Printing guidelines

FILAMENT-Eco Graphene

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 process. For a better grip of the filament and prevention of undesirable filament breakage, we recommend the use of a two-geared 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:**

Diameter	1.75 ± 0.15 mm
Printing temperature	170-180 °C
Stand-by temperature	30 ºC lower than the used printing temperature or <140 °C
Hot Pad temperature	50-60 °C

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

Printing speed	5-20 mm/s
Head travel speed	< 150 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 10ºC 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 warming up the nozzle to the indicated printing temperature and using cleaning filament to avoid clogging the nozzle with waste of any previously used materials. Additionally, we recommend warming up the filament at between 40-60 °C for at least 30 minutes before feeding.

Enable cold extrusion. COLFEED4Print materials require printing temperatures below the safety minimum temperature of most commercial printers. To overcome this feature in printers which firmware is based on Marlin, it is necessary to send a code line with the "M302" command to enable cold extrusion. If the machine used has its own firmware, you may want to contact your printer's technical service before feeding the material to ensure you can achieve the printing temperature conditions.

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 the parts avoiding delays larger than 1 minute. More time could induce a heterogeneous flow at the beginning of the printing process.

Depending on the type of 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.

Unload filament after printing

After printing, we recommend you unload the filament at the same temperature as it has been fed. Depending on the type of head it is posible to loosen up a screw on the extruder and vigorously pull the filament upward to avoid jamming the filament in the barrel. Alternatively, some printers allow to unload it automatically.

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.