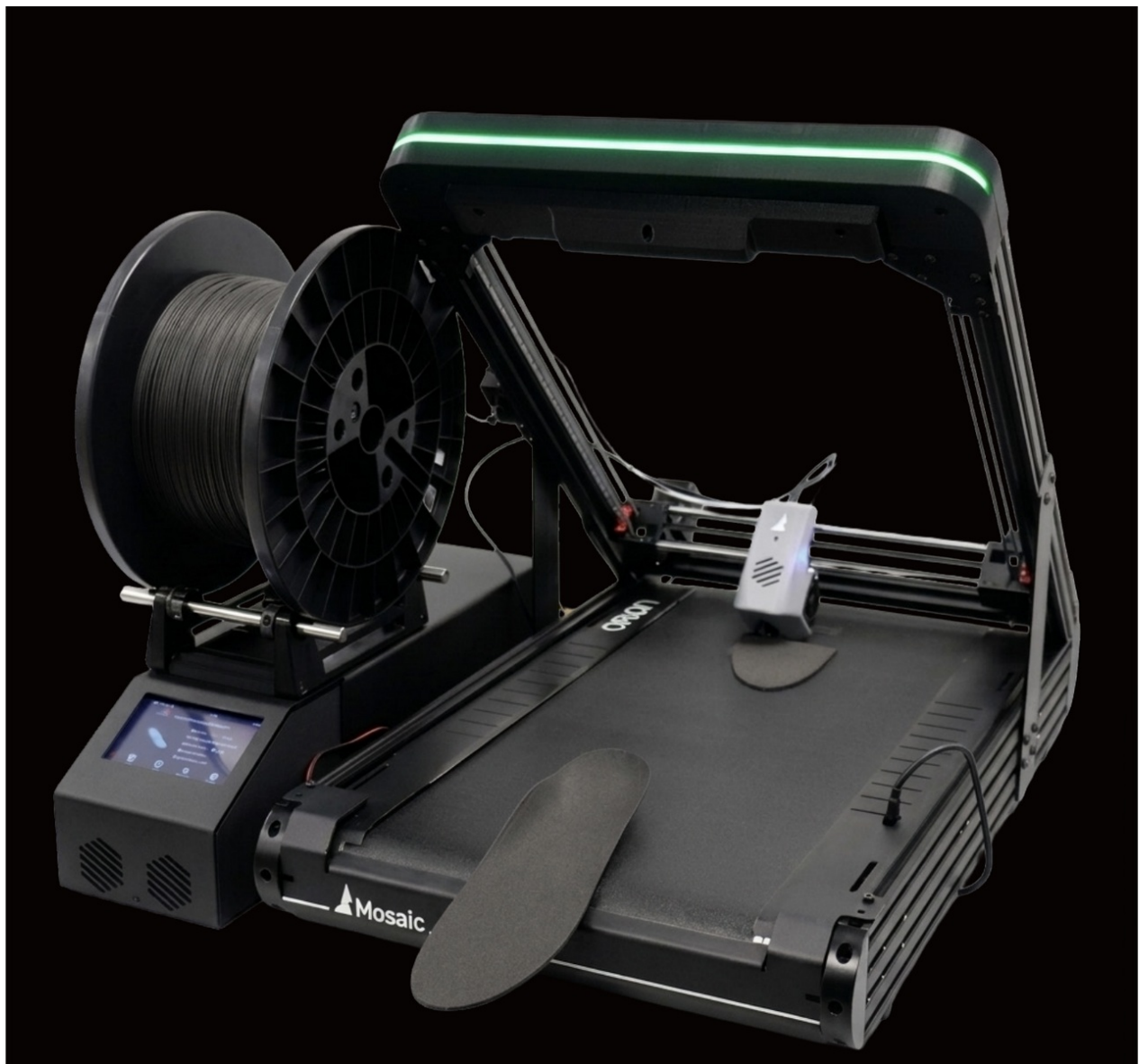




Printing with Aero

Complete procedure for setting up, preparing the bed, and successfully printing soft, accommodative orthotics using Mosaic Aero filament.

Written By: Mosaic Support



Step 1 — Understand Aero Material Settings



- Mosaic Aero is a flexible material engineered specifically for comfortable, cushioned orthotic insoles.
- Standard printing temperature for Aero is 240°C at the nozzle.
- The heated bed remains at room temperature (0°C) throughout the entire print.
 - ❗ The part cooling fan should run continuously at 100% to keep flexible walls crisp and prevent sagging on curves.

Step 2 — Prepare and Clean the Belt Surface



- Ensure the printer belt is clean, dry, and cool to the touch before printing.
- Wipe the printing area with isopropyl alcohol (rubbing alcohol) to remove any finger oils or dust.
- Apply a thin, even coat of nanopolymer bed adhesive mixed 50/50 with rubbing alcohol using the application brush.
 - ✦ One coat of adhesive typically lasts for about 3 insole prints before needing a light touch-up.

Step 3 — Load Aero Filament into the Printhead



- Mount the Aero filament spool onto the side spool holder so it feeds smoothly from the top.
- Snip the leading tip of the filament at a clean 45° angle with snips.
- Preheat the extruder nozzle to 240°C from the Temperature menu on the touchscreen.
- Feed the filament through the guide tube until you feel it enter the extruder gears, then tap Extrude on the screen until a smooth strand flows.

Step 4 — Finalize Project Settings



- Confirm that your models are aligned with our STL requirements
- Open your insole file in Stryde Print and select the desired Aero material profile (Single-density soft shell or Multi-density soft shell)
- After you've selected your models and project parameters, you're ready to slice. Stryde will automatically position the insole near the center-front of the belt, with the heel pointing toward the front origin.
 - ① Starting the print from the heel prevents rough surfaces on the heel cup.

Step 5 — Start Print and Monitor the First Layers



- Send the sliced file to Orion using Stryde Print or the Fluidt queue.
 - Watch the printer lay down the initial purge line and the first 3 layers.
 - Verify that the plastic squishes flat against the belt with no peeling or gaps.
- ⚠ If the plastic beads up round instead of flattening, adjust your Y-offset slightly closer to the belt via the Fluidt dashboard.