the closed support structures have to be broken up to be removed.

• Part of the spool model is enclosed by support structure.

• The support structure needs to be broken up to be removed.

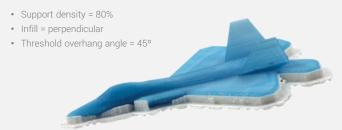polymaker

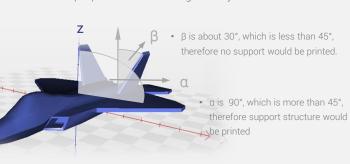# Printing with PolySupport™

Dual-Extrusion

Open Support

☐ Support Density: since open support can be removed in a single piece, we recommend that high support density (e.g. 40% or more) be used to obtained better surface finish. In some cases the density can even be as high as 100%.
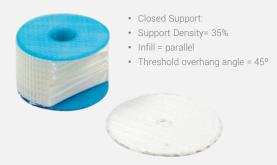
- Perpendicular infill facilitates a better surface quality

- The open support structures of F-22 Raptor fighter can be removed directly.

☐ Threshold Overhang Angle: most 3D printers can print overhang angles (the angle between the model and the Z axis) up to 45º, it is recommended that the threshold overhang angle be set at 45º. In this case support is only generated for parts of the model with overhang angles higher than 45º as seen in the illustration to the right.

- Support density = 80%
- Infill = perpendicular
- Threshold overhang angle = 45º

☐ Support Infill: in some slicer programs, the support infill can be set to have all the extruded lines of each layer either running in the same, parallel direction, or being perpendicular to the previous layer. The "parallel" infill facilitates removal, whereas the "perpendicular" infill usually leads to better support performance and surface quality. For open support, since the support structure can be removed relatively easily, perpendicular infill is generally recommended.

- β is about 30°, which is less than 45°, therefore no support would be printed.
- α is 90°, which is more than 45°, therefore support structure would be printed

- Closed Support:
- Support Density= 35%
- Infill = parallel
- Threshold overhang angle = 45º

Closed Support

☐ Support Density: It recommended that a support density in the range of 20%-50% is used for closed support. Using a support density that is too high will make the removal of the support structure difficult.

☐ Support Infill: for closed support, perpendicular support infill can often result in the difficulty of removing the support structure. Therefore it is recommended that the infill always be parallel for closed support.

☐ Threshold Overhang Angle: similar with open support, a threshold overhang angle of 45º is generally recommended.

- Easy to remove when break up the support structure.

polymaker

Single-Extrusion

PolySupport™ can be printed on both dual and single extrusion printers. When printing on a single-extrusion 3D printer, please make sure that proper printing temperature, speed and support settings are used.

☐ When setting the support density, support infill and threshold overhang angle: please follow similar guidelines for dual-extrusion printing.

☐ Since PolySupport™ exhibits lower interlayer adhesion than PLA, high model infill densities (e.g. > 50%) are recommended to obtain better part strength.
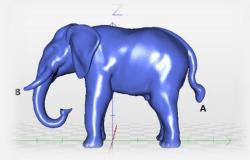
- Support density = 30%
- Infill = Parallel
- Threshold overhang angle = 40º

- Support density = 35%
- Infill = Parallel
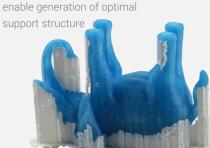- Threshold overhang angle = 45º

Tips & tricks for complex models

The elephant model to the right is relatively complex, and requires a large amount of support to print it right. The tip of the tail (A) and the tip of the ivory (B) are both weak points that can easily turn your model into a failed print. In this orientation, closed support is formed around the tip of the trunk which may make the removal difficult.

A solution for this is to change the orientation of the model by placing it up side down, like the picture to the left. This leads to multiple benefits are listed below.

- Placed up side down to enable generation of optimal support structure

☐ Both the tip of the tail and the tip of the ivory now do not need support.

☐ the support underneath the trunk now becomes open support, facilitating easier removal.

☐ the amount of support material needed has been greatly minimized.

The result? A perfectly printed, cute elephant!

- As-printed with support structure still on

- Printed model with support structure removed

polymaker

polymaker

Innovation Simplified