

# PA6CF20

Carbon Fiber Reinforced PA 6

Print Settings	
<b>Extruder Temperature</b>	285°C
<b>Bed Temperature</b>	50°C
<b>Bed Preparation</b>	Clean with isopropyl alcohol before and after every print.
<b>Cooling Fan</b>	No
<b>Heated Chamber</b>	No
<b>Closed Chamber</b>	Yes
<b>Chamber Temperature</b>	N/A
<b>Notes</b>	<ul style="list-style-type: none"> <li>Annealing is recommended, at 90°C for 2 hours in an oven.</li> </ul>

Material Management	
<b>Drying</b>	<b>Temperature</b> <ul style="list-style-type: none"> <li>70°C - 80°C</li> </ul> <b>Dry Time</b> <ul style="list-style-type: none"> <li>Heat only → 12 hours</li> <li>Heat + Vacuum → 3 hours</li> </ul>
<b>Recommended Support Material</b>	<ul style="list-style-type: none"> <li>SSU301</li> </ul>
<b>Storage</b>	<ul style="list-style-type: none"> <li>Moisture-free box.</li> </ul>
<b>Compatible Materials</b>	

Slicer Settings	KISSlicer (0.4mm)	KISSlicer (0.6mm)
<b>Material Profile Name</b>	PA6CF20-HT (0.4)	PA6CF20-HT (0.6)
<b>Layer Height</b>	0.18 – 0.2 mm	0.27 – 0.32 mm
<b>Loops</b>	5.5	
<b>Skin</b>	1.5	
<b>Infill</b>	16.7 %	

## Machine Setup & Procedure

A wide variety of filled materials are becoming more and more widely used for FFF printing. Where high strength and high heat deflection temperatures are needed, they are among the best choices available for additive manufacturing today. They also have a significant cost advantage over the emerging “super polymers” like PEEK, PEKK and Ultem, as well as being more printable and having a wider choice of support materials and strategies than the super polymers.

The primary downside to filled materials is their abrasiveness, which causes significantly increased wear on the nozzles and feed mechanisms, and the difficulty of handling and feeding due to their stiffness. The basic requirements to print most filled materials are:

- 1) Anti-abrasion nozzles (hardened nozzles).
- 2) Printer must be set in high temp mode for the appropriate nozzles if needed.
- 3) Print server must have extruder temperature range reset to match high temp requirements if needed.
- 4) Material must be dry. We always dry any nylon-based material over night before printing.

# PA6CF20

*Carbon Fiber Reinforced PA 6*

## Drying Materials

Nylons are among the more hygroscopic materials we print. It must be properly dried and kept dry, during the printing process for them to develop their full mechanical properties and print with high quality surfaces and accurate dimensions. Nylon Carbons should be dried, and then kept in our Material Management System or a dry box, to reduce its reabsorption of moisture. If the print surface quality begins to degrade, or stringing and "bumps" appear at seams, or on the surface, the material needs to be thoroughly dried again.

## Support Materials

Filled materials are generally paired with at least one support material. Use SSU301, which is water a soluble support.

## Build Surfaces

Be careful not to scratch or gouge the Diamond Plate. Always keep the scraper angle low and flat.

## Nozzles

Hardened nozzles are required. We use and recommend Tungsten Carbide nozzles made for 3ntr printers, which have the proper internal geometry for optimum material flow and surface finish.

## Filament Drive Components

Abrasive materials will prematurely wear filament drive gears as well. Please monitor your system for any potential filament feed slipping, which could indicate that cleaning is needed (monthly to quarterly cleaning as a preventative measurement is recommended under any circumstances) or a filament drive gear is worn enough to warrant replacement.

Consider drive gears a consumable with filled materials. How often they will need to be replaced is highly geometry dependent. We recommend keeping a spare motor with new feed gear handy to switch out whenever needed. This allows you to replace the drive gear on the stepper motor at your convenience, so it doesn't increase your down time.

## Setting & Configuration Changes Required

In addition to the hardened nozzles, you will need to put the extruders, that will be used for PA6CF20 (or any material requiring over 260°C), into high temperature mode, both physically, and in the printer's firmware configuration through the LCD menu.

**Note** - Extruders can be left in the high temp configuration, if desired. Surface finish degradation is minimal, and it makes it more convenient to print a wide range of materials, eliminating this step in the future. The exception is printing with the SPFU (soft polymer feed unit) and TPU85A, which require low temp mode. TPU 95A also prints with somewhat better quality in low temp mode.

Change the appropriate extruders to high temp (HITEMP) mode per the owner's manual instructions. The menu picks are:

Prepare > Hardware > Extruder Config > Set Nozzle(s) to1 > Hardware > Save > Save.

- 1) From the Hardware menu select extruders config:



- 2) Choose the extruder(s) you want to modify, then save (save) the configuration:



- 3) You will see two thermometers next to each nozzle you set to high temp (HITEMP) on the info screen once the settings have been changed and saved.

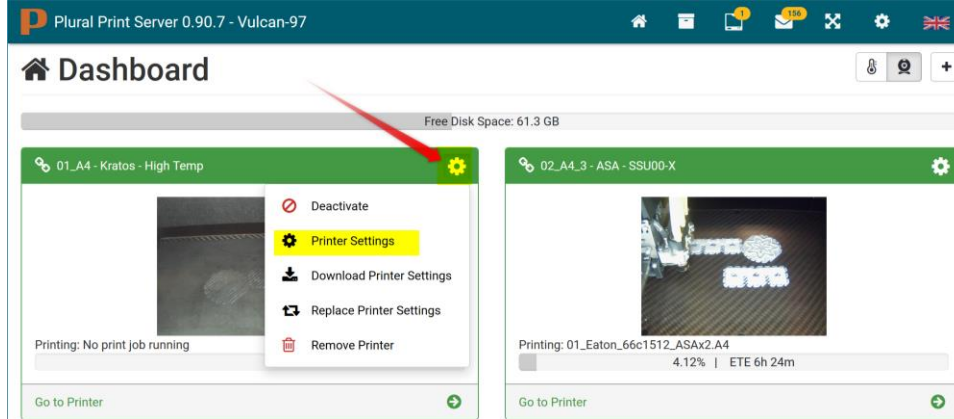


# PA6CF20

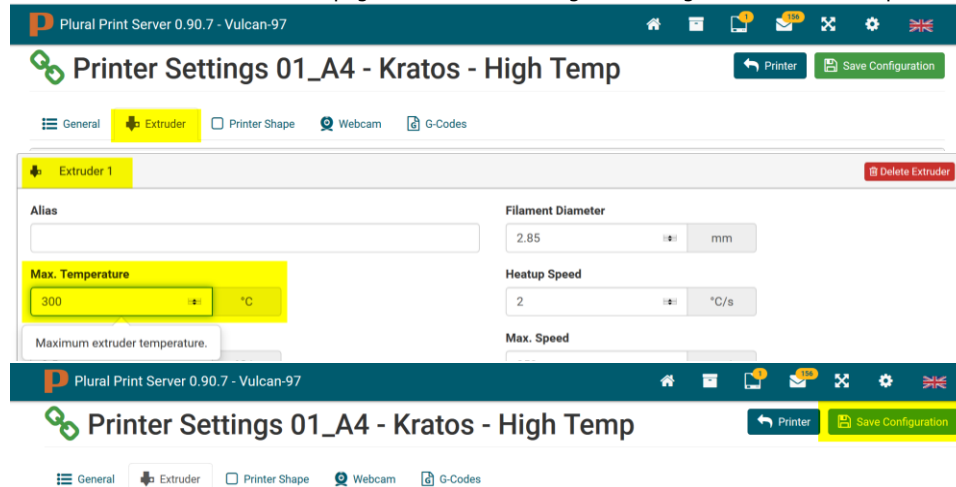
*Carbon Fiber Reinforced PA 6*

## Print Server Changes Required

- 1) From your print server's dashboard select Settings (Gear) for the printer you are changing:



- 2) Select the Extruder Tab and scroll down the page to Extruder 1 settings and change to the desired temperature.



- 3) Repeat for extruders 2 and 3 if desired, then Save Configuration.

## General Handling & Characteristics

- Most filled materials are stiff and brittle off the spool, and can be challenging to feed.
- Pre-drying will ensure the best quality prints.
- Loading the material while it is still warm from the dryer will also make it easier to handle and feed.
- As with most filled materials, nylon carbons have a bit of a sandpaper, slightly abrasive finish, but the surface finish and appearance are excellent, and the parts are very strong.

## Slicing Tips

Filled materials in general need Z layer settings to be on the lower side to achieve good Z bonding, overhang performance and good surface finish. Typically, we don't exceed 50-60% of the nozzle diameter for a Z layer setting. A good Adaptive setting (on the Style menu) with a 0.4mm nozzle, for example, would be 0.18 - 0.24mm with .2mm first layer. Fixed Z height of 0.2mm also works well.

## Post Processing

PA6CF20 requires annealing to reach its full mechanical properties. It should be annealed in a warm (~93°C / 200°F) oven for ~2 hours *after printing and before support removal*. We use a small toaster oven on its "warm" setting (confirm temperature isn't too high). After annealing the support material can be removed and part allowed to air dry.

# PA6CF20

*Carbon Fiber Reinforced PA 6*

## **Support Strategy & Removal**

SSU301 dissolves in plain warm water, but may take some manual rinsing and removal of the softened support material for complete and clean removal.

## **Cleaning Nylon Carbon from the Nozzles**

Filled materials are more difficult than most other materials to completely clean from a nozzle, generally meaning more passes with the clean program than you may be accustomed to with other materials. We find it typically can take between 4 and 6 cleaning cycles before the nozzle is completely clean and ready for other materials.