

Filament-Cer Al₂O₃

Printing Guidelines

Always maintain the storage conditions listed in the datasheet. On reception of the spool, check that the vacuum has not been lost during shipment and that there are no broken parts. If this is not the case, please contact our post-sales service immediately.

This appendix contains spool management instructions for a better printing experience and best quality results in your printed parts. The parameters and steps described below have been validated with different printers (brass nozzles, aluminum heating blocks and steel barrel with Teflon cannula) at a room temperature of 25°C. Lower temperatures will promote the brittleness of the filaments in the feeding zone. For a better grip of the filament and prevention of undesirable filament breakage, we recommend the use of two gears motor on the printing head.

Generating the .gcode file

Proceed as follows to generate the slice of the part to be printed:

1. Preparation of the material profile. Whether you have a slicing software with the option to save the settings for the materials profile, it will allow the machine to calculate the quantities and times necessary for printing according to the real composition of the filament in later printings. Consider the following key parameters as basics to include into the material profile:

Set the material profile

Density	2.41 g/cm ³
Diameter	1.75 ± 0.15 mm
Printing temperature	158-165 °C
Stand-by temperature	30 °C lower than the used printing temperature or <140 °C
Hot Pad temperature	30-60°C

2. Printer parameters. Configure the slice with the following recommended parameters.

Tested printing parameters

Printing speed	10-20 mm/s
Head travel speed	< 50 mm/s
Nozzle diameter	≥ 0.4 mm
Layer Height	≥ 0.15 mm (*)
Retraction	You can use the retraction during the layer changes with the values of filament distance and speed of the printer used for printing PLA filaments.
Build plate adhesion	The use of 1 or 2 skirts around the part allows the nozzle to be fully loaded. We recommend to fix these skirts at the same speed you have chosen to print.

** For 100% infill parts we recommend increasing the number of solid layers below and above the part, or increasing the shells until we get the required number of layers.*

To make several prints in a row, it is advisable to leave the head at 10°C below the printing temperature or at the same temperature as the print to avoid clogging of the nozzle due to the repeated cooling and heating of the filament. In this case, it is highly recommended to include a line of code in the .gcode files of the prints that purges the head before starting to print.

Feeding the filament before printing.

Warm the printer. Prior to printing, we recommend warm up to printing temperature and use cleaning filament to avoid clogging with waste of previous material. Additionally, we recommend to dry the filament between 40-60°C for 2 minutes before feed.

Enable cold extrusion. COLFEED4Print filaments require of printing temperatures below the safety minimum temperature of most of the commercial printers. To overcome this feature in printers whose firmware is based on Marlin, it is necessary send a code line with the "M302" command to enable cold extrusion. If the printer used has its own firmware, contact the technical service of your printer before feeding the filament to ensure you can achieve the printing temperature conditions. The use of a filament dryer box is recommended to feed the filament directly during printing.

NEVER FEED THE FILAMENT AT TEMPERATURES HIGHER THAN THE PRINTING TEMPERATURE AS THIS MAY DEGRADE THE MATERIAL AND CAUSE CLOGGING OF THE NOZZLE AND BARREL.

Launch the .gcode for printing

Once the filament has been fed at the printing temperature and the nozzle has been loaded, start printing of the parts avoiding delays larger than 1 minute. More time could induce a heterogeneous flow at the beginning of the printing.

Depending on the type printing bed we recommend the use of a low adhesion lacquer commonly used for PLA prints. This will be sufficient to avoid warping of the printed parts. For FCer-Al₂O₃ it is advised the use of aluminium beds without any surface treatment.

Download filament after printing

If you are going to stop printing, we recommend unload the filament at the same temperature as it has been fed. Depending on the type of head it is possible loose up the screw of extruder and vigorously pull the filament upward to avoid blocking the filament in the barrel. Alternatively, you can unload it automatically by taking into account the same consideration as for the thread feeding section.

Disclaimer: The technical information contained in this data sheet has been obtained by laboratory characterization and should be treated as a reference. Any description, data or printing recommendations given in this document may change without prior information and do not constitute the agreed contractual quality of the product. Printing conditions may vary depending on the 3D printer model used and part printed. Due to the numerous factors that may affect the data shown, we make no warranty of any kind, express or implied, as to the properties of the product or its suitability for a particular use.