Using PreForm 2.20 with Form 1+

formlabs

Table of Contents

Introduction

The Form 1+ is compatible with Firmware 1.4 and PreForm 2.20. Later versions of Firmware or PreForm are not compatible with the Form 1+. If you are unable to download compatible versions of Firmware or PreForm, please contact Formlabs Support or an authorized reseller for additional assistance.

Follow this guide to start printing with PreForm. More than just a walkthrough of how to send a digital model from your computer to the printer, this guide outlines all the aspects that go into a successful print. Learn how to import a file, support a model, orient a model, detect weak spots, adjust different layer thicknesses, and change advanced support settings. Using a variety of screenshots and examples to highlight dos and don'ts in 3D printing, you'll pick up best practices to apply in future projects.

Further information on the Form 1+ can be found on the Formlabs Support website under Main Topics > Form 1+. Troubleshooting information can be found in Frequently Asked Questions (FAQ).

> **NOTICE:** This guide only contains information specific to the Form 1+ and PreForm 2.20.0.

Designing a Printable Model

Consider the model's design as a primary factor for each print's success. These basic modeling guidelines help ensure good prints.

As you design your model in your favorite 3D CAD package, you will want to make sure it can successfully be printed. Some 3D CAD models are more difficult to print (and some are impossible on any printer). Read our design guide and then follow these basic guidelines when designing models for the Form 1+ .

Use Supports
Supports are an important feature to give each part a foundation and structure for accurate printing. While printing without supports can be tempting, especially for models with a flat surface, these are prone to shrinkage and compression at the layers closest to the build platform.

Allow Resin to Escape
A fully enclosed cavity (like a hollow sphere) will inevitably trap some resin inside while printing. Design such cavities with a drain hole. We suggest the minimum diameter of drain holes to be 3.5 mm.

> **TIP:** To conserve resin and decrease print time, print your model as a hollow object.
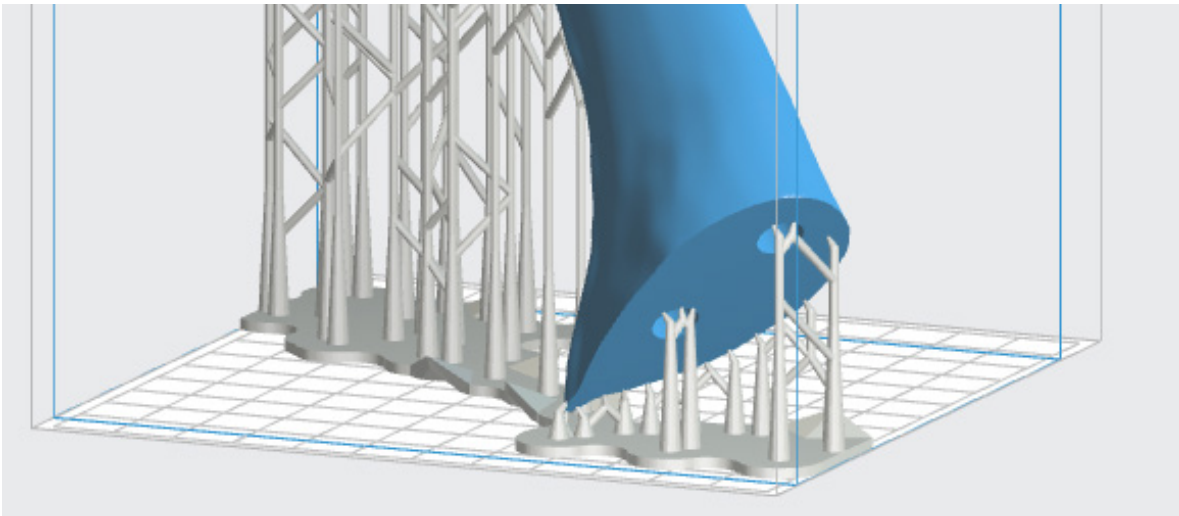
Consider Minimum Feature Constraints

The Form 1+ outperforms other desktop machines with features as small as 300 microns. Remember that pushing the limits of small details can result in failed prints. Very thin, wiry parts are prone to failure, for example. Of course, PreForm Software will not recognize features smaller than 300 microns, meaning they won't show up in your print.

### Use Internal Supports When Printing Internal Overhangs

Overhangs are features that require support during a print. PreForm Software generates supports so that an overhang doesn't fail to print. With the "internal supports" feature, PreForm can also create supports for internal overhangs (i.e., features that hang over other parts of the model). In order to have PreForm generate internal supports:

- Open the Support Generation dialogue.
- Click the "Internal Supports" checkbox.

internal

external
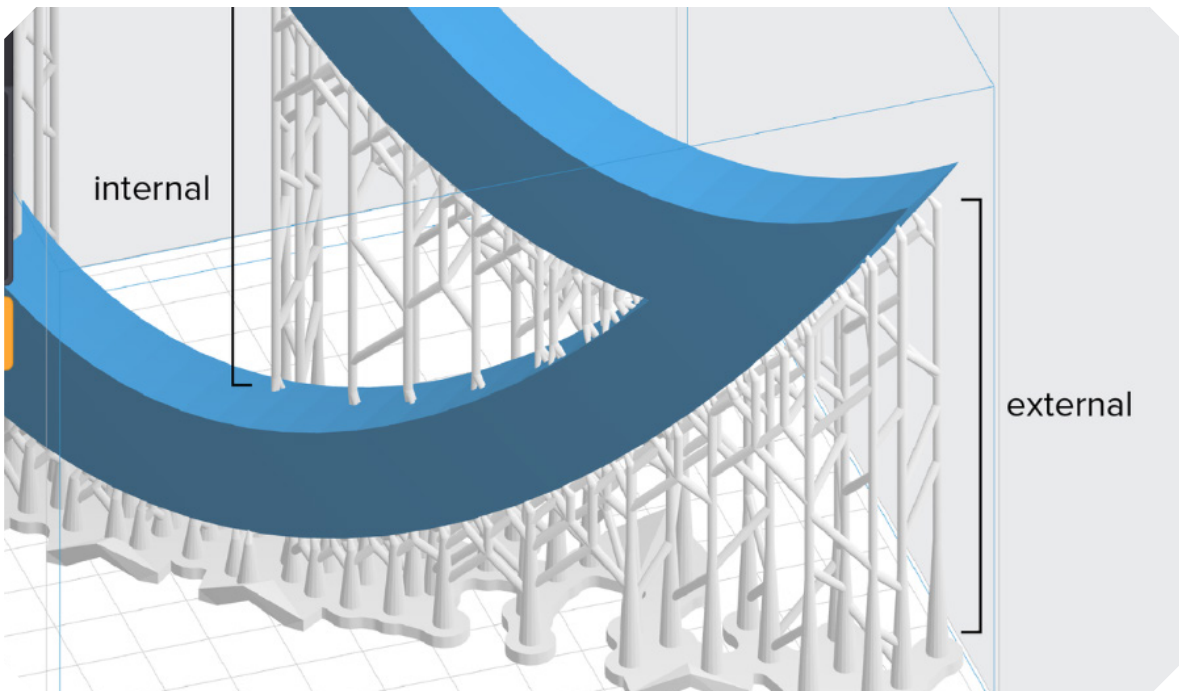
Consider the Print Volume

Prints have maximum dimensions of 4.9 × 4.9 × 6.5 in (125 × 125 × 165 mm) for the Form 1+. PreForm Software will help you resize your model to fit the build volume. For larger parts, consider designing them in multiple sections that can be assembled after printing.
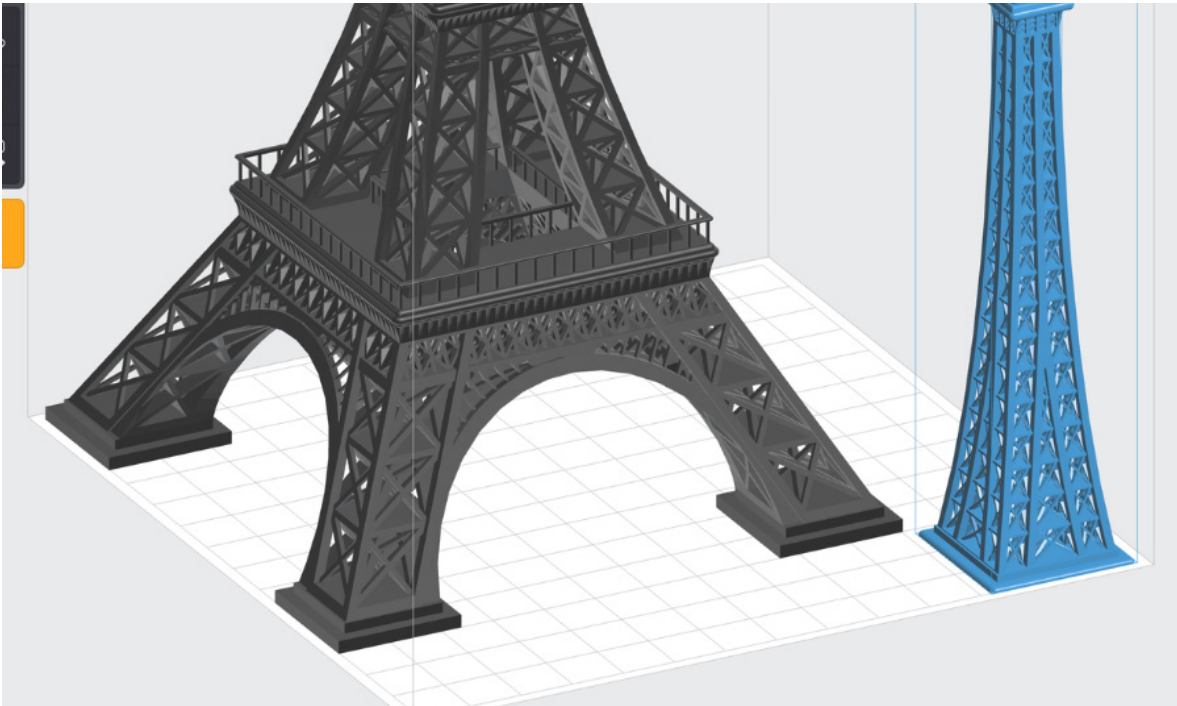
Use Single Body (or Mesh) Settings

If your final model is the combination of multiple sub-models, make sure to save it as a single body (or mesh) in your CAD package. Otherwise, there may be some irregularities in your print if the intersecting layers of the sub-models are not correctly recognized by PreForm Software.

Starting a Print

To start printing successfully, take time to learn and use PreForm software to setup each model and ensure a successful print.

You've set up your printer, filled the resin tank, and installed PreForm Software. You're ready to start printing! Make sure your printer is plugged into a power outlet and connected to your computer with a USB cable.

Open PreForm Software

The Print Setup window opens when PreForm launches. Use the material selection drop-down menus to select the material type and version for the specific print job.

> **TIP:** Select "Current Printer Settings" in the Material drop-down menu to apply the material settings last used on the printer. Change the layer thickness setting later by clicking "Print Settings" on the bottom bar.
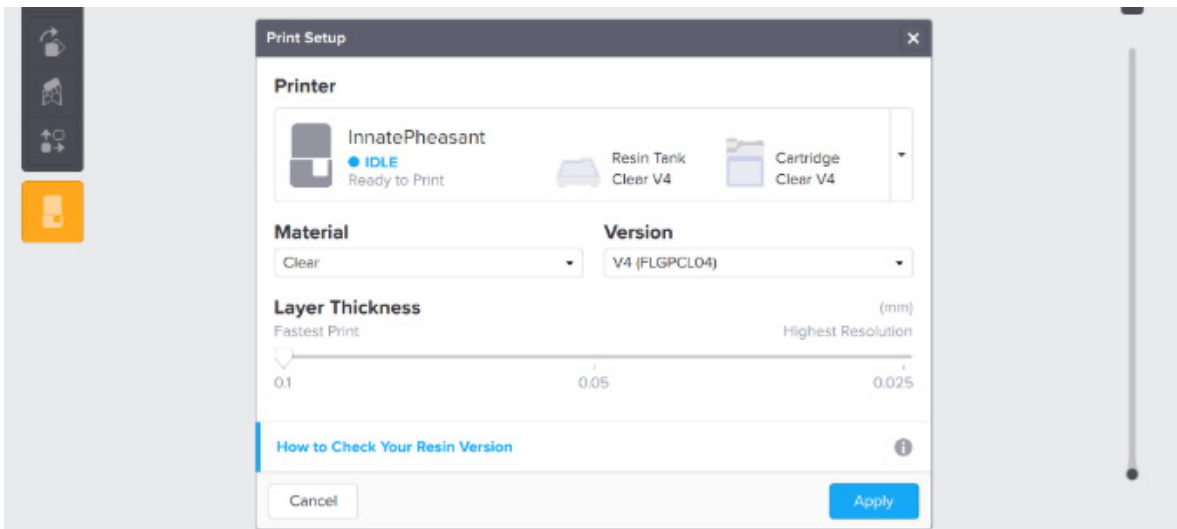
Formlabs Account Login

Log in to PreForm with a Formlabs account to link print jobs to the account and to enable Dashboard notifications for only prints linked to the account. The linked account name also appears for print jobs uploaded onto the printer while logged in.

> **TIP:** Select "Current Printer Settings" in the Material drop-down menu to apply the material settings last used on the printer. Change the layer thickness setting later by clicking "Print Settings" on the bottom bar.

Load a Model

PreForm Software supports STL and OBJ files, which you can export from most 3D CAD or animation packages. To load a model, drag and drop it into Preform or select "Open" from the File menu.

One-Click Print

Select the One-Click Print option to automatically orient, support, and layout each part. After sequentially completing all three print setup steps, PreForm opens the Print dialog window to confirm and send the job to a printer, so ensure that you've connected and selected the appropriate printer type, material type, and layer height before beginning One-Click Print. The specific printer can be selected by serial name from the Print dialog.

Change the View

Viewing your model from the correct angle can help you position things in just the right way. PreForm offers a couple of ways to control your view.

The View Menu

Hover over the View icon to reveal the view controls. The center icon will always take you to the initial view. The other icons rotate your view 90 degrees to the top, sides, or bottom. You can use the plus and minus buttons to zoom in and out.

> **TIP:** Right click and drag will rotate the view about the model. Right mouse and shift will pan. Use the scroll wheel to zoom in and out.

Mouse and Keyboard Controls

Control PreForm using the mouse and keyboard. Type '?' at any time in PreForm to view a list of keyboard shortcuts. Learning these keys will make part set-up fast and easy.

PreForm can also be controlled using third-party SpaceMouse hardware. PreForm automatically detects the SpaceMouse if the mouse device is connected when PreForm opens. Use the SpaceMouse to rotate, zoom, or pan the camera view. The first button on the SpaceMouse will reset the camera view, similarly to typing 'F' on the keyboard or selecting View > Home in the menu bar.

Model List

Use the Model List to find and select individual models in the scene by model name. To show or to hide the Model List, select or deselect "Show Model List" under the "View" menu. To turn on or to turn off the visibility of a model in the print scene, select or deselect the model in the Model List. Only the models visible in the scene will print.

Right-click any model in the Model List to do the following:
- View model properties and change the model name.
- Duplicate the model in the scene.
- Delete the model from the scene.

Model Properties

To view the model properties or to change the model name, right-click any model in the print scene or in the Model List and select "Model Properties".

"Model Properties" displays the following information:
- *File:* This field displays the file name of the original OBJ or STL file.
- *Name:* This field displays the model name, which is also used for the Raft Label. To change the model name, click the "Name" box to modify the text. Press Enter/Return or click outside the menu to save the new model name. Updating the model name also updates the Raft Label.
- *Layers:* This field displays the number of layers in the printed part.
- *Volume:* This field displays the estimated resin consumption for the printed part.
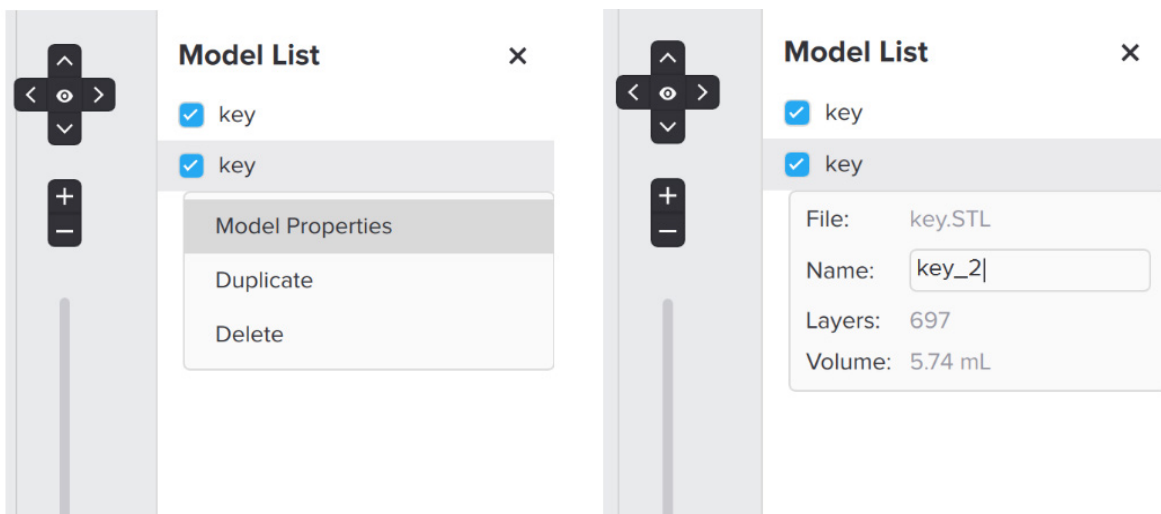
Toolbar Menu

Select a model by clicking on it. The model will turn blue and display the overlaid manipulator sphere with three circular axes to indicate that the model is selected. Once a model is selected, use PreForm Software features to manipulate the object.

> **TIP:** To toggle the visibility of the overlaid manipulator, select "View > Show/Hide Manipulators" from the PreForm menu.

Size or Scale

Before generating supports, set the size of each model. Use the left toolbar to input size values, or click and drag the arrow in the top-right of the manipulator that is overlaid on the model. Click the "Size" icon or press the "S" key resize the model from the toolbar.

Options for adjusting the size of a print in PreForm:
- *Direct resizing:* When any model is selected, click on the the arrow - above and to the right of the sphere - and drag to resize the model.
- *Scale factor:* change the size of your model by entering a specific factor from the Size toolbar.
- *Nominal dimensions:* Type a specific value into any axis in the Size toolbar to resize the model, with all axes' dimensions adjusting respective to the input value.
- *Units:* Toggle between millimeters and inches from the PreForm Edit menu. Configure the unit settings when exporting the file from the modeling program, before importing into PreForm.
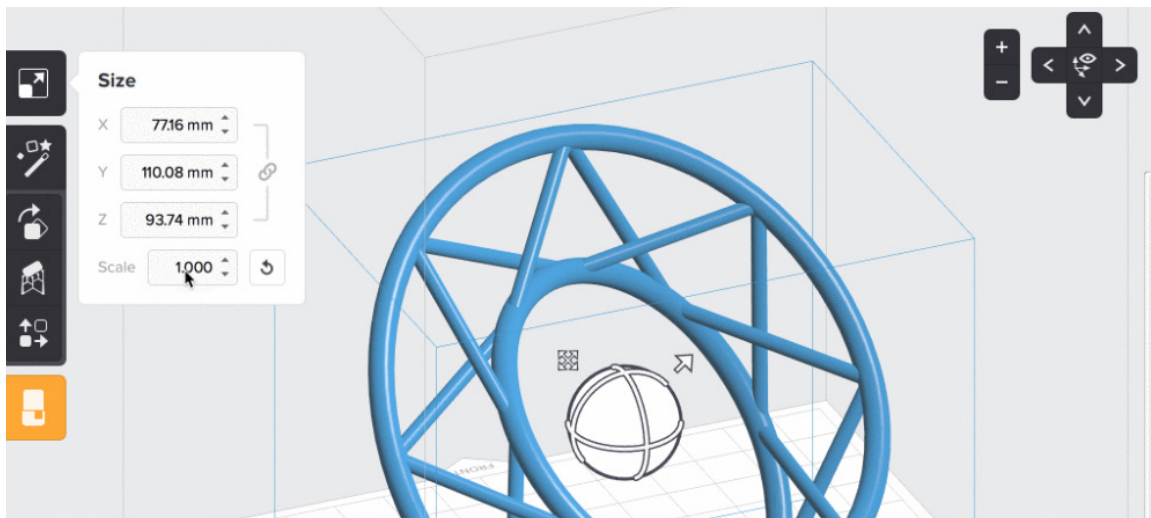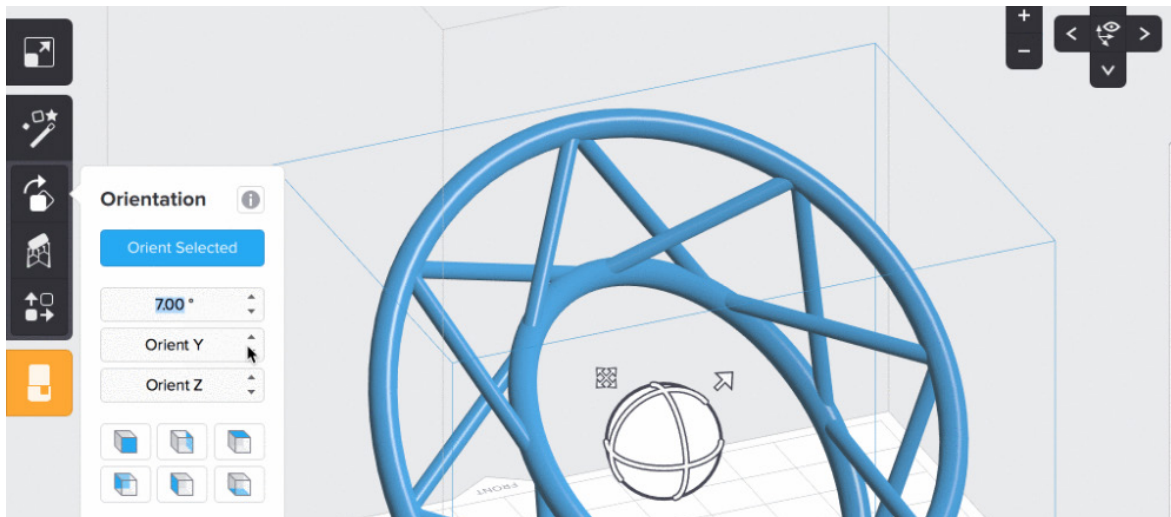

Orient

Models' orientation can be adjusted throughout the print set-up process. Note that changing the X or Y orientation will require regenerating supports. You can use the overlaid spherical manipulator, no matter what tool you have selected on the toolbar to the left of the screen. However, clicking on the "Orientation" (shortcut O) icon in the toolbar will give you additional options for adjusting the model's orientation.

The tools available for adjusting orientation in PreForm are as follows:

- Drag rotation: click to select a model, then click and drag the spherical manipulator to rotate the model freely. Select one of the circular rings to rotate the file about a singular axis.
- From the left toolbar:
    - *Orient Selected:* PreForm will compute an optimal orientation for printing.
    - *Plane-orientation:* Flip your model along each axis with the shaded cube icons.
    - *Axis orientation:* Rotate your model precisely by typing specific angles for each axis.
    - *Select Base:* Select a face to point toward the build platform.

> **TIP:** Model orientation can greatly affect the success of your print. Prepare your model in an orientation that is best suited for print quality on the axis you care about most. We recommend trying auto-orient for optimal results.

Layout or Translate
Configure prints across the build platform before sending the job to print. Use the toolbar, or move any model by clicking and dragging a selected part. Click the "Layout" icon or press the "L" key to reorient your view so that you look down on the build platform from above. From any tool, move the model around by clicking and dragging the four-directional arrow or any part of the model outside of the sphere. Choosing "Layout All" or "Layout Selected" will allow PreForm to recommend a layout for the current file set.

Duplicate
While in Layout mode, you can make one or more copies of the selected model by using the Duplicates feature. You can print as many models as will fit on the build platform. You can also use other PreForm features above to set a different size or alignment in the build area for each copy.

Import Other Models
Your printer can print multiple models at once, whether duplicates or different models. Import additional models the same way you imported the first model. Remove a model from the build area by simply selecting it and pressing delete or backspace on your keyboard.

Generate Supports

Almost all models need support structures for a successful print, especially those with overhangs. Learn more about what supports do for your model. Click the "Supports" icon or press the "C" key to access support structure features:

- *Generate Selected:* Generate supports using the current settings for a selected model.
- *Generate All:* Generate supports for all models in your scene.
- *Advanced:* Configure custom settings for flat spacing, raft thickness, slope factor, and height above raft.

> **TIP:** For trickier parts, it can take time to find the perfect orientation with optimal support settings. Use manual supports to selectively add and remove supports after auto-generating.
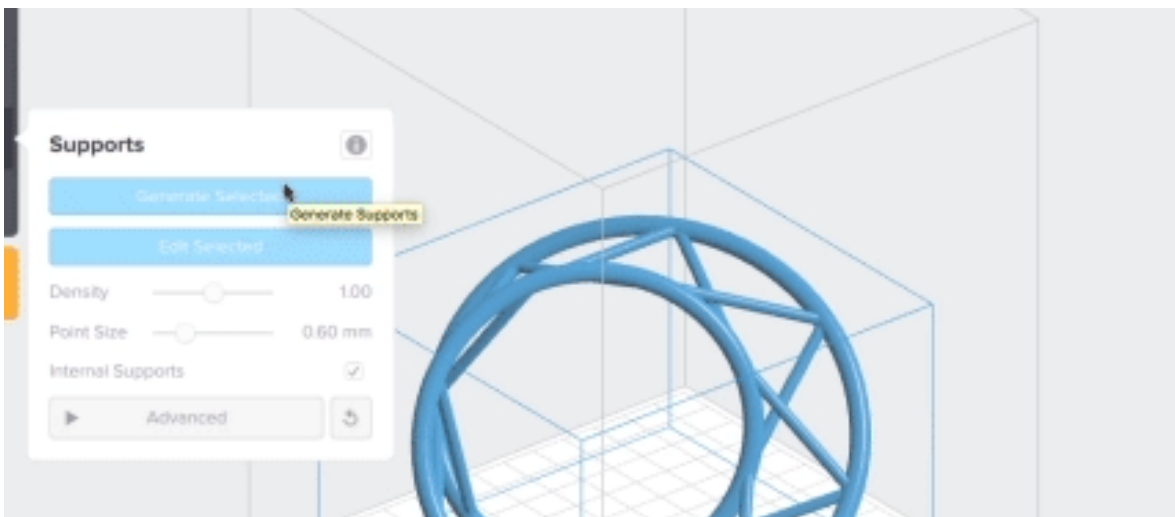
Printability

When a model is loaded into PreForm, algorithms analyze the model geometry and detect areas that may affect the printability, or the likelihood of a successful print. PreForm highlights three types of part features that reduce printability: minima, cups, and undersupported areas. PreForm runs a printability check for minima and cups before and after generating supports. PreForm checks for undersupported areas only after supports have generated.

Minima

A minimum point—plural: minima—refers to the lowest local point in any area of a model's geometry relative to the build platform. Unsupported minima may break off the part during printing and lead to structural defects, warping, or print failures, all of which can contaminate resin, damage the resin tank, or cause resin spills.

Minima are highlighted in red and will fail without additional support. Resolve unsupported minima on models using one of the following methods in PreForm:

- Add at least one touchpoint to the lowest part of the highlighted area using the Supports tool.
- Rotate the highlighted areas away from the build platform using the Orientation tool.

To switch the printability display for minima in PreForm, use the keyboard shortcut Alt+1 on Windows and Option+1 on macOS, or on the PreForm menu bar, click View > Printability Display, and select or deselect "Show Minima".
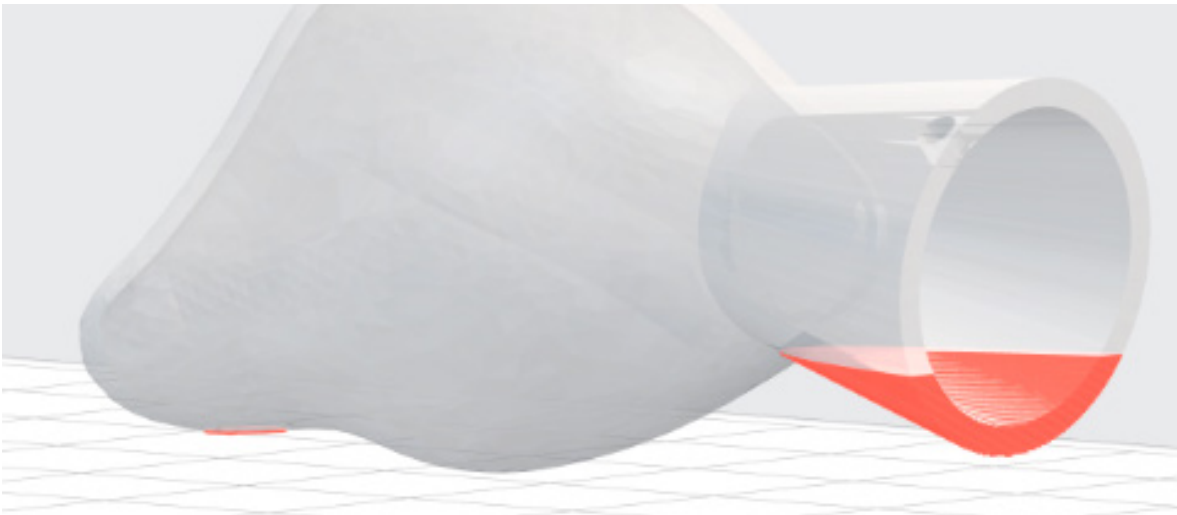

Cups

"Cups" refer to hollow volumes or cavities in a model. Hollow volumes and concave geometries require more careful attention to the print orientation. When the opening of a cup faces the resin tank, the cup traps air and resin as the build platform lowers the opening of the cup against the bottom of the resin tank. The resulting suction can cause separation between printed layers, structural defects, warping, and print failures, all of which can contaminate resin, damage the resin tank, or cause resin spills.

Cups that reduce printability are highlighted in orange. To resolve highlighted cups, re-orient the model using the Orientation tool so that the opening of the cavity either faces the build platform or is rotated at an angle, and then regenerate supports.

> **TIP:** For models that contain internal hollows or are enclosed volumes (for example, a hollow sphere), use CAD software to either fill in the hollow or add drainage holes in the hollow to minimize suction effects during printing and to avoid trapping resin inside the part.
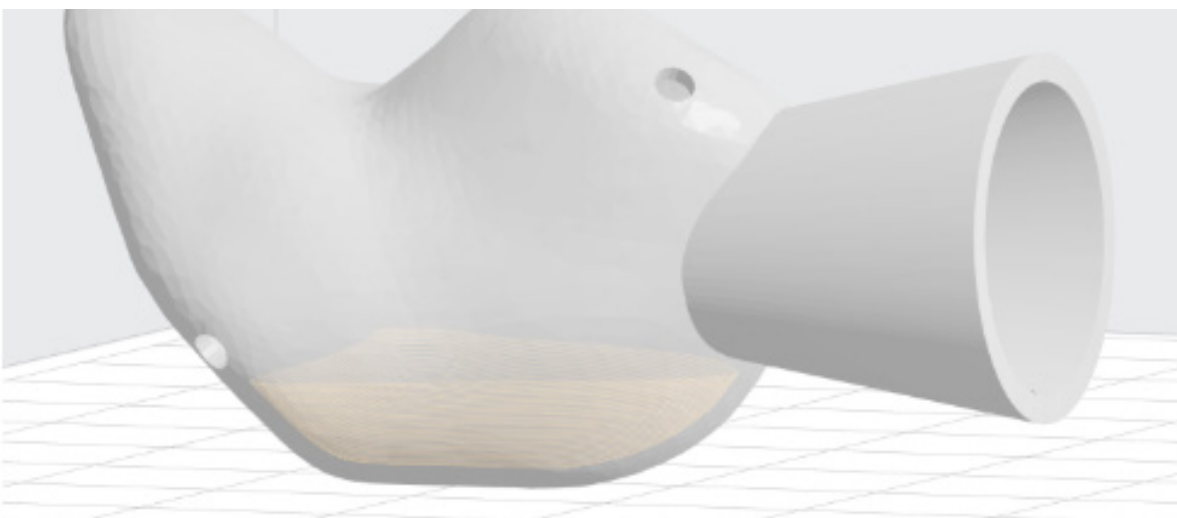
Undersupported Areas

"Undersupported areas" refers to areas that may require additional supports to print successfully. Undersupported areas are frequently overhangs (also known as "islands"), or features that would print without being attached to a raft or another layer.

Undersupported areas are highlighted with a red gradient. The intensity of the red indicates the overall impact on printability. A deeper red indicates a print failure is likely while a lighter red indicates possible structural defects or warping.

Resolve undersupported areas on models using one of the following methods in PreForm:
- Add additional support touchpoints to the highlighted area using the Supports tool.
- Rotate the highlighted areas away from the build platform using the Orientation tool to reduce overhangs.

Print the Scene

Upload a Print Job

With the printer connected to PreForm, click the orange printer button in the left toolbar to access the "Print" menu.

On the "Print" menu, select the appropriate printer by serial name, choose a name for the print job, and assign an account name for the job. Click "Log In" to set the account name from a Formlabs account. The job name and account name both appear on the jobs list on the touchscreen display once the job uploads to the printer.
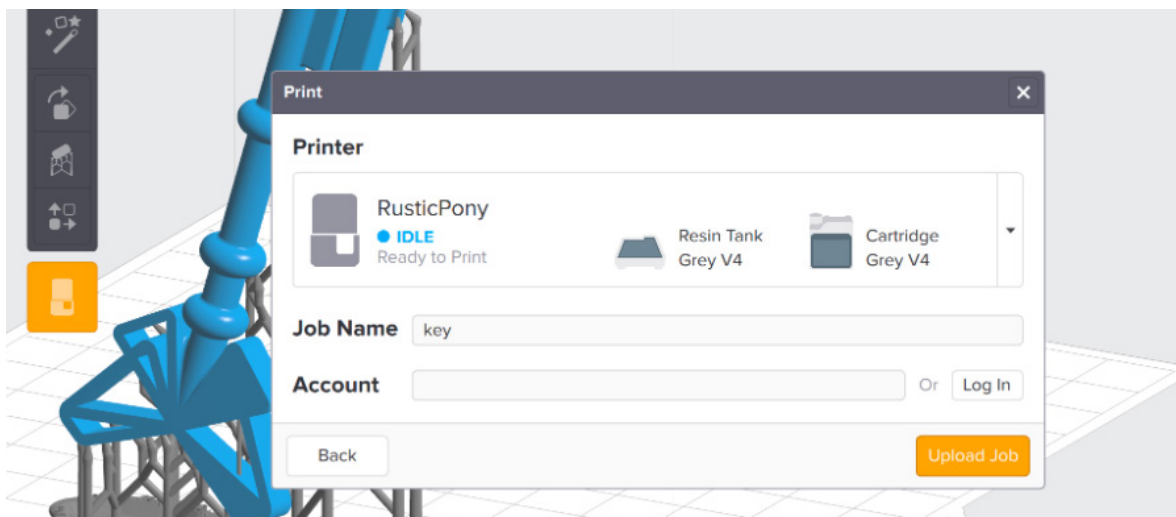
Log in to an existing Formlabs account to link print jobs to the account and to enable Dashboard notifications for only prints linked to the account. Log out to unlink the print job from an account. Pay attention to any pre-print reminders or warnings. Ensure the correct resin tank and cartridge are installed.

Click "Upload Job" to send the print job to the printer.

> **TIP:** For a Form 1+, click "Upload Job" once the resin tank is clean and the resin tank is filled with resin.
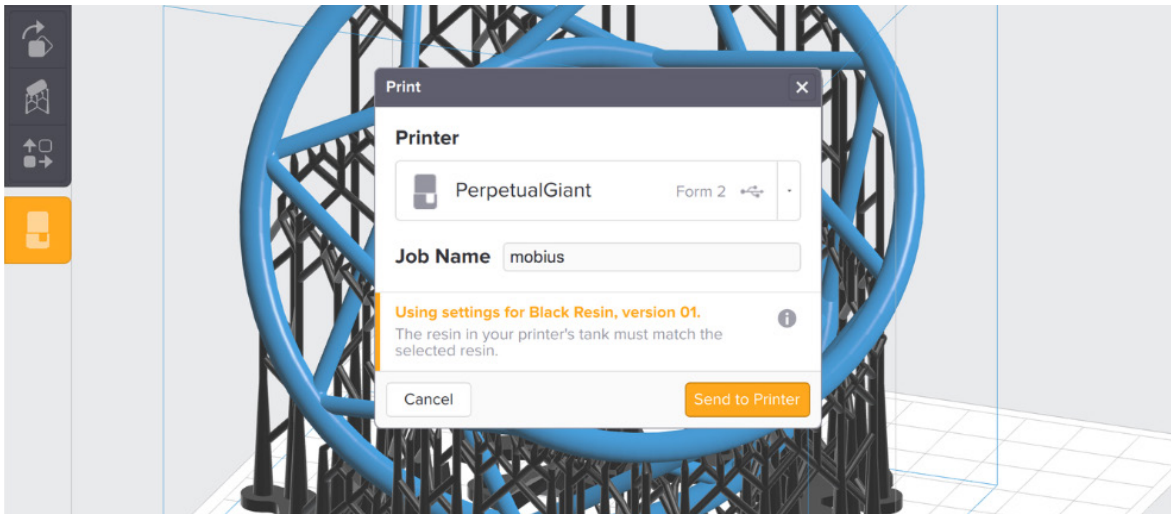
Complete the File Transfer

Once you click "Upload Job", PreForm displays a progress bar as each layer of your model uploads to the printer.

> **NOTICE:** Do not disconnect your printer from your computer or network connection until this process is complete. The file transfer can take several minutes. It is safe to disconnect the computer once PreForm confirms the upload of your print.

Confirm the Print

Confirm the print on the printer. Make sure that your printer is ready to print and then press the button on your printer to begin the print. Starting a Print

> **TIP:** For the fastest upload time, wait until PreForm completely uploads the file before confirming the print and pressing the button.
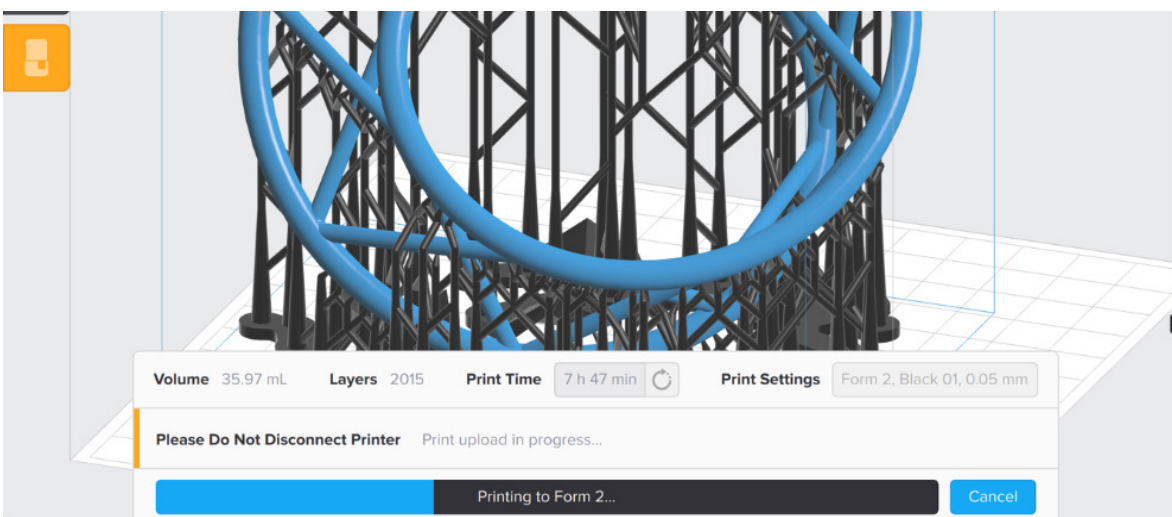
Orienting a Model

3D printing is as much an art as it is a science. For successful prints, learn the best ways to optimize each part's orientation.

XYZ Orientation
There are a few ways that you can change your model's orientation in the X, Y, and Z axes; see the suggestions in sections below for the best recommended orientations. The easiest way to rotate a print is to click-and-drag on the spherical manipulator that is overlaid on the model. Rotate any model more precisely in one of the three main axes by dragging one of the circular rings on the manipulator or by typing specific angles into the boxes in the Orientation toolbar. The Z axis rotation can be adjusted at any time - from the Orientation toolbar or using the Z axis rotation ring - without affecting the supports; supports must be re-generated if the X and/or Y angles are adjusted.

The X, Y, and Z rotation angles reset to 0 degrees after applying a rotation. This is because PreForm's orientation tools are relative, not absolute; any rotate change is applied relative to the model's current orientation, not its originating one. To see an example of this behavior, load a model and create a duplicate of it. Select the first copy of your model and apply a 90-degree X rotation, followed by a 90-degree Z rotation. Select the second copy and apply the same rotations in the opposite order: 90-degree Z rotation, then 90-degree X rotation. The result is different.

> **TIP:** If you hover the mouse over the X, Y, or Z rotation box in the orient tool, and move the mouse's scroll wheel up and down, the model will rotate smoothly around the corresponding axis.
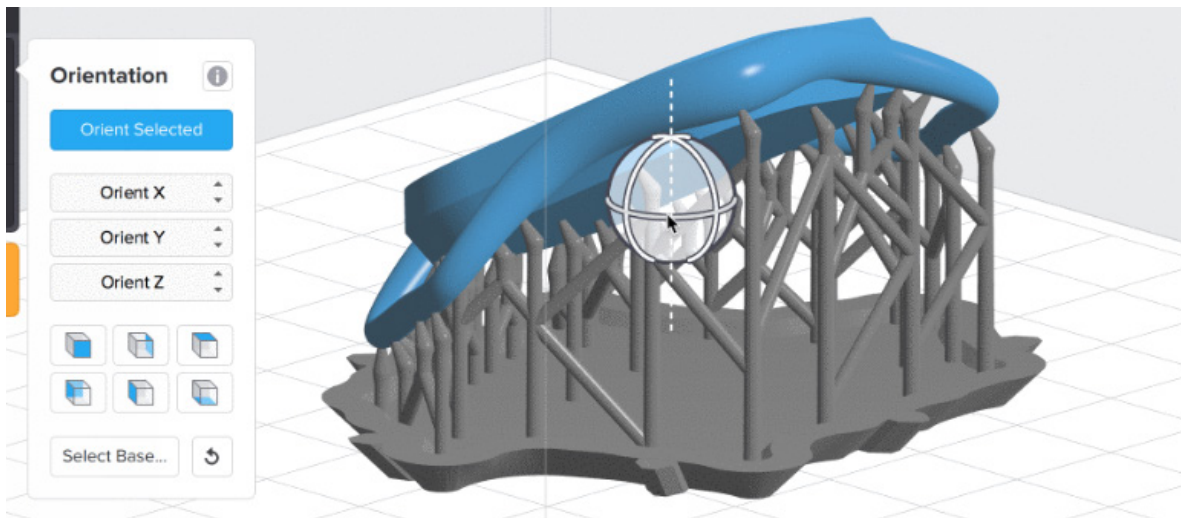

Vary Print Locations
Extend the life of your resin tank by placing your models in different locations on the build platform each time you print. This prolongs the life of the tank and improves print quality by distributing the amount of laser exposure throughout. Use the manipulator's translate feature

or the Layout toolbar to adjust a model's position.

Select Base

"Select Base…" is an orientation tool that allows you to select a face on your model to point down at the build platform. Click "Select Base…" and then click on a specific point on your part. PreForm will then rotate your part into position.

Once you click "Select Base…" your cursor will appear as an orange arrow when you hover over a model.
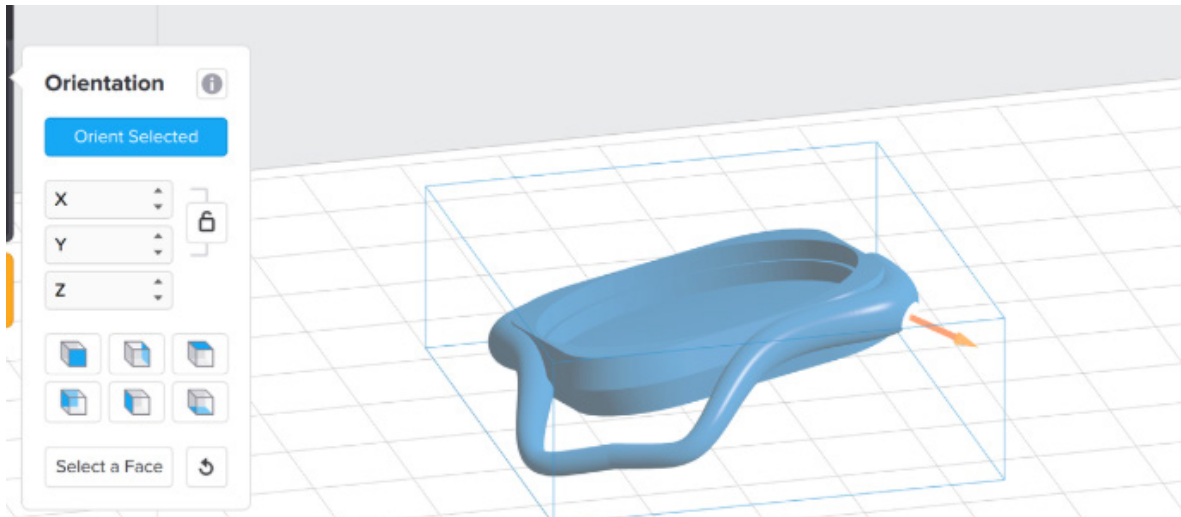
The part is rotated so that the clicked area faces the build platform.
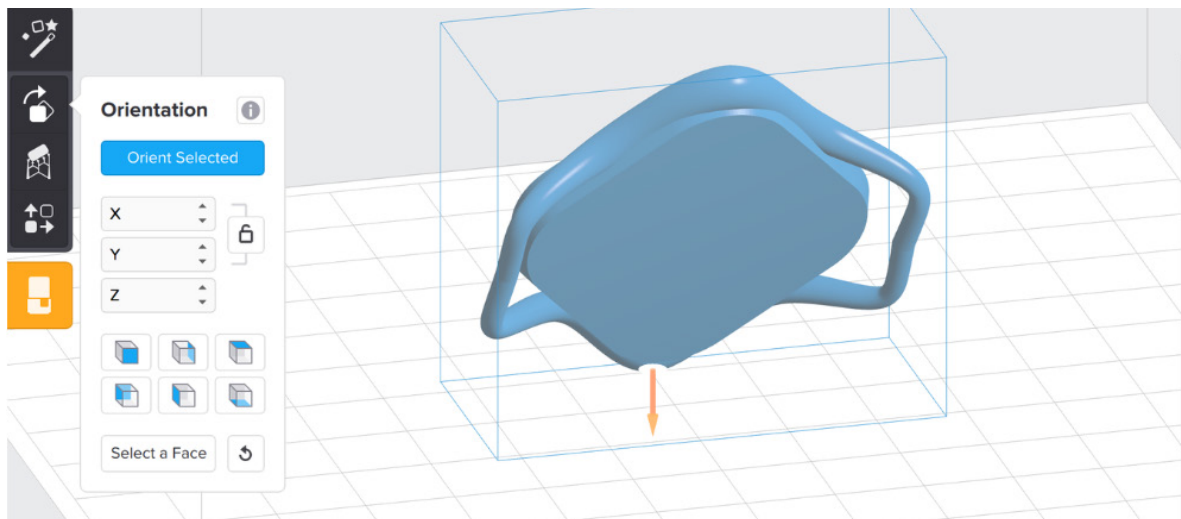
This is a quick way to orient parts when you have a definite "front" to your model. Selecting the underside as the base can help minimize the amount of post-print finishing you have to do, as the support marks will not appear on the cosmetic surface if it faces away from the build platform.

Tall Supports on Wiper Side

If printing a larger part at an angle, it is good practice to position the taller supports on the wiper side. In PreForm's default view, the wiper side is the right side of the build platform and

is labeled "WIPER SIDE".

Print Flat Surfaces at an Angle

Flat surfaces with support structures print much more successfully at an angle of at least 10–20°. You will notice that if you use the automatic orientation in PreForm by highlighting your part and clicking "Orient Selected" it will usually position your piece at an angle. After each layer, the printer performs a "peel" process, which literally peels the printed layer off the elastic surface of the resin tank. The forces involved can distort the extremely thin layer of a flat surface mounted on support structures. If a planar surface is oriented at an angle to the build platform, there is only a little overhang for each new layer. Furthermore, many thin-walled parts present significantly less area in any slice when printed at an angle.
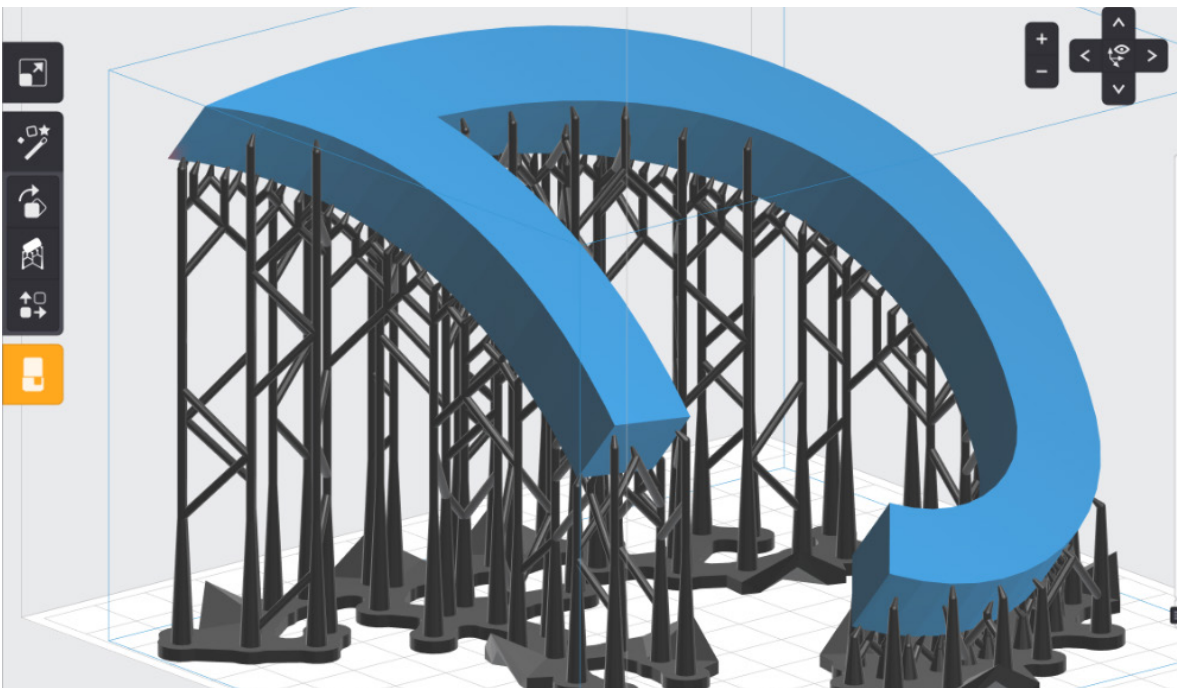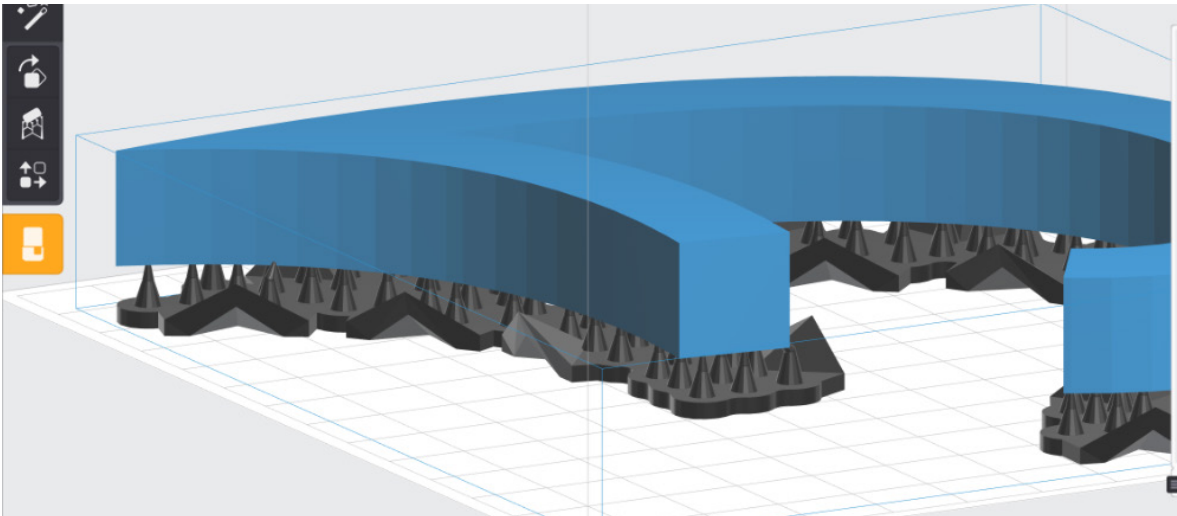
Orient to Support Overhangs

Parts print one layer at a time. Features that would print as part of a layer without being attached to the base or another section of the model are known as overhangs. Without support structures, such a feature would lose itself in the resin! PreForm Software automatically generates supports to fix this issue. By default, PreForm will also generate internal supports that build from one part of the model to another. You can disable this feature by unchecking "Internal supports" in the Supports menu.
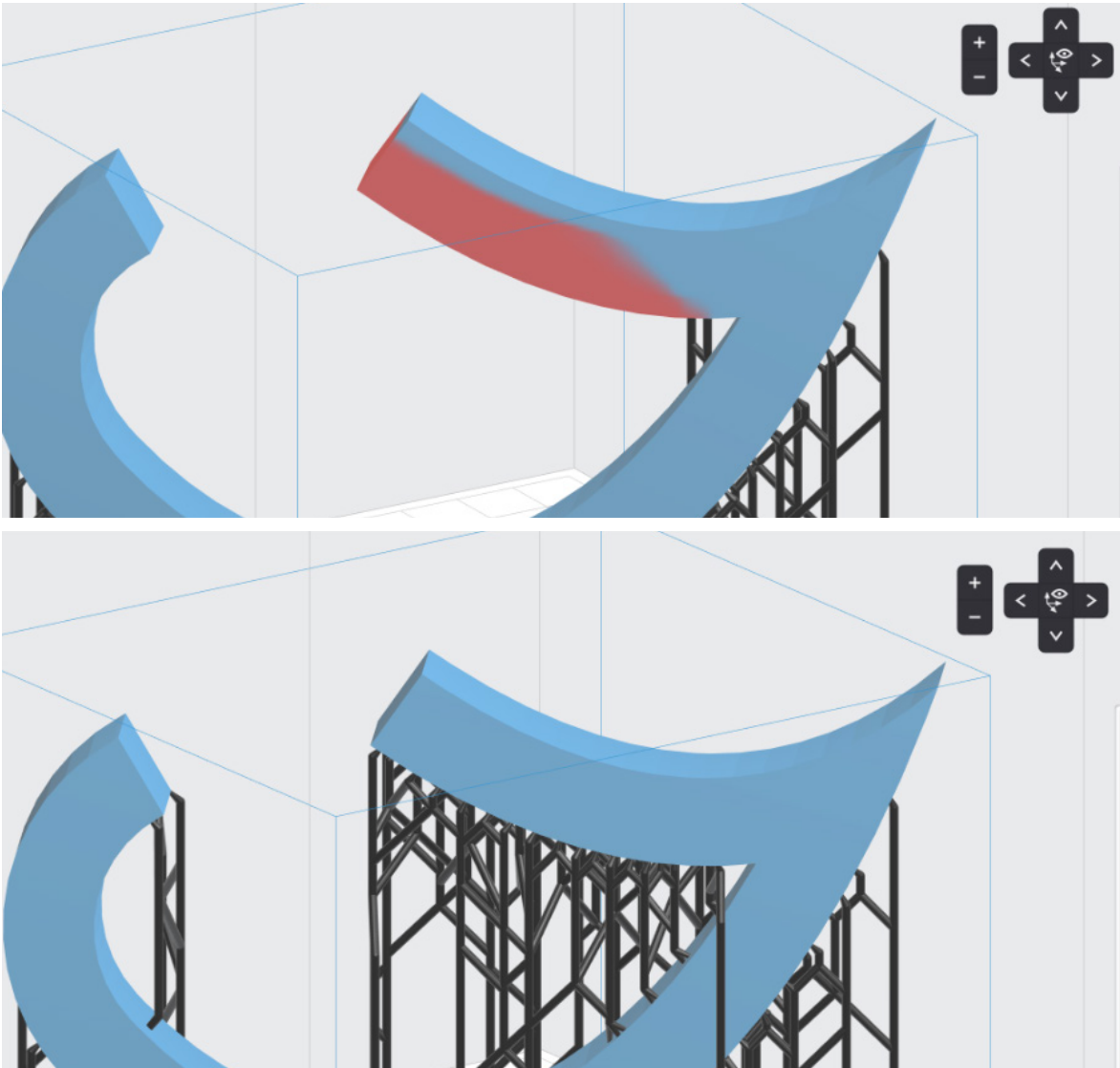
Orient to Support Minima

A minimum point—plural: minima—refers to the lowest local point in any area of a model's geometry relative to the build platform. Unsupported minima may break off the part during printing and lead to structural defects, warping, or print failures, all of which can contaminate resin, damage the resin tank, or cause resin spills.

Minima are highlighted in red and will fail without additional support. To resolve unsupported minima on models using the Orientation tool, rotate the highlighted areas away from the build platform.
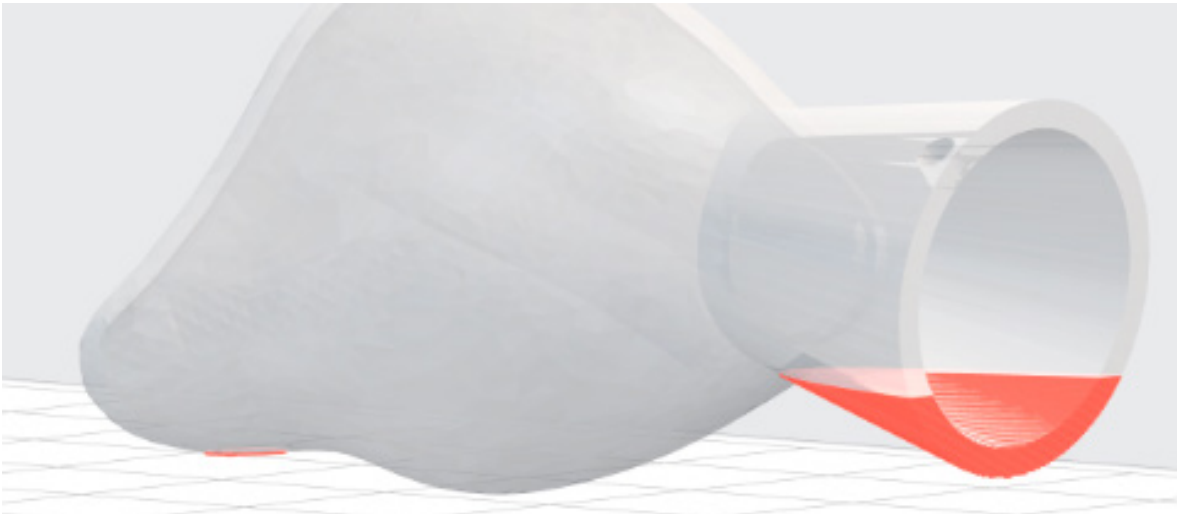
Orient to Prevent Suction Cups

Cups" refer to hollow volumes or cavities in a model. Hollow volumes and concave geometries require more careful attention to the print orientation. When the opening of a cup faces the resin tank, the cup traps air and resin as the build platform lowers the opening of the cup against the bottom of the resin tank. The resulting suction can cause separation between printed layers, structural defects, warping, and print failures, all of which can contaminate resin, damage the resin tank, or cause resin spills.

PreForm automatically highlights in orange any cups that may reduce the printability of a model. Use the Orientation tool to rotate cups so that the opening of the cavity either faces the build platform or is placed at an angle. Regenerate supports after orienting the model. Orienting cups at an angle or to face the build platform prevents a vacuum from forming between the part and the bottom of the resin tank during printing.

> **TIP:** For models that contain internal hollows or are enclosed volumes (for example, a hollow sphere), use CAD software to either fill in the hollow or add drainage holes to minimize suction effects during printing and to avoid trapping resin inside the part.
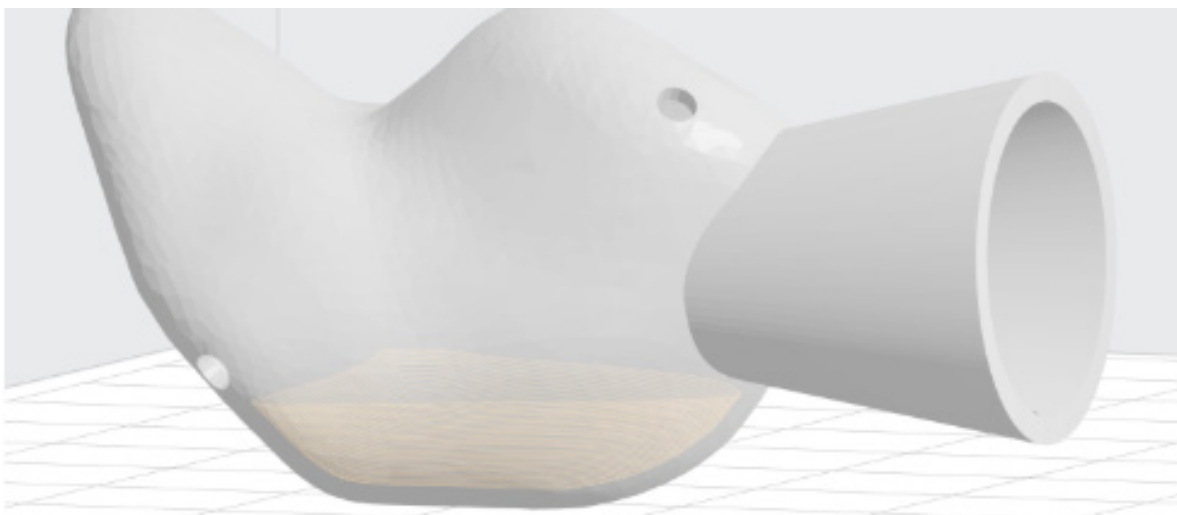
Save your Work

After all that work orienting your model, you may want to save the layout, or "job". PreForm Software will create a FORM file with all of your settings, making it convenient to reload or share the file exactly as you printed it. Do this by selecting "Save" or "Save as..." from the File menu.

Supporting a Model

Support structures improve print reliability on Formlabs 3D printers. Learn how to configure various print geometries for the best results.

PreForm offers basic and advanced support settings. This segment of our video series explains best practices for using PreForm to create support structures and the theory of supporting upside-down geometries.
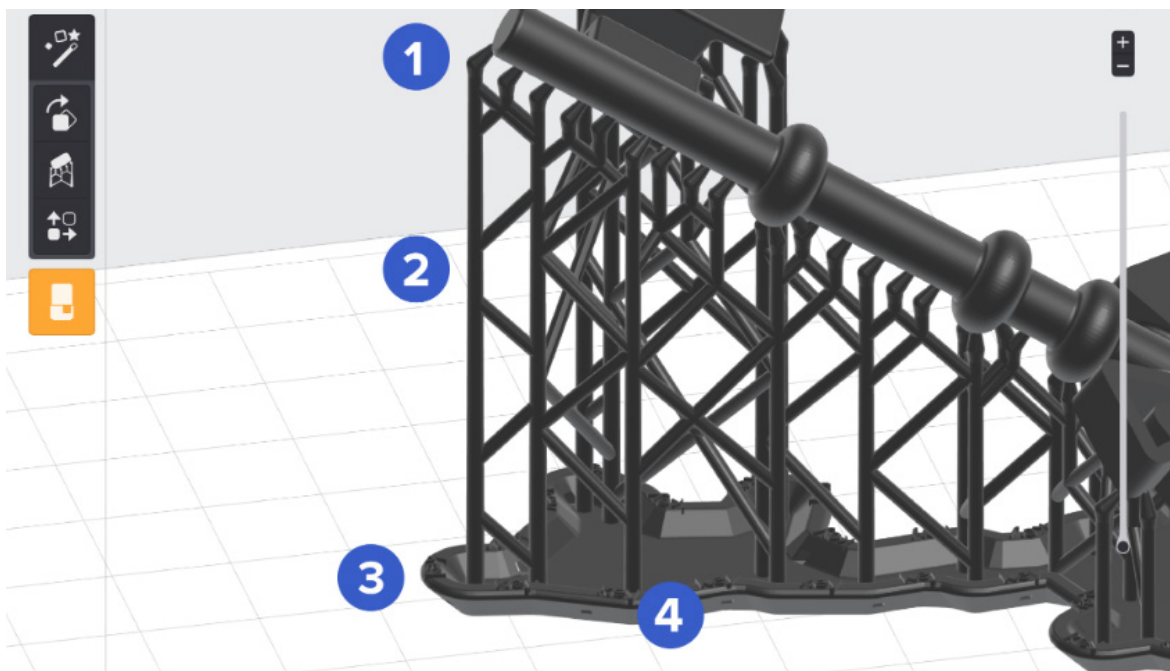
Support Structure Overview

Support structures function as scaffolding for the part during printing. The structures support the construction of parts and should be removed after the print is complete.

Supports consist of two core components: a flat raft and vertical shafts.

The raft ensures that the part adheres to the build platform. The support raft includes angled edges and rounded areas that aid part removal from the build platform. Squeeze flush cutters under rounded areas to lift the part from the platform.

The shafts emerge from the raft and connect to the part through small touchpoints. Remove supports by hand or with flush cutters when the print is finished.

Support structures include: (1) touchpoints, (2) scaffolding, and a (3) raft which has (4) raft label text extruded on the angled edges.

Overhangs and Shallow Angles

Using 3D printers to make parts without support structures severely limits the part geometries that will print successfully. Because 3D printing is an additive process, in which each layer is consecutively added to the previous layer, there must exist some material on which each layer can build. Accordingly, each layer can only be slightly larger than the previous layer without sagging. From a macroscopic perspective, parts must have steep angles in order to print properly without supports.
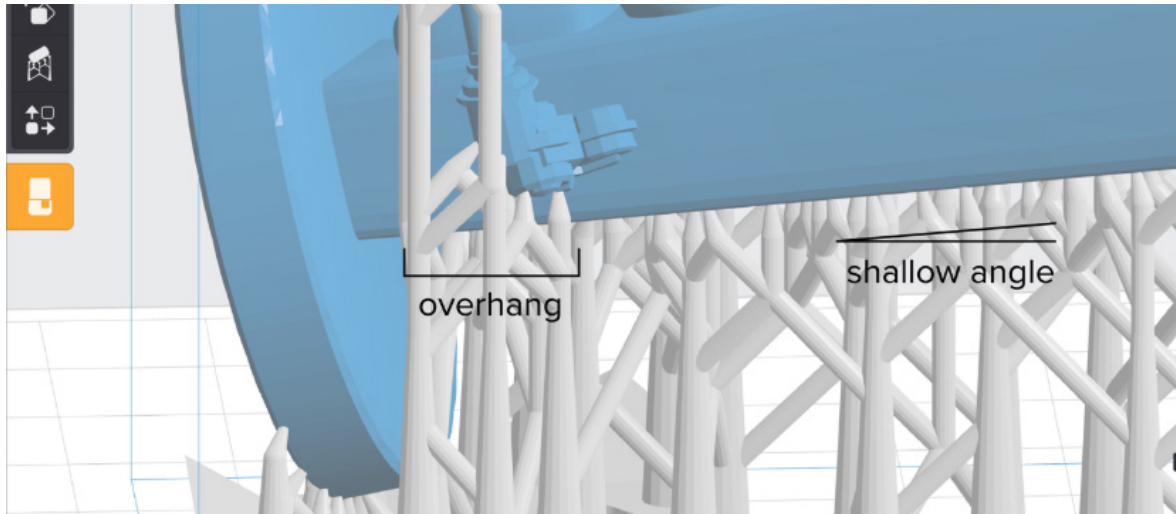
The support structures allow for parts to be printed at shallow angles. By touching the part via the touch-points along the shallow slope, the support structure gives it added strength, preventing unwanted twisting or other deformations. Additionally, the support structures can accommodate overhangs that would otherwise be unable to print.

Printing Without A Raft

While PreForm allows models to print without a raft, all parts that print without a raft will show Z-axis compression in the early layers. PreForm displays a warning if it detects that a model will print directly on the build platform and that the model would print more reliably with additional supports or a PreForm-generated raft.

Use the "On Build Platform" feature to resolve the warning when printing on the build platform and to enable automatic support generation for models that print directly on the build platform.

On Build Platform feature

Use the "On Build Platform" feature to print a model directly on the build platform with minimal supports and without a full raft. The "On Build Platform" option is located on the Supports menu in the left toolbar.

To print directly on the build platform with the "On Build Platform" option:
1. Use "Select Base" on the Orientation menu to orient the model so that a flat surface sits parallel to the build platform. Select a flat surface of the model to function as a base.
2. Select "Print on Build Platform" on the Supports menu.
3. Click "Generate Selected" to generate additional supports for the model geometry.
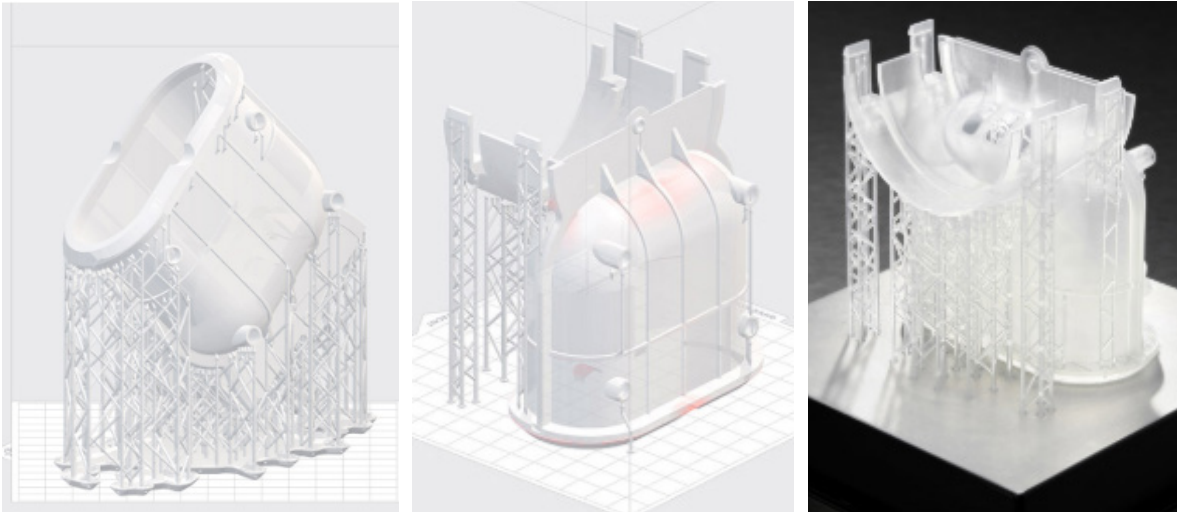
Considerations for printing directly on the build platform

- *Choose a flat surface of the model to function as a base:* When printing directly on the build platform, choose a flat surface of the model to function as a base during printing. Ensure that the chosen flat region of the model has a surface area sufficient to resist the suction forces experienced by the part during the printer's peel process. Experiment with different flat surfaces of each model to determine the best area to function as a base.
- *Vertical compression and Z-axis dimensional accuracy:* Formlabs printers compress the first several layers of every print to ensure adhesion to the build platform. This compression shrinks the vertical height of a print's early layers. When printing with a PreForm-generated raft and supports, the compression does not affect the part's dimensional accuracy because the compression only affects the generated raft and supports. When printing a model directly on the build platform, expect each model to be approximately 0.5 mm shorter than designed along the Z-axis in its first layers. The exact compressed height varies between printers.
- *Early layer unioning:* PreForm automatically revises the first 0.75 mm of all on-build-platform prints to increase adhesion. "Early layer unioning" combines and extrudes the shapes - the perimeter and fill area - of all layers contained in the first 0.75 mm to ensure that printed parts have a completely flat surface in contact with the build platform. Slide the Slicer through the early layers to view the uniform, combined geometry.
- *Model compatibility:* The "On Build Platform" feature works best for model geometries with at least one flat surface that can face directly parallel to the build platform. Shapes without any flat areas, such as spheres or organic shapes, are not recommended for the "On Build Platform" feature because these geometries will print with slightly flattened base layers due to early layer unioning and compression.
- *Part removal:* Take special care when removing parts printed with "On Build Platform." The flush cutters may chip the part. Separate the part from the build platform at the face that is least critical for the part's application or aesthetics.

Undersupported Models

When support structures are generated, Printability coloring is also added to the model, by shading red portions of the model that are unsupported. If a part of the model is a deep red, that portion of the model is unsupported and will most likely not print well. If a part of the

model is a lighter shade of red, that portion of the model is less-well supported and may have structural defects.

Typically a very unsupported region (deep red) is caused by an internal overhang or a feature too thin to be supported. In these cases, it is best to activate internal supports or try rotating the part to a different orientation. Auto-orient maximizes part Printability and can be very helpful for parts with unusual overhangs.

The example part shown here is unsupported and the top of the top arm will fail.
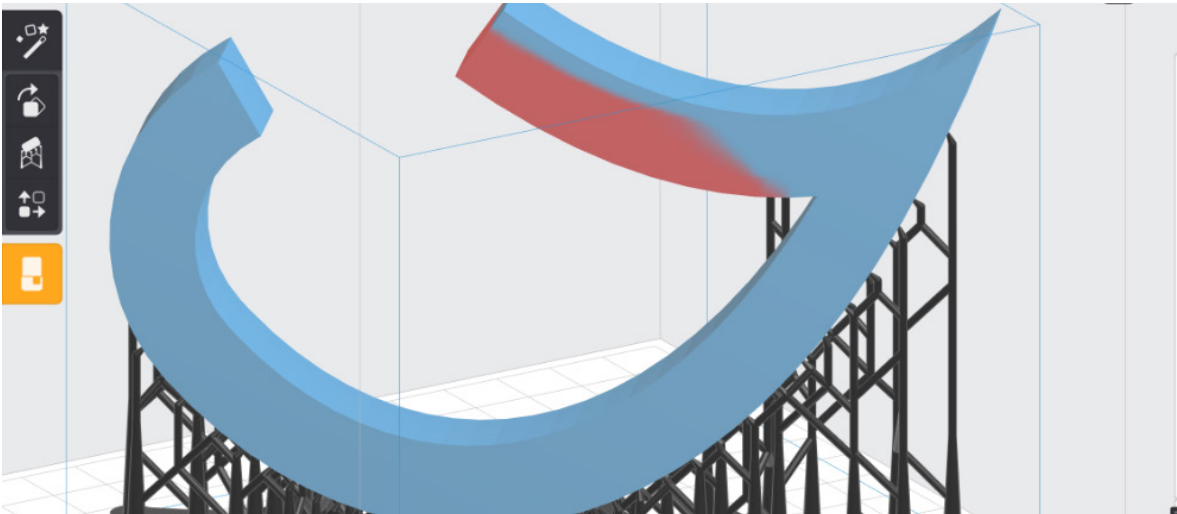
A lighter shade of red is less of a cause for concern. Very often, parts will print well despite the coloring. This light red serves only as a warning that there may be issues with the part orientation. The sphere shown with a faint red ring will likely print just fine.
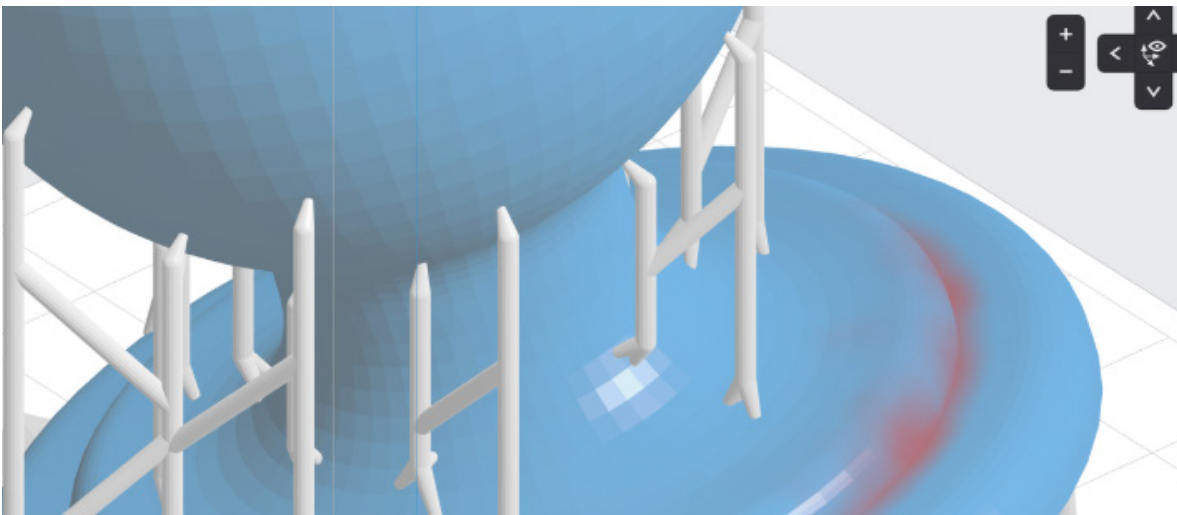
A fully-supported part will show no red. Make sure to examine your models closely, to identify any odd regions of poor Printability. Try re-orienting the part and generating new supports to find the ideal configuration.
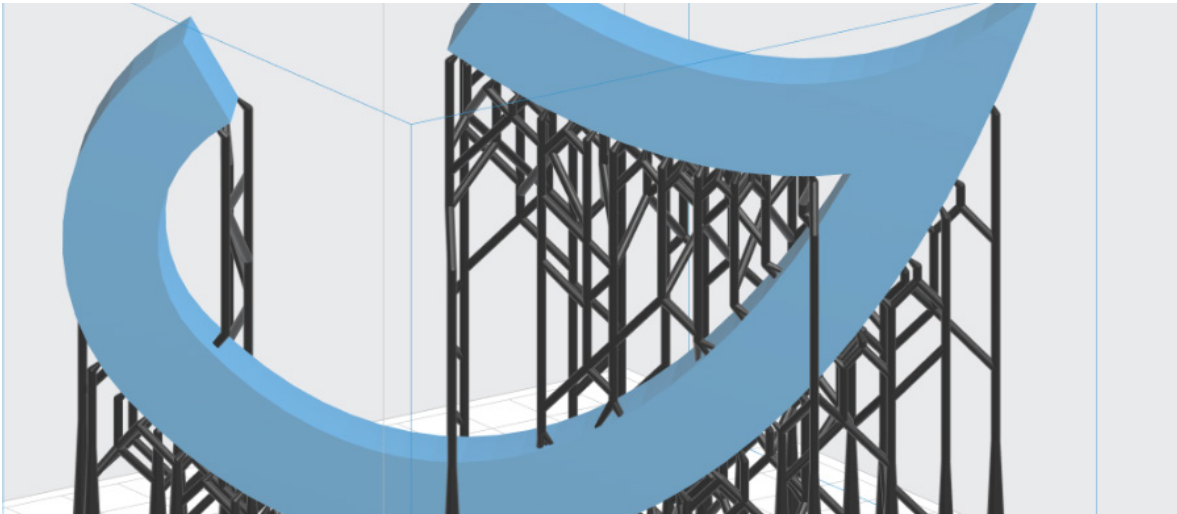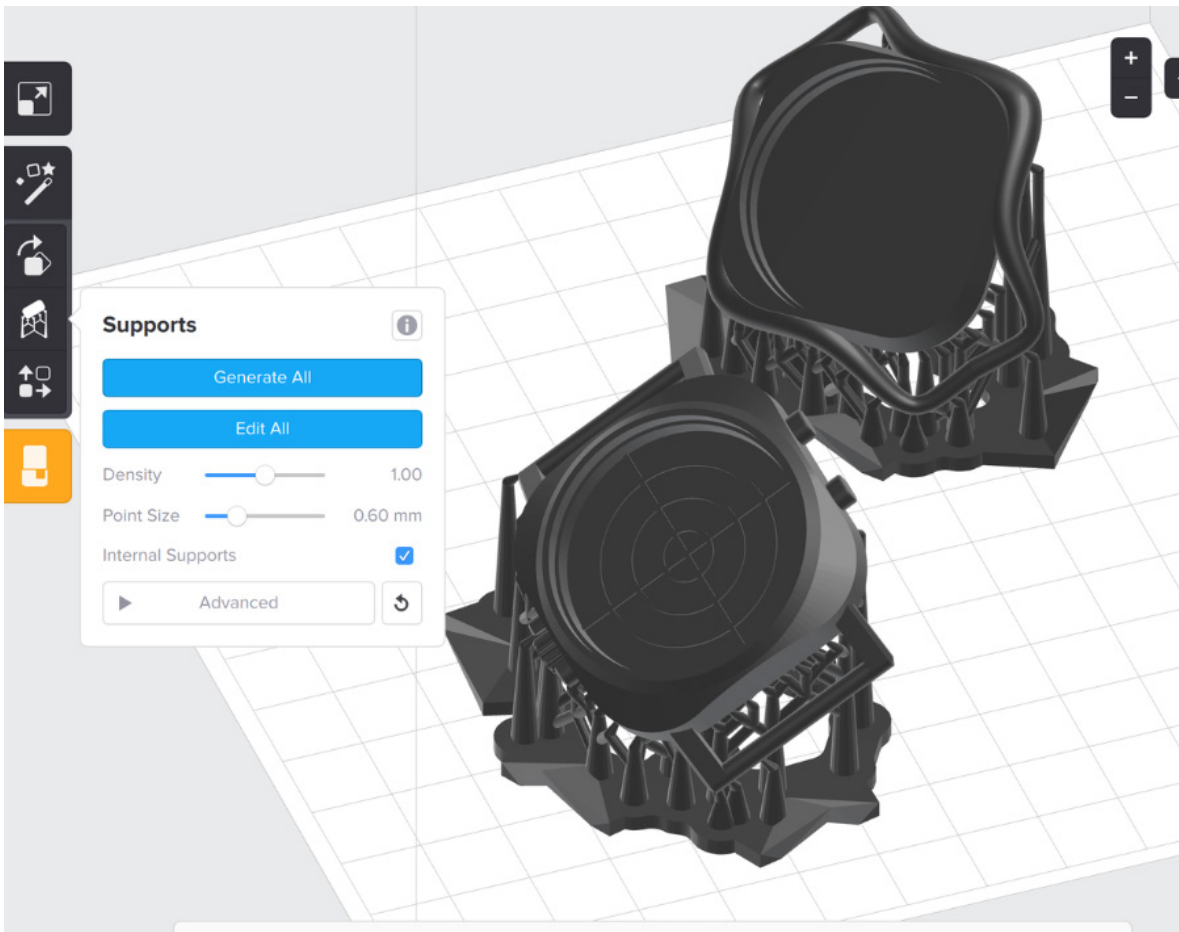
Generate Selected vs. Generate All

When you have positioned multiple .STL files in PreForm, you can generate supports for all those pieces at once, or individually for each part.
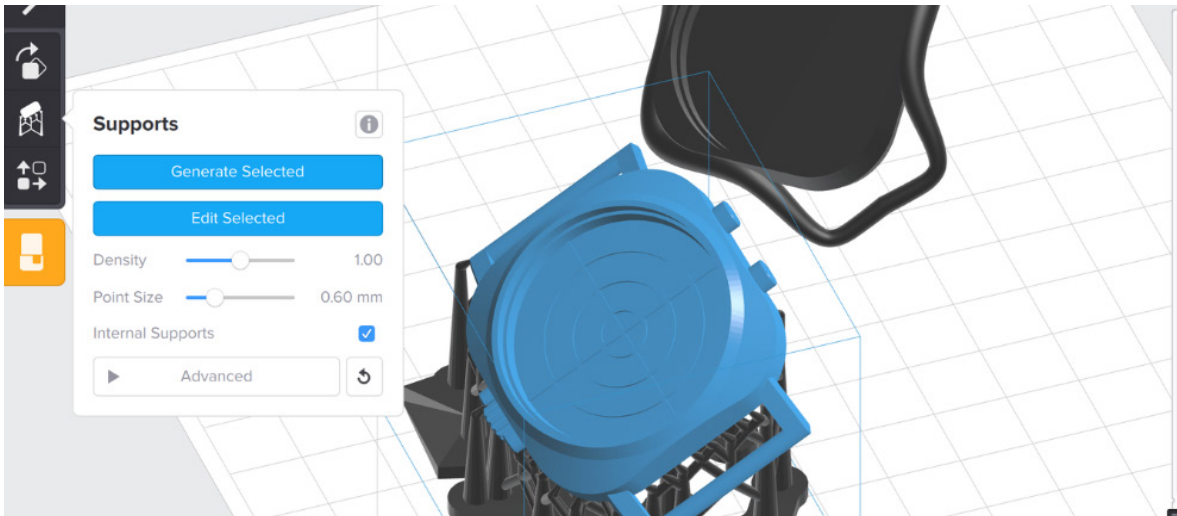
Generate All

Generate the same type of supports at once for all parts on the build platform by pressing "Generate All". Unless you have changed the Advanced Support Settings, this will use the default support settings.

Generate Selected

In the case where you might wish to vary the advanced support settings for each of these parts, and thus generate slightly different supports, you can generate a set of supports one part at a time using "Generate Selected".


Supports and Print Settings Compatibility

PreForm generates support structures based on the selected print settings, which include information such as layer thickness, material type, and material version. View and modify print settings in the toolbar at the bottom of the screen.

When you save a print job as a FORM file, PreForm saves the print settings and generated support structures. Reuse saved support structures and custom support locations when you load a FORM file into PreForm.

Opening and Importing FORM files
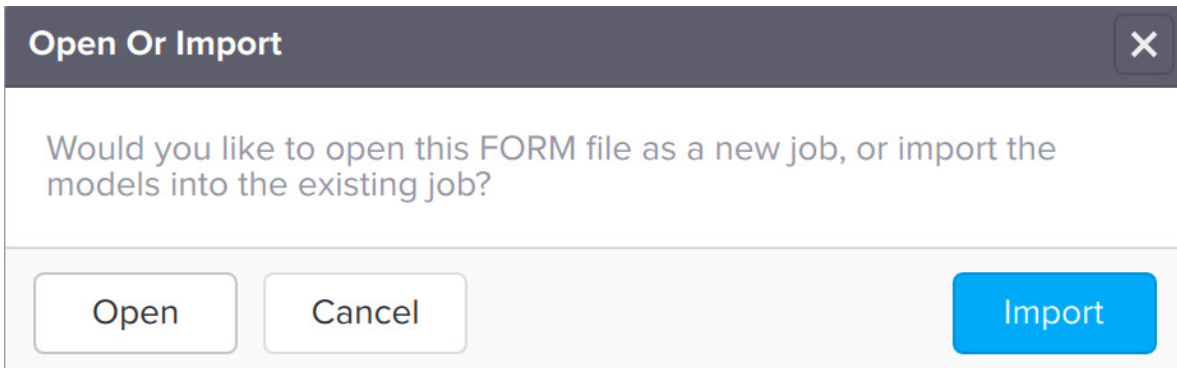
The "Open Or Import" box appears when you open a FORM file with saved print settings that are different from the existing print settings selected in PreForm.

Choose either to open the FORM file as a new job with the saved print settings, or to import the file into the existing job and apply the existing job's print settings.

- Click "Open" to clear the existing job and to open the FORM file as a new job. PreForm automatically loads the print settings and supports saved in the FORM file.
- Click "Import" to import the FORM file into the existing job. The "Import Support

Locations" box appears. You can choose either to import the saved support touchpoint locations or to delete and regenerate supports.

Importing Support Locations

The "Import Support Locations" box appears if the imported FORM file's print settings are incompatible with the print settings of the existing job.
- Click "Import" to maintain the saved support locations on the FORM file while regenerating supports according to the print settings for the existing job. PreForm only imports touchpoint locations. Use the Supports tool to manually edit the touchpoint size and to regenerate supports to match the new material settings.
- Click "Delete Supports" to clear all saved support settings from the imported FORM file. After the file loads in PreForm, regenerate supports manually using the Supports tool.

Adjusting Layer Thickness

Printing is fastest and produces the best prints with the right layer height. Choose the layer height with the material settings in PreForm.

You can use PreForm to quickly prepare and print your models without having to tweak an endless number of settings. With "0.05 mm" as a great default layer height, you can expect flawless prints with almost every model using Formlabs resin. This article looks at when you should use the other material settings, such as "0.025 mm" or "0.1 mm".

Changing the Layer Thickness
You can change the layer thickness by clicking on the button next to "Print Settings" on the toolbar at the bottom of PreForm.

This will bring up the "Print Setup" menu. Here you can set the layer thickness to a particular resolution (e.g., 0.1 mm) and the material to a particular type of Formlabs' resin (e.g., Grey).

The layer thickness refers to the height of each layer of resin cured to create your part. Since these measurements are fractions of a millimeter, they are often given in terms of microns (1 micron = 1 μm = 0.001 mm). To put that in perspective, 100 μm is approximately the thickness of a sheet of paper or a human hair; 7 μm is approximately the diameter of a red blood cell. The Form 1+ comes with settings for 0.025 mm, 0.05 mm, and 0.1 mm (25 μm, 50 μm, and 100 μm respectively).
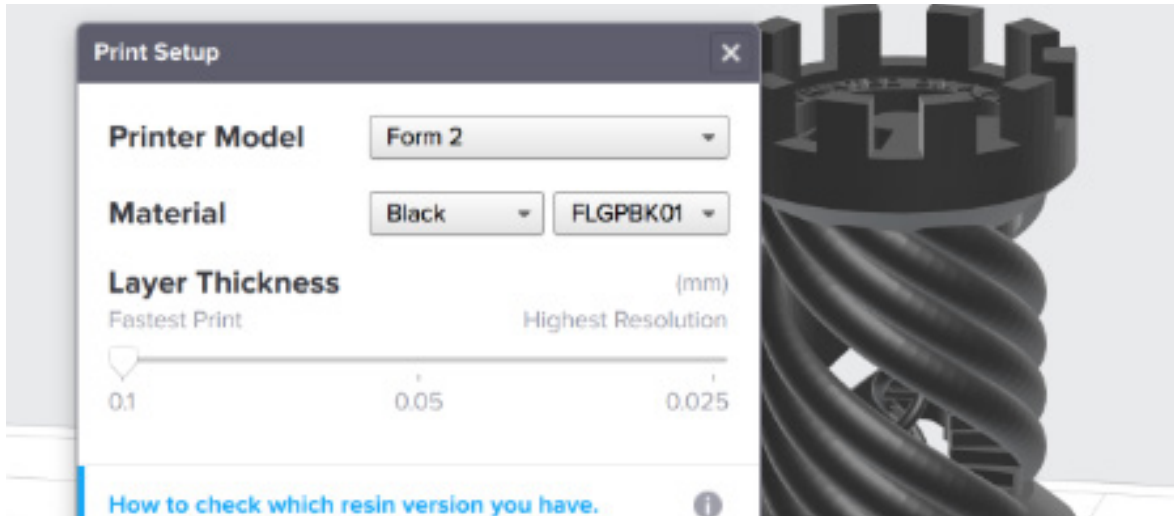
Speed
Thin layers significantly increase print time, when compared to the 0.1 mm layer thickness. Change the layer thickness in the Print Setup menu, then select the print time estimator in the bottom toolbar to approximate and compare print times with different settings.

When printing at 0.05 or 0.025 mm, the Form 1+ uses the 0.1 mm layer height print for the base and early support layers, then the printer switches to the selected layer thickness when printing parts of your model. When the base's raft or supports are large relative to your model, these early "coarse layers" can make a significant difference in print time.

Layer Thickness and Print Quality
Traditionally, extrusion-based 3D printers have emphasized the layer height (such as 0.1 mm or 0.27 mm) as the main indicator of accuracy or quality. One reason for this may be that 3D prints made with this technology typically have visible ridges between different layers. Smaller layer heights can help to reduce but not eliminate these defects. Layer height alone does not determine part quality. See the two rooks to compare the same model printed at 0.1 mm on the Form 1+ and an FDM machine.

*L-R: FDM Printer at 0.1 mm Layers, Form 1+ at 0.1 mm Layers*
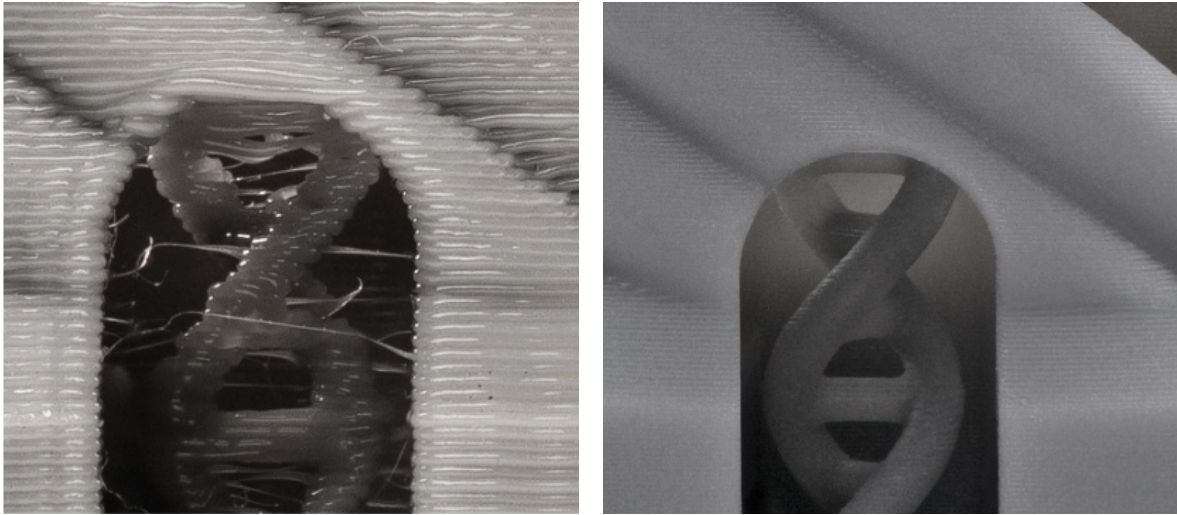
Images shown at 27x magnification

The 0.1 mm settings produce beautiful, smooth prints and are a great place to start. Because the Form 1+ uses a different and more accurate technology to create 3D prints, there's much less need to pursue smaller and smaller layer heights. Without a microscope it is difficult to identify Form 1+ parts printed using 0.1 mm layers from parts using 0.05 mm, or even 0.025 mm layers.

> **TIP:** Keep print time in mind! 0.1 mm prints finish in roughly a quarter the time of prints with 0.025 mm layers.

L—R: 0.1 mm, 0.05 mm, and 0.025 mm Layers
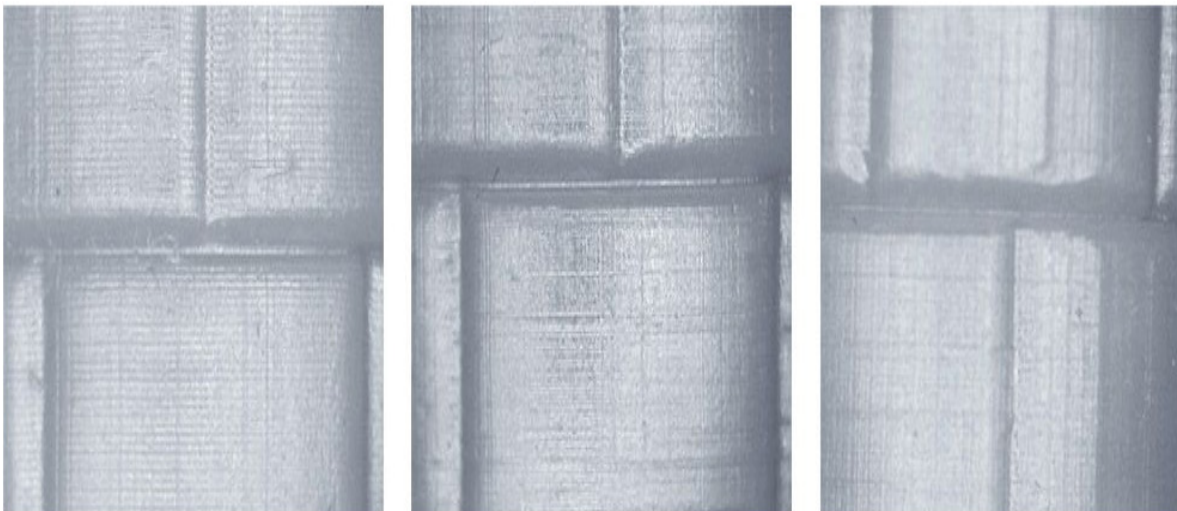
L−R: Print time approximately 2 hours, 4 hours, and 8 hours.

When to Use 0.025 mm
The Form 1+ can be used to capture incredibly small details in your model not possible on other desktop 3D printers. If you're not sure which setting to use, then the 0.05 mm setting is a great balance between capturing fine detail and having a print finish in a reasonable amount of time. The 0.025 mm setting should be reserved for models of exceptionally high resolution.

When to Use 0.2 mm
The Form 1+ can also produce reduced-resolution prints at a remarkable speed. This resolution is only available when printing with Formlabs Clear resin and is selected through the PreForm interface. Our 0.2 mm setting is perfect for moments when speed takes priority over surface quality. Printing at 0.2 mm resolution is typically twice as fast as printing at 0.1 mm and significantly faster than printing with an FDM printer at the same resolution.

L−R: 0.1 mm, 0.2 mm layers.
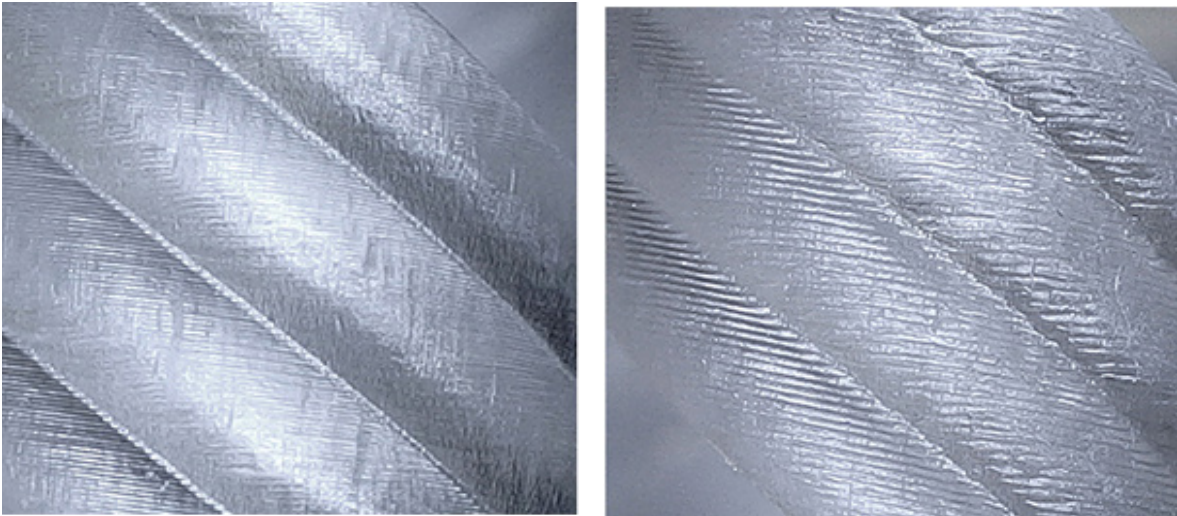
L—R: Print time approximately 2 hours, 48 minutes.

Editing Advanced Support Settings

Understanding how to configure each print makes printing easier. Advanced support structure options in PreForm offer greater control.

PreForm Software is designed to take the hard work out of generating support structures so that you can focus on creating great models and prints. 3D printing is as much an art as it is a science, so we've designed more advanced support structure options to give you greater control. Keep in mind that adjusting these settings can affect your print for better or worse.

Supports Menu
Click the Supports icon (third down in the PreForm toolbar) to access additional features.

The circular arrow button next to the advanced tab will restore the default setting values.
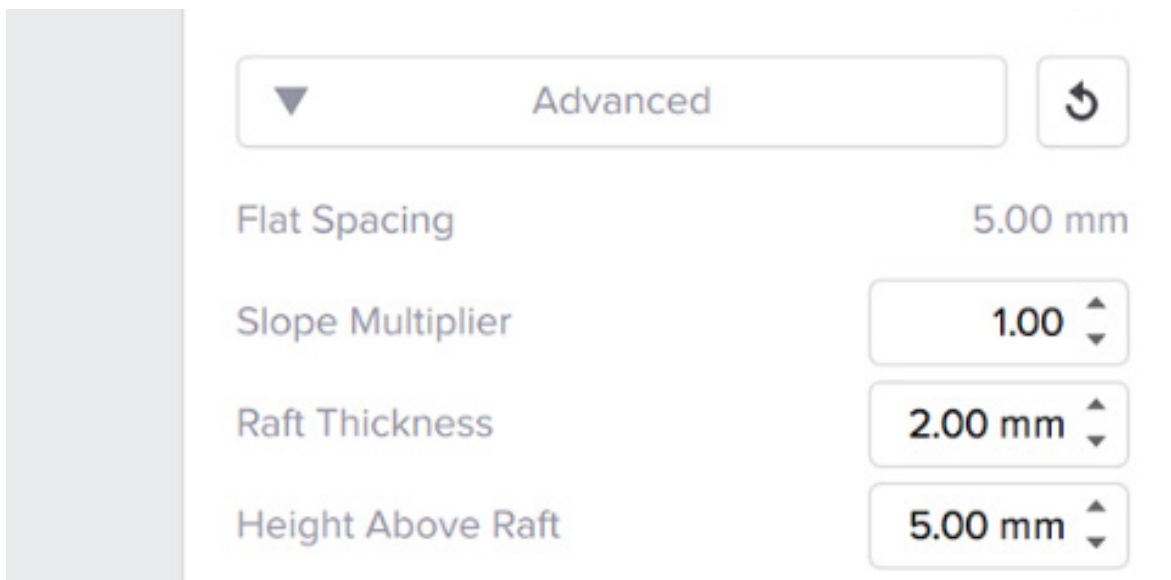
Density

This slider controls how closely-packed the support pillars are, and also affects how many contact points there are between the support structures and your model. Changing the density will also adjust the flat spacing value, which measures the distance between touchpoints on a flat, horizontal surface.

> **TIP:** Printing in flexible resin may require higher support density if your model has a large cross section. See our guide on Using Flexible Resin to ensure that your flexible models print successfully.

Touchpoint Size

Touchpoints are the locations where each support connects to the part. By default, this point has a smaller radius than the rest of the support structure, which allows for easier removal and fewer marks after printing. There are trade-offs to increasing or decreasing the touch point size:

- **Smaller touchpoints** make it easier to remove the support structures from the printed part, and will leave fewer marks on the build. However, there is a higher chance of the supports failing to hold up the part while printing, resulting in print failure.
- **Larger touchpoints** make for more security of the supports during printing, but the supports will be harder to remove from the finished part.

> **NOTICE:** Setting the touchpoint size below 0.40 mm may offer smoother print surfaces, but parts may separate from supports with a point size below 0.40 mm. For small touchpoints, increasing the density value may slightly improve the print quality. Gain printing experience and practice identifying critical support positions before lowering the touchpoint size.

Internal Supports

Checking this box will generate supports inside your model, to shore up overhangs and other difficult geometries.
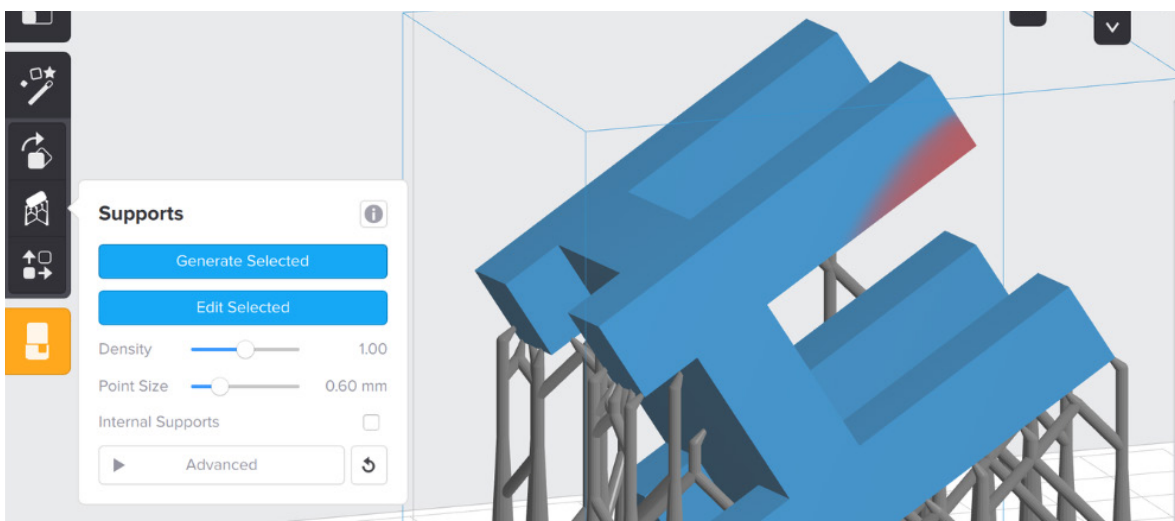
Without internal supports, the top legs of the sample table model are unsupported (indicated by the deep red region on the PreForm render) and may fail in a build. If this is the preferred orientation to print the part, it's necessary to add internal supports.
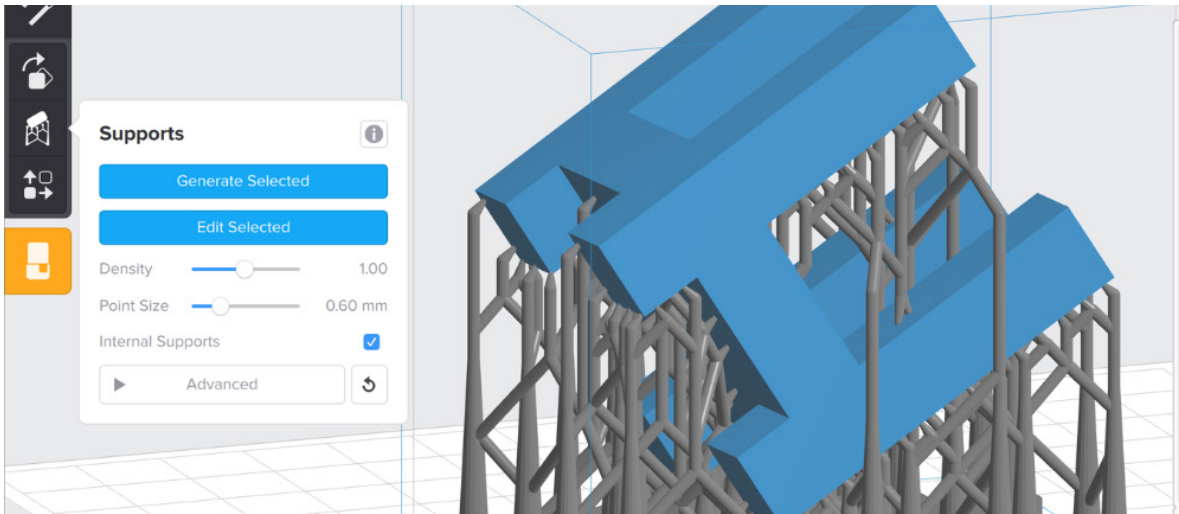
With internal supports added, the full structure is sufficiently supported and should print without issues.

> **TIP:** As supports are generated or edited, PreForm can help identify regions that may require additional support. While the red shading is a tool to guide support "neediness," you may find other support settings to be equally effective.

Raft Label

When generating supports, PreForm also generates the model name on the raft of the model. The model name repeats around the raft depending on the size of the raft and the length of the name.

To disable the default Raft Label feature in PreForm, click on the Supports tool and deselect "Raft Label".

> **TIP:** Use the Raft Label feature to keep track of different versions of printed parts.

To edit the Raft Label, right-click the selected model. A pop-up menu appears. Select "Model Properties", and click the "Name" box to modify the Raft Label. Press Enter/Return or click outside the menu to save the new model name and to update the Raft Label.

The Raft Label feature in PreForm supports most* language characters.
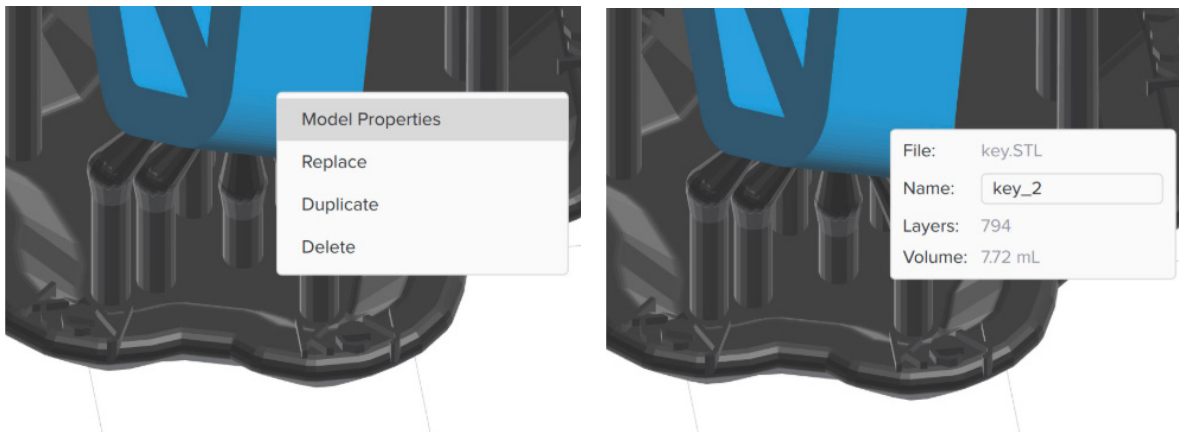
Editing Supports

PreForm allows support structures to be manually edited. Manual supports are especially useful for small features that require support to print successfully. Minima are highlighted in red and will fail without additional support. In the example of the snail, the shell required manually-added supports in order to be fully supported.

Click "Edit" on the Supports menu to open the Supports editor. Use the Supports editor to manually revise the support touchpoints auto-generated by PreForm. Use the pointer, and click on the model's surface to add and remove support touchpoints.
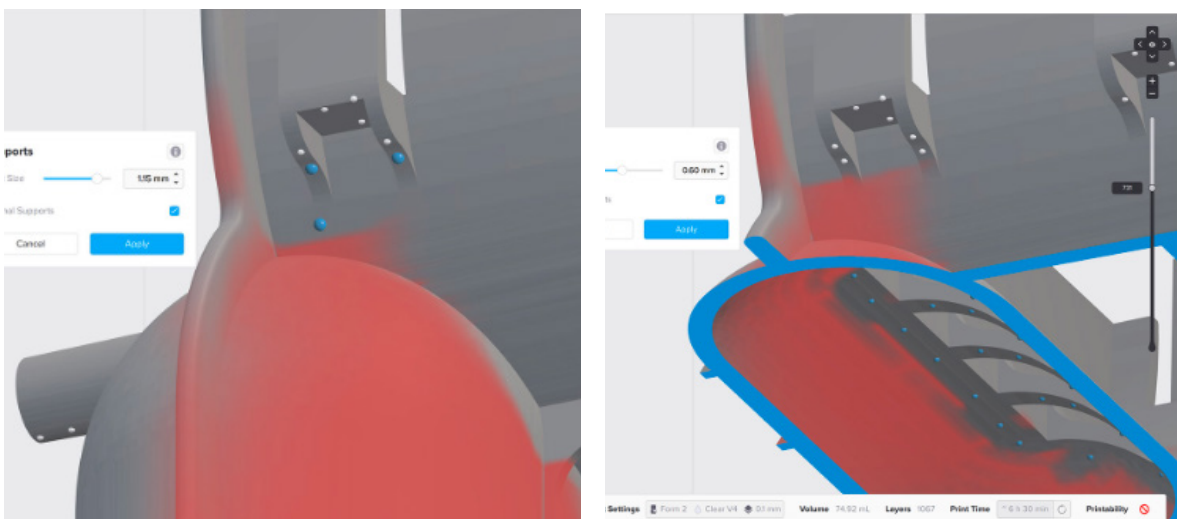
Click "Apply" to save edits.

To modify internal support touchpoints, select "Internal Supports".
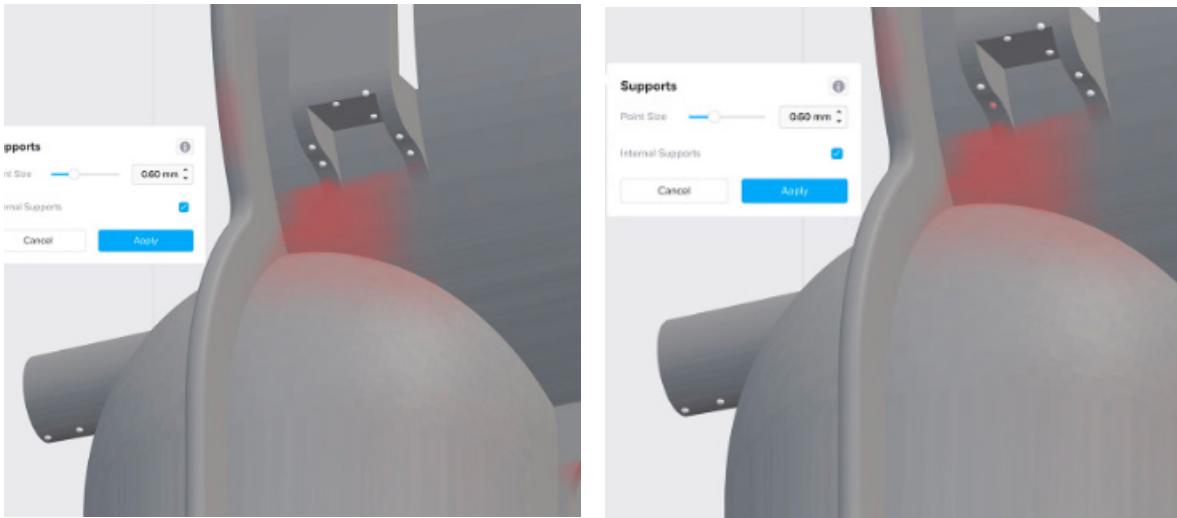
Red shading indicates your model needs additional supports, which can be added manually.

Hovering over a support will turn it red. Click once to remove.

The slicer tool allows you to see inside your model and place internal supports.

Adding and Deleting Manual Supports

Create supports by clicking on the surface of the model. To remove a contact point, click the contact point once. Supports can be added to any visible surface that faces the build platform, at intervals of 1.5 mm or larger. When manually adding supports, the touch-point size slider works identically to the automatic support generator. You can also edit touch point size for individual supports by holding down shift and scrolling on your mouse.

Internal supports can also be added, as long as the cavity or gap is visible and "Internal Supports" is selected in the Supports editor. The layer slicer is enabled in manual support editing mode, but is upside down to allow visibility into the model.

Editing multiple support touchpoints

Drag the pointer to select multiple touchpoints to apply group edits. Drag the slider, click the arrows, or type in a touchpoint size to adjust the point size for selected touchpoints. Press the 'Delete' key to delete selected touchpoints.

Applying Support Edits

When you are done manually editing the supports, click "Apply" to generate the additional supports. Once you exit the support editor, PreForm applies the edits to the model and reveals the newly generated supports and modifications to the raft. To continue revising individual supports and contact points, re-open the Supports editor.

Flat Spacing

Flat spacing is a measure of the spacing between support pillars. The measurement is set by adjusting the density slider. For standard resins, the default support density will generate a flat spacing measurement of 5.0 mm.
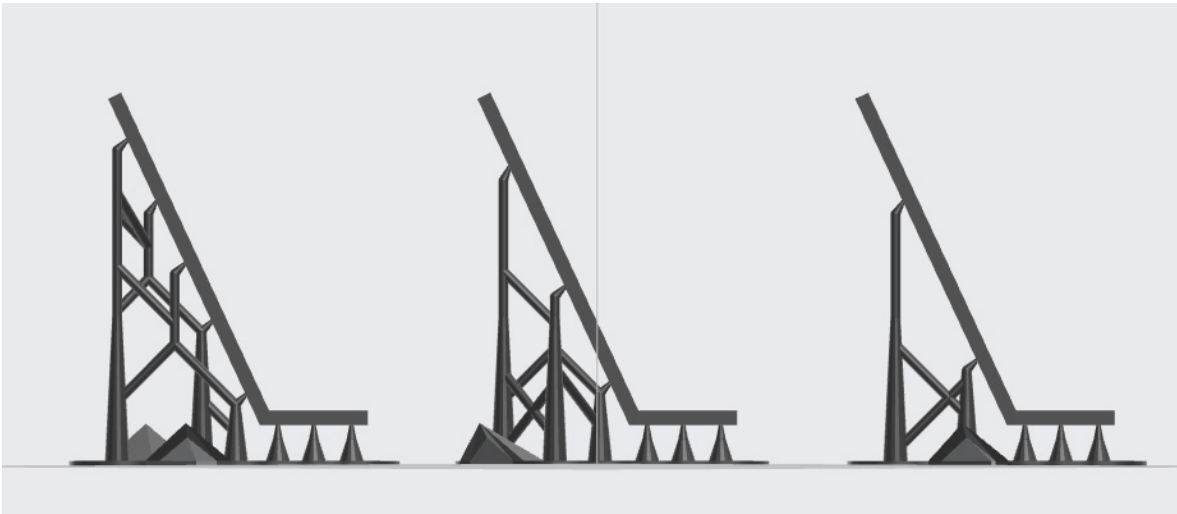
Slope Multiplier

Adjusting the slope multiplier changes the density of supports for sloped and angled surfaces. The default value for the slope multiplier is set to 1.0. The slope multiplier is a coefficient that can be set between 0.0 and 5.0 in increments of 0.01.

> **TIP:** Use the "Slope Multiplier" feature as an alternative to manually adding or subtracting individual support touchpoints for sloped and angled surfaces.

To change the slope multiplier, on the Supports tool menu, click on "Advanced" to reveal advanced support settings. Type the desired slope multiplier or click on the up and down arrows to change the slope multiplier in increments of 0.05.
- A low slope multiplier (that is, less than 1.0) will generate fewer supports on sloped surfaces, with more spacing between supports.
- A high slope multiplier (that is, greater than 1.0) will increase the number of supports on angled surfaces, with less spacing between supports.

From left to right: supports generated with the slope multiplier set to 2.0, 1.0, and 0.5.

Raft Thickness

All prints start with a raft that adheres to the build platform. Support structures and the models themselves are created on top of the raft layers. When adjusting the raft thickness, keep in mind that a thinner raft may affect support structure stability, while a thicker raft may be difficult to remove from the build platform. Because of normal compression that occurs between the build platform and tank during the first layers, the raft thickness that you set in the advanced support settings will usually be between 1.5 mm and 2.0 mm. The default raft thickness value is 2.0 mm.

Height Above Raft

"Height Above Raft" indicates the minimum height of support structures, or the shortest distance between the part and the raft. The default value and the Formlabs recommended minimum is 5.0 mm, and PreForm rejects any value less than 3.5 mm. If the Height Above Raft value is too small, supports on parts with large surface areas may fail or deform due to normal shrinkage.

Z-Compression Correction

Compression along the Z-axis occurs in the early layers of every print to ensure that a part properly adheres to the build platform. Printing directly on the build platform may result in a finished part being marginally shorter than designed. "Z-Compression Correction" compensates for differences in a model's height to maintain dimensional accuracy during printing. By default, "Z-Compression Correction" is set to a value of 0.75 mm.

Early Layer Merge

Uneven or bumpy surfaces cannot adhere to the build platform. Without a raft, parts with an uneven base fail if they are printed directly on the build platform. Level the base of a model to be flat and smooth with "Early Layer Merge" to ensure proper adherence to the build platform.