



FDM Nylon 12 Carbon Fiber

Overview

FDM Nylon 12 Carbon Fiber (12CF™) is a chopped carbon fiber-filled (35% by weight) polyamide 12, available on compatible Fortus® 3D Printer. The material has a high strength-to-weight ratio and a tensile strength on par with other high-performance FDM® materials. It has the highest specific modulus of any FDM material. Due to the unique directional orientation of carbon fibers (Figure 1), the material exhibits anisotropic properties that can be tailored

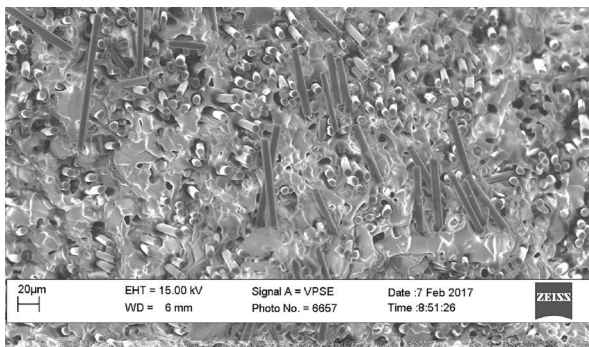


Figure 1: SEM image, top-down view of the filament cross-section showing the fiber alignment in the direction of extrusion.

similar to a composite laminate. This enables parts to have a higher uniaxial stiffness than injection molded parts of the same material. It opens up unique design freedoms, not present in either traditional thermoplastic or additive manufacturing.

These characteristics make FDM Nylon 12CF a good fit in a variety of functional prototyping and tooling applications in the following industries:

- **Aerospace** – UAV components and tooling
- **Transportation and auto** – bumpers, fixtures and brackets
- **Consumer goods** – performance sporting goods
- **General manufacturing** – tooling, fixtures and other manufacturing aids

Pre-processing – In addition to standard STL processing procedures, proper configuration of the modeler and modification of build parameters can dramatically impact the mechanical properties and build quality.

Machine preparation and printing – When packing parts for a job using Control Center™ software, proper placement and orientation of the part on the build platen and machine maintenance considerations are critical to success.

NOTE

The XYZ tip offset calibration is extremely important. Carefully follow the procedure outlined in Section 2.1.2 to prevent build failure or loss of extrusion due to improper tip offset calibration.

Post-processing – Proper part removal from the build chamber and proper support removal technique can improve user experience and quality of the resulting parts.

1. Pre-Processing Considerations

1.1 Prepare Part File Using Insight™ Software.

1.1.1 Part orientation

Because the FDM process results in anisotropic surface finish and mechanical properties, proper orientation is dependent on the part's function. For functional or production parts, build orientation should be selected based on the forces acting upon the printed part. By selectively orienting the part to avoid excessive stresses on thin geometries or in the z-direction, the mechanical properties of critical features can be optimized. For conceptual models or parts where aesthetics are most important, orientation should be selected to reduce the appearance of layer lines on critical surfaces. For more information, reference the **Best Practice: Orienting for Strength, Speed or Surface Finish**.

1.1.2 Build mode selection

The default build mode is **Normal**, which is recommended for most geometries (Figure 2). For thin-walled parts (sections thinner than 0.508 mm [0.2 in.]), or with small individual parts that show signs of the part melting when printed using the default build mode, utilize the **Thin wall mode**. The **thin wall mode** reduces the oven temperature and the likelihood of melting on thin walls and small individual parts.

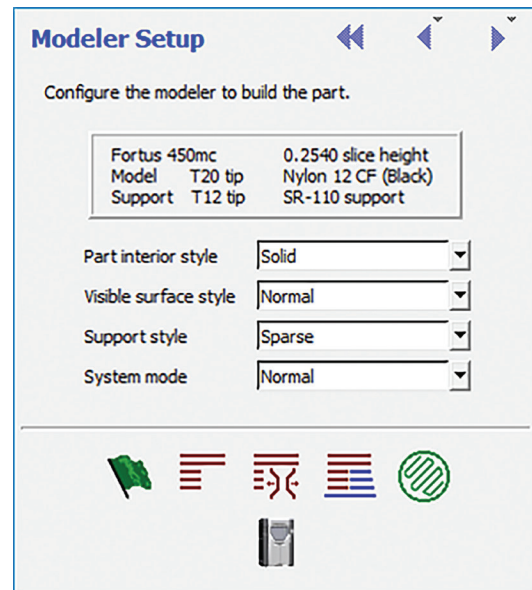


Figure 2: The System Mode, Part Interior Style, and Support Style can be selected in Modeler Setup.

1.1.3 Build style selection

The default build style for FDM Nylon 12CF is **Solid**. The **Sparse** fill style is also available and will reduce material consumption.

NOTE

Sparse - double dense is not a fill style option for FDM Nylon 12CF. Instead, users should create a **Custom Group** and select the **Alternate sparse fill style** as **Sawtooth**.

1.1.4 Toolpath parameters

It is recommended to use the default raster and contour widths of 0.508 mm (0.020 in.).

NOTE

Avoid using contour widths thinner than the defaults as it can result in inconsistent surface quality and finish.

1.1.4.1 Raster angle

The default raster angle is 45° with a delta angle of 90° every layer. Modification of the raster and delta angle may improve the mechanical properties as measured in certain directions, but may also result in part curl in some combinations.

NOTE

For thin flat parts and sections (less than 10 layers) and an even number of layers, symmetry about the mid-plane must be maintained by using **Custom Groups** (Figure 3). Create a new **Custom Group** and modify the raster **Start angle** to be equal to 0° or 90°. Add the midplane to the newly defined **Custom group**; maintain the default raster angles for all other layers.

1.1.4.2 Seam control

The seam location is where the material extrusion toolpath starts and ends on each closed part curve. The recommended seam placement method is **Align** or **Align to nearest** as it yields better surface quality. Seam Control is accessed from the **Toolpaths** menu. For more information, reference the **Best Practice: Optimizing Seam Location**.

NOTE

For thin parts, avoid placing seams directly on the edge of the part.

Odd number of layers			
	Layer	Alternating	Midplane Symetry
	1	45	45
	2	-45	-45
	3	45	45
	4	-45	-45
Midplane	5	45	45
	6	-45	-45
	7	45	45
	8	-45	-45
	9	45	45

Even number of layers			
	Layer	Alternating	Midplane Symetry
	1	45	45
	2	-45	-45
	3	45	45
	4	-45	-45
Midplane	5	45	45
	6	-45	45
	7	45	-45
	8	-45	45
	9	45	-45
	10	-45	45

Figure 3: Schematic showing the difference in midplane symmetry for an odd and even number of layers. To maintain symmetry about the midplane when there is an even number of layers, a change in raster angle may be required.

1.1.4.3 Z Shrink Height

The Nylon 12CF shrink factor is calibrated to be good for most use cases, but the amount of Z shrink can vary based on the geometry and mass of the part. Customers should be aware of the shrink issue and check their critical parts for out-of-spec measurements.

After printing, tanking, and rinsing the part, measure the height, especially any critical areas. If the part height is unacceptable you can adjust the Z shrink factor, in the Insight software only, using the following equation.

$$\text{New Shrink Factor} = \frac{\text{Nominal Value}}{\text{Measured Value}} * \text{Current Shrink Factor}$$

Choose only one critical height to perform this calculation. The current shrink factor can be found in the **Toolpaths** drop down menu, **Setup**, and then **Advanced Settings**. It is called **Part Z shrink factor** under the **Additional Settings** section.

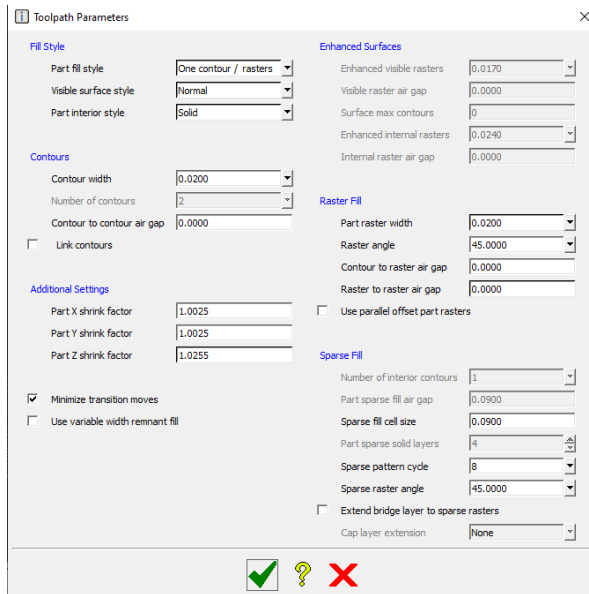


Figure 3a: Finding the shrink factor in the Toolpath Parameters menu.

Once the new shrink factor is calculated, either open the original part in Insight or process it in Insight with the original settings and enter the new shrink factor in the same pop up location. Re-generate toolpaths, save the CMB and print the part. (Note that all parts in a pack must have the same Z shrink factors.) Once the part is finished, tanked, and rinsed, remeasure the part. It may take more than one iteration of shrink factors to achieve an acceptable height.

The other factor that can affect the height of Nylon 12CF parts is moisture absorption. To find out more about both of these phenomena, refer to the Height Accuracy of FDM Nylon 12CF Material White Paper.

1.1.5 Support parameters

Due to the mismatch in the coefficient of thermal expansion between the model and support material, the support parameters must be set to prevent build issues or support-removal issues.

1.1.5.1 Use model as support

In the **Support Parameters** dialog box, the checkbox for **Use model material where possible** is selected by default (Figure 4).

This reduces the impact of the mismatch in the coefficient of thermal expansion between the model and support materials on the part. Additionally, the time that the model tip is the inactive tip is shorter, which also minimizes the potential for extraneous model material to be deposited into the part.

NOTE

For parts greater than 76 mm (3 in.) in height, the **Use model material where possible** option must be enabled due to the differential in shrink rate between the model and support materials.

1.1.5.2 Support style selection

The support style is configured in *Modeler Setup*. The default support style is **Sparse**. **Sparse** or **Basic** support styles are recommended for most builds. The exception is if the part contains internal channels or cavities where the model material used as support may become trapped and difficult to remove. For parts with internal channels and cavities, the recommended support style is **Box** (Figure 5).

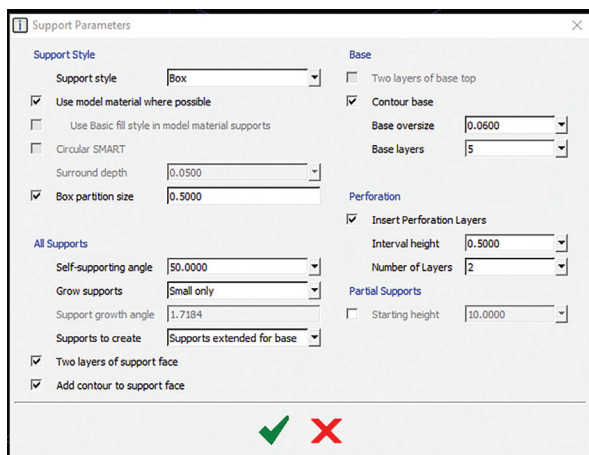


Figure 4: Changes to the Support defaults can be made in the Support Parameters, including Box partition size, Perforation interval and Layers of perforation.

1.1.5.3 Box support style

Box support is similar to sparse supports, but consists of boxes instead of continuous raster. There is no closed toolpath-perimeter curve around the box supports. Box supports will build more slowly than sparse supports. By default the **Box partition size** is set to 12.7 mm (0.5 in.) which limits the length of the partitioned supports.

When using the **Box** support style with the **Use model material where possible** option selected, there is an option to **Partition box supports**. When this is enabled, the user has the option to select the **Box partition size** from a dropdown menu. When the option for **Partition box supports** is enabled, the default partition size

is 12.7 mm (0.50 in.). This is the recommended partition size unless the part geometry requires otherwise. Using this option can improve both the manual and automated support removal processes.

NOTE

The use of a partition size smaller than the default is not recommended unless it is required for the partitioned model-as-support to be removed from an internal cavity. In this case use the largest partition size that will still allow the supports to be removed.

For parts that contain delicate or fine features, use a partition size of 6.35 mm (0.25 in.).

1.1.5.4 Perforations

When using model as support, in the *Support Parameters* dialog box ensure that the checkbox for **Insert Perforation Layers** is selected with an interval height of 12.7 mm (0.50 in.) and **Number of Layers** set to 2 (Figure 5). For models that have nearly enclosed cavities, set the **Number of Layers** to 3 and adjust the intervals so that the “boxes” of model-as-support can be removed from the cavity once the soluble support material is dissolved.



Figure 5: A FDM Nylon 12CF part printed with sparse supports, using model as support where possible and two layers of perforation with a 0.5-inch interval.

NOTE

Take care when performing the XYZ tip offset calibration because a poor calibration can result in long dissolution times for the perforation layers in box supports, or incomplete perforation. Both will result in difficulty removing the support from internal cavities.

1.2 Build Preparation Using Control Center Software

1.2.1 Part placement due to the airflow

Due to the airflow configuration of the Fortus system, proper part placement can prevent and alleviate quality issues.

For the Fortus 380mc and 450mc platforms:

- **Single Parts** – Place near the center of the platen. For thinner parts the part should be placed in the front right corner of the build platen. For thicker geometries, the part should be placed near back left corner of the platen.

NOTE

For single parts that are less than 12.7 mm (0.50 in.) in cross-section, add a full height Sacrificial Tower to the pack or build an additional copy of the part. This will result in improved surface quality compared to the single part.

- **Multiple Parts** – Place in the center and move concentrically outwards.
- **Mixed Pack** – Place the tallest parts in the center and move concentrically outwards. Place parts with thick-wall sections to either side (left or right) of the platen. Avoid placing thin-walled parts to the rear of the platen.

For the Stratasys F900 platform:

- **Thin Parts** – Place in the front right corner of the build platen. For thicker geometries, the part should be placed near back left corner of the platen.

1.2.1.1 Sacrificial Tower

The Sacrificial Tower is the first part built that improves the quality of the other parts in the pack. The Sacrificial Tower is automatically placed in the right-front corner of the packed parts assembly,

but can be repositioned by the user. The use of a **Full height** Sacrificial Tower is the default setting and is recommended with all FDM Nylon 12CF builds.

STEP 1: On the **Pack** tab, click on the packing **Options** button.

STEP 2: In the **Sacrificial Tower** options drop-down menu, ensure the **Full height** option is selected.

STEP 3: Click “OK” to confirm your selection.

2. Machine Preparation And Printing

2.1 Machine Preparation

Proper machine preparation is important for safeguarding the machine against damage, but also to ensure good build quality. Follow regular maintenance outlined in the appropriate Fortus System User Guide. Also, ensure that each of the following items is completed when starting a job.

2.1.1 Maintenance

STEP 1: Clean the platen, empty the purge bin and vacuum the build chamber.

STEP 2: Inspect the gantry area, specifically the flexible heat shield that isolates the head and gantry area from the oven. If there is any black filament powder accumulating in the heat shield, carefully vacuum the area.

NOTE

You may need to move the Y carriage assembly around in order to vacuum all of the filament powder. Take care not to disrupt or strike any of the electrical components or the belts.

STEP 3: Verify the flicker and brush assembly is clean and calibrated at the correct height.

NOTE

The purged FDM Nylon 12CF material can buildup on the flicker; be sure to clear this material before starting the next job.

STEP 4: Verify the compatible head assembly is installed.

NOTE

It is necessary to install a special head assembly on the Fortus 450mc™ or the Stratasys F900 3D Printers when using FDM Nylon 12CF.

This head is capable of running ASA or FDM Nylon 12CF model materials. When this head is not installed in the Fortus 450mc, 900mc or Stratasys F900 3D Printer, it should be stored in the provided docking container (Figure 6).

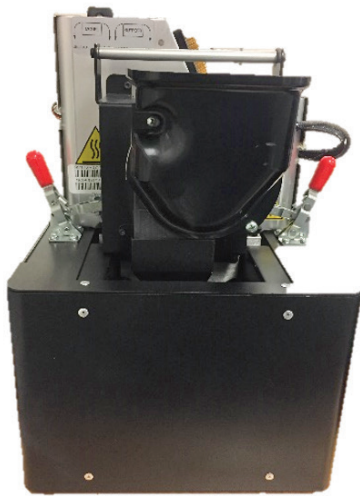


Figure 6: A compatible head assembly is required for FDM Nylon 12CF. The head assembly that is not in use should be properly stored in the docking container.

STEP 5: Install the appropriate model and support tips. Ensure that the tip shrouds are clean and undamaged.

Slice Thickness	Model Tip	Support Tip
0.254 mm (0.010 in.)	T20C	T12 SR-100

2.1.2 Load material

STEP 1: Load the model and support materials.

NOTE

The FDM Nylon 12CF filament is more brittle than other model materials and filament that is out of the canister may break during load and unload if it's bent or impacted. Some black filament powder may be present under the drive blocks during load and unload of the filament.

TIP

The base nylon 12 material is hygroscopic. Load material into each available material bay to prevent excess ambient moisture from entering the filament tubes and degrading build quality.

If the machine is left in an idle state for multiple days, it is recommended you unload the filament from the machine and properly store it until printing resumes. This will prevent tip plugging and diminished part quality resulting from moisture-contaminated filament.

STEP 2: Install a nylon build sheet and verify that vacuum is present.

TIP

For best results, place the build sheet onto the platen in the concave-down orientation if there is a set to the sheet. If there is not a set to the sheet, place the sheet in the oven with the matte/rough side facing down.

NOTE

Build sheets that have been exposed to moisture will make it difficult to establish vacuum. Ensure that the nylon build sheets are properly stored in the sealed foil bag, with desiccant.

STEP 3: Perform XYZ tip offset calibration. Repeat until the support layer **measurement** is within ± 0.0127 mm (± 0.0005 in). Be sure to inspect the profile and look for flat edges on the top and bottom (Figure 7). Repeat the calibration

procedure if the top and bottom edges appear rounded. Failure to do so may cause poor adhesion between model and support layers, leading to part curl, overfill or tip plugging.

NOTE

Part quality and support removability are directly related to accurate system calibration. X and Y offset must be within tolerance (± 0.05 mm [± 0.002 in.]). The Z slice variation is ideally within ± 0.0127 mm (± 0.0005 in.).

The calibration part used for FDM Nylon 12CF is modified to print faster (Figure 8). Although the appearance of the number labels and calibration box is different, the calibration is still performed in the same manner as other materials on the Fortus.

A blade can be used to assist in removing the support layer for measurement if it becomes difficult to peel off the entire layer.

STEP 4: Start the job.

2.2 PRINTING

While the part is printing, the oven door and top cover are locked for user safety. The touchscreen user interface will provide updates on the amount of material remaining as well as an estimate of when the current job will be complete. In the event of an error, the machine will typically pause, allowing the user to assess the situation and recover the build. One specific build pause warning that can typically be recovered is a “Model filament clogged in tip” (Figure 9).

NOTE

Do not press play and resume the build if you have the build pause warning “Model filament clogged in tip.” Instead, please follow the process in Section 2.2.1, “Recovering from ‘Model filament clogged in tip’ error” to properly recover from this error and resume the build.

2.2.1 Recovering from “Model filament clogged in tip” error

STEP 1: Verify the build pause warning message of “Model Filament clogged in tip” and press the check mark.

STEP 2: Select the **Tips Menu** from the touchscreen user interface to access the **Tip Controls** (Figure 10).

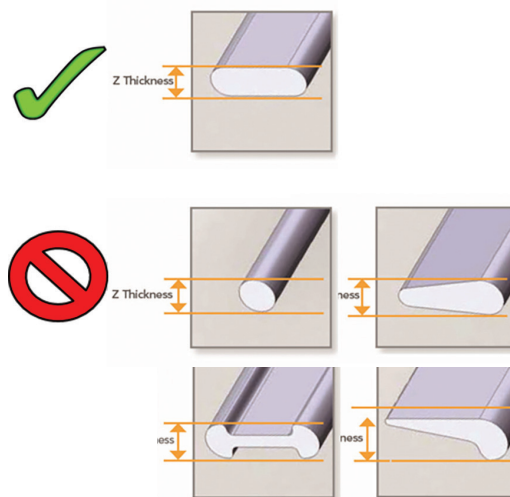


Figure 7: The calibration part used for FDM Nylon 12CF is modified to print faster.

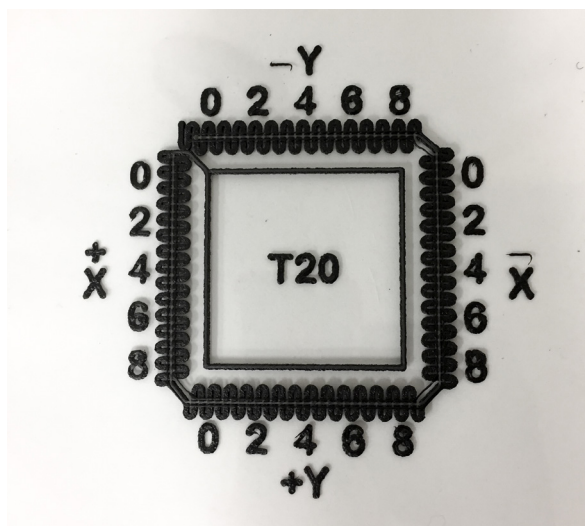


Figure 8: The calibration part used for FDM Nylon 12CF is modified to print faster.



Figure 9: When printing with FDM Nylon 12CF material the build pause warning “Model filament clogged in tip” should be resolved using the procedure in Section 2.2.1 of this document.

Figure 10: The Tips Menu is in the Navigation bar at the bottom of the screen and provides access to the Tip Controls.

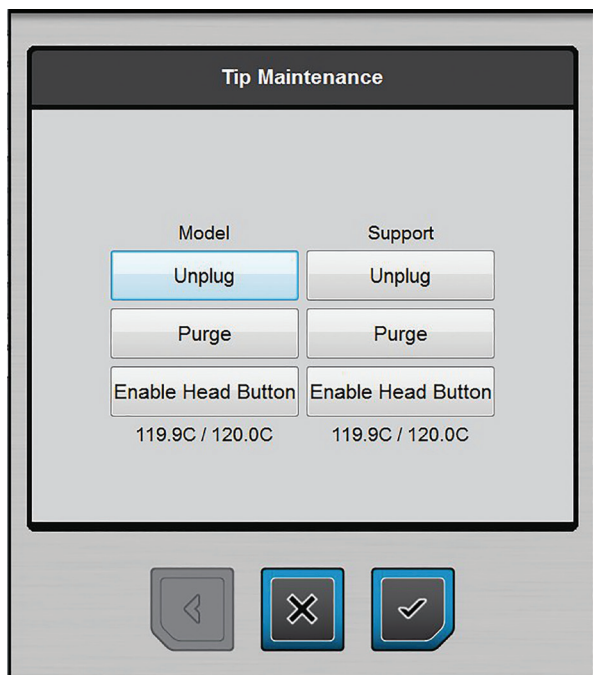


Figure 11: The Tip Maintenance Page has the option to Enable Head Button for the model filament.

STEP 3: Press the **Tip Purge/Unplug** button to access the **Tip Maintenance** page (Figure 11).

STEP 4: On the touchscreen interface press the **Enable Head** button for the model material (Figure 11).

NOTE

After pressing this button the tip will begin to heat to a predetermined set point temperature. Once the tip reaches temperature, open the top cover and begin using the tip's head button.

STEP 5: Open the top cover of the machine to gain access to the Head buttons.

NOTE

The head can be pulled closer to the user to allow easier access to the fitting and the head buttons.

STEP 6: On the head assembly, press the model "UNLOAD" button to move the model filament away from the tip (Figure 12).

STEP 7: Use the model-side tube fitting to release the model filament tube (Figure 13).

STEP 8: On the head assembly, press the model "LOAD" button to advance the filament out of the tube for inspection.

STEP 9: Look for a kink in the filament (Figure 14); use a pair of clippers to trim the entire kinked portion off the end of the filament at a 45-degree angle. Trim at least 50 mm (2 inches) of filament. After trimming, ensure that the filament end does not protrude out of the filament tube more than 2.54 mm (0.1 in.).

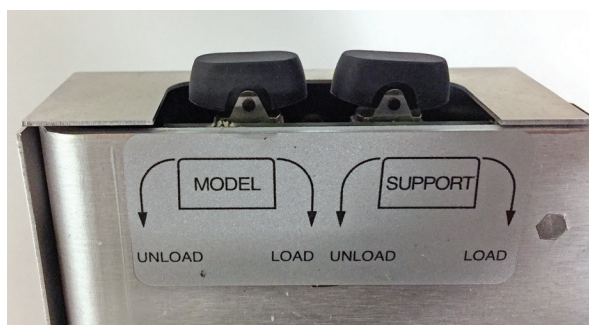


Figure 12: The model-side UNLOAD/LOAD rocker button is located on the top left of the head assembly.

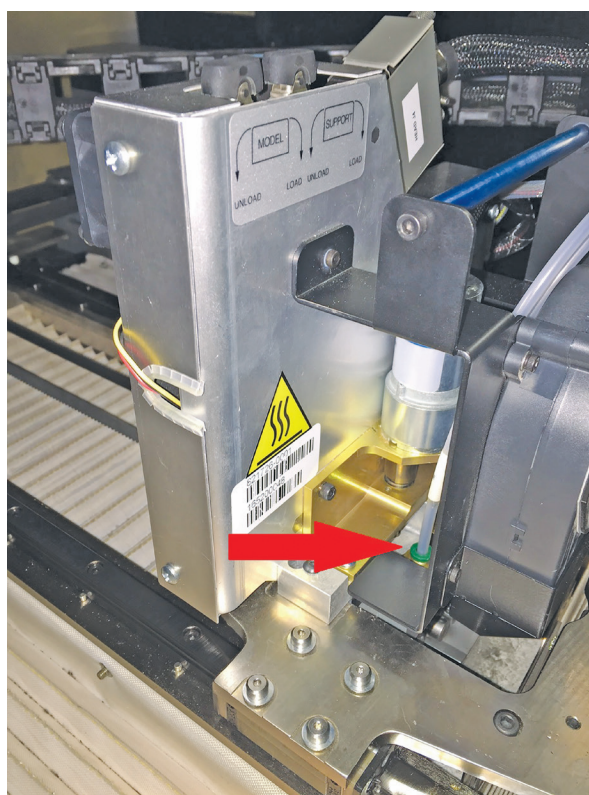


Figure 13: The model-side tube fitting is used to release the filament tube.



Figure 14: A section of filament that shows a kink that can correspond with the build pause warning “Model filament is clogged in tip.”

STEP 10: Reseat the model filament tube into the fitting and firmly push it in. Give the model filament tube a light pull to ensure it is locked in place.

NOTE

The hardened fittings on the FDM Nylon 12CF compatible head accepts more filament tube than the standard fittings. Be sure the filament tube is properly installed.

STEP 11: On the touch screen interface check that the head buttons are still enabled; if they are disabled, press the Enable Head button for the model material.

STEP 12: On the head assembly, press the model “LOAD” button to advance the filament. Look at the model tip in the oven and confirm that material is being purged.

STEP 13: Repeat Step 12 of this procedure until material is visibly purged from the model tip.

NOTE

If after several attempts, no material visibly exits the model tip, you will need to replace the model tip.

STEP 14: Close the Top cover.

STEP 15: Press the **Play** button to resume the build.

NOTE

There may be a visible line at the layer which the pause took place. The visibility of this line can be minimized by executing the procedure quickly and soon after the initial pause.

2.3 Remove Part

Remove the build sheet and part from the build chamber. Wear appropriate personal protective equipment (PPE) when handling because the parts may be sharp, posing a danger of minor scrapes, scratches or splinters. To prevent part warping,

allow the part and build sheet to cool before removal. The build sheet can become quite brittle as it cools down and may break off in pieces when removing the part from the sheet.

TIP

The part may sometimes appear to change geometry as it cools. It will typically return to the nominal geometry upon soluble support removal.

2.4 Remove Support

2.4.1 Soluble support removal

Large support structures may be manually removed before placing the part in the support removal tank, but this is not required. Parts with the recommended **Use model material where possible** option enabled should be placed into a mesh bag or container, then placed into a tank of WaterWorks™ solution set at 70 °C (158 °F) to properly rehydrate the part. Although the soluble support may be fully dissolved before this, leave it in the heated tank for a minimum of four hours. The mesh container prevents the non-soluble model-as-support from floating freely and damaging the impeller of the tank. After the parts are removed from the WaterWorks solution, they should be rinsed thoroughly and dried.

2.4.2 Support removal technique

Some manual support removal may be required even after the soluble SR-110™ support is dissolved away. Remaining supports have a weak mechanical bond to the model and can be removed with pliers or a pick.

2.5 Finishing

Parts can easily be sanded to make the surface smoother or more uniform in appearance.

3. Safety

Observe manufacturer recommendations for safety, material handling and storage. This information can be found in the Safety Data Sheet (SDS).

Recommended PPE

- Gloves
- Protective eyewear

4. Tools & Supplies

- Chisel
- Magnetic handle pick set
- Pliers
- Needle-nose pliers

5. Materials & Consumables

- FDM Nylon 12CF (Fortus Plus)
- SR-110 (Fortus Plus)
- T20C Tips
- T12 SR100 Tips
- Nylon foundation sheet

6. Software

- Fortus 380mc Carbon Fiber Edition system: Insight Version 12.2 or newer, Control software version 5.10.6438 or newer, GrabCAD Print version 1.20 or newer
- Fortus 450mc system: Insight version 11.2 or newer, Control software version 5.8.6414 or newer, GrabCAD Print version 1.19 or newer
- Fortus 900mc system: Insight version 12.0 or newer, Control software version 3.27.0 or newer, GrabCAD Print version 1.19 or newer

7. Printers

- Fortus 380 Carbon Fiber Edition
- Fortus 450 (licensed and configured for FDM Nylon 12CF)
- Fortus 900mc (licensed and configured for FDM Nylon 12CF)
- F900 (licensed for FDM Nylon 12CF)

The information contained herein is for general reference purposes only and may not be suitable for your situation. As such, Stratasys does not warrant this information. For assistance concerning your specific application, consult a Stratasys application engineer. To ensure user safety, Stratasys recommends reading, understanding, and adhering to the safety and usage directions for all Stratasys and other manufacturers' equipment and products. In addition, when using products like paints, solvents, epoxies, Stratasys recommends that users perform a product test on a sample part or a non-critical area of the final part to determine product suitability and prevent part damage.



서울특별시 금천구 가산디지털1로 19(가산동 670-2),
대동테크노타운18차 302호

TEL:02-6929-4117

E-mail:prototech_ae@prototech.co.kr

ISO 9001:2015 인증