

# Tullomer™ Multi-Material Printing Procedure

---



## Background

This guide outlines an experimental procedure for multi-material printing with Tullomer™ using Bambu Studio for an H2D. We use Tullomer for the part infill and another material for the part walls to take advantage of Tullomer's outstanding in-plane strength.

This procedure uses an in-development feature in Bambu Studio that allows users to assign features to different filaments, but this result can be replicated for any dual-nozzle printer by manually modifying the part in any CAD software to make a shell/core multi-material part.

# 1 Prepare materials

Tullomer is a very stiff filament and is not supported for Bambu Lab AMS, so it should be loaded as an external spool.

Consider desired material properties when selecting a wall material. Materials with high interlayer strength (such as PA, PC, or PET) aid Tullomer in the Z-direction by constraining it within a rigid shell.

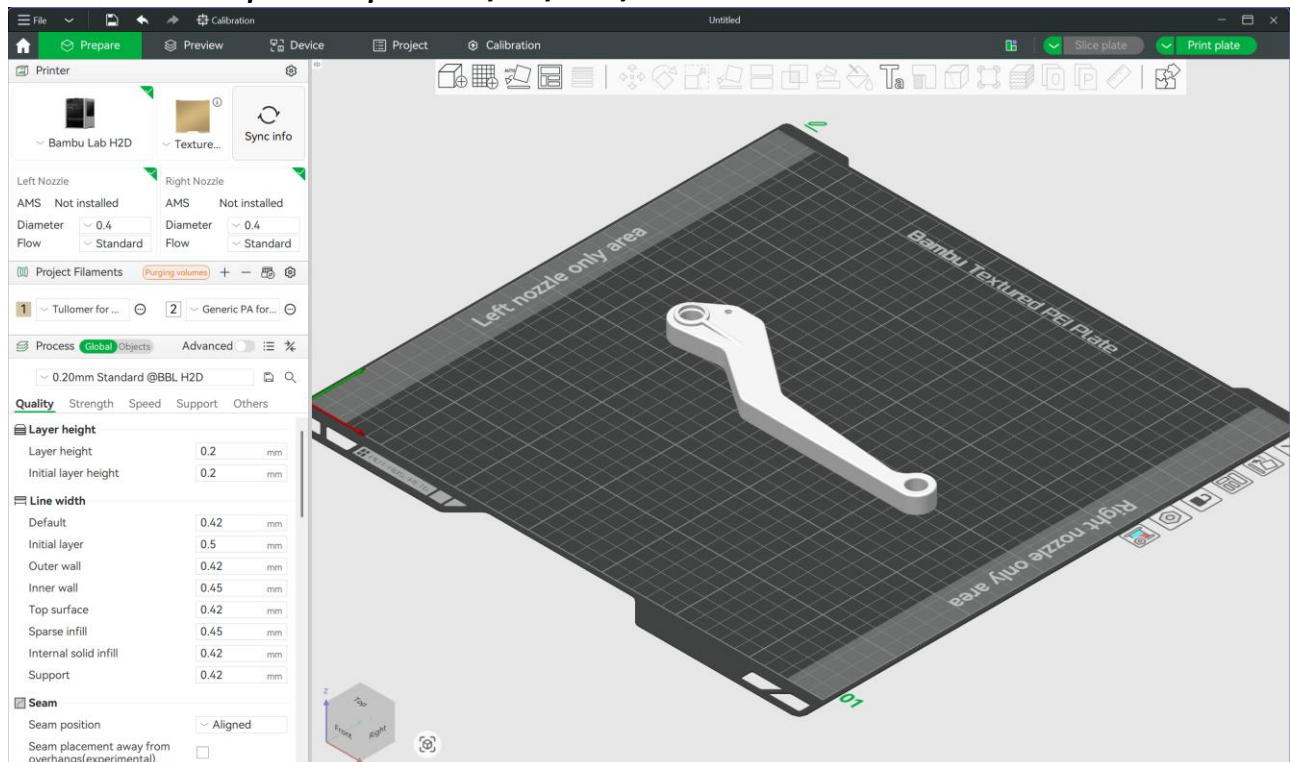
## Material setting recommendations

- *Tullomer nozzle temperature:* 300 °C to 330 °C
- *Wall material nozzle temperature:* Refer to the material and brand temperature recommendation
- **Important note:** When printing with materials prone to absorbing moisture, such as PET or PA, **leave the printer door open** or **reduce the chamber temperature to 20 °C to 40 °C**. This minimizes risk of heat creep that may cause swelling inside of the extruder which can seriously damage a printer.

# 2 Import part file

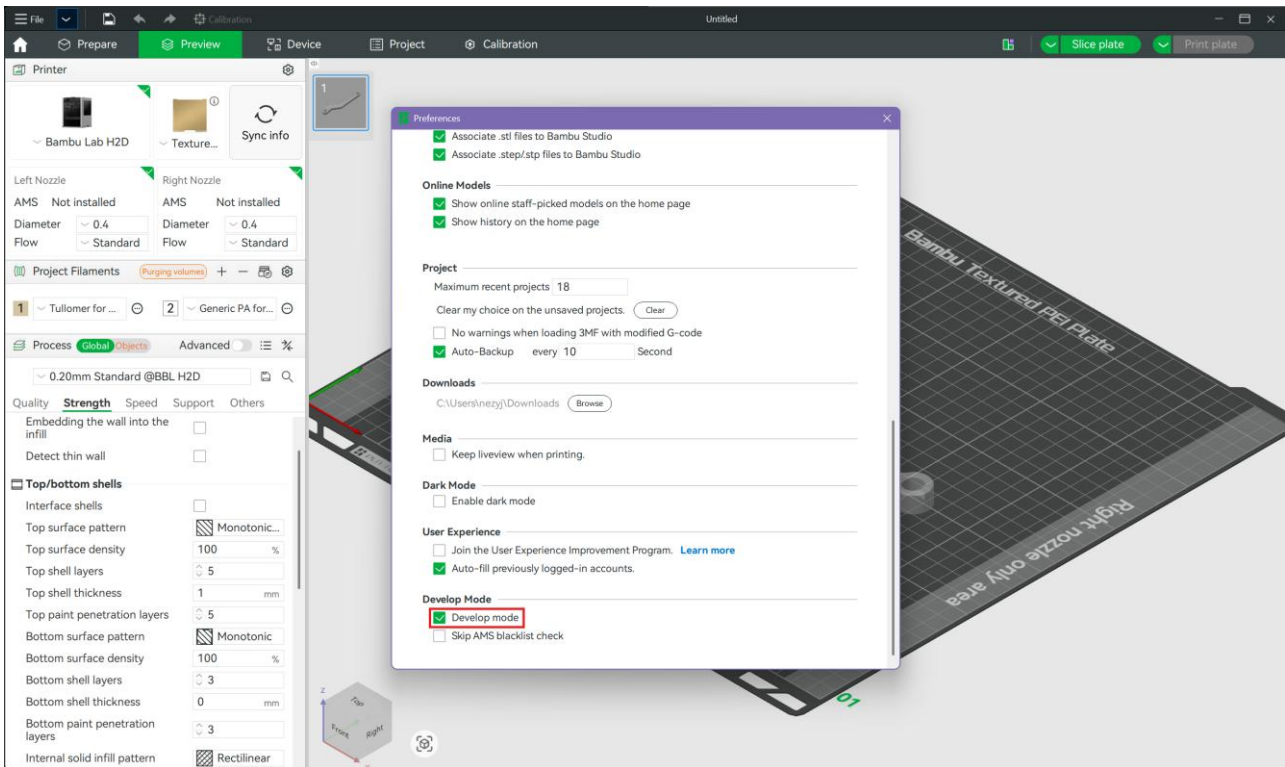
Import and arrange your part into Bambu Studio as normal as a 3D printable file, such as **.stl**, **.step**, or **.3mf**

**Ctrl + I** or **File > Import > Import 3MF/STL/STEP/SVG...**



### 3 Switch to Develop Mode

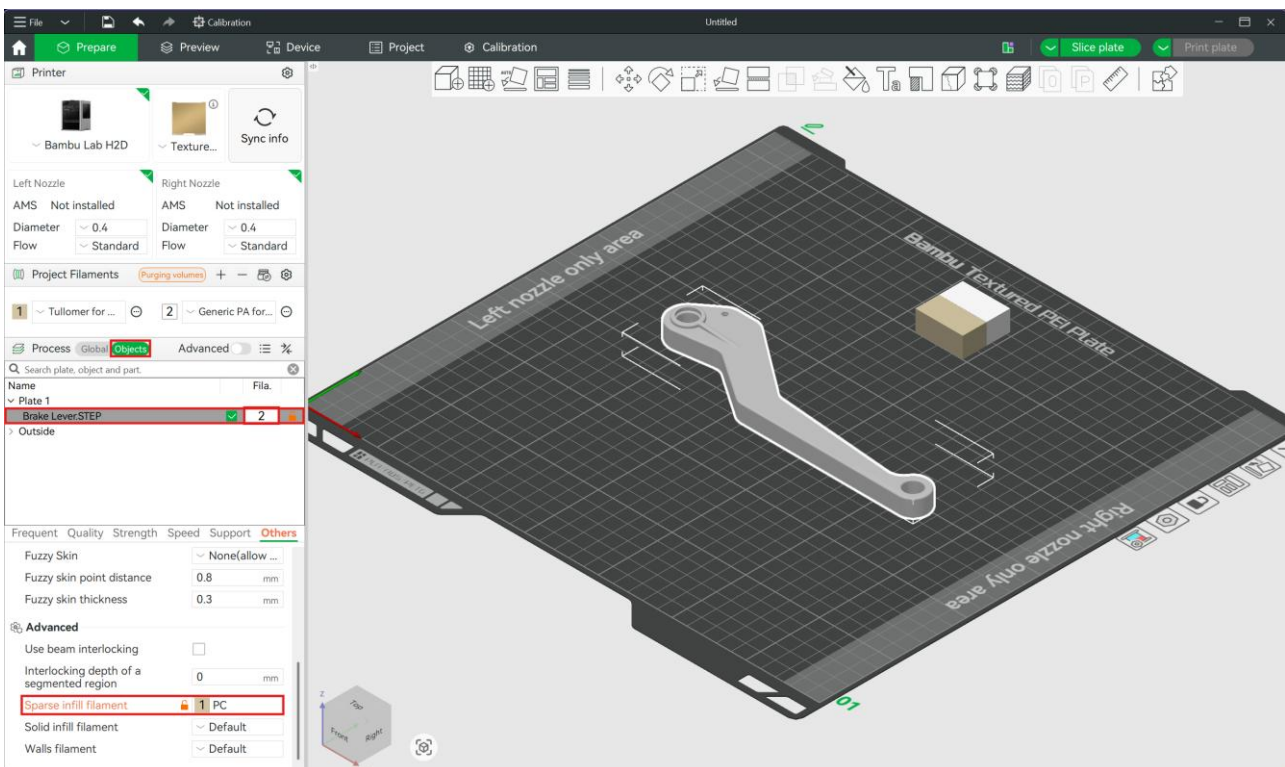
**Ctrl + P** or **Preferences > Develop Mode**



### 4 Assign filament

Under **Objects**, select the part and assign the part to your wall material.

With the part selected, in the **Others** tab, under the **Advanced** settings, assign **Sparse infill filament** to Tullomer.



## 5 Adjust settings

### Print setting recommendations

- *Wall loops: 2-4*
- *Sparse infill density: 100%*
- *Sparse infill pattern: Concentric*
- *Top/Bottom shell thickness: 1-2mm*

## 6 Slice plate and verify results

