fillamentum forMAKERS

VERSION UPDATED IN 8/2024

SO LET'S CREATE AWESOME

# 3D PRINTING GUIDE
# PLA CRYSTAL CLEAR

Plug in and print.
Compatible with high speed printers.

Second life
Sustainable spool

## BASIC OVERVIEW

HARDNESS

IMPACT RESISTANCE

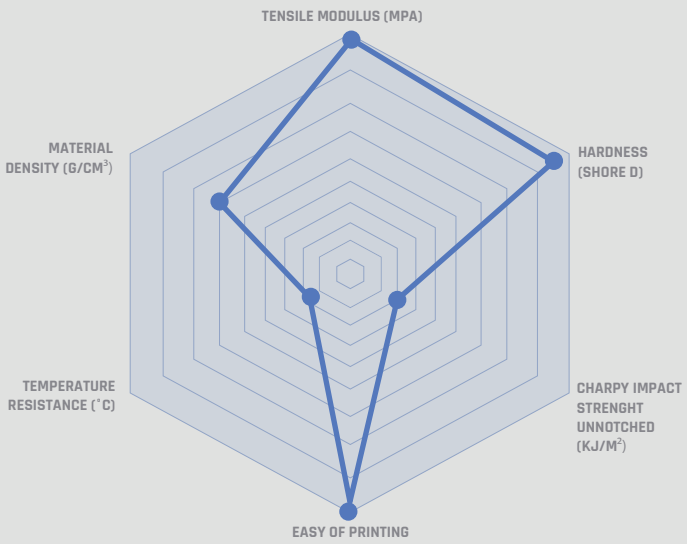
FLEXIBILITY

EASY OF PRINTING

WEATHER RESISTANCE

WEAR AND ABRASION RESISTANCE

## DETAILED VIEW




## BASIC NON HIGH-SPEED PRINTERS SETUP

**Print Temp:**
195 – 230 °C

Slightly higher than standard PLA. We recommend increasing it by 5 - 10°C than you are used to with PLA Extrafill.

**Bed Temp:**
0 – 55 °C

**Printing Speed:**
30 – 70 mm/s

Avoid too high speeds to reduce internal fogging and ensure a clearer finish.

**Cooling Fan:**
0 – 30 %

Start at 0 - 30 % for initial layers, gradually increasing to 100 % from the 3rd - 5th layer for reduced warping and clearer print results.

## HIGH SPEED PRINTERS SETUP

**Print Temp:**
200 – 240 °C

**Bed Temp:**
0 – 55 °C

**Printing Speed:**
30 – 300 mm/s

Avoid too high speeds to reduce internal fogging and ensure a clearer finish.

**Cooling Fan:**
0 – 30 %

Start at 0 - 30 % for initial layers, gradually increasing to 100 % from the 3rd - 5th layer for reduced warping and clearer print results.

DISCLAIMER:
Drying: highly recommended 50 °C for 4 hours
Storage 15 – 25 °C with low humidity.

## RECOMMENDED BED SURFACES AND ADHESIVES

**Surface Preparation:** Glass bed (for smooth finish), PEI sheet, etc.
**Bed Heating and Brim/Skirt:** Use adhesives such as 3DLac or Magigoo PLA

DATASHEETS AND MORE...
24/7 AVAILABLE

## ADDITIONAL TIPS FOR PLA CRYSTAL CLEAR

### 1 Achieving Clarity:

**To maximize transparency,** ensure clean extrusion by reducing retractions and avoiding excess cooling in the initial layers. For better visual clarity, print with thicker layers (e.g., 0.2 – 0.3 mm).

**Print with „Vase mode“** if possible and reduce infill to avoid internal structure visibility.

**Use higher temperatures** towards the max range (around 240 °C) and ensure uniform cooling to prevent internal layer haze.

### 2 Preventing Zits and Blobs:

**Enable linear** advance or pressure advance in your slicer to manage extrusion consistency. Lower the flow rate slightly (e.g., 90 - 95 %) toreduce material buildup that causes zits and blobs.

**Retraction tuning is crucial:** try retraction distances around 1 – 2 mm and speeds around 25 – 40 mm/s. Avoid retracting too much as it can introduce air bubbles and disrupt clarity.

### 3 Moisture Sensitivity:

**PLA Crystal Clear,** like other PLAs, can absorb moisture, leading to popping and foggy prints. Store in a dry box with desiccant when not in use. We highly recommended dry the filament at around 50 °C for 4 - 5 hours.

### 4 Nozzle Recommendations:

**A standard 0.4 mm nozzle works,** but for clearer results and to minimize potential clogging, a 0.5 mm or larger nozzle can improve transparency by allowing for better flow

### 5 Avoiding Print Failures:

**Start with a solid** adhesion method to prevent curling, especially for large or flat objects, and use a skirt or brim to keep edges from lifting.

**Calibrate your flow rate** and temperature carefully to avoid under-extrusion, which can result in layer separation starting on complex prints.

### 6 Vase Mode:

**Print temperature and speed:** To achieve a homogeneous layer, it is better to print at the upper limit of the temperature range and more slowly than usual. This helps prevent the formation of tiny air bubbles that could compromise transparency and better layer bonding.

**Ensure sufficient cooling** between layers on tall models to prevent distortion. Reduce cooling for smaller models. The thickness of the walls should be set slightly wider than you are used to in your "vase mode" due to better cohesion of the layers.

**Layer height:** For better smoothness, choose a low layer height, ideally 0.1 mm or less. This improves the overall appearance of the print and creates a smoother surface.

**Postprocessing:** For PLA Crystal Clear, gradual grinding (from a coarser grain of around 400 to a fine 2000) and subsequent polishing is proven. The use of special clear varnishes or epoxy resin can increase the transparency of the surface.

## TROUBLESHOOTING TIPS

**Layer Separation and Weak Bonds:** Increasing the extrusion temperature towards 230 °C can improve layer adhesion and reduce brittleness. Reduce speed and fan.

**Cloudy or Hazy Prints:** If prints are consistently foggy, lower the print speed and ensure the filament is dry.

**Warping on Large Prints:** Increase bed adhesion with Magigoo or PVA glue stick or use a brim for better stability on larger prints. And and increased bred temperature.

**Clogging Issues:** If you encounter clogging, consider increasing the nozzle size or lowering the print temperature slightly to prevent filament from burning.

## ARE YOU MISSING THE RIGHT ANSWER?

CHOOSE THE PLACE YOU'D LIKE TO CONNECT WITH US.

CLICK FOR FILLA FELLA CHAT BOT
24/7 INSTANT ANSWER

HELPDESK@FILLAMENTUM.COM
REACTION TIME 12 - 16 HOUR

CALL US IF YOU WANT +420 725 463 731
BETWEEN 8 AM AND 4 PM CENTRAL EUROPEAN TIME.

**WE GUARANTEE THE BEST QUALITY WITH CPK PROCESS MEASUREMENT.**

At Fillamentum, we go beyond achieving a lower filament diameter. We focus on CPK (Process Capability Index) could be known as a Sigma within Industry. It is a crucial measure that ensures every spool of filament meets the highest standards. Here is Why CPK is essential for you and why it is more important than just diameter.

WE PROVIDE FILAMENT INSPECTION

RECOMMENDED

3D Printing Industry AWARDS 2021 WINNER OF 2021