

3D printing at a larger scale LFAM requires special attention to the printer and its settings. This handbook is designed to help you master the unique requirements of large-format 3D printers, specifically the Ginger V1.3 Beta 3D Printer. You'll learn how to manage potential challenges and optimize your printing process.

THE PROCESS

- 1. **Design Geometry:**
  - Create the 3D model using your preferred design software.
- 2. **Slice the Geometry:**
  - Import the model into your slicer software and prepare it for printing.
- 3. **Transfer File to Printer:**
  - Save the sliced file and transfer it to the printer via SD card or wireless connection.
- 4. **Preheat Printer & Bed:**
  - Begin preheating both the printer and the bed to the appropriate temperatures.
- 5. **Extrude/Purge Material:**
  - Extrude and purge material to ensure the nozzle is clean and ready.
- 6. **Start Print:**
  - Initiate the print from the printer's interface.
- 7. **Set Mixer Settings & Flow:**
  - Adjust the mixer settings and flow rate as needed during the print.
- 8. **Print:**
  - Allow the printer to complete the job, monitoring if necessary.
- 9. **Cool Down and Take Print:**
  - Let the print cool down, then carefully remove it from the bed.

STARTING A PRINT

Follow these steps to begin a print:

- 1. **Power On:** Turn on the printer.
- 2. **Preheat:** Start heating the bed and the extruder.
- 3. **Set Temperature:** Adjust the temperature as needed, then press the button to confirm.
- 4. **Auto Feeding:** Turn auto feeding "on."
- 5. **Fan Control:** Ensure fans are turned "off."
- 6. **Bed Preparation:** Apply glue, tape, or other adhesive to the print bed.
- 7. **Purge:** Purge material to clean the extruder.
- 8. **Home the Printer:** Home the printer to ensure proper positioning.
- 9. **Select File:** Choose your print file and start the print.
- 10. **Mixer Settings:** Set the mixer ratio (e.g., "93/7%").
- 11. **Flow Adjustment:** Adjust the flow rate during the brim phase.
- 12. **Cooling:** Turn fans on after the first 4 layers (set between 60-100%).
- 13. **Bed Control:** Turn off the bed when needed.
- 14. **Print:** Enjoy your print!

INFORMATION

3D PRINTING PRODUCTION PLATE

Date	Operator	File				
Mixer	Flow	Slicer				
		Grasshopper	Cura	Prusa	Orca	
Material			Nozzle			
	PLA	PETg	ABS	ASA		Adapter
Zone 1						2 mm
Zone 2						3 mm
Zone 3						5 mm
						8 mm

Pict. 3D-Printing Production Plate for the Ginger Printer to 3D-Printing

IMPORTANT PRE-PRINT CHECKS

Before starting a print, ensure the following:

- 1. **Nozzle Size:** Verify that the nozzle size matches your slicing settings.
- 2. **Nozzle Cleanliness:** Clean the nozzle before homing the printer.
- 3. **Power Setup:** Avoid plugging the bed and printer into the same power socket.
- 4. **Bed Leveling:** Confirm that the bed is properly leveled.

GEOMETRIE TYPES

- **Polysurface:**
  - A combination of surfaces that are joined together. Can be open or closed.
- **Surface:**
  - A single, continuous face. Typically open.
- **BREP (Boundary Representation):**
  - Represents the shape of a solid object, can include surfaces and edges. Can be open or closed.
- **Mesh:**
  - A collection of vertices, edges, and faces that defines the shape of a polyhedral object.

CHARACTERISTICS

- **Open/Closed Versions:**
  - **Open:** Geometry with gaps or holes, often requires closing or additional processing before slicing.
  - **Closed:** Watertight geometry, ready for slicing.
- **Surface Properties:**
  - **Top Open:** Ideal for prints where the top remains exposed.
  - **Top Closed:** Suitable for sealed, solid prints.
  - **Top Smooth:** Best for designs requiring a flat or curved top surface.
  - **Top Angled:** Requires careful slicing to manage steep angles or overhangs.
  - **Undercuts:** Requires specific slicing strategies to avoid print failures.

DESIGN RULES

- **Continuous Toolpath:** Ensure the toolpath is continuous to minimize retractions and maintain smooth printing.
- **No Seams or Bridging:** Design your model to avoid seams and bridging for better surface quality and structural integrity.
- **Brim or Skirt:** Always include a brim or at least a skirt to enhance bed adhesion and stabilize the print.

RECOMMENDED SLICING METHODES

1. **Simple & Fast:**
  - **Software:** Orca
  - **Use for:** Standard geometries, quick prints, straightforward shapes.
  - **Features:** Delete pre-generated start and end G-code, suitable for planar slicing.
2. **Custom & Complex:**
  - **Software:** Grasshopper
  - **Use for:** Customized prints, non-planar geometries, complex forms.
  - **Features:**
    - Adaptive slicing
    - Custom brim, skirt
    - No seams

SOFTWARE OVERVIEW

- **Cura**

Note: **STEP files** are not supported in Cura. Make sure to convert your models to a compatible format (e.g., STL or OBJ) before importing them.
- **Orca**

Important: When using Orca, **delete everything** in the start and end G-code sections to customize the print process and avoid conflicts with the printer’s specific settings.
- **Grasshopper**

**Custom Slicing:** Grasshopper allows for advanced, custom slicing methods, providing greater control over the slicing process. This is ideal for complex geometries and specialized print requirements.

LAYER SIZE RULES

- **Layer Width to Layer Height Ratio:**
  - The ratio of layer width to layer height should be **at least 2:1** or higher. This ensures proper layer adhesion and print stability.
- **Maximum Layer Height:**
  - The layer height should be no more than **60% of the nozzle diameter**. Staying within this limit ensures that each layer bonds properly to the previous one and maintains print quality.
- **Minimum Layer Width:**
  - The layer width should be at least **150% of the nozzle diameter**. This provides sufficient coverage and ensures that each pass of the nozzle overlaps correctly with the previous one.

GENERAL SLICING TIPS

- **Travel Moves**
  - a) **Continuing Toolpath:** Ensure the toolpath is continuous to minimize unnecessary retractions and reduce print time.
  - b) **Combine Prints:** If possible, combine multiple prints into a single job to optimize travel moves and reduce idle time.
  - c) **Use Artificial Ooze Walls:** Implement artificial ooze walls to manage excess filament during travel moves and reduce stringing.
- **Support Structures**
  - Dynamic Z-Axis Lifting:** Consider dynamically lifting the Z-axis during travel moves to avoid collisions and improve print quality, especially with complex geometries.
- **Overhangs**
  - Standard Printing:** Maintain a maximum overhang angle of 45° to ensure proper support and prevent sagging.
  - 45° Printing:** For prints oriented at 45°, consider achieving steeper overhangs up to 90°.
  - Stepover Rule:** Apply the stepover rule to manage overhangs effectively, ensuring that each layer provides adequate support for the next.
- **Bridging**
  - Avoid When Possible:** Minimize the use of bridges, as they can lead to poor print quality. Design your model to eliminate or reduce bridging whenever possible.
- **Infill**
  - Integrate into Design:** Incorporate infill patterns directly into your design to optimize strength and material usage, ensuring better structural integrity and aesthetics.
- **Corners**
  - Round or Step Divide:** Round sharp corners or divide them into steps to reduce stress concentration and improve print quality.
  - Nozzle Diameter Consideration:** Keep the corner radius at least 0.5 times the nozzle diameter to ensure smooth transitions.
- **Double Beads**
  - Rule of Thumb:** Follow these guidelines for double beads to achieve optimal layer bonding and wall thickness:
    - **Layer Width:** Use 0.8-0.9 times the layer width.
    - **Nozzle Diameter:** Set the layer height/width to 1.2-1.4 times the nozzle diameter for consistent extrusion and wall strength.

MATERIAL	TYPE	FOUR
----------	------	------

Nateruworks PLA Pelletes	220	210	205	Gleitmittel	93/7%
--------------------------	-----	-----	-----	-------------	-------

## ALL-PLAN DET

Mr. C. J. ...

This image shows a blank sheet of white paper with horizontal ruling lines. The lines are evenly spaced and run across the width of the page. There are no margins, text, or other markings on the paper.

# GINGER HANDBOOK - TROUBLESHOOTING DURING PRINTING

## MIXER SETTINGS

The mixer controls the ratio of materials fed into the extruder. Based on our experience, we recommend a 93/7% mix ratio.

## COOLING GUIDLINES

Proper cooling is crucial, especially for overhangs. However, avoid using cooling during the initial layers to ensure the polymer sticks to the bed and prevents warping.

- **First Layers:** No cooling for the first 2-4 layers.
- **Cooling Rates:** Use 60% cooling for most prints; increase to 100% only for strong overhangs.

## SPECIFIC PROBLEMS

### First Layer Does Not Stick / No Adhesion:

- **Bed Leveling:** The bed might not be properly leveled. Check for frame misalignment, readjust, or re-run the auto-leveling process.
- **Z-Offset:** Verify the Z-offset and adjust if necessary. Ensure the print bed is heated and apply adhesive spray.
- **First Layer Speed:** Slow down the speed of the first layer compared to the rest.
- **Flow Rate:** Increase the flow rate for the first layer.

### Warping:

- **Brim:** Add a brim to increase surface area and prevent warping.
- **Securing the Print:** Use screws or clamps to hold the print in place, apply more adhesive, or use a permanent print surface like tape or fleece.
- **Cooling:** Set the cooling to 0% for the first 2-4 layers to improve adhesion.
- **Bed:** Turn on the Printbed to app. 60°C.

### No/Insufficient Material from Extruder:

- **Purge:** Purge a significant amount of material to clear any blockage.
- **Flow/Speed Adjustment:** Adjust the flow rate or print speed to ensure proper extrusion.

## NOTES

---

---

---

---

---

---

## HOMING

Homing is essential to ensure the correct movement of the Z-axis. Always check if there's any leftover material on the nozzle before homing to prevent damage to the build plate.

## BED LEVELING

Regularly auto-level the bed through the menu and save the settings in the EEPROM. Don't forget to set and save a new Z-offset. Additionally, manually measure the frame's distance from the bed. If it's uneven, power off the printer and manually adjust each motor until all four Z-axis motors are at the same height.

### Polymer Droplets on the Print:

- **Nozzle Maintenance:** Apply new Teflon tape to the printer's nozzle to ensure a tight seal.
- **Mixing Issues:** Properly mix the liquid masterbatch to prevent pellets from sticking together.
- **Drying:** Ensure the material is properly dried before printing.

### Material Popping, Hissing, or Visible Steam at the Nozzle:

- **Drying:** Dry the material thoroughly to prevent moisture-related issues.

### Poor Print Quality:

- **Material Dryness:** Ensure the material is dry before printing.
- **Temperature Adjustment:** Adjust the extrusion temperature up or down to optimize print quality.

### Printer Crashes into the Part:

- **Clearance Issues:** The clearance between the extruder and the part may not be sufficient, especially with non-planar printing. This can cause the extruder or bed-leveling sensor to touch the print.
- **Design Adjustment:** Consider redesigning the part to make it larger or adjust the height to prevent collisions.