

This is a reference list of 3D printing parameter settings used for printing with PETG+CF (PET-G with carbon fiber) filament for the Prusa iMK3S+ printer.

Please note that the correct settings may vary depending on the actual printer and filament brand used.

---

### Temperatures:

First layer: 230 degrees  
Other layers: 240 degrees  
Bed - first layer: 70 degrees  
Bed - other layers: 65 degrees

### Cooling:

Fan speed min.: 30% or 100%  
(see note 3)

Fan speed max.: 50% or 100%  
(see note 3)

Bridges fan speed: 100%  
Disable fan for the first: 3 layers  
Full fan speed at layer: 5

### Speed:

Perimeters: 45 mm/s  
Small perimeters: 15 mm/s  
External perimeters: 15 mm/s  
Infill: 45 mm/s  
Top solid infill: 40 mm/s  
Bridges: 15 mm/s  
Gap fill: 15 mm/s  
Travel: 160 mm/s  
First layer speed: 20 mm/s

### Filament and retraction:

Extrusion multiplier: 0.9  
Travel lift height: 0.4 mm  
Retraction length: 1.0 mm  
Retraction speed: 35 mm/s  
Retraction extra length: 0.1 mm  
Travel lift height: 0.4 mm  
Retract on layer change: yes  
Wipe while retracting: yes

### Other settings:

Infill: 100%  
Seam position: nearest  
Number of perimeters: 2  
  
Avoid crossing perimeters  
No brim  
No supports

**Additional notes:**

1. Always complete individual objects first: don't print several objects at once, layer-by-layer.
2. Use only very well-dried filament (This is critically important. PETG+CF easily absorbs moisture, and printing with damp filament is very imperfect and unsuitable for astrophotography).
3. Often, for PETG filaments, even with the addition of carbon fiber, a cooling fan speed of 30%-50% (min-max) is used. High cooling speeds (100% cooling fan speed) are rarely used here, as they can cause weaker layer adhesion in some filaments. However, fast and powerful cooling reduces unwanted shrinkage during cooling immediately after printing. Therefore, in prints for astrophotography, where dimensional accuracy is often extremely important, after first 3 layers printed with no fan, for critical elements (such as the U-shaped filter drawer core), you should consider turning the cooling fan to 100%.

After printing, always check that the manufactured part has no delamination or other defects. If 100% cooling is too much in your case (printing errors, layers separation, threads are not consistent), reduce the speed, but always check the dimensional accuracy of the result after printing.

To sum up, you can print non-critical components with airflow as low as 30%, but for components where accuracy is extremely important (such as the drawer core), you should try to set the fan speed as high as possible.