

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