

# Orca / Bambu Slicer 0.2 + 0.8er Nozzle Einstellungen

**Hier hab ihr eine grobe Vorlage, wenn ihr mit einer 0.2er und 0.8er nozzle drucken wollt.**

*Vielen dank an Flashtu, der diese Einstellungen erarbeitet hat.*

## **0.2 mm Nozzle Slicer Software Settings**

Layer Height - 0.08 mm First Layer Height - 0.12 mm

Layer/Extrusion Width - 0.15 - 0.25 mm

Extrusion Multiplier - Can be adjusted between 0.95 - 1.05

Speed - Same as with 0.4mm nozzle

Temperature - Same as with 0.4mm nozzle Exotic Materials (Wood, PETG, TPU) and Home Made

Filaments - NOT Recommended (Increased risk of nozzle clogging even with bad quality PLA)

## **0.8 mm Nozzle Slicer Software Settings**

Layer Height - 0.6 mm First Layer Height - 0.6 mm

Layer/Extrusion Width - 0.6 - 1 mm

Extrusion Multiplier - Can be adjusted between 0.95 - 1.05

Speed - Lower than with 0.4mm nozzle Temperature - Slightly higher than with 0.4mm nozzle

Exotic Materials (Wood, PETG, TPU) and Home Made Filaments - Recommended (reduced risk of nozzle clogging)

---

Revision #1

Created 16 June 2023 19:21:37 by Robin

Updated 16 June 2023 19:24:45 by Robin