Advanced Overview to Improve
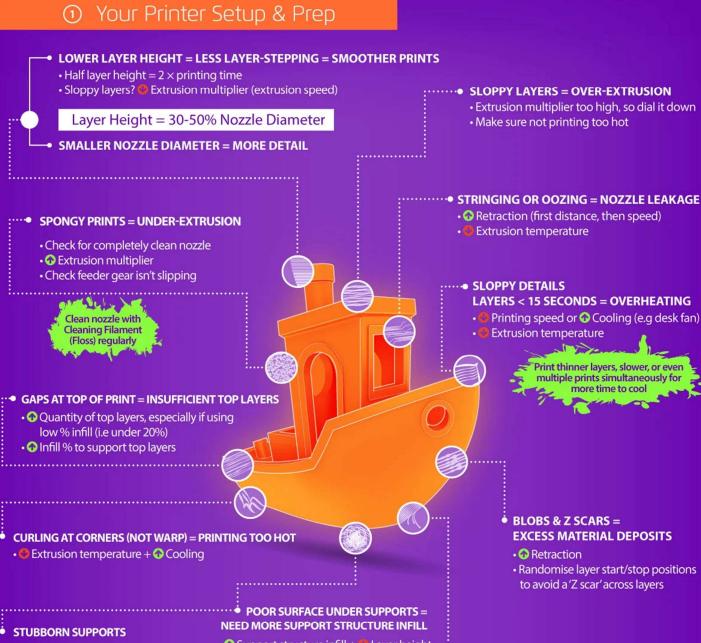# Your 3D Print Finish Quality

rigid.ink

So, your printer's now working 100% reliably, but your prints aren't as nice as they could be.

Here's how to make your 3D prints look not-so-3D printed...

① Your Printer Setup & Prep

LOWER LAYER HEIGHT = LESS LAYER-STEPPING = SMOOTHER PRINTS
- Half layer height = 2 × printing time
- Sloppy layers? ⬇ Extrusion multiplier (extrusion speed)

Layer Height = 30-50% Nozzle Diameter

SMALLER NOZZLE DIAMETER = MORE DETAIL

SLOPPY LAYERS = OVER-EXTRUSION
- Extrusion multiplier too high, so dial it down
- Make sure not printing too hot

SPONGY PRINTS = UNDER-EXTRUSION
- Check for completely clean nozzle
- ⬆ Extrusion multiplier
- Check feeder gear isn't slipping

Clean nozzle with Cleaning Filament (Floss) regularly

STRINGING OR OOZING = NOZZLE LEAKAGE
- ⬆ Retraction (first distance, then speed)
- ⬇ Extrusion temperature

SLOPPY DETAILS
LAYERS < 15 SECONDS = OVERHEATING
- ⬇ Printing speed or ⬆ Cooling (e.g desk fan)
- ⬇ Extrusion temperature

Print thinner layers, slower, or even multiple prints simultaneously for more time to cool

GAPS AT TOP OF PRINT = INSUFFICIENT TOP LAYERS
- ⬆ Quantity of top layers, especially if using low % infill (i.e under 20%)
- ⬆ Infill % to support top layers

BLOBS & Z SCARS =
EXCESS MATERIAL DEPOSITS
- ⬆ Retraction
- Randomise layer start/stop positions to avoid a 'Z scar' across layers

CURLING AT CORNERS (NOT WARP) = PRINTING TOO HOT
- ⬇ Extrusion temperature + ⬆ Cooling

STUBBORN SUPPORTS
- Make supports easier to remove:
  ⬆ Upper Vertical Separation Layers in slicer

Removing supports can leave rough surfaces

POOR SURFACE UNDER SUPPORTS =
NEED MORE SUPPORT STRUCTURE INFILL
- ⬆ Support structure infill + ⬇ Layer height

TIP: Use receipt paper to level bed, thinner gap leaves smoother bottom print surface.

BETTER OVERHANGS / BRIDGES =
⬇ LAYER HEIGHT + ⬆ COOLING + ⬇ PRINT SPEED
- With care, over 45° can be achieved without supports