

Complete Guide to Photocuring 3D Printing for Dentistry: Printing Parameters

Dental 3D printing has grown from an experiment into a core production method for modern dental labs and clinics. Whether you make permanent crowns, surgical guides, denture bases, or orthodontic models, the final quality comes down to three things: choosing the right resin, dialing in your equipment parameters, and knowing how to troubleshoot when problems arise. This guide combines Zonmed's resin-selection experience with photocuring workflow training material, following the complete production chain — material selection → slicing → machine setup → parameters → monitoring → post-processing → troubleshooting → equipment maintenance — and flags the key parameters, operating points, and pitfalls at each step to help you deliver clinic-ready parts stably and reproducibly.

Part 4 · Equipment Settings & Printing Parameters

4.1 Exposure Calibration (Test-Block Method)

Exposure time need not be guessed — calibrate it with a standard test block. Using a standard 4 mm test block as an example:

- If a 2.5s single-layer exposure yields a test block smaller than 3.96 mm — exposure is too low; increase single-layer exposure in 0.5s steps.
- If a 2.5s single-layer exposure yields a test block larger than 4.05 mm — exposure is too high; decrease single-layer exposure in 0.5s steps.

Adjust gradually until the test block falls within the 3.96–4.05 mm passing range, then set the production single-layer exposure accordingly.

4.2 Layer Thickness

50 μm (0.05 mm) is the dental standard, balancing speed and accuracy. Drop to 25–30 μm only when marginal fit or fine occlusal detail demands it, accepting longer print times. Thicker layers leave visible stair-stepping on curved surfaces.

4.3 Bottom / Transition Layers

Bottom layers anchor the model to the platform; typically 5–8 layers with longer bottom exposure, then a gradual transition to normal exposure to avoid over-curing and brittleness in the first layers. Insufficient bottom layers is a leading cause of detachment.

4.4 Post-Curing

Post-curing completes polymerization, bringing the resin to its final strength, hardness, and biocompatibility. First clean and dry thoroughly, then cure in a UV chamber at the manufacturer's recommended time and temperature (commonly 1–3 minutes, flipping every half minute); some materials require post-curing at about 60 °C for best performance. Under-curing leaves residual monomer and reduces biosafety; over-curing makes parts brittle and yellowed. Always follow the resin's IFU.

4.5 Orientation

- Tilt crowns, bridges, and slender parts 25–45° to reduce cross-section per layer and lower peel stress.
- Keep critical fit surfaces (margins, intaglio) away from supports and facing away from the platform.
- Angle flat surfaces to avoid trapping resin or forming adhesion with the release film.

4.6 Support Strategy

- Support load-bearing and unsupported areas; avoid placing supports on visible or fit-critical surfaces.

- Use finer contact tips on delicate features and thicker supports on heavy overhangs to prevent flexing during peel.
- Use enough supports to resist peel force, otherwise overhangs warp or tear away mid-print.
- Leave a small gap between support and model for easy finishing and fewer marks.

4.7 Peel Force

Peel force is the resistance when a cured layer separates from the release film (LCD) or vat (SLA); it rises with cross-sectional area and layer count. You can effectively reduce it by lowering the cross-section per layer (via orientation), keeping resin temperature stable, and avoiding large flat areas that create suction. The slicer's peel/separation parameters adjust how fast the platform lifts away — slower separation means lower peak force.

4.8 Lift & Retract (Lift Speed)

- Lift speed — Slower reduces peel force and protects delicate features; faster shortens time but risks tearing.
- Retract speed — Can be faster than lift, but too fast traps bubbles; allow a short rest before the next exposure so resin settles.

On DLP / LCD printers, a two-stage lift (slow breakaway, then fast travel) is a reliable way to balance speed and success rate.

4.9 Parameter Downloads

Don't reinvent validated settings. Download the Zonmed PRS parameter guide matching your printer brand and resin, load it into your slicer, and adjust only small variables. Per-brand guides cover Ackuretta, Anycubic, ASIGA, Creality, Dekema, Dentafab, Dreve/Otoplastik, Elegoo, Flashforge, Formlabs, HeyGears, NextDent, Phrozen, RapidShape, Rayshape, Shining3D, SprintRay, Uniformation, and UNIZ.