

3D Industries – Flexi Filament Printing

1 Preamble

This article explains how to print with Flexi filament.

Flexi filament is PLA based plastic that is rubber like, very strong but can be squeezed and stretched.

Objects printed have stretchy and squeezable properties so that the printed object is tough yet flexible can be squashed and when distorted remembers its original shape.

There are many uses for object print in Flexi plastic.

3D Industries provides sample objects and config.ini files with details of other settings for the W J and P models. These will enable the printing of flexi objects. To print further objects the supplied config.in file should be used for slicing and if any changes are made then the config.ini file should be exported to a different name and the original not changed. Remember the config.ini file should only be changed by modifying the settings in slic3r and exporting the file. NEVER make changes to the config.ini file directly.

2 Flexi filament

This is more expensive the normal PLA or ABS filament and often is only available in 200 gm spools. Where the filament is often wound on small spools that are very full so the filament comes to the top these smaller spools and there can be a tendency for the filament to ‘escape’ and be entangled. So if possible the filament should be rewound onto a larger empty spool.

Flexi filament properties mean that is clunky to the touch and difficult to push in a straight line, I can also twirl and twist.

3 Printing with Flexi filament

The method and settings used to print with Flexi filament are described using 3DIndustrie series 6 printers but should apply to other printers where they have the facilities indicated. It is important to realise that many printers will be unable to print properly even at all in flexi because they do not have adjustable extruders or they have all metal or restrictive hot ends.

3.1 Filament spool

Ensure that the filament can be withdrawn with little force from the spool and that there are no issues with entanglement

3.2 Filament cleaning

It is very important to ensure the outside of the filament is wiped but that the wiping action does not place too much friction on the movement. Sponge material is the ideal medium for this.

3.3 Filament path

The filament should follow a path from the cleaner to the extruder in a graceful arc with no sharp corners that can deform and impeded the filament travel.

3.4 Extruder settings

The filament push and pull is achieved by it passing through the extruder feed mechanism composed of a bearing smooth outer surface the presses the filament against a knurled cog on the stepper motor shaft. The pressure exerted on the filament has to be strong enough to be able to draw the filament from the spool and to push it through the hot end. Normally the feed mechanism, depending on the object print will both push and retract the filament so that the pressure on the filament must not cause any damage as a result. Another result of too much pressure is that the filament can have a tendency to wrap itself around the knurled cog.

3.5 Hot end requirements

The plastic filament entering the hot end travels through to the melt point whereupon the unmelted filament. Causes already melted plastic in the reservoir to be ejected through the nozzle.

It is important that the filament path through the hot end up to the melt point follows a non metallic channel, preferable PTFE but definitely not metallic. A warm or hot feed channel or throat can soften the filament and introducing friction

3D Industries – Flexi Filament Printing

between the filament and the metal. As a result the filament can try to resist the extruder pushing action with the filament being compressed and a tendency to crinkle or deform.

3.6 Bed requirements

Flexi filament seems to adhere best to plain glass with a spray adhesive. The use of a 5mm brim provides increased adhesion to the glass and enables easy removal without damage.

4 Set up and testing

Heat the nozzle to 230 C. Note that although the final temperature used will be around 220 C it is important to ensure that too low a temperature acts as a partial blockage so we start with a high temperature that can be reduced later. Feed the filament from the spool holder through the cleaner and ensure that there is very little force required to pull the filament through.

Reduce the pressure on the extruder between the knurled cog and pressure bearing and feed the filament through the extruder and through to the hotend, it must be possible to easily push the filament into the hotend and observe extruded filament from the nozzle.

Gradually increase the pressure in the filament in the extruder until it just strongly pulls the filament and sends it into the hot end and extrusion is observed.

When this is successful start a test print and ensure the extrusion to the bed as a first layer is successful. It may be necessary to increase the Z-gap slightly (to reduce back pressure) and to increase the bed temperature for the first layers.

4.1 Printing speed

Start testing by using a print speed half of that normally used for PLA on that machine

4.2 Fans and cooling

Generally fans not running or running at half speed depending on the object being printed, is required

4.3 Test objects

- Cube (partially printed) is used to check
- Hollow cylinder with a single layer perimeter

4.4 Config.ini

The config.ini files are available for the W J and P models.

The printing of Flexi filament is best performed on glass with adhesive and using a 5mm brim.

Significant base settings are:

- Bed temperature 60C first layer 50 C afterwards
- Nozzle temperature 220 C
- Skirt
- Brim
- Cooling fan settings as: off.
- Printing Speed 50%

These settings will need to be modified according to the object being printed. For example a calibration vase object where a single perimeter layer forms the walls then a print speed of 50% and no cooling is required, but for a more solid object a faster print speed can be used. In this case the nozzle temperature should be monitored as always when printing fast to ensure that the temperature is maintained and setting it higher if required.

The print speed, bed and Nozzle temperatures, the cooling fans can all be modified in real-time during the printing.

3D Industries – Flexi Filament Printing

4.5 Trouble shooting

This can be performed using critical step approaches.

The most common issue is the filament not causing filament ejection.

Ensure the nozzle temperature is 230 C and move the carriage to the side or front and using the LCD controller and “**move axis**” menu ensure that the filament can be extruded then reduce the temperature to that used for the printings.

If the filament is not extruding follow steps in **Investigation A**

If the filament is extruded follow steps in **Investigation B**

4.5.1 Investigation A

- Loosen the pressure adjuster on the extruder
- Ensure the temperature of the nozzle is around 230C
- Pull the filament out of the extruder, cut the end and insert it again.
- Push the filament down until it can be seen extruding from the nozzle
- Tighten the pressure plate and use the LCD controller:
- From the status display press the knob select the “**Utilities Menu**” and press to confirm.
- Select “**Move Axis**” and press to confirm
- Select “**0.1mm**” and select to confirm
- Select “**Extrude**” and press to confirm
- Rotate the knob clockwise and continue to do so and check if filament is being extruded. Holding the filament entering the extruder at the top will show how strongly the filament is being pulled.

If the above is successful then switch the printer off and then on. Start up and set the nozzle temperature to 230C and try to print a cube. If this is successful try to print at 220 C.

Note that the main reason for failure of the flexi print in a working system is that the nozzle temperature is not sufficient to create a sufficiently molten pool of flexi plastic and that the less than molten pool acts as a partial lockage and the flexi filament is unable to push into the molten pool sufficiently to cause plastic to be ejected through the nozzle. The printing speed – the speed at which unmelted filament is pushed into the hot end can also be a factor here.

If the above steps do not resolve the issue send an email to or contact 3D Industries support.

4.5.2 Investigation B

The filament is being extruded OK so follow the steps for your problem as indicated below. If your problem is not listed here send an email to or contact 3D Industries support.

Object not being extruded onto the glass bed

This is normally due to:

- Extruder pressure adjustment not correct – test extrusion as above
- Nozzle temperature not high enough
- Z gap not correct (normally too small)

Filament not sticking to the glass bed

This is normally due to:

- Bed not hot enough
- Not enough adhesive applied to the bed
- Z gap not correct (normally too big)
- Nozzle temperature not high enough
- Environment – air movement around the object

Filament stops extruding after part of the object has been printed

This is normally due to:

- Nozzle temperature not high enough
- Extruder pressure adjustment not correct – test extrusion as above
- Printer printing too fast
- Environment – air movement around the object

3D Industries – Flexi Filament Printing

Layers not sticking to previous layers

- Z gap not correct (normally too big)
 - Nozzle temperature not high enough
 - Printer printing too fast
 - Environment – air movement around the object
-