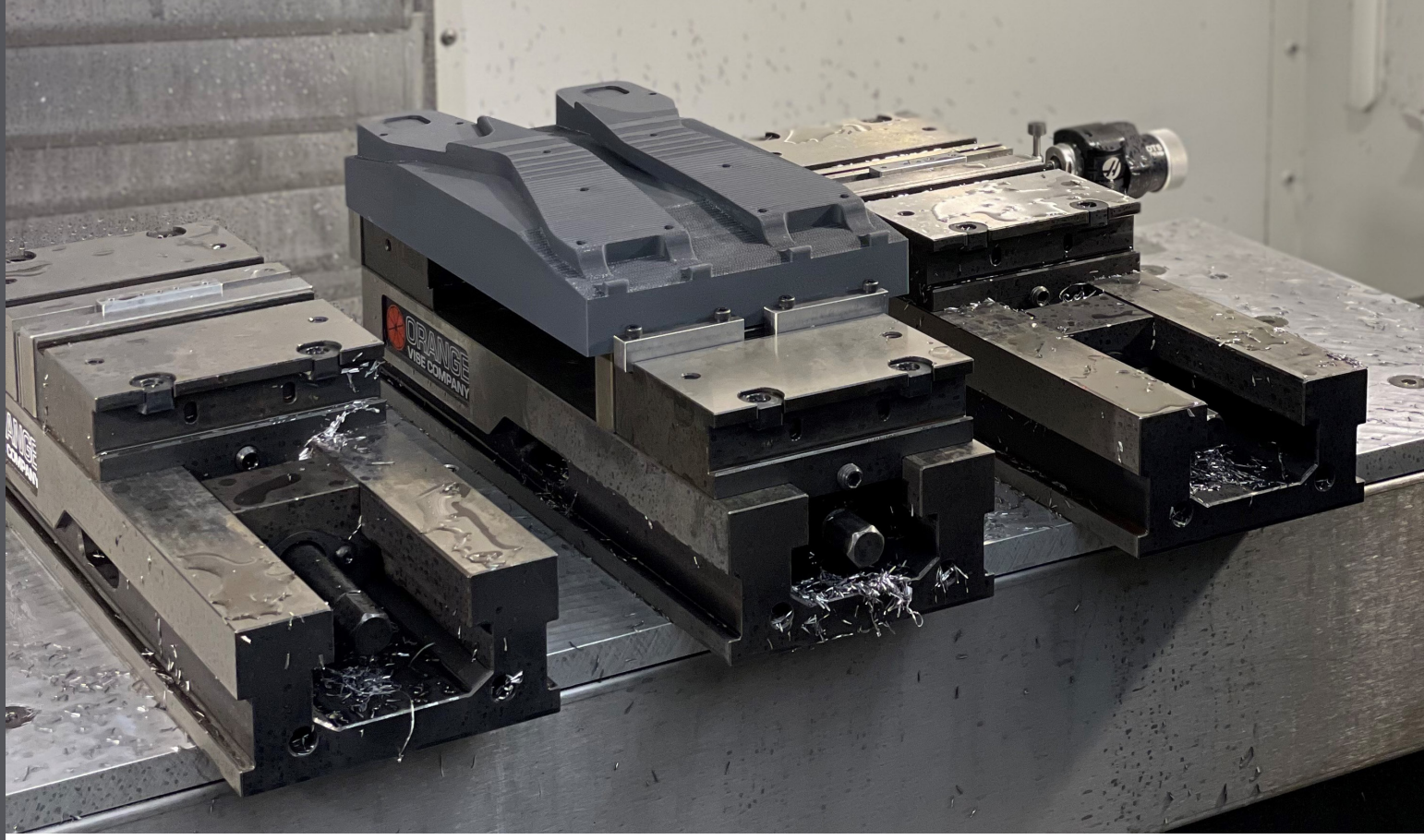

FDM Nylon-CF10
Material Guide

Introduction

FDM® Nylon-CF10 is a combination of a nylon blend base material and 10% chopped carbon fiber. This provides a stiff, strong material with good chemical resistance, suitable for tooling and machining fixtures, among other applications requiring these material properties. FDM Nylon-CF10 is available on the F190™CR and F370®CR composite printers.

System Overview and Compatibility

Nylon-CF10 is currently available with 0.007 in. (0.178 mm), 0.010 in. (0.254 mm) and 0.013 in. (0.330 mm) layer heights. It can be used with QSR Support and SUP4000B™ breakaway support. Nylon-CF10 requires the standard build tray and uses an F123 Series™ composite-ready (CR) extrusion head for the model material and a standard F123 Series printer extrusion head for QSR and SUP4000B support materials. The F123 Series CR extrusion head has a hardened T20 nozzle. Nylon-CF10 is available in 90-cubic inch spools and QSR and SUP4000B support materials are available in standard 60-cubic inch spools.

Nylon-CF10 is an available material option for the F370CR printer starting in Insight™ software, version 16.5, and for the F190CR and F370CR using GrabCAD Print™ software, starting with version 1.66. SUP4000B will be available starting with Insight version 17.5 and GrabCAD Print version 1.78.

stratasys

Part Design

Designing parts that use FDM Nylon-CF10 follows a similar process as designing other FDM® parts, and design for additive manufacturing (DFAM) guidelines should be followed (e.g., utilizing self-supporting angles where possible, observing minimum wall thicknesses, allowing proper clearance for assemblies, etc.). A general list of DFAM guidelines can be found within the Fused Deposition Modeling (FDM) Design Guidelines document. The minimum recommended plate thickness for Nylon-CF10 is 0.2 in. (5.1 mm). The minimum recommended wall thickness is two contours, based on the default single contour widths (Table 1).

Support materials are used to support the model material in areas of overhang for stability. Self-supporting angles (angles greater than 55 degrees from the build platen for 0.007 in./0.178 mm and 0.013 in./0.330 mm layer heights, or greater than 50 degrees for 0.010 in./0.254 mm layer height) should be used whenever possible to reduce the need for support material. The QSR soluble support adheres well to the Nylon-CF10 model material and is difficult to remove manually, although some portions of larger pieces can be removed this way. As a result, QSR support material is most effectively removed by tanking in a support removal detergent. Consider this when designing your parts to possibly minimize the amount of required support material.

When your model's geometry doesn't have recesses and other features that need the benefit of soluble support, SUP4000B breakaway support is a beneficial alternative. The support can be removed faster in most cases since it avoids the tank dissolution time. It is also useful for avoiding curl and warp on thin parts that can result from exposure to the moisture of the immersion bath.

The mechanical performance of nylon materials at different temperatures can be impacted by moisture. In laboratory testing to observe the effect of moisture uptake on glass transition temperature (Tg), the dry sample (< 0.04% moisture) had a Tg above 100 ˚C (212 ˚F) compared to a saturated sample (> 5.7% moisture) with an estimated Tg closer to 60 ˚C (140 ˚F). This could be an important consideration when selecting materials and designing for specific service temperatures and conditions.

Table 1. Minimum recommended wall thickness based on default single contour widths.

Slice Thickness	Default single contour width	Minimum recommended wall thickness
0.007 in. (0.178 mm)	0.020 in. (0.508 mm)	0.040 in. (1.02 mm)
0.010 in. (0.254 mm)	0.020 in. (0.508 mm)	0.040 in. (1.02 mm)
0.013 in. (0.330 mm)	0.026 in. (0.660 mm)	0.052 in. (1.32 mm)

Part Processing Considerations

When selecting Nylon-CF10 material in either Insight software or GrabCAD Print, the default settings for each slice height are recommended. The default settings for all layer heights is a single contour, with +45/-45 degree solid raster fill for model material. The default support style is SMART.

Because the FDM process results in anisotropic surface finish and mechanical properties, proper orientation depends on the part's function. For jigs, fixtures, grippers, functional or production parts, build orientation should be selected based on the forces acting upon the printed part. Optimize the mechanical properties of critical features by selectively orienting the part to avoid excessive stresses on thin geometries or in the Z-direction. For conceptual models or parts where aesthetics are most important, choose an orientation to reduce the appearance of layer lines on critical surfaces.

In the case of thin geometries (< 0.2 in. (5.1mm)), there is a possibility that after support removal, the part may warp (similar behavior is observed in molded nylons). There are several processing steps that can minimize or prevent post support removal warping:

- When possible, based on the geometry of the parts, use SUP4000B breakaway support material to avoid the moisture exposure of the soluble support removal process.
- Print using 0.007 in. (0.178 mm) layer height.
- Reduce raster length.
    - Rotate the part in the XY plane to an orientation with shorter rasters
    - Modify the raster angle globally (Insight software or GrabCAD Print)
    - Modify the raster angle on individual layers (Insight software only)
    - Rotate the flat parts by a few degrees about the X or Y axes.
- Change part orientation to orient long thin features along the Z-axis.
- Thicken thin parts if possible.
- Choose a different infill such as HD Sparse.
- For thin flat parts and sections (<10 layers) and an even number of layers, maintain the symmetry of the raster angles about the mid plane (Figure 1). This can be accomplished utilizing ***Custom Groups*** in Insight software to change the raster ***Start angle*** of the midplane layers.

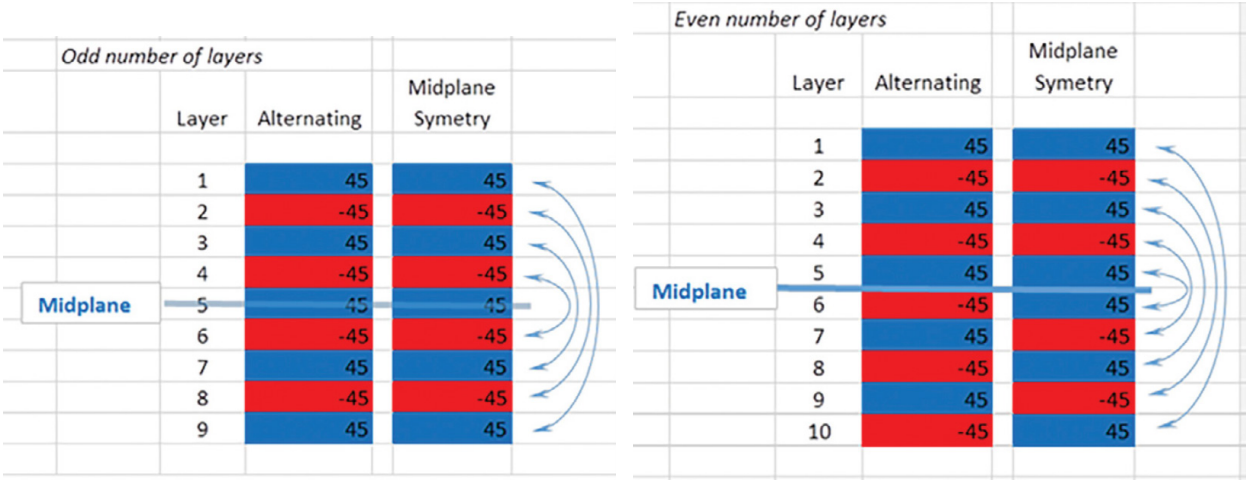

Figure 1-A/B. Schematic showing the difference in mid-plane symmetry for an odd and even number of layers. To maintain symmetry about the mid-plane when there is an even number of layers, a change in raster angle may be required.

In the case of solid blocky geometries (due to the hygroscopic nature of the nylon matrix material) parts can retain moisture from the tank (~10 wt% increase) which can cause parts to swell in the Z direction by over 0.080 in. (2.03 mm) on a 2 in. (50.8 mm) part. This swelling can be limited or eliminated by adjusting the toolpath settings (Figures 2 and 3) to add two additional contours to most of the part (three contours total) while also maintaining a single contour on the top and bottom layers (Insight) or all cap layers (GrabCAD Print).

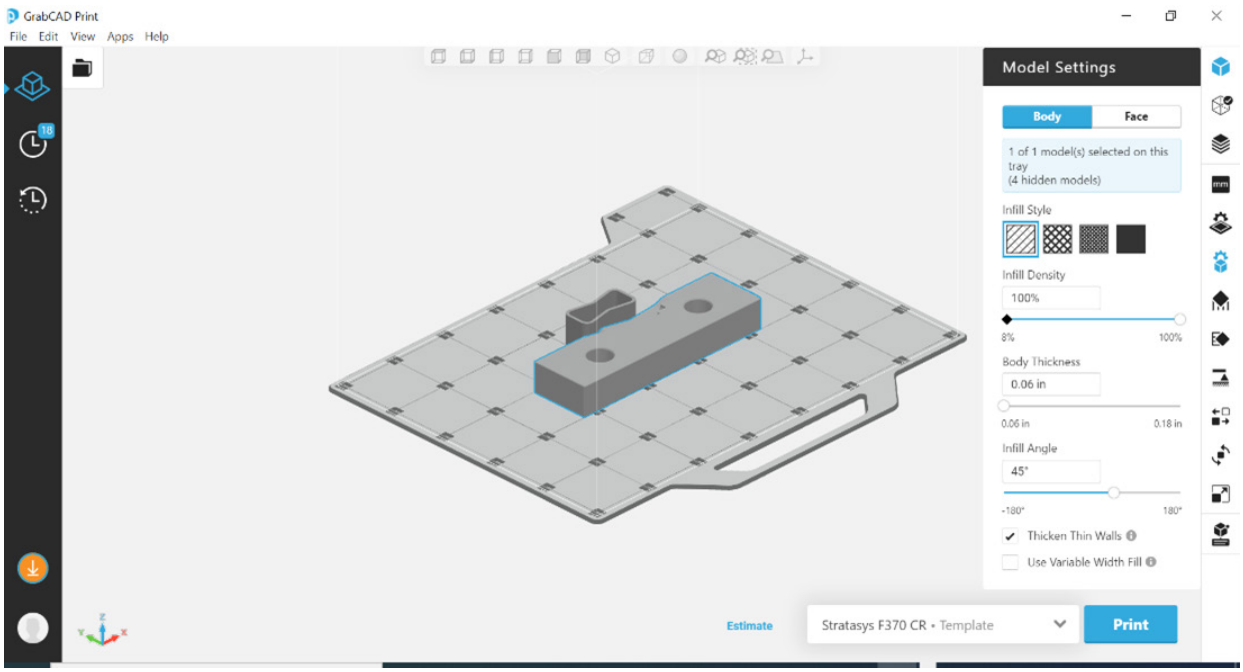

Figure 2. Adjusting the Toolpath settings in GrabCAD Print to reduce post support removal tank swelling in blocky geometries.

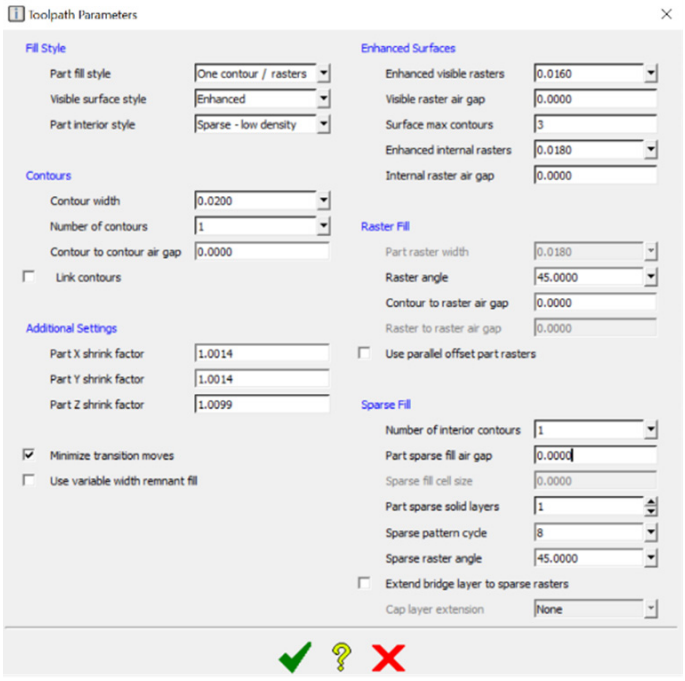

Figure 3. Insight Toolpath Parameters are adjusted to reduce or eliminate post support removal tank swelling.

Insight Software

STEP 1: Configure the modeler.

STEP 2: Open, orient and slice the STL file.

STEP 3: In ***Toolpath Setup,*** open ***the Toolpath Parameters*** menu.

STEP 4: Change the ***Part interior style*** to ***Sparse – low density.***

STEP 5: In the ***Sparse Fill*** section,change the ***Part sparse fill air gap*** based on the chosen slice thickness (Table 2). Change the ***Part sparse solid layers*** to ***1***

Table 2. Part sparse fill air gap to reduce post tank swelling.

Slice Thickness	Part sparse fill air gap
0.007 in. (0.178 mm)	0.000 in. (0.000 mm)
0.010 in. (0.254 mm)	0.000 in. (0.000 mm)
0.013 in. (0.330 mm)	0.000 in. (0.000 mm)

STEP 6: Click the checkmark to save the changes to the parameters.

STEP 7: Process the part

GrabCAD Print

STEP 1: Select your printer or template printer.

STEP 2: Import your model and configure the tray settings.

STEP 3: Adjust the ***Model Settings:***

Change ***Infill Style*** to ***Sparse.***

Change ***Infill Density*** based on the chosen slice thickness (Table 3).

Verify the ***Body thickness*** is the equivalent of 3 contour widths.

Table 3. Sparse infill density to reduce post tank swelling

Slice Thickness	Infill Density
0.007 in. (0.178 mm)	100%
0.010 in. (0.254 mm)	100%
0.013 in. (0.330 mm)	100%

STEP 4: Process the part.

When preparing packs in Control Center™ or GrabCAD Print, the default settings add a full height purge tower. This is strongly recommended to ensure the best part quality.

System Preparation

Extrusion head life

The system should be prepared using the heads and build tray as mentioned in the system overview section of this document. A manual tip calibration must be performed when switching from a different material to Nylon-CF10, and when replacing the heads at the end of their recommended life. The life of the F123 Series CR extrusion head is 1500 hours of printing. The user will receive a warning after 1350 hours. Due to geometry differences in the tip of the F123 Series CR extrusion head, automatic calibration can be used to get closer to a correct XY tip offset, but a manual calibration is still required; for more details review the F123 Series User Guide.

Material autochangeover

When preparing the F370CR printer, consider whether an autochange of model material is acceptable. If not, ensure that a spool with sufficient material is installed and loaded to the extrusion head. If an autochange is acceptable, try to match material lots (noted on the material packaging) to minimize any visible difference in color in the printed part based on the spools being utilized.

0.007 in (0.178 mm) layer printing

When printing with 0.007 in. (0.178 mm) layer thickness, the extrusion temperature is increased. This print setting is more sensitive to moisture than the other slice heights. Oozing may be observed even though parts printed in thicker slice heights using the same spool had perfect print quality. If oozing is observed and impacts part quality, switch to a new spool of Nylon-CF10 material or dry the spool before printing using 0.007 in. (0.178 mm) layer thickness.

Handling low-filament spools

Due to the stiffness of the material, when the spool gets below ~5% remaining, the filament will start loosening on the spool when unloaded. Although the spool will be loose, it is still usable, and should be stored by clipping the end of the filament into the retention clips along the spool's perimeter or taping the filament to the perimeter. Ensure that only the end of the filament fed into the bay drive is secured in either manner for storage.

Material Handling/Moisture Management

Nylon-CF10 is moisture sensitive and should be kept in its original packaging or sealed carrier storage bags when not in use. Letting the material sit in ambient air for an hour could cause it to become wet, resulting in poor print quality or loss of extrusion. If the material has been idle in the machine for more than 24 hours, discard the material in the tubing between the bay and the head and reload the material. Even with continuous use the material will only remain dry enough for printing for three weeks in the sealed material bay. Past three weeks, the material should be removed from the printer and dried before printing is resumed. To dry the material, insert the spool into an oven at 70 ˚C (158 ˚F) for a minimum of 4 hours. Drying overnight is the longest recommended time for exceptionally wet material.

Support Removal

Larger pieces of soluble support structures may be manually removed to expedite the process before placing the part in the support removal tank, but this is not required. Parts may be placed into a mesh bag, container, or placed directly into a tank of WaterWorks™ solution set at 50 °C (122 °F) until soluble support is fully dissolved (typically 1 hour minimum). The mesh container protects the part from collisions with other parts or the sidewalls of the tank as it circulates. To prevent warping or damage, avoid placing weight or pressure directly on parts while they are in the tank. Make sure the parts are fully submerged and avoid placing weight or pressure directly on parts while they are in the tank. Rinse the parts thoroughly and dry them after they are removed from the WaterWorks solution.

Dry parts lying flat; an oven set at no higher than 50 ˚C (122 ˚F) may accelerate drying time. Decrease the oven temperature if any warping is observed. A vacuum oven is another alternative to decrease drying time.

Some warping or swelling of the parts may be observed after the parts are dried. For mitigation strategies, see the Part Processing Considerations section of this document.

As mentioned previously, using SUP4000B breakaway support when possible is an effective way to avoid warping due to nylon's hygroscopic nature.

Post Processing

Nylon-CF10 can be sanded, media blasted, bonded, machined, drilled, receive inserts and more, similar to other thermoplastics. In addition, the parts can be dyed black using commercially available dyes; for this process ensure that the tank temperature is not above 50 ˚C (122 ˚F).

Stratasys Headquarters

7665 Commerce Way,
Eden Prairie, MN 55344
+1 800 801 6491 (US Toll Free)
+1 952 937-3000 (Intl)
+1 952 937-0070 (Fax)

1 Holtzman St., Science Park,
PO Box 2496
Rehovot 76124, Israel
+972 74 745 4000
+972 74 745 5000 (Fax)

stratasys.com
ISO 9001:2015 Certified