# Origin Operations and Applications Guide

DOC-30013 Rev. E

www.stratasys.com

Comments: c-support@stratasys.com

# Revision History

| Revision | Release Date | Description of Changes |
|---|---|---|
| A | Nov. 2023 | Initial release |
| B | Aug. 2024 | • Added a Workflow section<br>• Added Optimizing Anchor Points and Generating Support Structure, including Extra Island Support, Max strut length, and other software UI changes<br>• Added Optimizing Model Accuracy, updated the sections, and added Through-cure Mitigation (Z-compensation)<br>• Printing Slice Stacks added<br>• Cross-section View tool added<br>• Measure tool added |
| C | Sep. 2024 | • Added Origin Cure Guidelines<br>• Added Online Resources<br>• Added Collections<br>• Added Snap Model Vertices<br>• Updated document name to Origin Operations and Applications Guide<br>• Updated Maximal Actual Part Volume (was called Build Volume) |
| D | Oct. 2024 | • Added printer specs for materials<br>• Added P3 Stretch 80 material<br>• Updated Origin Cure program names and adding new programs<br>• Updated Collection names in images |
| E | Nov. 2024 | • In Support Profile for Stratasys Material, added Conservative profiles<br>• Defined Preferred, Validated, and Open collections of material profiles<br>• Updated Origin Cure Guidelines<br>• Added Inline Line of Anchors |

# Table of Contents

# Overview

The Stratasys® Programmable Photopolymerization (P3) process is unique for the geometrical freedom it offers. It is capable of printing fine features, as well as large cross-sections, with excellent surface quality comparable to injection molding, all while being able to keep supports to a minimum.

Like traditional manufacturing processes, there are guidelines that result in the highest yields, optimal quality, and reduced labor.

Models that could be designed specifically for additive manufacturing can be designed using familiar traditional techniques like filleting, adding ribs to reduce the number of supports, and increase production throughput.

At the same time, whether you have Designed for Additive Manufacturing (DfAM) model or existing geometry models, following a few orientation and support generation guidelines can ensure the best quality models with minimal supports. Together with tools for optimizing model accuracy, you can expect the highest success rate and the best possible outcome.

This Origin Operations and Applications guide is intended to help you understand Origin's capabilities, select candidate models for additive production, and learn how to optimize model preparation for the best throughput and quality on Origin. The GrabCAD Print™ images were captured in version 1.93 and later.

# Print Preparation Workflow

The pre-processing workflow includes adding a model, selecting the appropriate materials and support profiles, optimizing dimensional accuracy of the model, arranging the models, and printing.

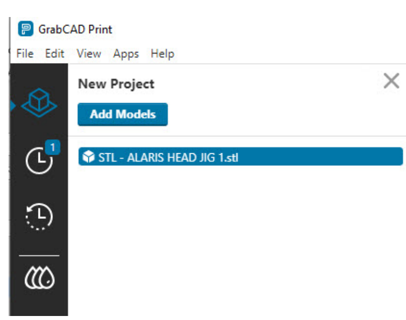
## Adding a Model

Import your model into GrabCAD Print.

See "How to Choose the Right Model for Additive Manufacturing (AM)" (page 54).

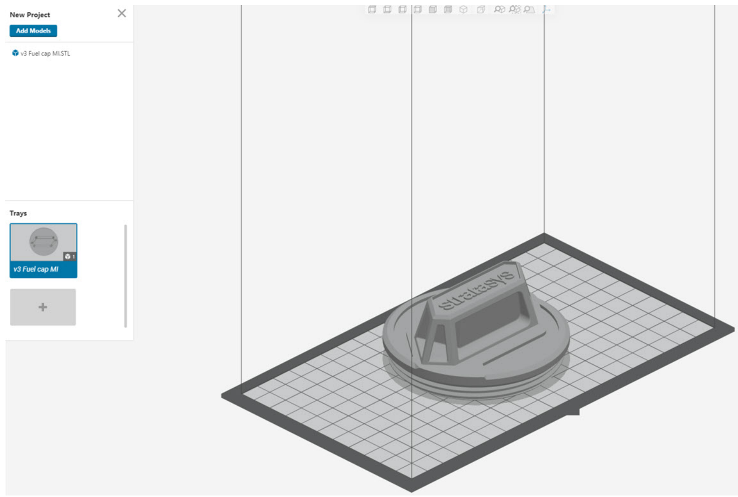
1. In GrabCAD Print, do the following:
    a. In the *Print* view, click **Add Models**.

Figure 1: Adding a model



b. Browse and select a file (for example, an *.stl file).

Figure 2: Imported model



c. If needed, click **Repair All Models**.

After the model is inserted, GrabCAD Print analyzes the model. If there are other errors, resolve them.

d. Select the model.

The *Anchor Points* and *Support* tabs are now available.

# Selecting a Material and Support Profile

To ensure ultimate performance and quality, Stratasys provides a robust range of advanced 3D printing materials and filaments for the following distinctive tiers:

- *Stratasys Preferred*—Preferred by Stratasys for its customers for the highest performance applications. These materials are engineered to provide the best combination of material and printer performance and are developed either by Stratasys or third-party material partners.
- *Stratasys Validated*—Materials validated by Stratasys with basic reliability testing to accelerate the expansion of material options available in the marketplace. These materials may or may not be exclusive to Stratasys.
- *Open*—Unvalidated materials accessed via an OpenAM. These materials may offer unique attributes and the potential to address new applications but they have not received validation testing or optimization relative to performance and functionality on a Stratasys printer. Specifically created for tooling, functional prototyping, jigs & fixtures, and end-use production parts, the Stratasys materials & filaments meet the most demanding industry standards.

All the *Validated* and *Preferred* materials in our material library undergo an optimization process aimed at accuracy, and deliver satisfactory results in most situations. Occasionally, due to factors such as geometry and workflow, printed results may vary from expectations. GrabCAD Print for Origin printers offers several tools to optimize the dimensional accuracy of a 3D-printed model. For more details, see the “Optimizing Model Accuracy” (page 36).

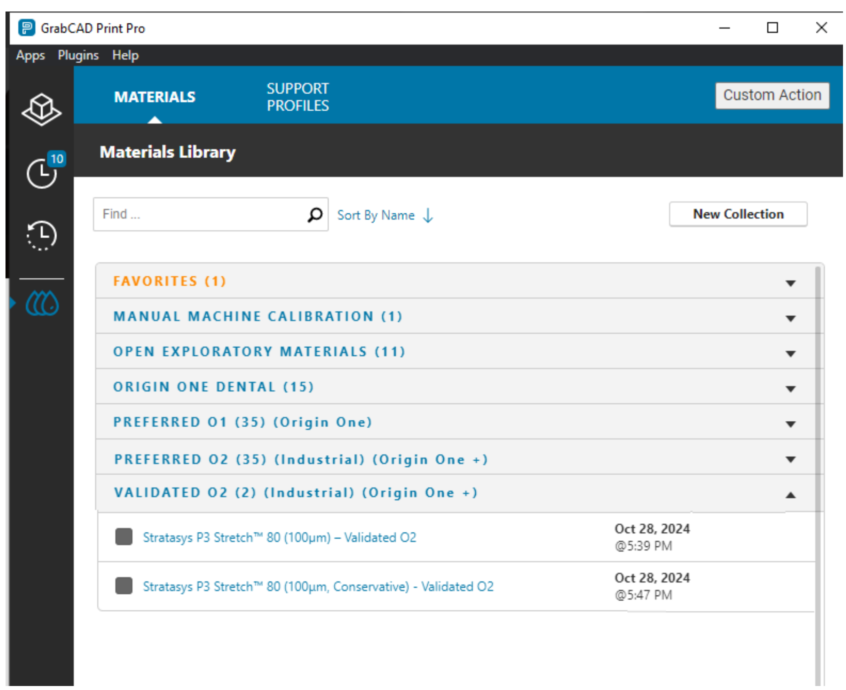
### To select a material and support profile:

1. In *Materials > Materials Library*, do any of the following:
   - Browse to find a material by clicking any of the pre-defined collections, and select one of the materials that is displayed.

     For example, in *VALIDATED O2 (Industrial) (Origin One +)*, you can select the *Stratasys P3 Stretch™ 80(100 µm) - Validated O2* validated material profile.
   - Use a filter to find a material by using the *Find* filter.
   - To create a new collection by clicking **New Collection**.

     You can also create a new material and share it by clicking *Create Material* in the collection.

Figure 3: Selecting materials

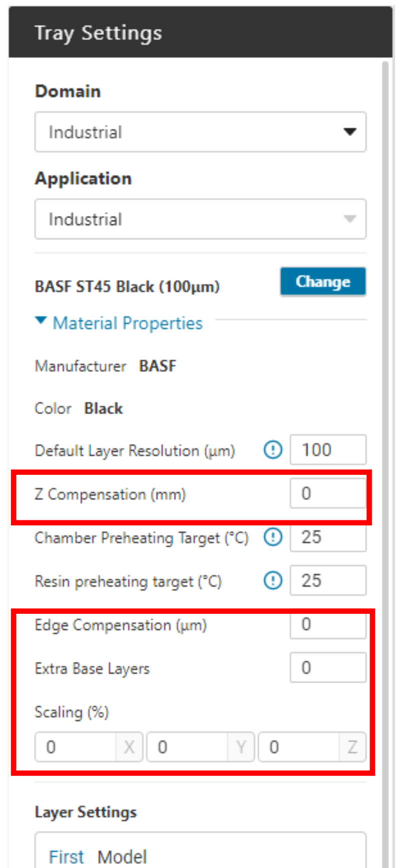



2. From the right menu under **Tray Settings** , set material properties by clicking **Material**.
3. In the *Tray Settings* tab, do any of the following:
   - To change the settings of the selected material, click **Change**.
   - To improve model accuracy, select **Properties** and set any of the following parameters: see “Optimizing Model Accuracy” (page 36):

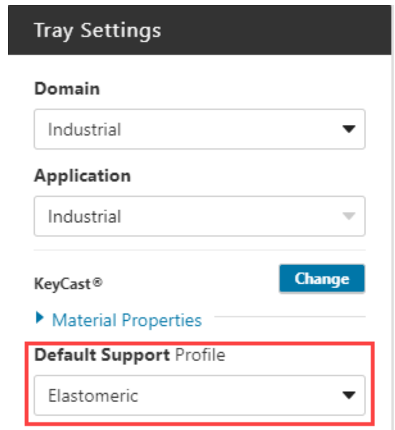
- Z-Compensation
- Edge compensation
- Extra Base Layers
- Scaling (%)

Figure 4: Tray Settings—Accuracy-related



e. Under *Default Support Profile*, select the support profile for all models on the tray. You can change the support profile for individual models, if necessary.

Figure 5: Tray settings - Default Support Profile




Materials are categorized by their characteristics, which correspond to different support profiles. The recommended support profiles for the Stratasys materials are pre-defined for the listed materials. For information about the recommended support profile that corresponds to the Stratasys-Preferred and Validated materials, see Table 1.

Table 1: Support Profile for Stratasys Materials

| Support Profile | Stratasys Material | Description |
|---|---|---|
| Rigid/<br>Rigid - Conservative | • Uncur3D ST45<br>• Loctite® 3955<br>• P3™ Deflect™ 120<br>• Loctite® IND403<br>• Somos® WeatherX™ 100 | Thin support structure and tip diameter |
| Light Tough/<br>Light Tough - Conservative | • Loctite® IND3172<br>• Loctite® IND3843<br>• Loctite® IND405<br>• Loctite Dura™56<br>• Loctite® MED413<br>• Loctite® MED412 | • Thicker support structure than Rigid<br>• Same tip diameter as Rigid |
| Ultra Tough | • Somos® QuickGen™ 500 | • Same thickness of support structure as Rigid and Light Tough<br>• Longer tip diameter than Rigid and Light Tough |
| Elastomeric/<br>Elastomeric - Conservative | • Loctite® IND402<br>• P3™ Stretch™ IND475<br>• P3™ Stretch™ 80 | Thickest support structure and tip diameter, compared to the other support profiles |

# Changing a Support Profile

You can change a support profile, if relevant. In some cases, changing the settings can reduce the manual work that is required to generate an effective support structure. For example, settings can be applied to generate more anchor points automatically, rather than having to manually add the anchor points (see "Best Practices" (page 54)).

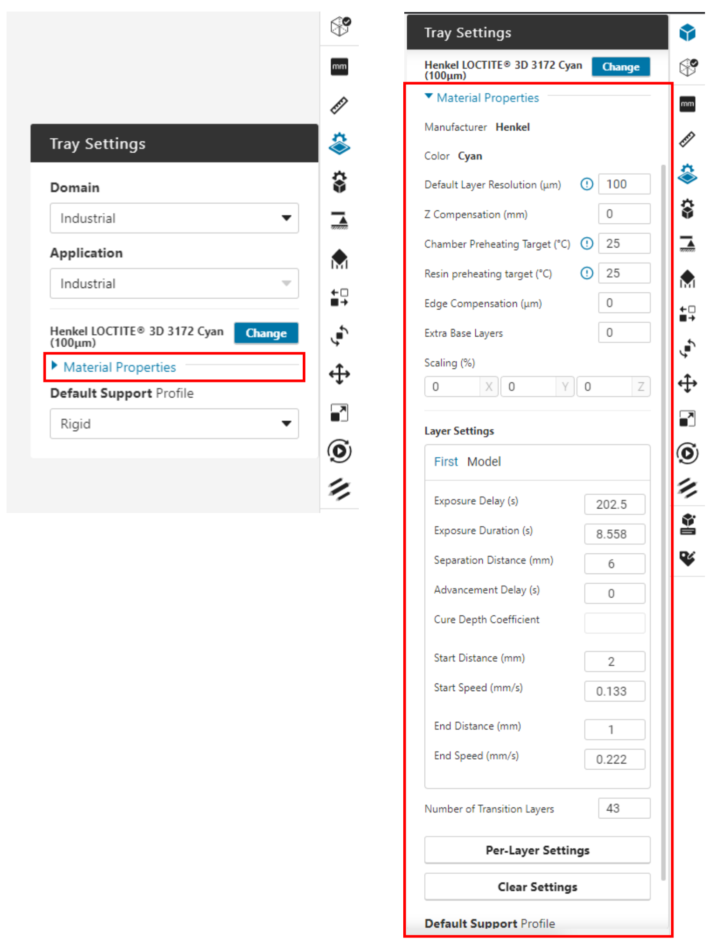
The following options are available for changing a support profile:

- Customize and save a support profile, see "Customizing a Support Profile" (page 11).
- To edit settings on-the-fly for a specific model without creating a new support profile, see "Customizing a Support Profile Per Model" (page 13).

## Customizing a Support Profile

1. In the *Materials & Profiles* view, select *SUPPORT PROFILES*.
2. Do the following:
   a. Browse to the *Support Profiles* list, and set the *Material Properties*, if needed.

Figure 6: Tray Settings - Minimize and expanded *Material Properties*



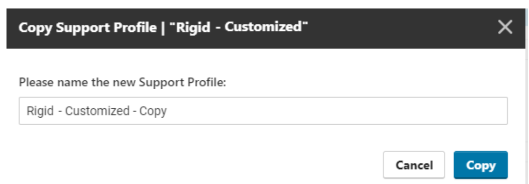
b. *S*elect a pre-defined support profile that most closely represents the support style you would like to mimic. For information about the following pre-defined support profiles, see Table 1:

- Rigid/Rigid - Conservative
- Light Tough/Light Tough - Conservative
- Ultra Tough
- Elastomeric/Elastomeric - Conservative

c. In the *Support Profile* screen, click **Copy** .

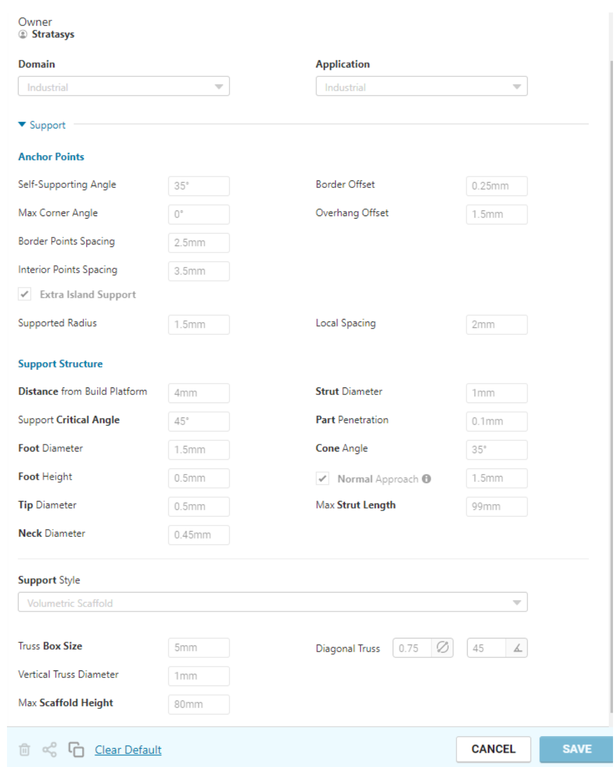
d. In the *Copy Support Profile* dialog box, update the name and click **Copy** .

Figure 7: Copy Support Profile




e. In the *Support Profile* screen, click , to expand the *Anchor Points* and *Support Structure* settings.

Figure 8: Support Profile



f. Set the *Anchor Points* settings, as described in "Anchor Points Settings" (page 18).

g. Set the *Support Structure* settings, as described in "Support Settings" (page 23).

> - For information about how the support structure is affected by the settings, see "Best Practices" (page 54).
> - Although multiple *Support Styles* are available in the support structure settings, only *Volumetric Scaffolds* have been optimized for printing with P3 Origin Industrial printers and resins.

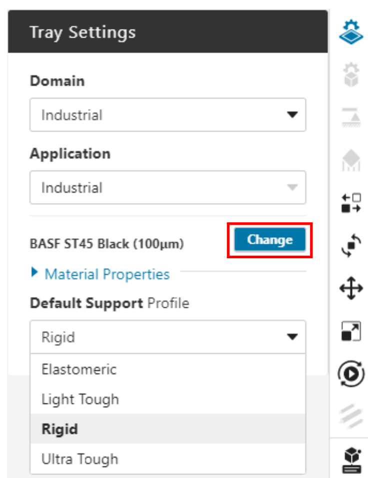
h. Click **SAVE**.

3. In the *Tray Settings* tab, select the *Default Support Profile* that you just created:

a. Return to the *Print* view .

b. Click **Change** and select the material.

Figure 9: Selecting the *Default Support Profile*




## Customizing a Support Profile Per Model

You can edit *Support Profile* settings while viewing a model to customize the result. These settings are only applied to one or more selected models and are not saved for future use.

1. In the *Print* view, select a support profile.
2. In the *Anchor Points* tab, modify the *Anchor Points* settings, as described in "Support Settings" (page 23).
3. In the *Support* tab, modify the *Support* settings, as described in "Support Settings" (page 23).

# Orienting the Model

You can orient the model automatically or manually. For more information about orienting models by customizing support profiles, see "Model Orientation" (page 56).

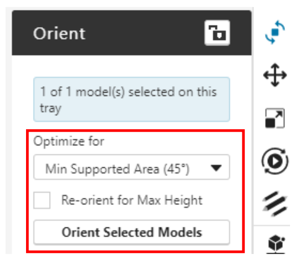
## Orienting Automatically

1. In the *Orient* tab, do the following:
   a. Under *Optimize for*, select the type of orientation you require.
      - Min Supported Area (*angle*)—Orients the model according to the *Self-Supporting Angle* that is set in the selected support profile.
      - Min Height—Orients the model to have the lowest Z height.
      - Max Models—Orients the model to fit as many models as possible.

   b. Click **Orient Selected Models**.

      The model is oriented according to the *Optimize for* setting.
   c. Select *Re-orient for Max Height* for large models to fit the tray size.

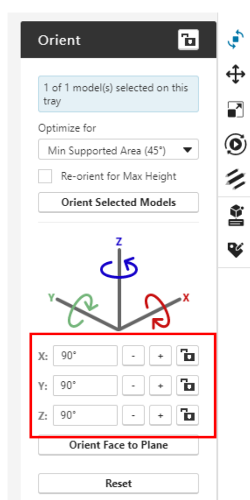
Figure 10: Optimize for different orientation types




## Orienting Manually

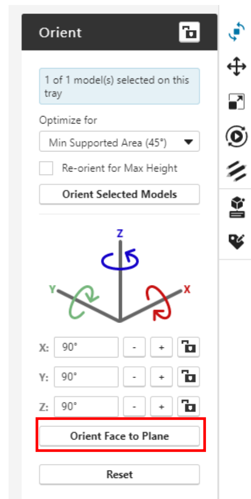
1. To manipulate the model manually, do any of the following:
   - In the *Orient* tab:
     - For any of the axes, enter the angle of rotation around the corresponding axis.

Figure 11: Setting angles of rotation around axes



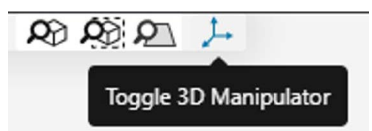
- Click **Orient Face to Plane** to set a surface (face) to be aligned to a selected plane, select the face and select the plane to which the surface should be aligned.

Figure 12: Setting Orient Face to Plane




- Use the *Toggle 3D Manipulator* tool on the top tool bar to rotate and move a model.

Figure 13: Selecting Toggle 3D Manipulator




# Optimizing Anchor Points and Generating Support Structures

The support generation feature comprises generating anchor points and a support structure for printing 3D models in P3 Origin Industrial printers.

1. *Generate Anchor Points* ("Anchor Points Settings" (page 18))—After an analysis of the model geometry using the support profile settings, anchor points are positioned on areas of a model that require structural support to enable print success. You can modify the support anchor positions.
2. *Generate Support Structure* ("Support Settings" (page 23))—After verifying the anchor point positions and approving the support profile settings, support structures are generated accordingly.

A support profile includes a set of *Anchor Points* and *Support* settings that determine the outcome of the support structure for all models on the tray. You can change the support profile for individual models, if necessary.

Besides the material, the geometry and orientation of the model can affect the effectiveness of a selected support profile. If a recommended support profile is not suitable for a certain model, GrabCAD Print enables you to copy a support profile and edit it according to your specifications.

For example, profiles define the size and geometry of the connections and breaking points to the model. Adjusting the size and geometry of these connections can make an impact on the print quality and witness marks left on a model. These vary depending on the model geometry and material being printed. The marks can be sanded off, but the goal is to require minimal labor while leaving the desired surface quality.

## Optimizing Anchor Points

The *Anchor Points* settings determine the placement of the anchor points on selected models. After the *Anchor Points* settings are defined, generate the anchor points and inspect the anchor point positions. For recommendations about improving the stability of the model, see “Optimizing Anchor Points Manually” (page 22).

> Before generating anchor points, check that the *Support Profile* in *Model Settings* matches the desired profile. GrabCAD Print™ supports using different profiles per model, regardless of the *Tray Settings*.

1. In the *Anchor Points* tab, do the following:
   a. Click **Generate Anchor Points**.

      Calculations may take some time to create the anchor points and display them. This depends on the geometry and file size of the model.

      - For information about the anchor point settings, see “Anchor Points Settings” (page 18).
      - For information about the anchor points actions, see Table 2.

Figure 14: Model with anchor points

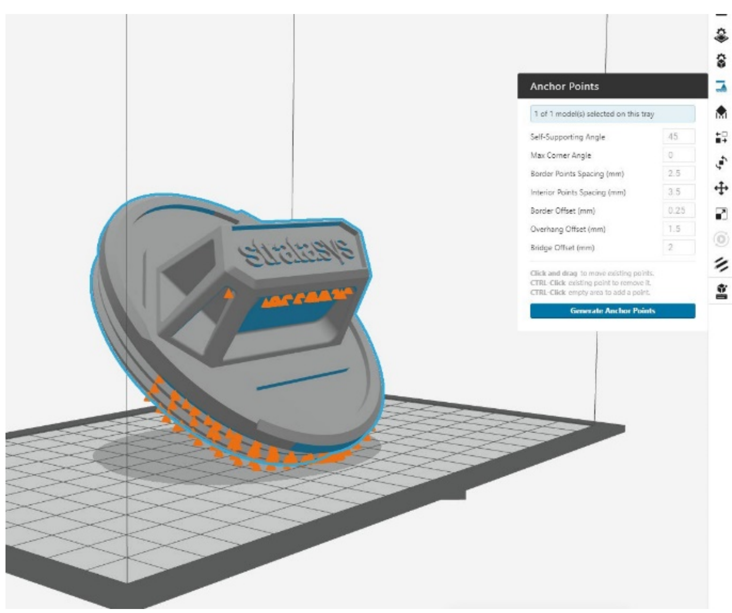
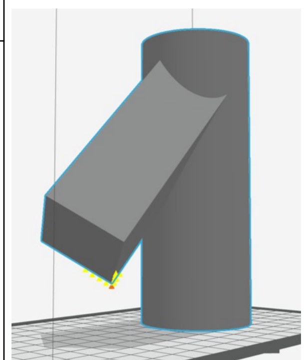
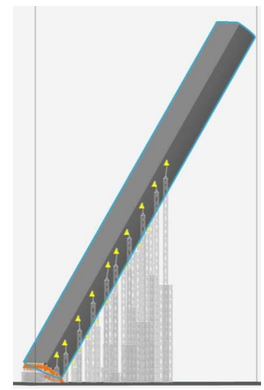
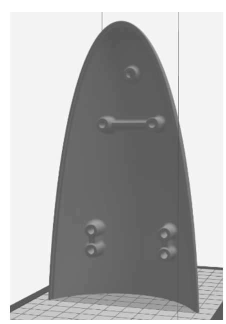


Table 2: Anchor Point Actions

| Parameter | Description | Example 1 | Example 2 |
|---|---|---|---|
| Click and drag | Moves an existing anchor point to a new position. | Figure 15: Manual anchor points added  | Figure 16: After manual anchor points added, support structure is generated  See "Generating the Support Structure" (page 22). |
| CTRL-Click | Adds or removes an anchor point, as follows:<br>• A selected location adds an anchor point.<br>• A selected point removes the anchor point. | | |
| Generate Anchor Points | Generates anchor points positioned on areas of a model that require structural support according to the *Anchor Points* settings. | Figure 17: Model before anchor points generated  | Figure 18: Model after anchor points generated  |

b. Inspect the anchor points placement.

If the desired outcome is not met, you can make changes to the settings in the *Anchor Points* settings and generate the anchor points again (see "Support Settings" (page 23)).

c. For a stable printed model, make changes to the anchor points, as described in "Optimizing Anchor Points Manually" (page 22).

For recommendations about using anchor points, see the following in "Best Practices" (page 54).

## *Anchor Points Settings*

The *Anchor Points* settings, which are part of the support profile, determine the placement of anchor points on the selected models. You can modify the *Anchor Points* settings and also change the positions manually, when relevant.

For information about generating the support, see “Support Settings” (page 23).

Figure 19: Anchor Points

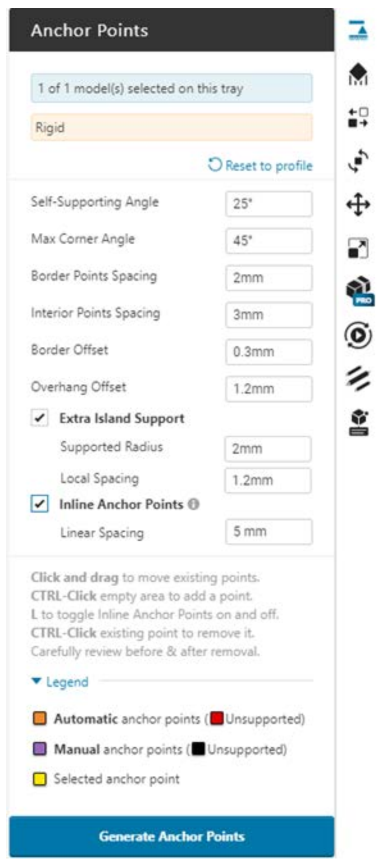


Table 3: Anchor Point Settings

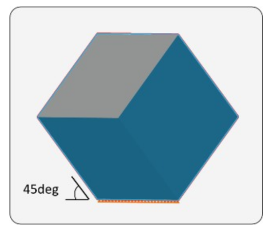
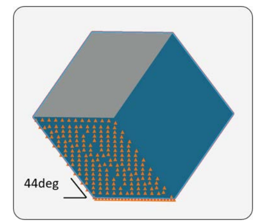
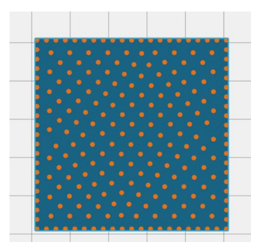
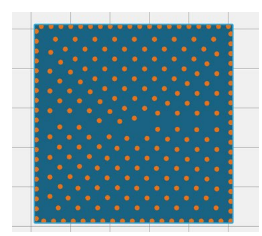
| Parameter | Description | Example 1 | Example 2 |
|---|---|---|---|
| Self-Supporting Angle | The minimum allowed angle between the surface and the horizontal, which can be printed without support structures.<br>• For surfaces with larger angles, anchor points may need to be added manually.<br>• For surfaces with a lower angle, anchor points are added automatically. | Figure 20: Angle equal to the self-supporting angle (45°)<br> | Figure 21: Angle (44°) less than the self-supporting angle (45°)<br> |
| Max Corner Angle | For surfaces that are parallel to the platform, anchor points are placed in corners when the defined angle is below the angle of a corner. | Figure 22: 89° Max Corner Angle – places anchor points at the true corners<br><br>**Note:** Anchor points are placed at the corners of models when there is a corner with an angle greater than the setting:<br>• For 90° corners, this includes settings from 1°–89°.<br>• For 60° corners, this includes settings from 1°–59°. | Figure 23: 90° Max Corner Angle – places anchor points at interior points away from the true corners.<br><br>**Note:** Anchor points are placed near the true corners. For angles that are the same or greater than the corner angle:<br>• For 90° corners, set to 90°and greater.<br>• For 60° corners, set to 60° and greater.<br>• For all geometries, set to 0°. |

Table 3: Anchor Point Settings (Continued)

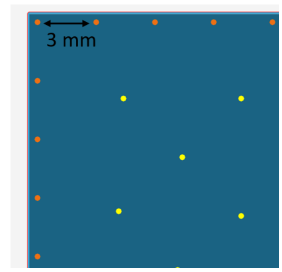
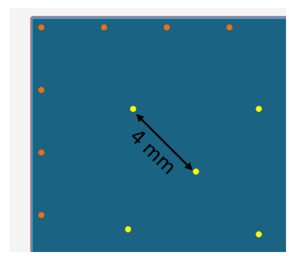
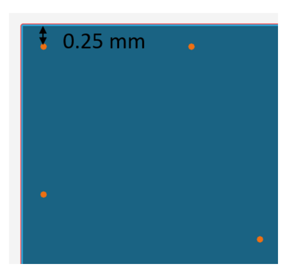
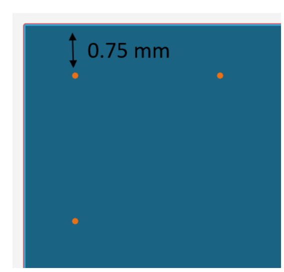
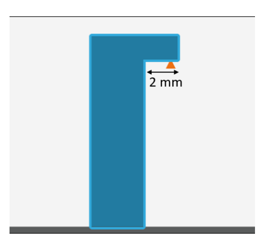
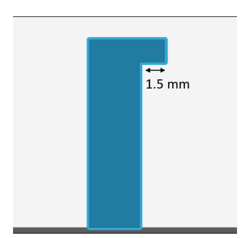
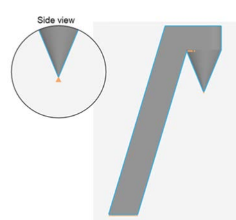
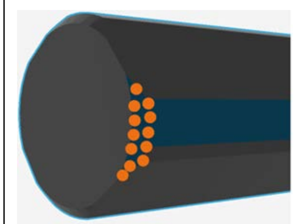
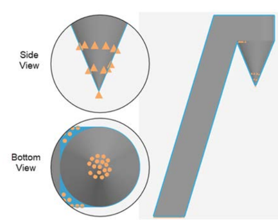
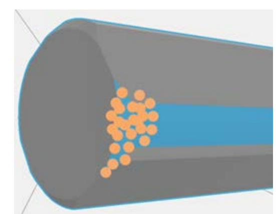
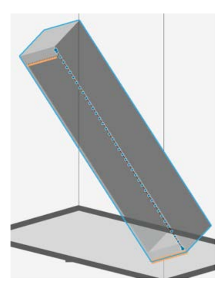
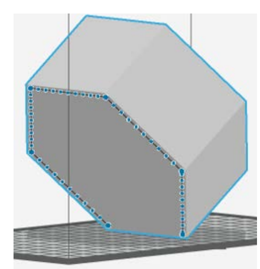
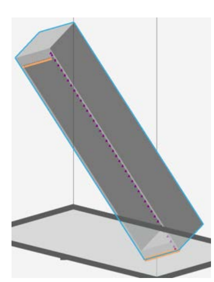
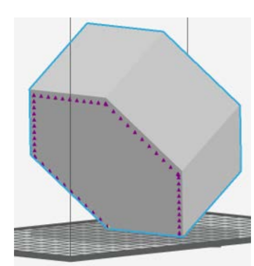
| Parameter | Description | Example 1 | Example 2 |
|---|---|---|---|
| Border Points Spacing | Sets the anchor point spacing from point-to-point along the border of a model. | Figure 24: Border Points Spacing – 3 mm   | |
| Interior Points Spacing | Sets the anchor point spacing from point-to-point within the interior area of a surface of a model. | Figure 25: Interior Points Spacing – 4 mm   | |
| Border Offset | The distance between the border anchor points and the edge of a surface.<br>The value is usually the radius of the tip point of the support structure. See "Optimizing the Border Offset" (page 75). | Figure 26: Border offset –0.25 mm   | Figure 27: Border offset –0.75 mm   |
| Overhang Offset | The maximum overhang length from the previous slice without needing support. | Figure 28: Overhang requires support when the Overhang Offset (1.5 mm) is less than the overhang (2 mm)   | Figure 29: Overhang does not require support when the Overhang Offset (1.5 mm) is the same as the overhang   |

Table 3: Anchor Point Settings (Continued)

<table>
<tr><th>Parameter</th><th>Description</th><th>Example 1</th><th>Example 2</th></tr>
<tr><td rowspan="2">Extra Island Support</td><td rowspan="2">When selected, the following parameters ensure that islands receive adequate support (even the ones above the self-supporting angle), as follows:<br>• Supported Radius—The radius around the first evidence of an island receives extra anchor points.<br>• Local Spacing—The defined spacing between the additional anchor points receives extra support.</td><td>Figure 30: Extra Island support is Off<br></td><td>Figure 31: Extra Island support is Off<br></td></tr>
<tr><td>Figure 32: Extra Island Support is On<br></td><td>Figure 33: Extra Island Support is On<br></td></tr>
<tr><td rowspan="2">Inline Anchor Points</td><td rowspan="2">After generating automatic anchor points, this parameter can be selected to enable anchor points to be generated in a line, as follows:<br>• Linear Spacing—Enter the defined distance between the anchor points.<br>• Start and End points—Click to place the first anchor point and double-click to place the final anchor point.<br>See “Adding a Line of Anchor Points to Stabilize Models” (page 73).</td><td>Figure 34: Simple—Placed start, stops, and an end-point<br></td><td>Figure 35: SimplePlaced start, stops, and an end-point<br></td></tr>
<tr><td>Figure 36: Multiple stops—Manually-added anchor points<br></td><td>Figure 37: Multiple stops—Manually-added anchor points<br></td></tr>
</table>

## Optimizing Anchor Points Manually

Review the anchor points that were added automatically. You can add anchor points or modify anchor point placement, if needed. For example, remove support from a critical area or add support for a model with heavy geometry.

1. To change the anchor points manually, do any of the following:
   - To move existing anchor points, *click and drag* the points.
   - To remove a point, press *CTRL-Click* on the point.
   - To add a point, press *CTRL-Click* on the position to add the point.
2. Optimize anchor points according to the following model characteristics:
   a. "Adding Support to Sharp Islands" (page 70)—If there are islands in the model, verify there is enough support.
   b. "Stabilizing High Models" (page 71)—If some parts of the model are high or heavy, some additional support may be needed.
   c. "Using the Cross-Section View Tool" (page 76)—If you need to view inside the model.
   d. "Using the Measure Tool" (page 77)—To follow the requirements for overhangs.

## Generating the Support Structure

After reviewing and updating the anchor points, generate the support structure automatically and inspect the support generated. If anchor points are shown in red, the structure cannot be generated in that position. In some cases, you need to make changes to the anchor points, and then regenerate the Support structure.

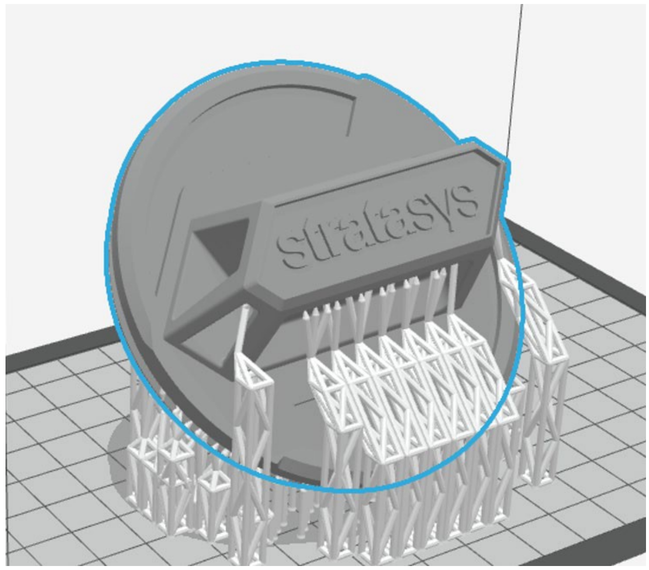
1. In the *Support* tab, do the following:
   a. Select the model on the tray.
   b. Click **Generate Support Structure**.

      Calculations may take some time to create the structures and display them. This depends on the geometry of the model.
      - For information about the support settings, see "Support Settings" (page 23).
      - For information about the support actions, see Table 5 (page 30).

   c. If there are anchor points in red, verify that the support is sufficient.
      - If the support is sufficient, there is no other action necessary.
      - If the support is not sufficient, continue with step 2:
2. Do any of the following:
   - Move the support anchor points to a different position, as described in "Optimizing Anchor Points Manually" (page 22).
   - Modify settings in the *Support* tab, as described in "Support Settings" (page 23).

- Add a strut manually between parts in the model, as described in "Using the Part-to-Part Support Tool" (page 71).

Figure 38: Support structure generated




3. If you made any changes to the anchor points, in the *Support* tab, click **Generate Support Structure**.

> Due to the complex process used for generating support structures in GrabCAD Print, the *Hide (don't print)* feature may have unintended consequences. If a model is hidden and then unhidden, support structures need to be generated again.
>
> **Tip:** As an alternative, transfer models out of the viewing area to focus on only one model at a time.

## Support Settings

The *Support* settings determine how the support structure for the model can be generated based on the generated anchor points and the model geometry. After examining your support structure, you can make changes by doing any of the following:

- Modify *Anchor Points* settings and re-generate *Anchor Points* and support structures.
- Manually change anchor point positions and re-generate only the support structures.

For information about generating the anchor points, see “Anchor Points Settings” (page 18).

Figure 39: Support Settings

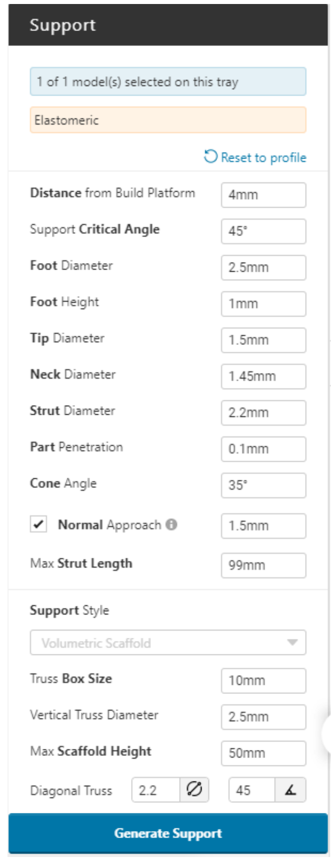


Table 4: Anchor Point Settings

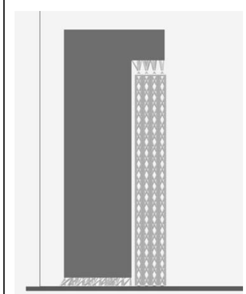
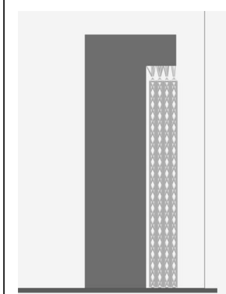
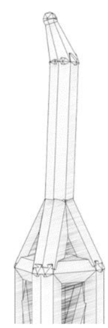
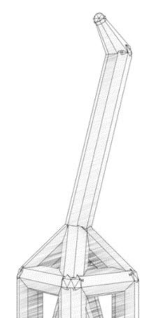
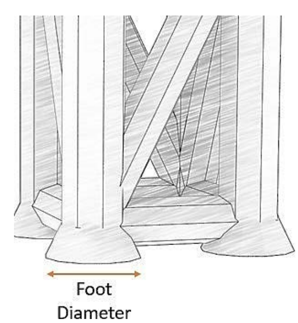
| Parameter | Description | Example 1 | Example 2 |
|---|---|---|---|
| Distance from Build Platform | Controls the vertical gap between the model and the build platform.<br>**Note:** When this is set to 0, the model is flat on the build platform with no support structure to the anchor points on these surfaces. | Figure 40: Distance from Build Platform – 4 mm<br> | Figure 41: Distance from Build Platform– 0 mm<br> |
| Support Critical Angle | Sets the minimum angle between the horizontal and the support structure for which the structure can be built.<br>**Note:** If the angle is less than this angle, then no support structure is built at this point. | Figure 42: Support Critical Angle 60°<br> | Figure 43: Support Critical Angle 45°<br> |
| Foot Diameter (mm) | The bottom of the support structure where it contacts the build platform. | Figure 44: Foot diameter of the support structure<br> | |

Table 4: Anchor Point Settings (Continued)

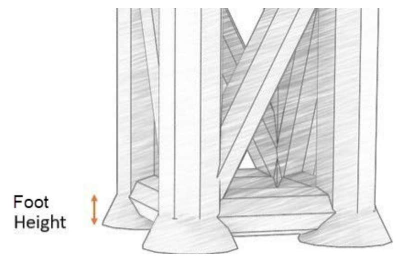
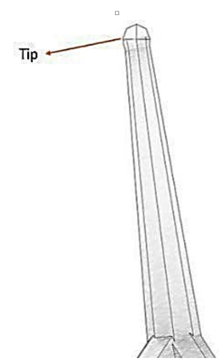
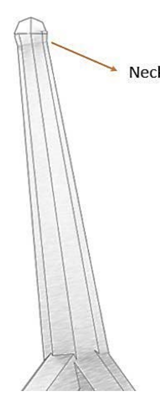
| Parameter | Description | Example 1 | Example 2 |
|---|---|---|---|
| Foot Height (mm) | The height of the foot. This controls the bottom of the support structure. | Figure 45: Foot height of the support structure   | |
| Tip Diameter (mm) | The thickness of the contact point between the model and the support structure.<br>**Note:** The tip shape is a sphere, and the contact point of the support structure and the 3D model is at the sphere diameter.<br>For more information about defining the tip diameter, see "Adjusting the Tip Diameter" (page 76). | Figure 46: Tip diameter of the support structure   | |
| Neck Diameter (mm) | Located under the tip sphere and acts as the breaking point of the support structure. | Figure 47: Neck diameter of the support structure   | |

Table 4: Anchor Point Settings (Continued)

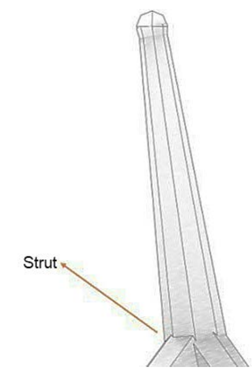
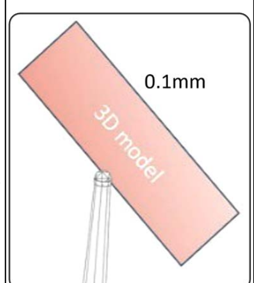
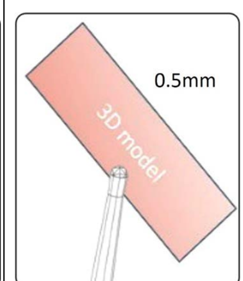
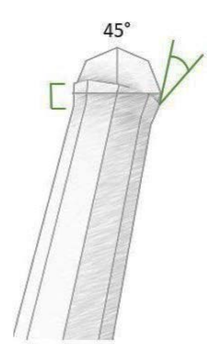
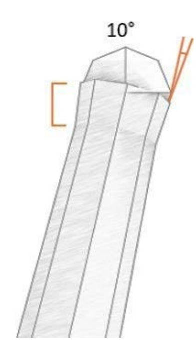
| Parameter | Description | Example 1 | Example 2 |
|---|---|---|---|
| Strut Diameter (mm) | The lowest part of the top pole and the longest diameter of the top pole. | Figure 48: Strut diameter of the support structure<br> | |
| Part Penetration (mm) | Distance from the surface of the model to the end of the tip points within the model.<br>• The position opposite the *Cone Angle* does not affect the neck position.<br>• It might be useful to increase *Part Penetration* when *Normal Approach* is off, but there is a steep surface to support. | Figure 49: Part Penetration is 0.1 mm<br> | Figure 50: Part Penetration is 0.5 mm<br> |
| Cone Angle | The angle between the neck and the tip (the sphere diameter).<br>• This controls the distance between the breaking point and the tip point. A lower angle makes this distance from the 3D model greater than a higher angle.<br>• This controls the appearance on the model surface after the removal of the support structure. | Figure 51: Cone angle of 45°<br> | Figure 52: Cone angle of 10°<br> |

Table 4: Anchor Point Settings (Continued)

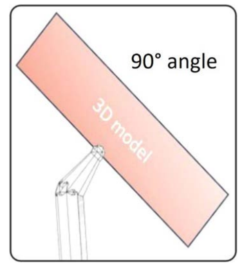
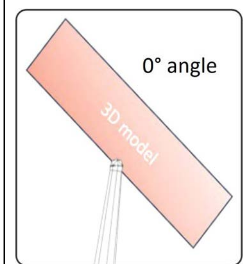
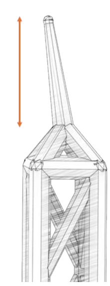
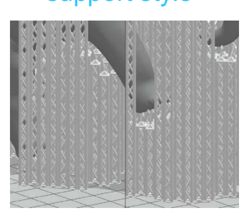
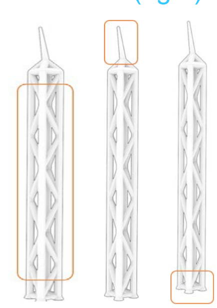
| Parameter | Description | Example 1 | Example 2 |
|---|---|---|---|
| Normal Approach | Determines how the tip point touches the surface and the value of the final tip length.<br>• If selected (on), the tip approaches the surface at a 90° angle.<br>  • An approach to the normal can make the tip point *cleaner*.<br>  • However, it might be a bit less stable because of the forced angle that it needs in order to reach the surface.<br>• If not selected (off), the tip does not approach the surface at a 90° angle. | Figure 53: Approach of the tip is a 90° angle<br> | Figure 54: Approach of the tip is not a 90° angle<br> |
| Max Strut Length | The maximum length of the struts whether they are used as part-to-part support or positioned on top of a scaffold structure .<br>• A higher maximum strut length enhances the support structure's ability to reach difficult anchor points. However, setting this value too high can compromise the stability of the struts.<br>• A lower maximum strut length may lead to more angled struts or red anchors, indicating that GrabCAD Print could not effectively reach the support points. | Figure 55: Strut length in a part-to-part<br> | Figure 56: Strut length in a scaffold structure<br> |
| Support Style | *Volumetric Scaffold*—Controls the values of the scaffold support structure.<br>**Note:** Other *Support Styles* have not been optimized for P3 Origin Industrial printers and resins. | Figure 57: Volumetric Scaffold support style<br> | Figure 58: Main (left); Neck and Tip (middle); Pedestal and Foot (right)<br> |

Table 4: Anchor Point Settings (Continued)

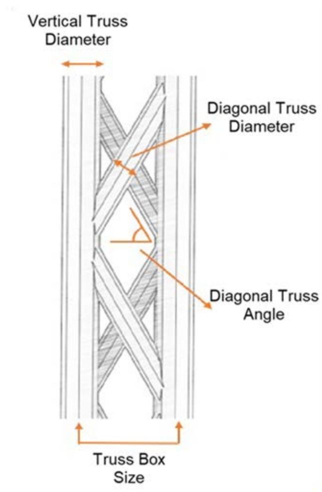
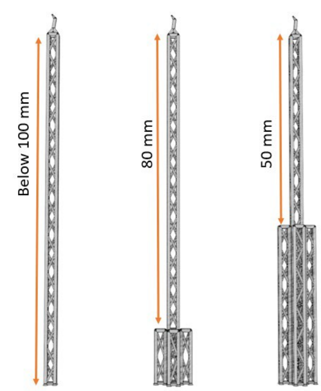
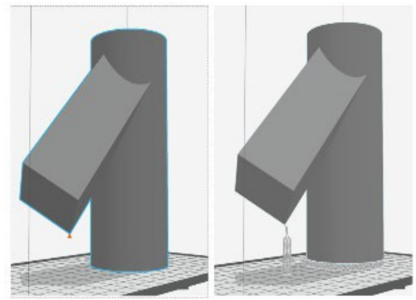
| Parameter | Description | Example 1 | Example 2 |
|---|---|---|---|
| Truss Box Size (mm) | The length of each box wall for the square which creates the truss box.<br>This is measured from the center of a truss to the center of a truss. | Figure 59: Parts of the volumetric scaffold: vertical truss, diagonal truss, and truss box<br><br> | |
| Vertical Truss Diameter (mm) | Thickness of a vertical truss element of a scaffold. | | |
| Diagonal Truss | Defines the *Diagonal Truss Diameter* and *Diagonal Truss Angle*. | | |
| Max Scaffold Height | • Maximum height for a *lone* scaffold to remain as one.<br>• For heights that are greater than this value, more scaffolds are added. | Figure 60: Max Scaffold Height=100 mm (left); 80 mm (middle); 50 mm (right)<br><br> | |

Table 5: Support Action

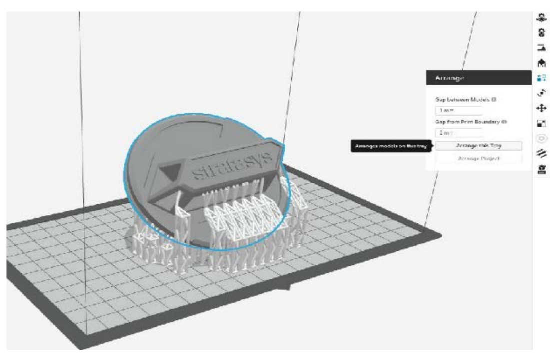
| Parameter | Description | Example 1 | Example 2 |
|---|---|---|---|
| Generate Support | Generates support structures for models according to the anchor points positions and *Support* settings. | Figure 61: Generated support structure automatically, based on defined anchor points and Support settings  | |

# Arranging the Tray

The *Arrange this Tray* function optimizes the position of multiple models on a tray by using the settings for a gap between models and a gap from the print boundary.

1. In the *Arrange* tab, click **Arrange this Tray**.

Figure 62: Arranging the tray



# Printing the Model

After you are satisfied with the support structure and arrangement of the models, you can send the tray to the printer.

1. In GrabCAD Print, in the bottom-right corner, click **Print**.

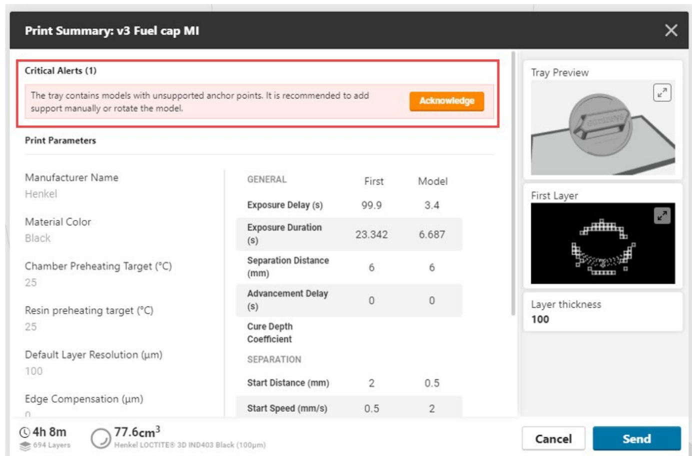
Figure 63: Printing the model

2. Review the *Print Summary* window.

   When there are red anchor points on the model, an alert is visible in this window.

Figure 64: Critical Alerts




3. Do one of the following:
   - If you confirm that you want to send to print in this case, click **Acknowledge** and then **Send**.
   - If you do not confirm, then you can update the anchor points manually, as described in "Optimizing Anchor Points Manually" (page 22).

# Printing a Slice Stack

You can import slices that were created by applications like Netfabb, and 3D print the file.

- The tray must be clear in order to insert the slices.
- Changing the printer after importing the slice stack will close the job and require you to insert it again.
- Saving the project file is not possible.
- The slice naming should be "slice_xx.png", where xx is a number.
- Pay attention that the file type that is required when importing the slice stack is a zip file, not a regular folder that contains the png files.
- If the Print Summary screen window doesn't show a picture of the first layer, it means the zip file is probably missing the first image - "slice_1.png".

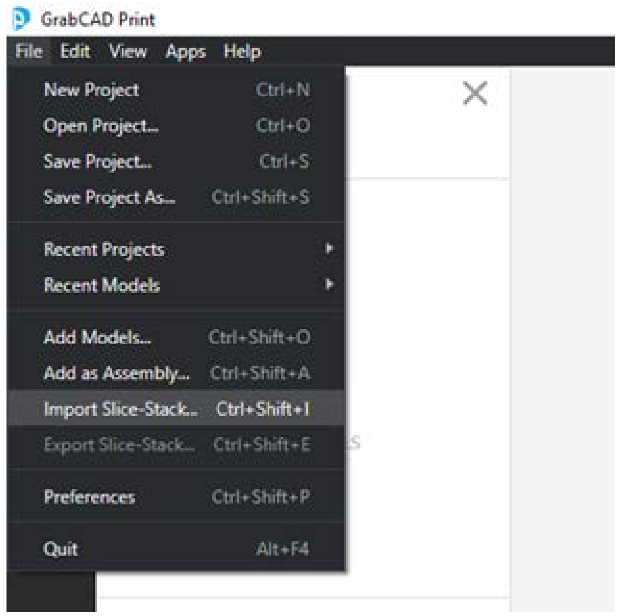
1. Select the Origin printer to print on.

Figure 65: Selected printer

2. Insert the slices:

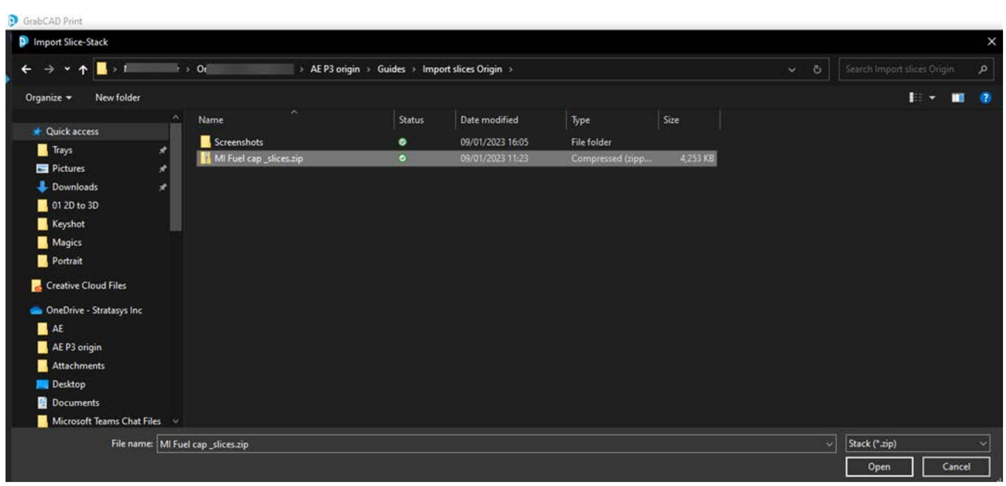
   a. From the **File** menu, select the option I*mport Slice-Stack*.

Figure 66: Imported model




   b. Select the zip files that contain the PNG files, and then click **Open**.

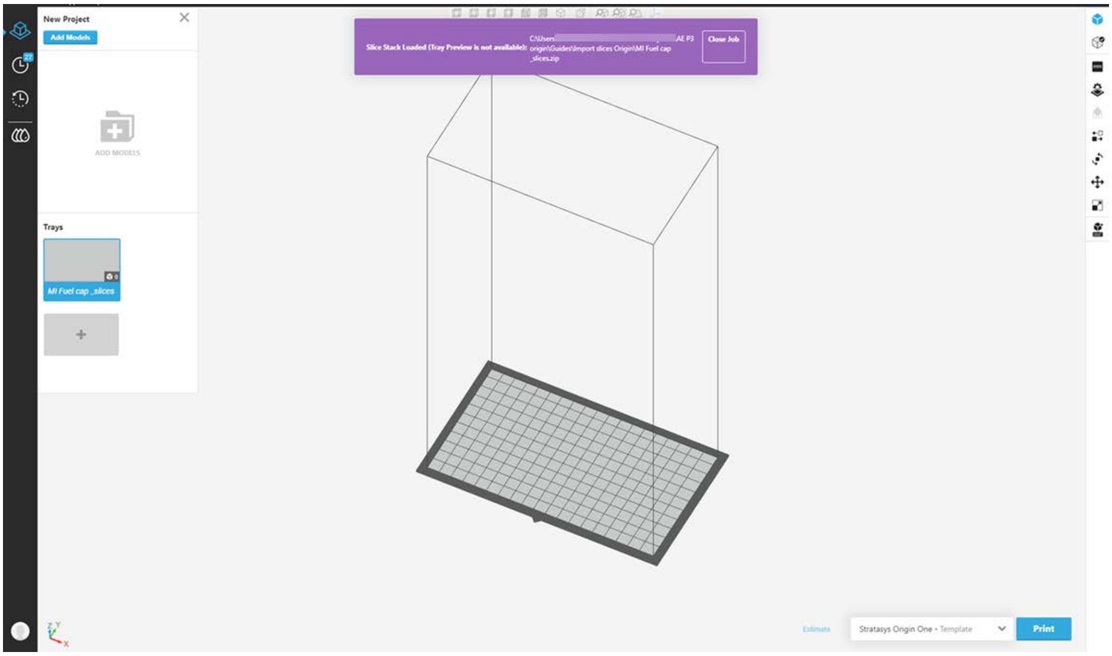
Figure 67: Imported model



A purple banner appearing on top of the screen indicates that the slices are imported.

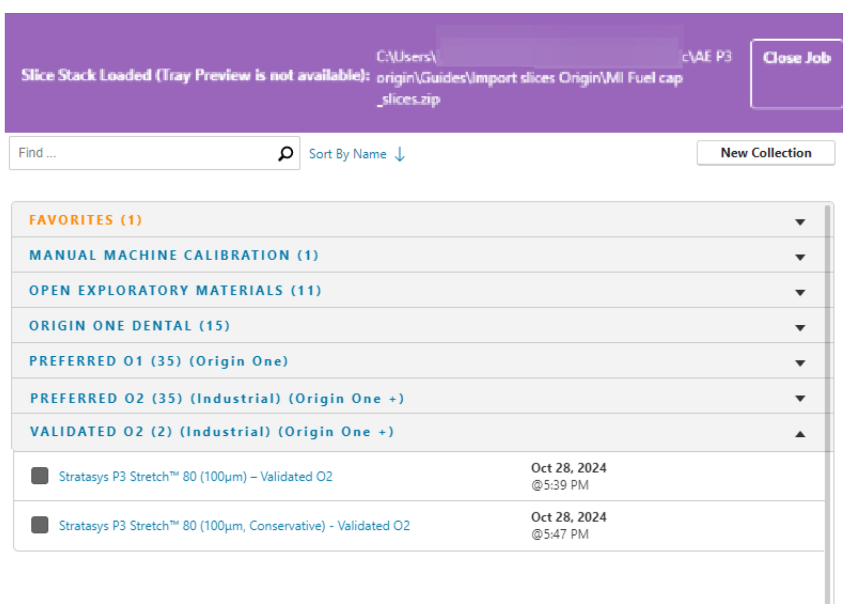
Figure 68: Slices are imported




c. To discard the slices in the purple banner, click **Close Job**.

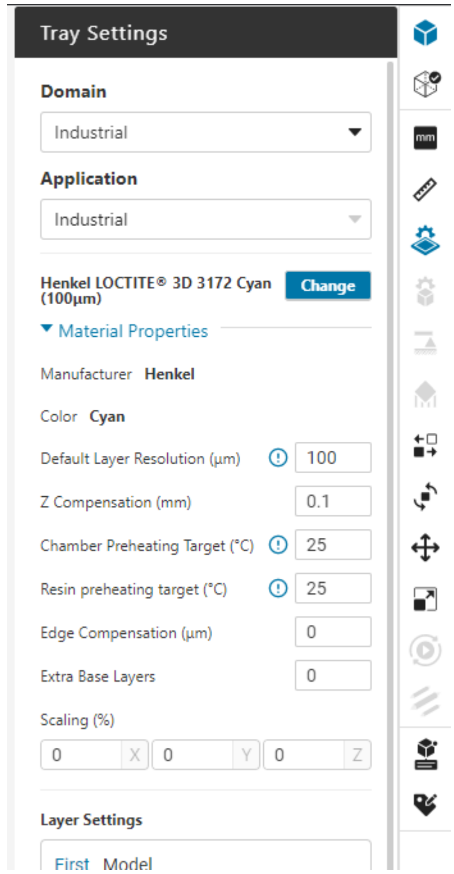
3. Select the material to print.

Figure 69: Assigning a material to the model



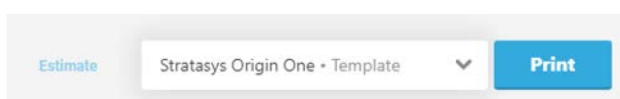
4. If necessary, modify the tray settings.

Figure 70: Tray Settings for stack



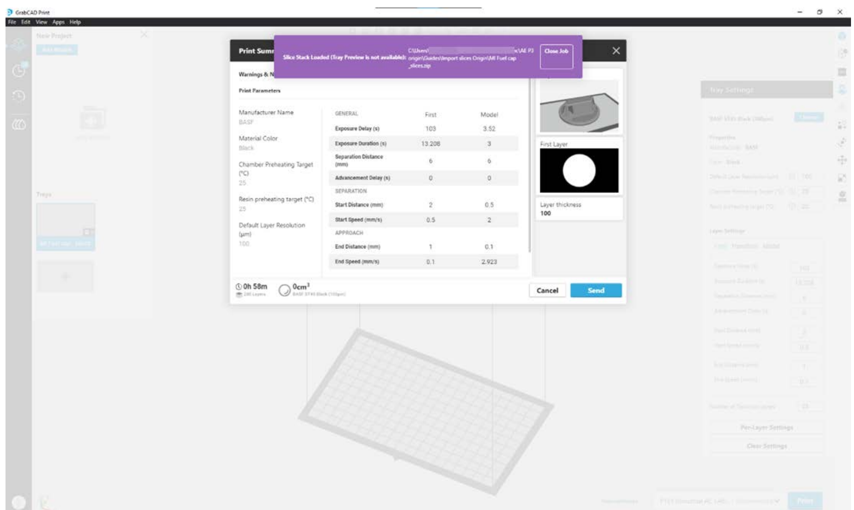

5. When you are ready click **Print**.

Figure 71: Selected printer



A print summary window appears and show details about the print setting. In addition, you'll see a thumbnail of the model you inserted and the first PNG slice of the print.

Figure 72: Print Summary with the thumbnail image of the first PNG image slice



6. Review the summary and click **Send**.

Figure 73: Print Summary missing Tray Preview

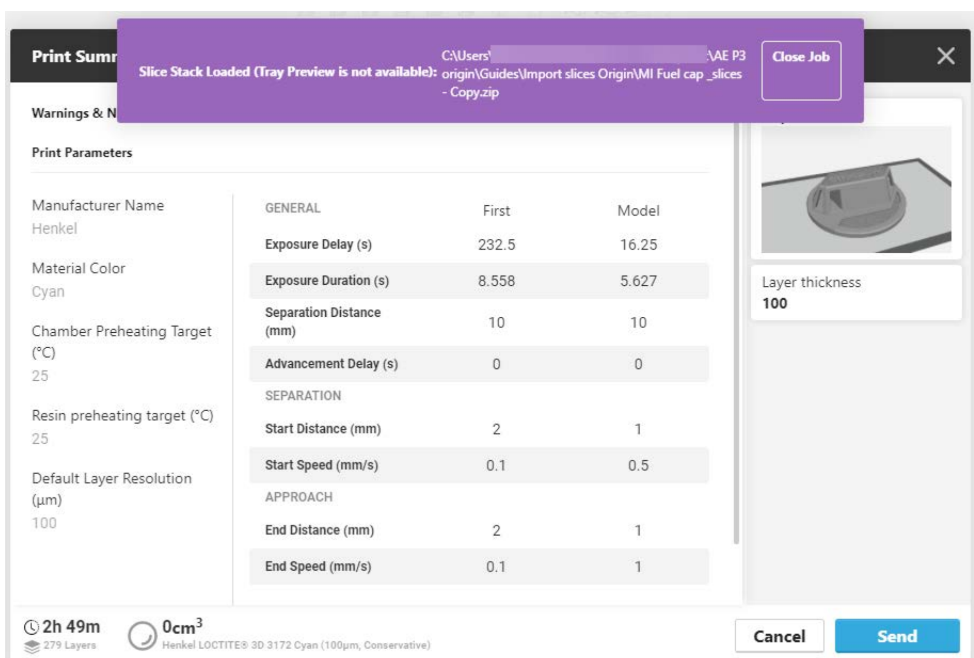
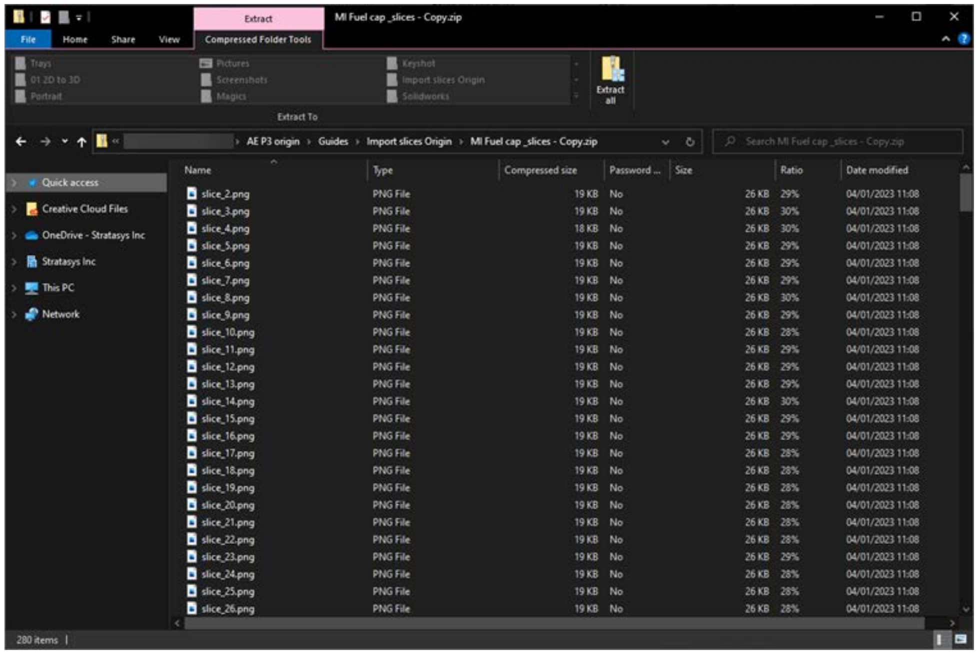


Figure 74: Extracted files from the zip folder-missing slice_1.png



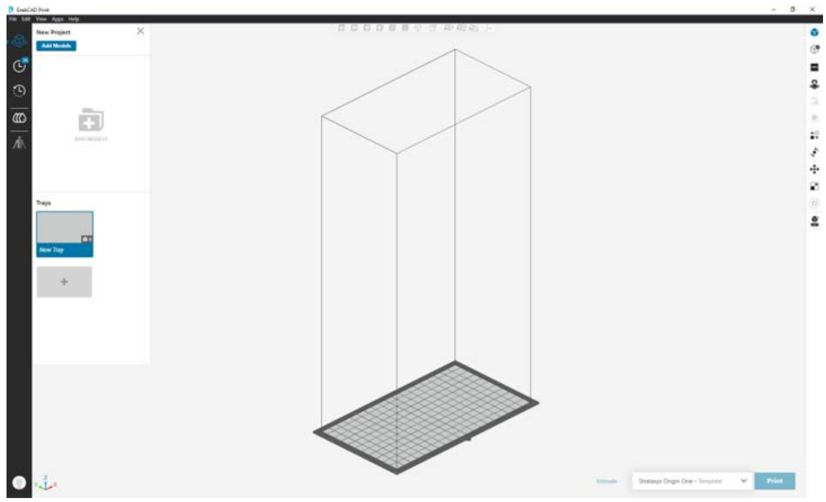

# Optimizing Model Accuracy

GrabCAD Print for Origin printers offers several tools to optimize the dimensional accuracy of a 3D-printed model. All the validated and preferred materials in our material library go through an in-depth optimization process that should give satisfactory results in most situations, see the "Origin Printer Specifications for Stratasys Materials" (page 53).

Occasionally, for reasons such as geometry and workflow, printed results may vary from expectations. This section describes how to optimize the following settings to improve model accuracy:

- Z-Compensation, as described in "Through-Cure Mitigation" (page 37).
- Edge compensation, as described in "Edge Compensation" (page 44).
- Extra Base Layers, as described in "Extra Base Layer" (page 48).
- Scaling (%), as described in "Material Scaling" (page 50).

Figure 75: GrabCAD Print



Be aware that multiple factors can affect the final dimension of a model, such as:

- Model geometry
- Printed material
- Different printers
- Calibration
- Placement of the model on the build platform
- Removal from the build platform
- Handling of the model while cleaning
- Curing strategy—choice of solvent, amount of time in the solvent, drying time, post-cure equipment, intensity, and duration

Therefore, in order for these tools to be useful, it is necessary to perform the same workflow that was used without the tool. By doing so, the chances of successfully improving the model accuracy is increased.

# Through-Cure Mitigation

The Through-Cure phenomenon is a common issue that can occur in vat polymerization 3D printers. In DLP 3D printing, a digital projector is used to cure a liquid photopolymer resin layer by layer to create a solid object. The projector shines light through a transparent bottom surface of a vat containing the liquid resin, curing the resin where the light hits it.

The through-cure problem occurs when the light from the projector cures the resin not only where it is intended, but also through the previously cured layers. This means it hardens more liquid than the size of the layer height. Therefore, the thickness of the overhang below is greater than the original dimensions. Each material profile contains a different amount of through-cure, depending on resin chemistry.

Figure 76: Print orientation

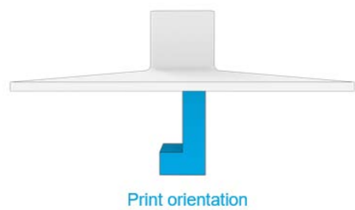
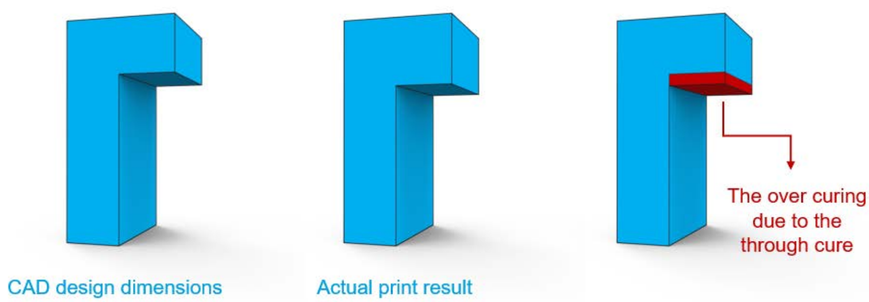



Figure 77: Over-curing due to through-cure




The way to mitigate and compensate for this issue is by reducing the thickness of relevant features and accounting for the extra resin that will cure during the print process. By fine-tuning these parameters, Z compensation ensures each layer is accurately formed, reducing the risk of over-curing and enhancing the accuracy of your 3D prints.

Figure 78: Effect of light penetrating deeper than the layer height

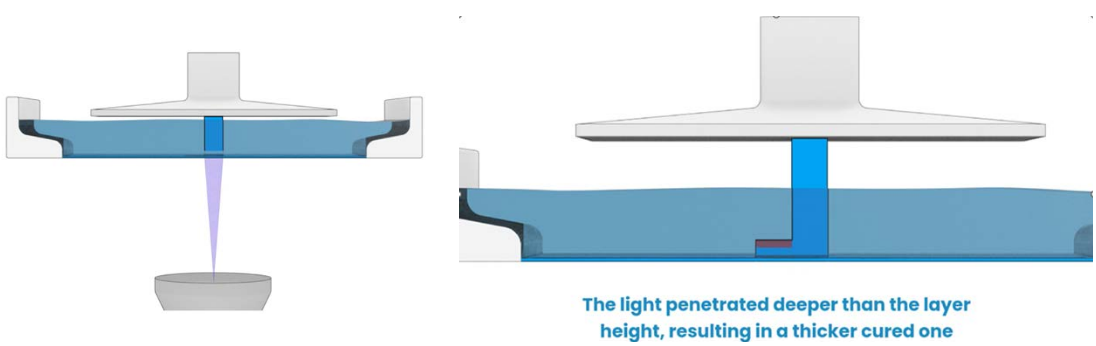


Figure 79: Over-curing on the Z-axis in an accurate thickness

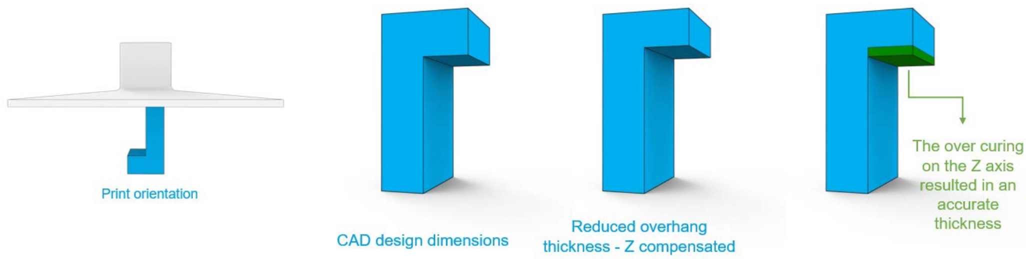
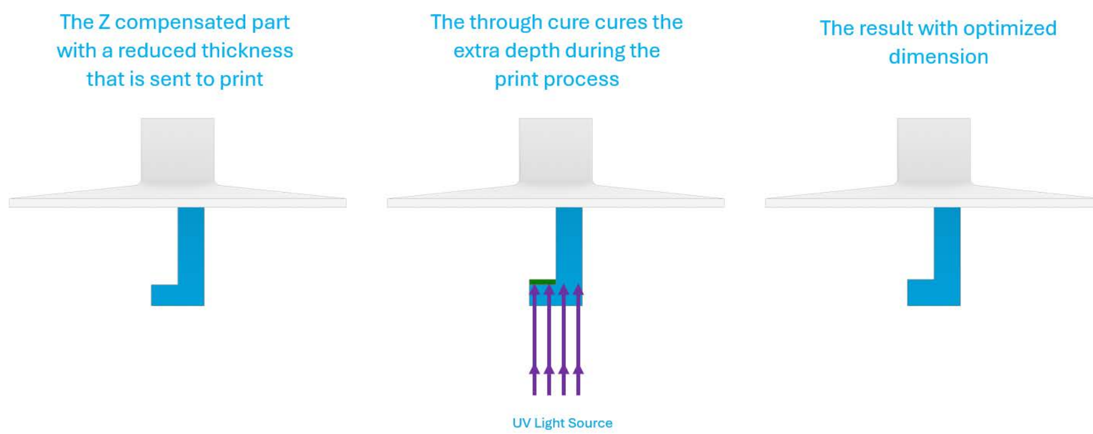
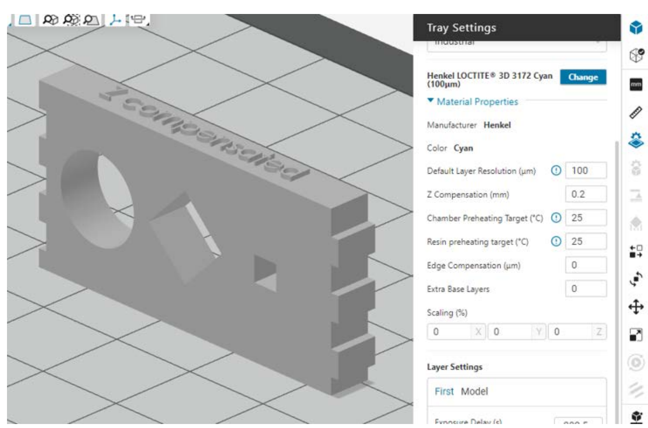



Figure 80: Through cure to optimize the dimensions




For example, the Loctite IND3172 Cyan material has an expected through-cure value of 0.2 mm. To account for this, we enter the same value in the specified field.

Figure 81: Over-curing on the Z-axis in an accurate thickness




> The maximum possible value is constrained by the *Part Penetration* parameter found in the *Support* window. Ensure you adjust this parameter to the desired value.
> This step is necessary even if there are no supports.

This part contains multiple shapes and each of them is impacted by the through-cure. By using the Z compensation, we can optimize the accuracy of the part.

**Nominal dimensions:**

- Circular hole 10-mm diameter
- Diamond hole 10-mm height
- Cubic hole 3-mm height
- Overhangs 3-mm height

Figure 82: Over-curing on the Z-axis in an accurate thickness

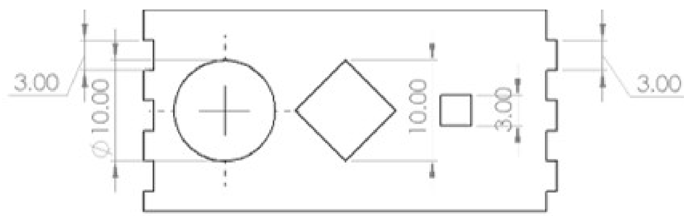



Figure 83: Down-facing features nominal deviation of about 0.18 mm

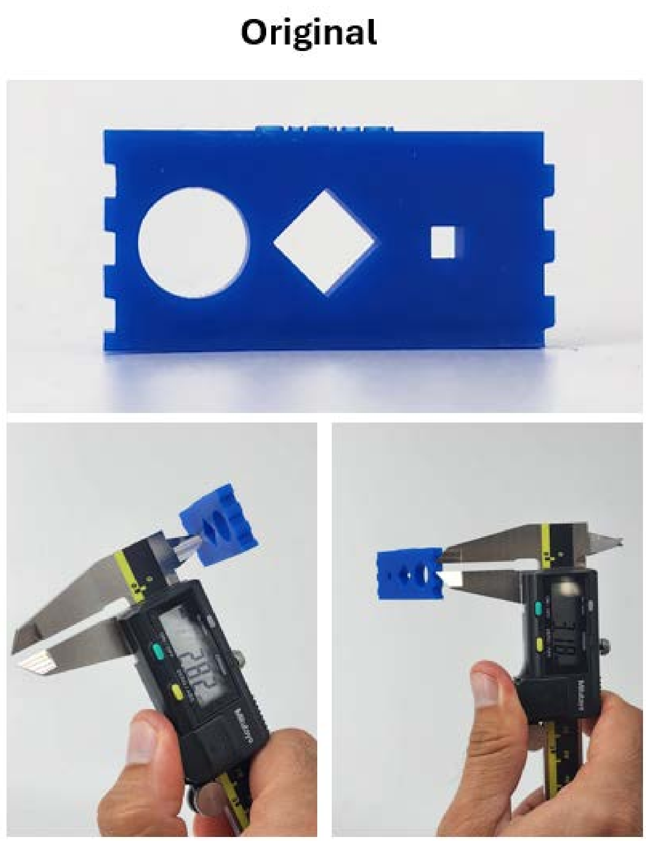
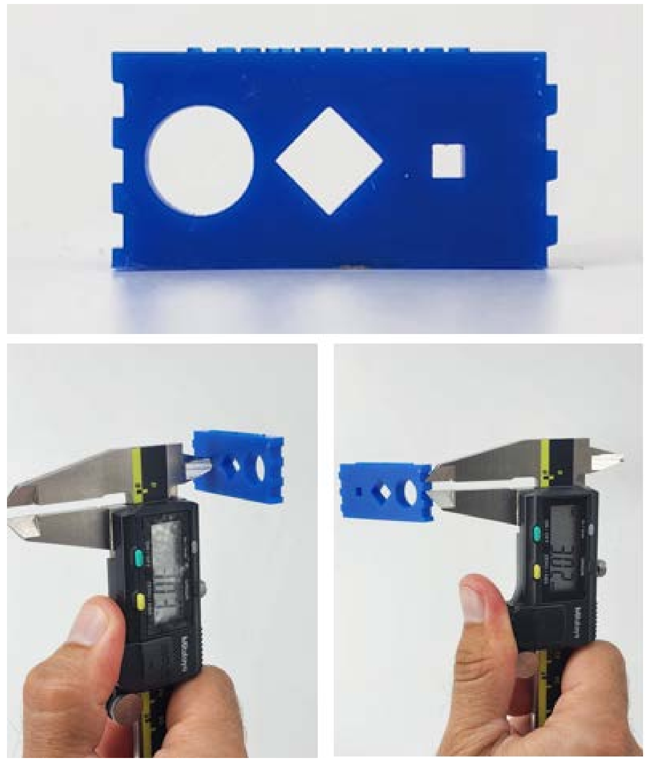


Figure 84: Down-facing features nominal deviation of about 0.03 mm



**Fine features limitation:**

- When printing thin features along the Z-axis, using the Z-compensation tool might eliminate them.
- The model below includes overhangs ranging from 0.05 mm to 0.5 mm in thickness. Applying a Z-compensation value of 0.2 mm resulted in the loss of all overhangs in the range of 0.2 mm thick and under.

Figure 85: Original—All 0.1-mm and above thick overhangs are printed successfully

Figure 86: Modified Z-compensated (0.2)—All overhang that are 0.2-mm thick or thinner were gone due to the Z-compensation

> The 0.05-mm thick overhang was not printed on both jobs due to the fact of printing with a 100-µm layer height.

To mitigate the through-cure effect according to the model geometry:

- Simple geometry—Orient the model to be flat on the build platform, if possible.
- Complex geometry—Orient the model for key features.

Figure 87: Re-orient to overcome through-cure

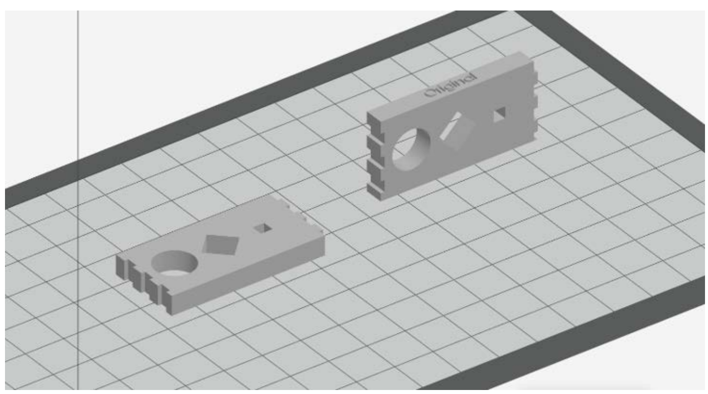



Table 6 shows the expected through-cure for each of our validated and preferred materials. For best results, measure the printed model from the specific printer in use and compensate, respectively.

Table 6: Through-cure measurements by material

| Resin | Expected Through-Cure (mm) |
|---|---|
| Henkel Loctite® 3955 (100 µm) | 0.1 |
| Stratasys® P3™ Dura™56 (100 µm) | 0.2 |

Table 6: Through-cure measurements by material (Continued)

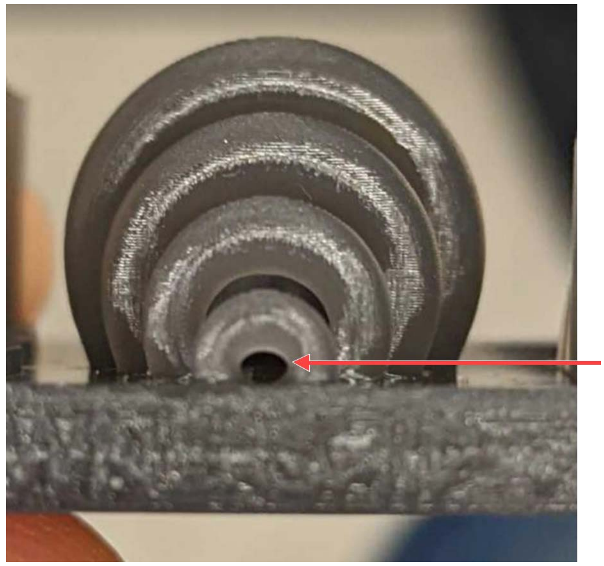
| Resin | Expected Through-Cure (mm) |
|---|---|
| Henkel Loctite® IND3172 Cyan (100 μm) | 0.2 |
| Henkel Loctite® IND3172 Grey (100 μm) | 0.2 |
| Henkel Loctite® IND3843 Black (100 μm) | 0.2 |
| Henkel Loctite® IND3843 Clear (100 μm) | 0.1 |
| Henkel Loctite® IND3843 White (100 μm) | 0.2 |
| Henkel Loctite® IND402 (100 μm) | 0.1 |
| Henkel Loctite® IND403 (100 μm) | 0.2 |
| Henkel Loctite® IND405 (100 μm) | 0.3 |
| Henkel Loctite® MED412 Clear (100 μm) | 0.2 |
| Henkel Loctite® MED413 Clear (100 μm) | 0.2 |
| Henkel Loctite® MED413 White (100 μm) | 0.3 |
| Stratasys® P3™ Deflect™120 (100 μm) | 0.2 |
| Stratasys® P3™ Stretch™ IND475 (100 μm) | 0.3 |
| Stratasys® P3™ Stretch™ 80 | 0.2 |
| Somos® QuickGen™ 500 (100 μm) | 0.1 |
| Somos® WeatherX™ (100 μm) | 0.2 |
| Somos® WeatherX™ (50 μm) | 0.25 |
| BASF UltraCur3D ST45 Black (100 μm) | 0.1 |
| BASF UltraCur3D ST45 Clear (100 μm) | 0.3 |

Printing with a different layer, modified exposure time, or uncalibrated projector will affect the results of the expected over cure.

Two commonly observed phenomena regarding through-cure:

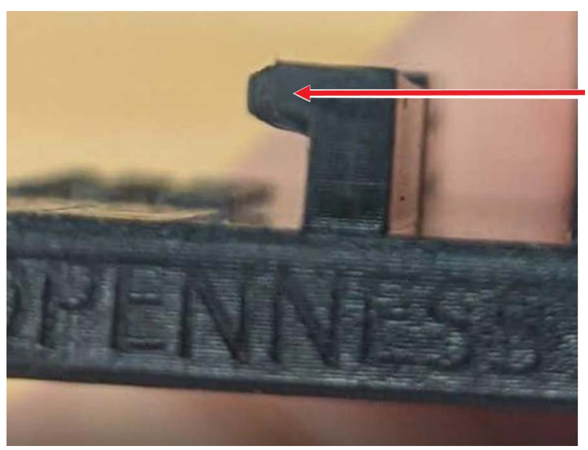
- Horizontal curves that lose roundness, see Figure 88

Figure 88: Circles out of round



- Overhangs that become thicker than designed, see Figure 89 (page 44).

Figure 89: Thick overhangs become big in Z direction

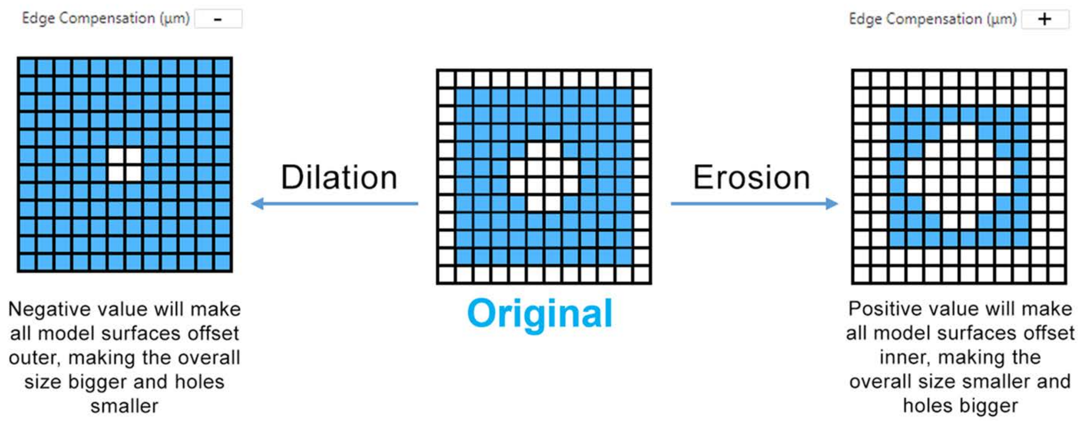


# Edge Compensation

In a 3D-printed model, fine details, such as small holes, can lose accuracy due to excess material around the edges caused by light scattering, incomplete solidification, or surface tension. To mitigate this inaccuracy, use edge compensation, which is a method to offset a specific number of microns from the edges of the projected image of each slice. This ensures precise replication of fine details in the final print.

Follow the minimum through-hole size guidance found in the Material Processing Guide for each material to reduce the chance that fine details lose definition and result in inaccurate size and shape during the liquid photopolymer printing process.

Figure 90: Edge compensation



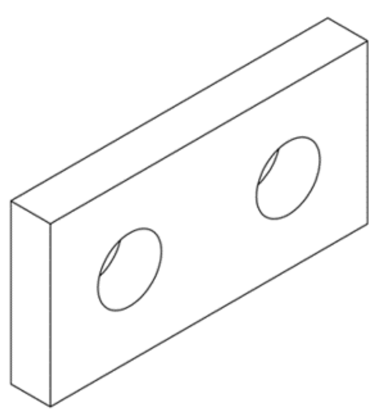

Each square represents a pixel from the projector. The blue represents squares that are illuminated and will cure the resin.

## Edge Compensation Versus Model Scaling

In situations where a model has holes that are either too small or too large, scaling the model proportionally is a viable solution to achieve the appropriate hole size. Unfortunately, this approach results in a shift of the hole locations, which can be undesirable.

Alternatively, edge compensation can be employed to preserve the original position of the hole centers while adjusting their size. By implementing this technique, the holes will remain in the same relative location despite any applied scaling adjustments.

Figure 91: Edge compensation applied



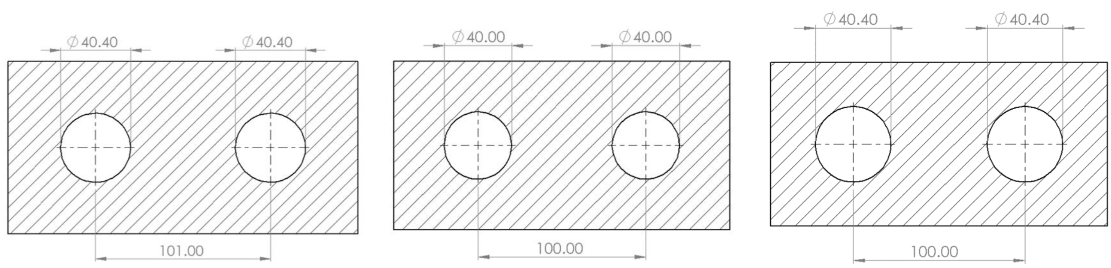
In the Figure 92 (middle), the holes are initially 40 mm in diameter. The objective is to adjust the size and make the holes larger.

In the bottom left picture, the entire image was scaled up to enlarge the holes. However, this resulted in a shift of the hole locations. The gap between the centers of the holes increased to 101 mm, which exceeds the original 100-mm gap.

To address this issue, edge compensation was applied. This technique allows for enlarging the holes while maintaining their original positions.

> **Important:**
> Using edge compensation affects all surfaces, including the overall size of the model. In this example, the model's overall dimensions will be reduced as a result.

Figure 92: Edge compensation applied




| Scaled up. The holes are bigger, but the distance between centers has changed. | Original | Edge-compensated. The holes are bigger, but the distance between centers has not changed |
|---|---|---|

## Original CAD

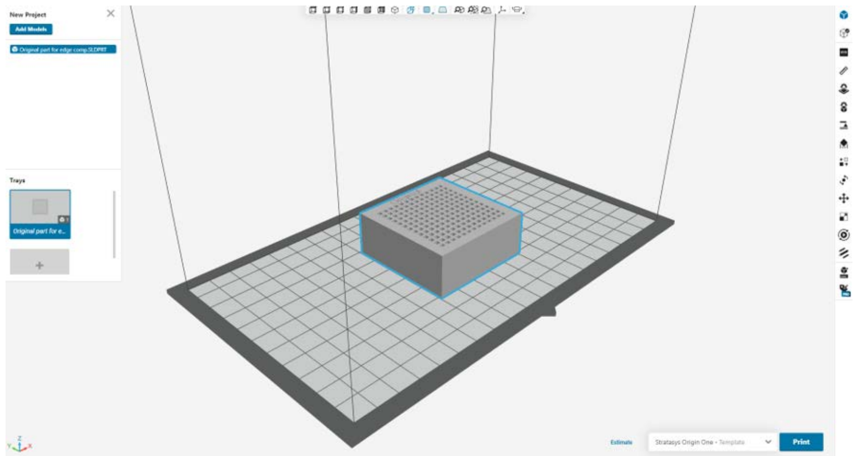
The designed diameter of the holes is 40.40 mm, but the actual printed holes with original scaling have a 40.00-mm diameter. To compensate for the difference, the edge compensation value is set to 200 microns.

A positive value removes pixels from the model contours and therefore, enlarges the holes. As hole diameter is affected by the contour on either end, the hole size in the example above is increased by 400 microns (0.4 mm), which is twice the edge compensation value.

## CAD Example and Results

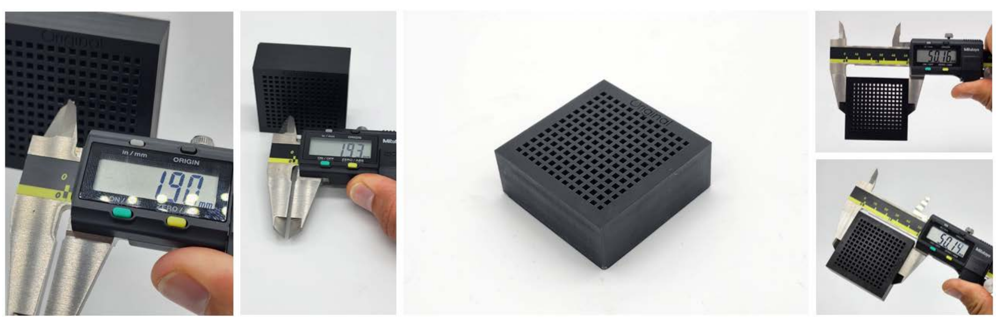
A CAD model is a 50-mm cube on X and Y, with squared holes of 2x2 mm.

Figure 93: CAD example for edge compensation



The designed square holes are 2x2 mm, but the actual 3D-printed sizes range is 1.9-1.93 mm.

Figure 94: CAD example for edge compensation

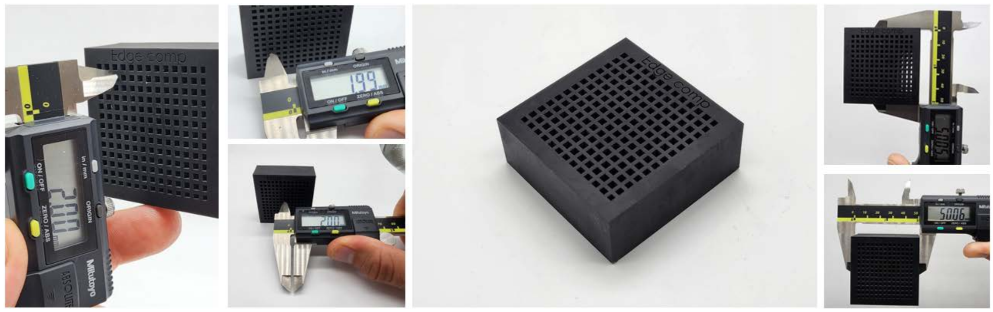


To correct this, we set the edge compensation value to -2, which shifts the surfaces inward by 0.1 mm, effectively enlarging the holes.

In the following images, you can see the results of using the edge compensation feature. We managed to optimize the square hole and print it accurately as per the original design.

Figure 95: Results of using edge compensation



# Extra Base Layer

Z shrinkage occurs in the 3D printing process when liquid resin cures and solidifies into a denser state, reducing the model size. The shrinkage varies based on the geometry, material, and workflow. Typically, the most significant shrinkage is in base region of the model, although it can occur throughout the model.

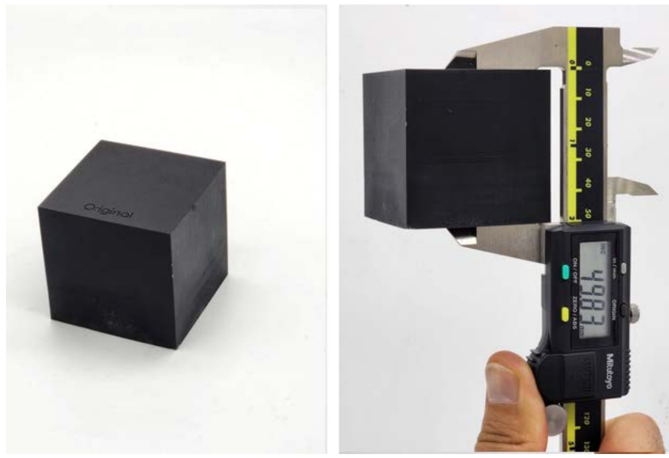
Longer exposure times and higher forces during the first layer and transition region can cause thinner layers, increasing shrinkage in the first 1-2 mm of the print. To overcome this, use the Extra Base Layer tool, which clones the first layer and adds it to the bottom of the model.

This technique can mitigate the shrinkage that typically occurs in the base region, resulting in a more accurate and precise final print.

**Important:**
The shrinkage on the base layers affects only models that are printed directly on the build platform, if the model is held above the build platform with a support structure that is taller than 2 mm, the base region shrinkage is insignificant.

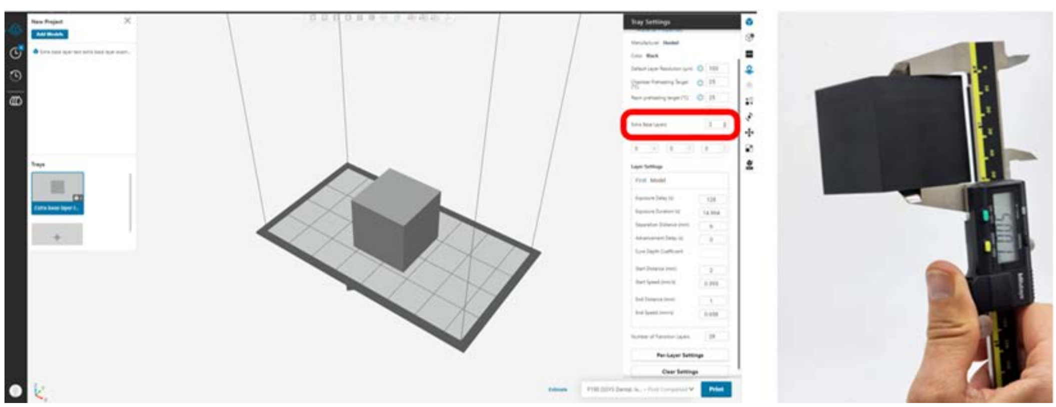
Despite setting the CAD design dimension to 50 mm, the final 3D-printed model measured 49.83 mm on the Z axis. The print was produced using a layer height of 0.1 mm. To compensate for this difference, add two additional base layers.

Figure 96: Z shrinkage




By incorporating two additional base layers, we achieved optimized dimensional accuracy in the Z axis of the printed model.

Figure 97: Extra base layer



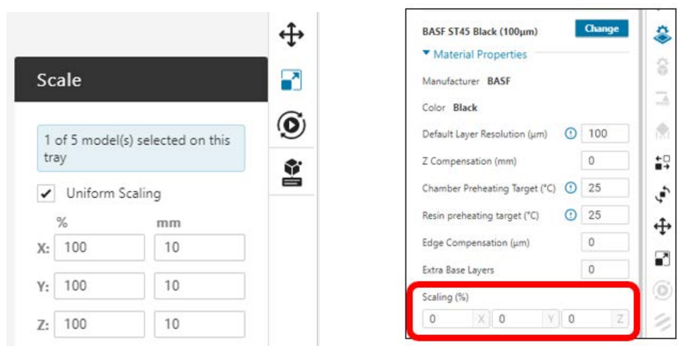
# Material Scaling

The shrinkage of the model beyond the base region (1-2 mm) is proportional to the Z height of the model and varies with different materials and geometries.

For information about the *Scale* tool, see “Scale Tool for Part” (page 62).

Typically, use *Scaling Material* to counteract a shrinkage or growth factor, while the *Scale* tool acts more as a traditional scale feature, see “Key Differences Between Material Scaling and the Scale Window” (page 64).

Figure 98: Scale window and Material Scaling dialog




Stratasys-certified material profiles incorporate X, Y, and Z shrinkage compensation factors, but the actual shrinkage varies with different geometries. The *Scaling (%)* option in GrabCAD Print allows you to adjust the scaling factor for the entire job, and then fine tune shrinkage as needed. This tool enables you to:

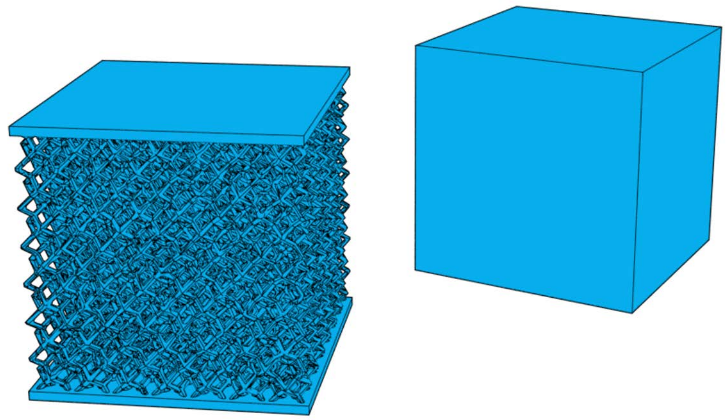
- Increase the scale up to 1% of the size
- Decrease the scale down to 0.5% of the size.

OpenAM users have a wider range of modifications:

- Increase the scale up to 2% of the size
- Decrease the scale down to 1% of the size.

This is because they explore and develop material profiles, which requires more meaningful modification than fine-tuning a validated or preferred material.

Figure 99: Material scaling



## Original Dimensions Printed

To scale a printed model along the Z-axis, first measure the Z-axis height. In this example, the printed Z-axis height is 50.22 mm. Next, divide the nominal Z value by the measured Z value to get the Z scale factor: 50 / 50.22 = 0.995.

> In this example, the Z height is higher than designed, which is different from as opposed to the model shown in the “Extra Base Layer” (page 48) section. The dimension of a model is derived from material and geometry.
>
> The lattice cube was printed flat on the build platform, and despite that, the separation forces during printing and the material post-processing, resulted in a higher Z height.

The Z-axis height of the printed model is measured at 50.22 mm. To calculate the necessary modification, do the following:

1. Divide the nominal Z value by the measured Z value to get the Z scale factor.

   Therefore, we divide the correct dimension by the measured printed result: 50 / 50.22 = 0.995

2. Enter the percentage you would like to increase or decrease.

   For example, to scale down a model to 99.82% of its original size, set the scaling percentage to -0.18%.

   The following show the percentages to scale down:

   - For X: 50 / 50.09 = 99.82—

     Scale down by -0.18%
   - For Y: 50 / 50.06 = 99.88—

     Scale down by -0.12%
   - For Z: 50 / 50.22 = 99.562—

     Scale down by -0.438%

Figure 100: Entering percentages to scale up/down (after calculating)

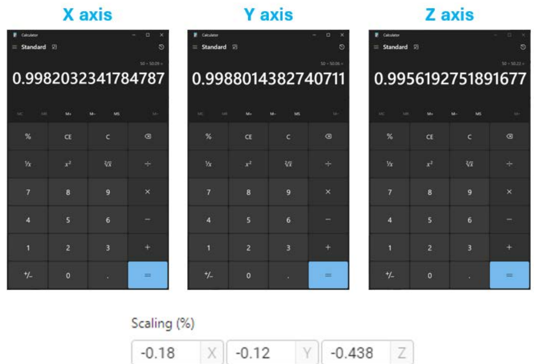
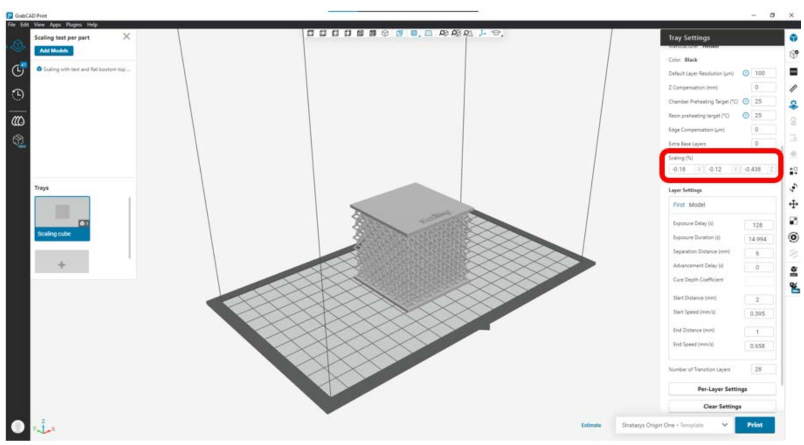
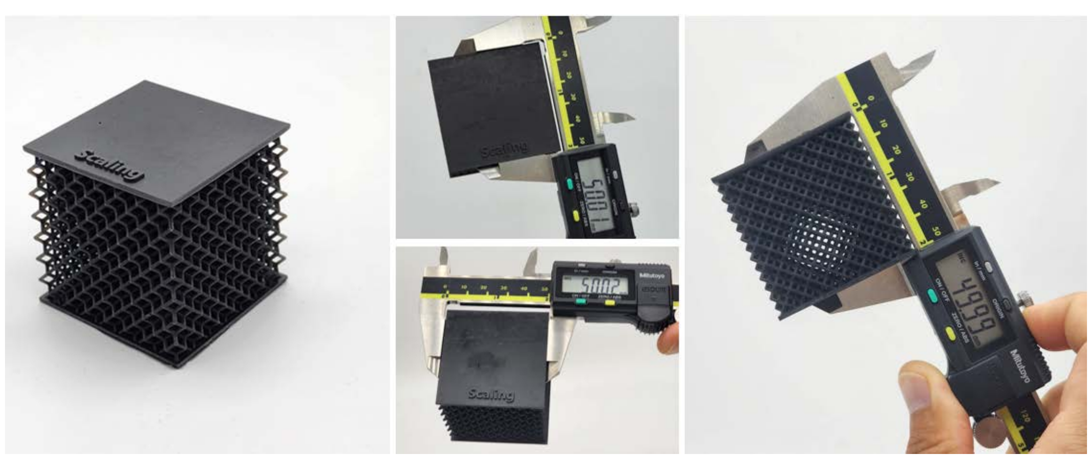



Figure 101: Material scaling in GrabCAD Print



Any changes in the material scaling is not visible on screen.

Figure 102: Scaled printed model



# Origin Printer Specifications for Stratasys Materials

By understanding the printer specifications that relate to Stratasys Validated and Preferred materials, you can better predict which materials will work well with the printer and how to optimize settings for each material. This enables you to select the right material for specific models to ensure that you achieve the optimal print quality and efficiency, see Table 7.

- Accuracy capabilities are dependent on the material and model geometry.
- Specifications are valid for less than 5 mm wall thickness, for bulky models, and require the addition of thermal post-cure according to the material processing guide of the specific material.
- To achieve the best accuracy for high-precision parts, use accuracy optimization tools in GrabCAD Print.
- Optimal model accuracy is achieved at the center of the model.

Table 7: Origin Two Printer Specifications for Validated & Preferred Materials

| Stratasys Material | Material Type | XY |
|---|---|---|
| BASF P3™ Stretch™ 80 | Validated | ± 90 µm up to 25 mm +1.5 µm per additional mm |
| Loctite® FST 3955 | Preferred | ± 100 µm up to 25 mm +1 µm per additional mm |
| Loctite® IND402 | Preferred | ± 90 µm up to 25 mm +1 µm per additional mm |
| Loctite® IND3843 Black/White/Clear | Preferred | ± 100 µm up to 25 mm +1 µm per additional mm |
| Loctite® IND3172 all colors | Preferred | ± 75 µm up to 25 mm +1.5 µm per additional mm |
| Uncur3D ST45 | Preferred | ± 75 µm up to 25 mm +1.5 µm per additional mm |
| P3™ Deflect™ 120 | Preferred | ± 75 µm up to 25 mm +1 µm per additional mm |

Table 7: Origin Two Printer Specifications for Validated & Preferred Materials

| Stratasys Material | Material Type | XY |
|---|---|---|
| Loctite® P3™ Dura™56 | Preferred | ± 75 µm up to 25 mm +1 µm per additional mm |
| Somos® WeatherX™ 50 µm/100 µm | Preferred | ± 80 µm up to 25 mm +1.5 µm per additional mm |

# Best Practices

## How to Choose the Right Model for Additive Manufacturing (AM)

When considering whether or not to use additive manufacturing to create end-use models, consider existing production models and identify use cases where it is cost-effective and more efficient to print the models.

Some general considerations include:

- *Customization and Stock Keeping Unit (SKU) variation*—Print models and products customized to users, companies, or specific customer segments. These include automotive connectors, footwear, or dental products.
- *Reducing weight*—Increase product life, reduce material usage, fuel, and maintenance costs with lattice structures and topology optimization.
- *Tight tolerances and fine features*—Improve fit and function with tolerances that exceed those of typical injection molded or CNC models. Typically, small, accurate polymer models are more suitable for additive manufacturing.
- *Low-volume production*—Avoid or postpone the high fixed cost of injection molds by printing models directly.
- *Time to market*—Launch products faster and incorporate feedback quickly to accelerate innovation.

### Before You Print Checklist

When identifying a model or design to take advantage of additive manufacturing, it’s important to answer these questions first:

- Is a suitable material available for the application?
- Will the model fit inside the build volume?
- Are there any features or walls smaller than 200 µm (0.2 mm)?
- Are there overhangs?

- Are supports needed on any critical surfaces?
- Are there areas where resin can't escape?

## Maximal Actual Part Volume

Due to material shrinkage, the printed parts cannot reach the full printer build volume. Therefore, in Origin Two and Origin One+, you can only send to print a part whose part size is a bit smaller than the full printer build volume, representing the actual part size that will be printed.

Origin Two and Origin One+ use scaling factors in the material profile to compensate for the shrinkage and ensure model accuracy.

The dimensions are as follows:

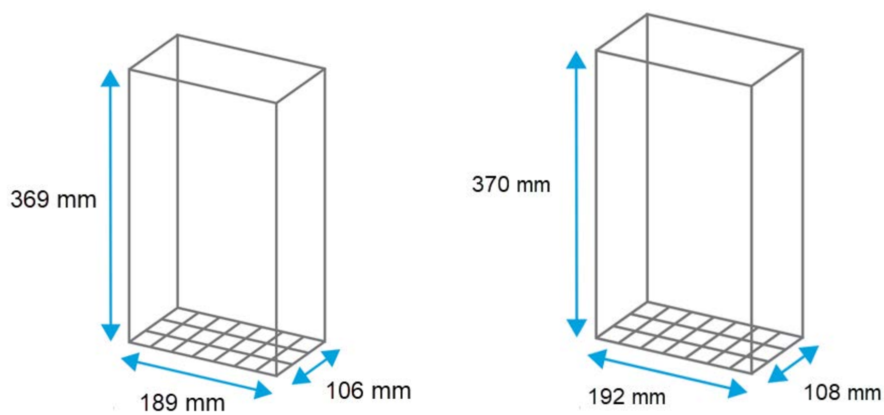
- All Origin printers have the same full printer build volume:

  192 mm x 108 mm x 370 mm (7.56 in x 4.25 in x 14.57 in).
- The maximal actual part volume for all Origin printers is:

  189 mm x 106 mm x 369 mm (7.44 in × 4.17 in × 14.52 in).

  Therefore, this is the part size you can send to print on the Origin Two and Origin One+ printer. In Figure 103, the maximal actual part volume is on the left and the full printer build volume is on the right.

Figure 103: Left: Maximal part volume; Right: Full printer build volume




# Choosing a Material

After the model to be printed has been identified, the next step is selecting the right material. Review the Stratasys Origin material portfolio and each material's physical properties to select the best one for the application.

Various materials require specific printer settings. Typically, rigid materials can handle thinner walls compared to soft materials and will require fewer supports.

The Origin printer is highly configurable. For example, long and thin models made from a soft material may require a longer delay between exposures. The printer settings can be easily modified to accommodate for this requirement.

Table 8: Material and properties

| Material Category | Description | Properties | Unique Print Considerations |
|---|---|---|---|
| Rigid | Rigid materials typically have the highest green strength, so they become mechanically strong during the print process and can also print faster. | • Stiffness<br>• Strength<br>• Toughness<br>• Thermal resistance | Rigid materials achieve the smallest feature sizes, biggest overhangs, and greatest detail, with minimal supports. |
| Light Tough<br>Ultra Tough | Tough materials live between rigid and elastomeric. With moderate moduli and increased elongation, these materials demonstrate high impact strength. | • Strength<br>• Toughness<br>• Moderate flexibility<br>• Moderate elongation | Tough materials require moderate supports on overhangs, yet are still capable of achieving fine detail. |
| Elastomers | Elastomers are stretchy and highly flexible. The flexible nature of these materials requires the most supports and slowest print speeds. | • High elongation<br>• Highly flexible<br>• Resiliency | Elastomers require the most support on overhangs and tall models, to ensure geometric accuracy and model quality. |

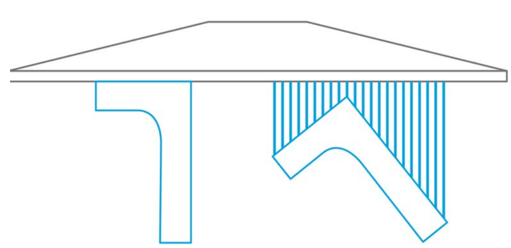
# Model Orientation

Model orientation (Figure 104) is important to consider when designing a model and preparing a print as it directly impacts the final model in several ways. Orient the model so that it is stable.

For build platform adhesion and model support, the important questions are:

- Is there a flat surface that can be placed on the build platform?
- If not, is there adequate surface area attached to the build platform?
- Which surfaces need to be support-free?
- Is there an orientation that requires fewer supports?
  - The part on the left is oriented directly under the build platform and has a stable center of mass, so it doesn't require supports in the this orientation.
  - The part on the right is oriented such that it needs supports.

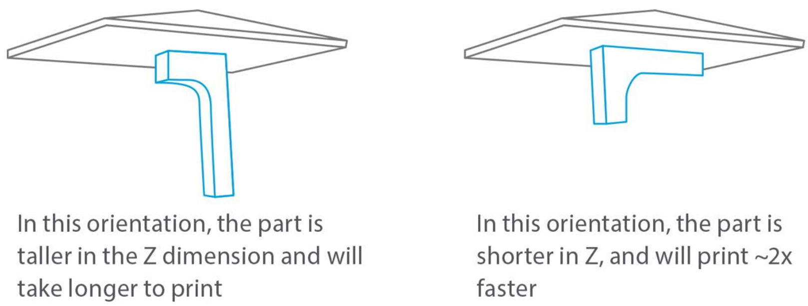
Figure 104: Model orientation



## Orientation for Print Time

Print time on an Origin printer is primarily dependent on material and the Z dimension height of the model. Compared with other additive technologies, geometry is less of a factor in print time, see Figure 105.

Figure 105: Orientation versus time

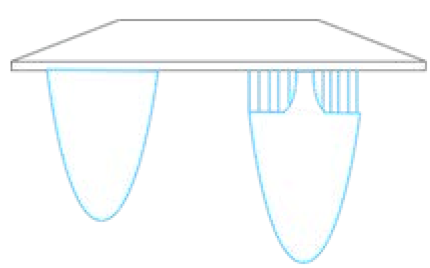



## Orientation and Separation Forces

Separation forces occur between layers, and are caused by the build platform moving vertically in the resin during the build process. After each layer is exposed, the build platform raises. This allows new resin to flow in preparation for the next layer. When the build platform moves up, the newly cured layer is pulled off the membrane. The Origin printer uses a unique proprietary separation mechanism that drastically reduces these forces; however, if the part is not firmly attached to the build platform, it can be damaged or completely fall off during printing. When orienting the part, it is important to attach enough material to the build platform to support the part for the duration of the build.

- If the part is designed such that it has a planar face that can be placed directly on the build platform, orient the part that way.
- If the part has a planar face that could be placed directly on the build platform, but the area of that face is too small to hold the entire mass of the part, place additional supports around that face to provide a supplemental support structure.
- If there is no planar face that can be oriented toward the build platform, create enough supports to attach different features of the part to the build platform. The supported area of the part must be strong enough to keep the part attached to the build platform during the entire build process. See Figure 106.

Figure 106: Supports to mitigate separation forces



## Orientation for Surface Finish

The Origin printer is capable of printing parts with high accuracy, small features, and a beautiful surface finish. Surface orientation can affect the surface finish, and while the differences are subtle, depending on the use case, they may matter.

If a surface has to be as smooth as possible, the best way to achieve this is to keep the surface horizontal (parallel) to the build platform. Surfaces oriented towards the build platform will appear smoother than surfaces oriented away from the build platform. Another option is to decrease layer thickness because thinner layers result in a better surface finish. The trade off to this is that print times will increase.

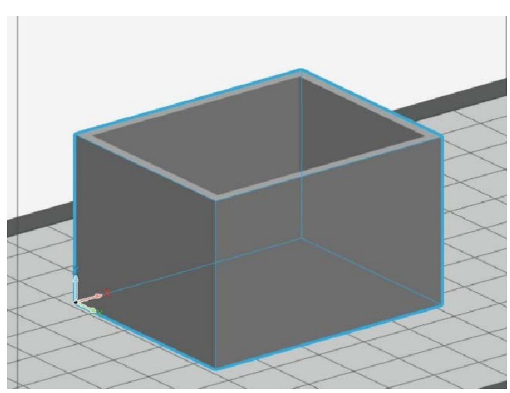
The next best surface quality will be a curved or flat part that is angled to the build platform; however, any part printed at an angle will show subtle layer lines. The most challenging surface orientation is a flat surface that is perpendicular to the build platform. This orientation will result in subtle layer lines in the Z direction.

Figure 107: Orientation versus surface quality


## Orienting a Model by Adjusting the Self-Supporting Angle

You can orient the model by the *Self-Supporting Angle*. For models such as a box shape, you can minimize the support structure (either created automatically or manually). For information about the Self-Supporting Angle, see “Self-Supporting Angle” (page 19).

Figure 108: Box-shaped model



For the same model, different orientations affect the number of automatically-added anchor points, as described in the following examples:

- For a *Rigid* pre-defined support profile, the model is oriented on a tip and one anchor point is created automatically.

  For a successful print, many anchor points need to be added manually, see Figure 109.

- For a *Rigid* customized support profile, the model is oriented at 35° and many anchor points are created automatically.

  For a successful print, there can be a minimal number of anchor points added manually, which requires less manual effort, see Figure 110.

Figure 109: Rigid support profile—Pre-defined–oriented on tip

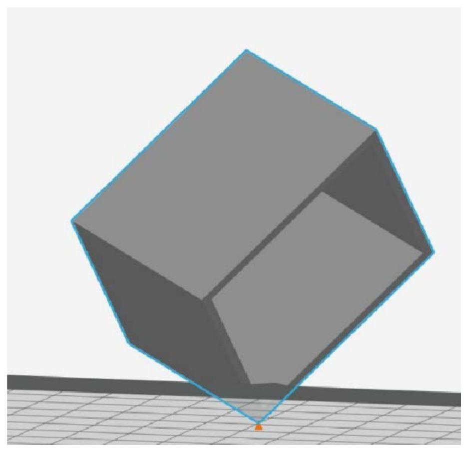
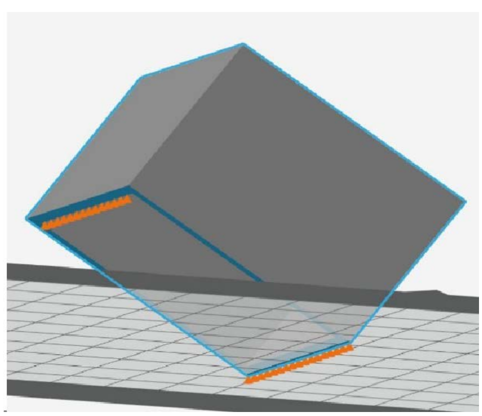


Figure 110: Rigid support profile—Customized–oriented to 35°



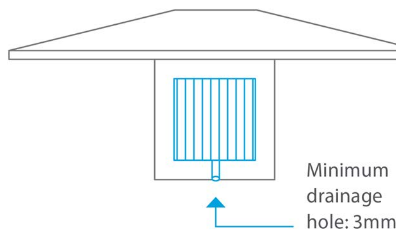
The following table summarizes the examples:

Table 9: Same Model with Different Orientations

| Support Profile | Orientation | Anchor Points Added Automatically | Missing Anchor Points | Example |
|---|---|---|---|---|
| Rigid – Pre-defined | Oriented on a tip | One | Many | Figure 109 |
| Rigid – Customized | Oriented at 35° | Many | Fewer | Figure 110 |

## Enclosed Volumes

Figure 111: Enclosed volume




An enclosed volume will not allow resin to escape during the printing process. This will result in a failed build due to uncured resin being trapped in the final part. Create a resin drain hole in all parts with enclosed volumes.

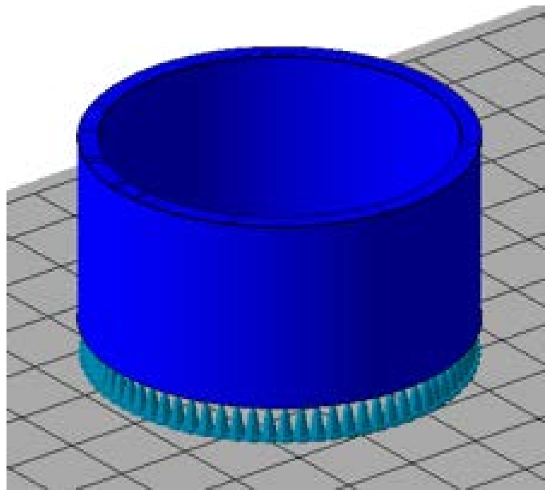
## Unvented Volumes

A part with an unvented volume has a hollow region that creates a pressure differential depression between the printed part and the build platform. Unvented volumes inhibit needed resin flow and create suction forces, which may lead to part failures.

Without changing the design, the only way to print this model successfully is to add supports. Otherwise, you can design a small vent hole to break the vacuum in the part or reorient the part to avoid the vacuum when the vent hole cannot be implemented. When you change the design, position vents so that they are formed early in the print.

In this example, the part above the supports is a hollow structure that creates an unvented internal volume. No matter which way the part is oriented, there will still be an unvented region. In this case, raise it on supports with vented sides. These will eliminate the pressure differential.

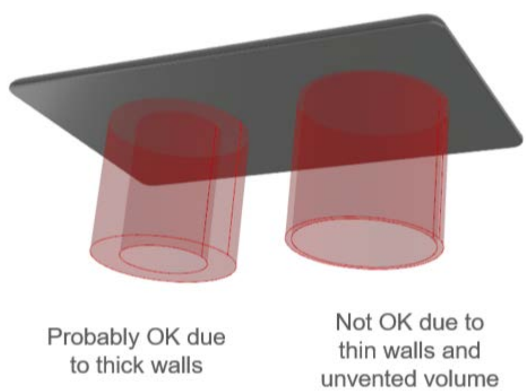
Figure 112: Unvented volumes



In these examples, both models are unvented and the wall thickness can determine if venting is necessary:

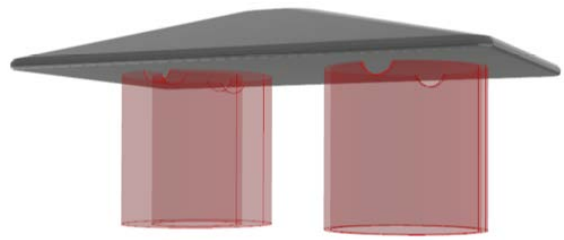
- Models with thick walls may not need venting, depending on the thickness, see the example on the left.
- Models with thin walls need venting, as shown in the example on the right.

Figure 113: Unvented examples



In these examples, both models are vented:

Figure 114: Vented examples



OK due to venting

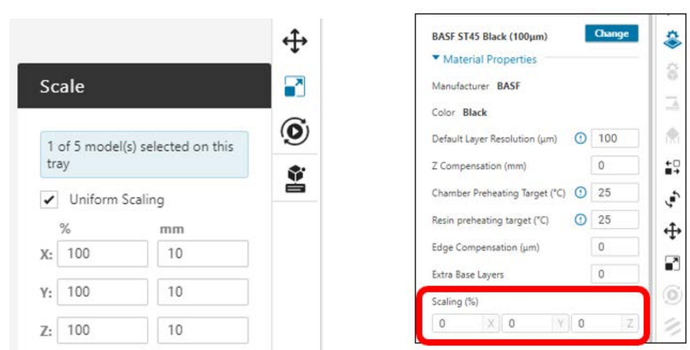
# Scale Tool for Part

If the build platform has multiple parts and you want to adjust the scaling for each part individually, you can do so by using the *Scale* tab. The settings function in a similar way as the *Scaling* option in the *Tray Settings tab* ("Material Scaling" (page 50)), but has more ways to manipulate scaled parts.

For information about the *Material Scaling* tool, see "Material Scaling" (page 50).

Typically, use *Material Scaling* to counteract a shrinkage or growth factor, while the *Scale* tool acts more as a traditional scale feature, see "Key Differences Between Material Scaling and the Scale Window" (page 64).

Figure 115: Scale window and Material Scaling dialog




- To modify a part, select the desired model and adjust the scaling accordingly.

  By default, the *Uniform Scaling* checkbox is checked. Multiple parts on the build platform can be scaled differently.
- To adjust the scaling of individual parts on a build platform containing multiple parts, access the *Scale* tab.

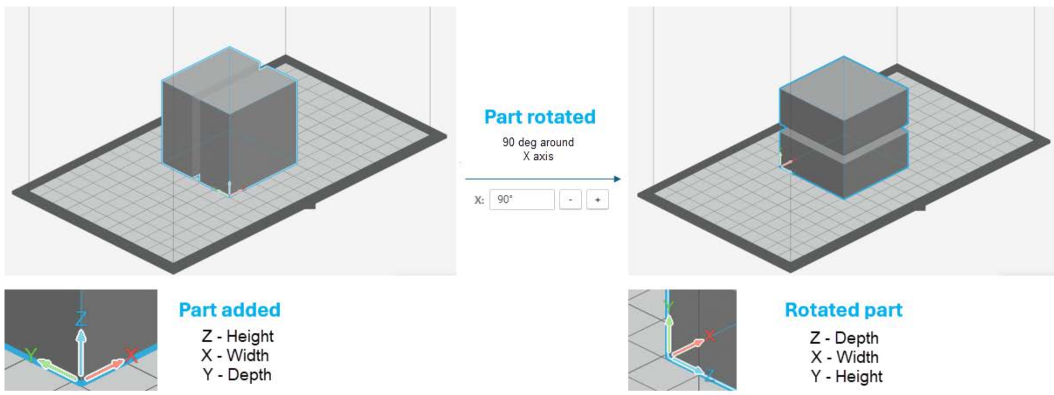
  

## Understanding the GrabCAD Print Orientation Logic

When a part is added to GrabCAD Print, it receives the XYZ axes according to the imported orientation. Rotating it inside GrabCAD Pring won't affect the axis designations. The axes of the part rotate as well.

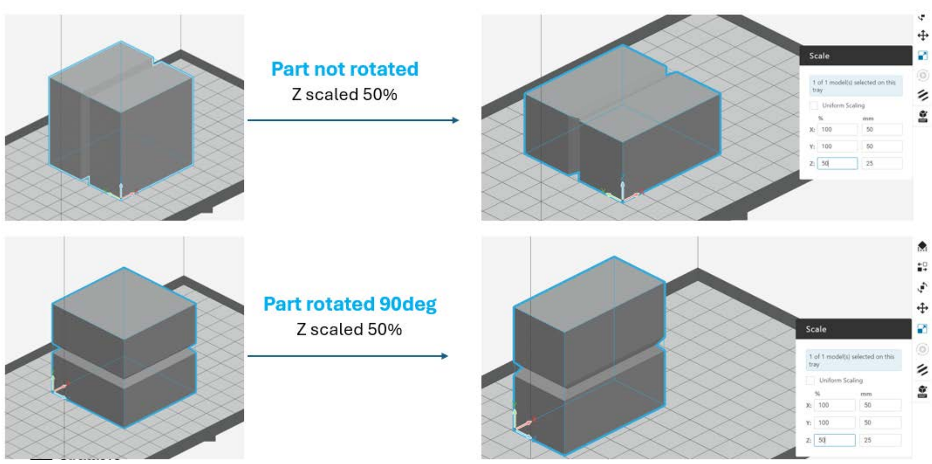
> In practice, if you rotate a model and scale it, make sure you're modifying the intended axis.

Figure 116: Rotate part




The following examples show the effect of the *Scale* tool on the Z axis for a cube that wasn't rotated compared to one that was rotated:

Figure 117: Part not rotated versus rotated part



### Key Differences Between Material Scaling and the Scale Window

- The definition of the axis is not the same for the Material Scaling and the Scale tool:
  - The Material Scaling is based on the tray axis.
  - The *Scale* tool is based on the part axis.

> **Important:**
> The axis designations does not change even when the part is rotated.

- *Material Scaling* affects all the parts that will be printed, while the *Scale* tool can be applied per part modification.
- When using the *Scale* tool per part, the changes are seen on screen.
- The *Scale* tool percentage is displayed in 100%.
- The *Scale* tool also displays the percentages as measurement units.
- *Scaling Material* has a range of +1% to -0.5% and, while the **Scale** tool doesn't have limits.

See also: "Material Scaling" (page 50)

## Theory of Support Requirements

### Support Considerations

The nature of bottom-up photopolymer systems requires supports for some geometries, such as overhangs, islands, and undercuts. Supports are removable, disposable structures that stabilize features like overhangs, as well as any feature that is not supported by the part geometry itself. Engineers and designers can reduce the amount of support required by creating flat surfaces that can be directly adhered to the build plate or by creating self-supporting geometries.

### Supports Defined

Supports are printed material that is used to provide structural integrity to the part during the print process so that the print is successful. In Digital Light Processing (DLP) printing supports are formed as a structure of thin features printed in the same material as the model.

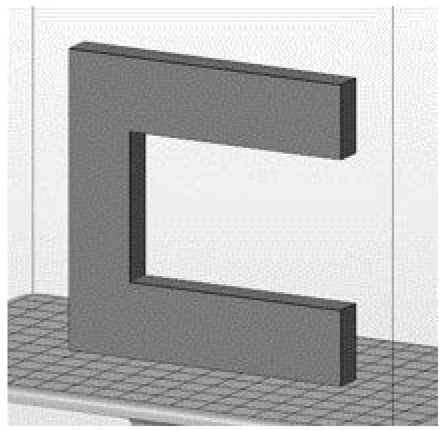
### When Supports Are Needed

- Supports are needed for large overhangs, bridges, islands, and shallow angles. The dimensions of these features that need supports are material-dependent.
- If a part or feature is heavy, robust supports are needed to keep it stable and secured to the build platform throughout the print.
- If a part or feature has large cross-sectional areas, separation forces will be larger, so robust supports are needed to keep the part stable and secured to the build platform.
- Sometimes tall and thin parts may need supports to keep the part from wobbling during print.

- It is best to reduce or eliminate supports whenever possible to improve surface quality and reduce post-processing.
- Some parts, in certain orientations, will need supports. Others are self-supporting.
- Sometimes changing the orientation or changing the design will make a part self-supporting.

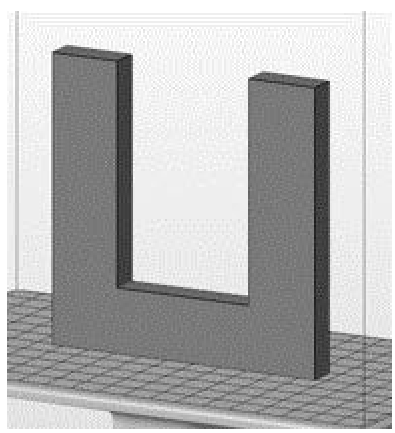
In the example below, the overhang is too large and will not print.

Figure 118: Large overhang



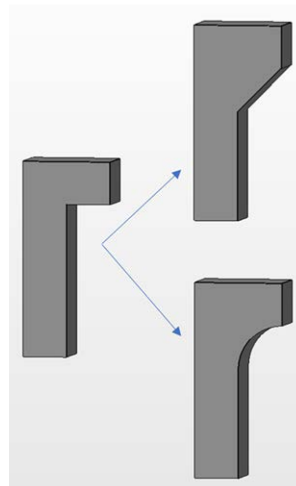
The orientation can be changed so that no supports are required.

Figure 119: Changed orientation



If the orientation can't be changed due to certain constraints, sometimes altering the design will work for removing required supports. The overhang below can be printed without supports by using a chamfer or fillet the overhang.

Figure 120: Altered design



## Structures Requiring Supports

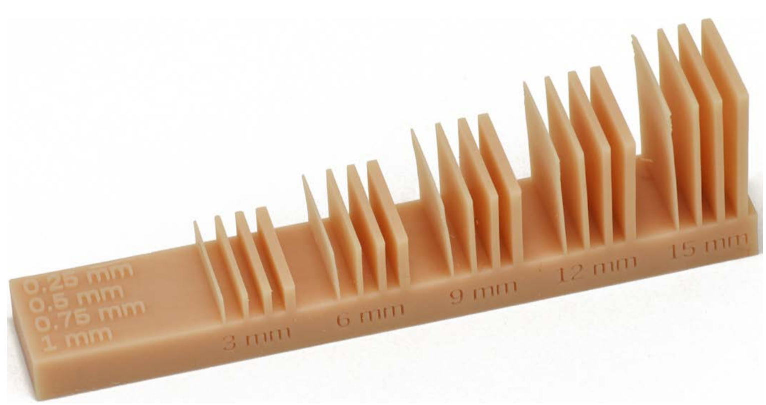
The sections below describe instances where supports are required. They include critical angles, overhangs, bridges, islands and unvented volumes. Material specific capabilities for unsupported geometries are defined in the Material Processing Guide for each material.

### *Unsupported Wall Thickness*

Unsupported wall thickness is dependent on wall height, aspect ratio, and print direction, as well as material type.

- Minimum: 0.25mm
- Recommended: 0.5mm

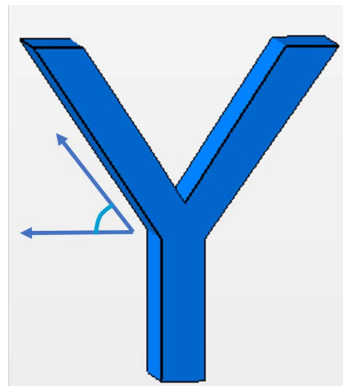
Figure 121: Unsupported wall thickness



## Critical Angles

Each material has a different critical angle at which it can print. This is the minimum angle from horizontal that does not require supports. Any angle below this will require supports. This feature is sometimes called an angled overhang.

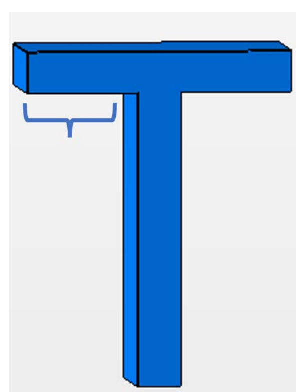
Figure 122: Critical angle



## Overhang

An overhang is a feature on one side of a part that is supported by the part itself. A horizontal overhang is perpendicular to the build platform that does not have a feature to hold it up. See Figure 123. Each material has a short horizontal distance from the part that can print successfully. For example, the maximum unsupported overhang length for ST45 Black is 1mm. Any overhang beyond this distance will require supports.

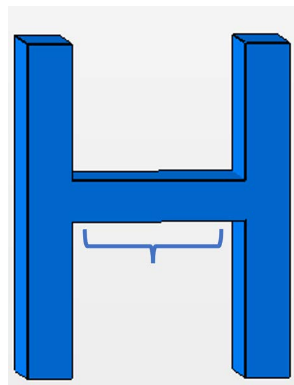
Figure 123: Overhang



### *Bridge*

A bridge is a feature that is supported on its ends but not in the middle. Bridges can be longer than horizontal overhangs but may droop as the span becomes too big. Each material has a short horizontal distance from the part to part that can print successfully. Any bridge wider this distance will require supports.

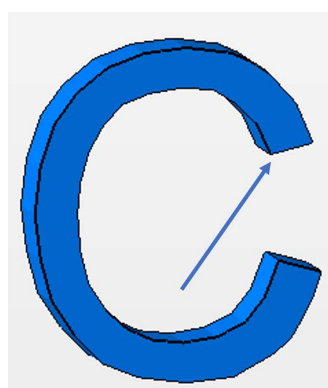
Figure 124: Bridge



### *Island*

An island is a feature that is completely unsupported. When the print gets to the first layer of an island, the newly exposed island region will not have anything to secure it during separation. The island region will stick to the glass and cause the print to fail.

Figure 125: Island



## If Supports Aren't Used

If supports aren't used where necessary, the part can fail in various ways. These include misaligned layers, cured resin stuck in the tray, a partially printed model fallen from the build platform, warped or missing features, or dimensional accuracy problems.

## Using More Supports

Supports can properly attach all of the features to the model or the build platform. The supports will also hold these features in place when they are pulled off the membrane after each layer is printed. Supports also provide structural integrity and stability during print. This enables a successful print and better model accuracy.

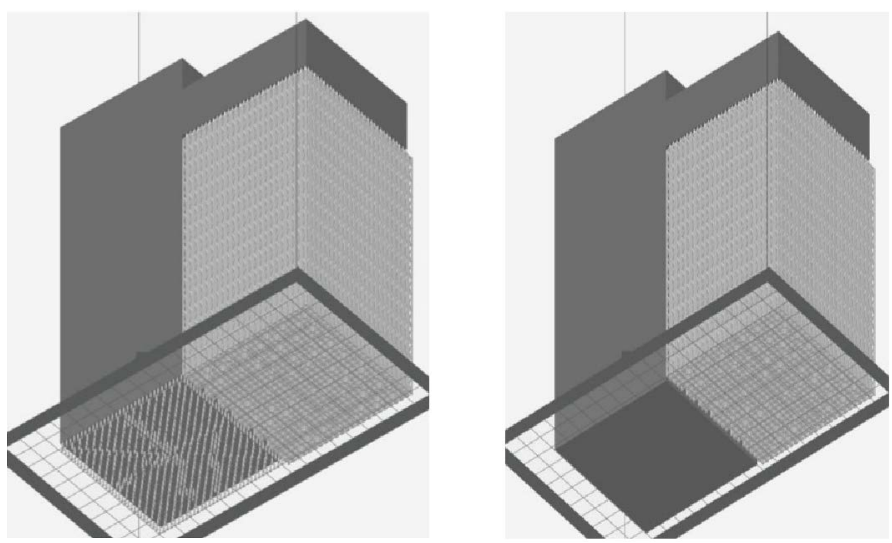
## Using Fewer Supports

Supports must be removed afterwards. They leave small blemishes on the part that must be tolerated or subsequently removed by post-processing the part. Supports also consume additional material.

# Printing Directly on the Build Platform

For printing big bulky models directly on the platform, create a customized profile and set the *Min Height Above Plate* = 0. For information about the *Min Height Above Plate* setting in the *Anchor Points*, see "Distance from Build Platform" (page 25).

Figure 126: Pre-defined support profile-4 mm (left); Customized support profile-0 mm (right)



# Adding Support to Sharp Islands

An island is the first layer of a model that is raised above the platform.

Extra Island Support parameters can be configured, as described in “Extra Island Support” (page 21).

When adding anchor points automatically, an anchor point may be placed only on the first slice, instead of adding more anchor points. This could result in inadequate support in islands, which affects print quality. The following 2 examples result in only one automatically generated anchor point at the tip:

- The angle between the model and the build platform is greater than the *Self-Supporting Angle*
- The overhang length is greater than the *Overhang Offset*

To overcome these challenges, after the support anchor points are generated, examine the islands to ensure they are sufficiently supported. If necessary, add anchor point, as described in “Optimizing Anchor Points Manually” (page 22).

Figure 127: Default support generation – anchor point added automatically (left); generated support structure (right)

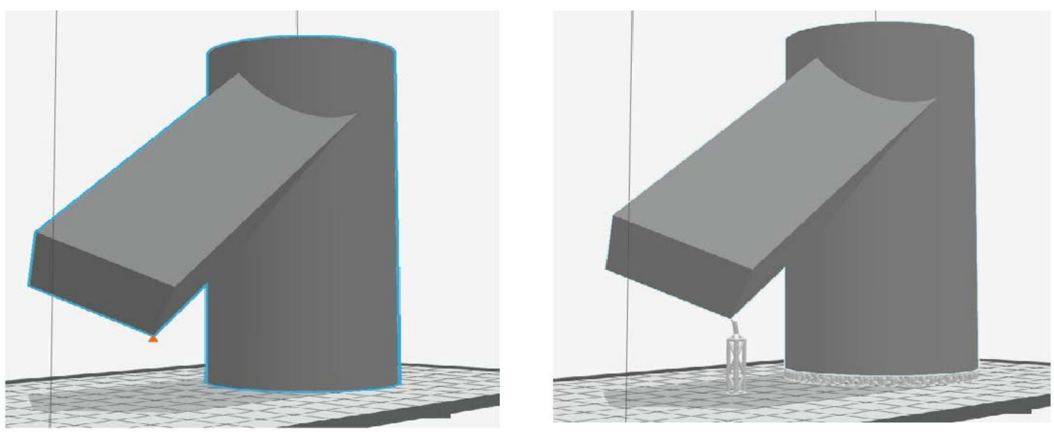
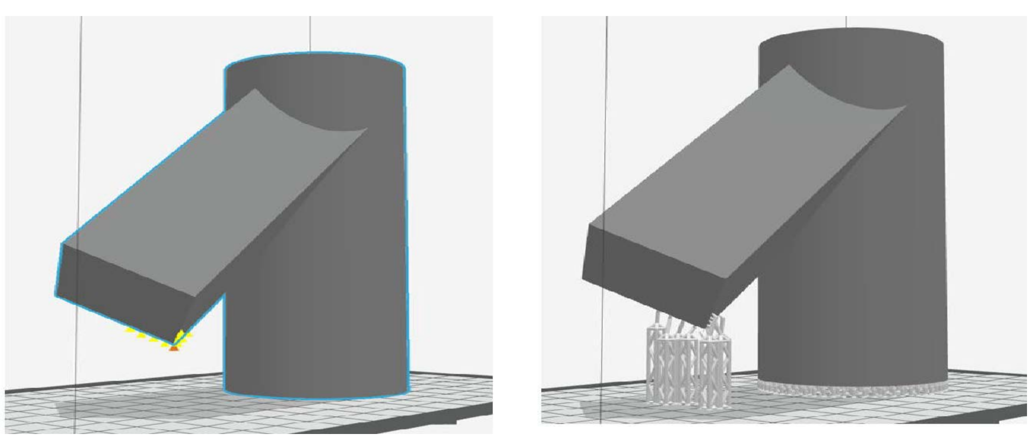


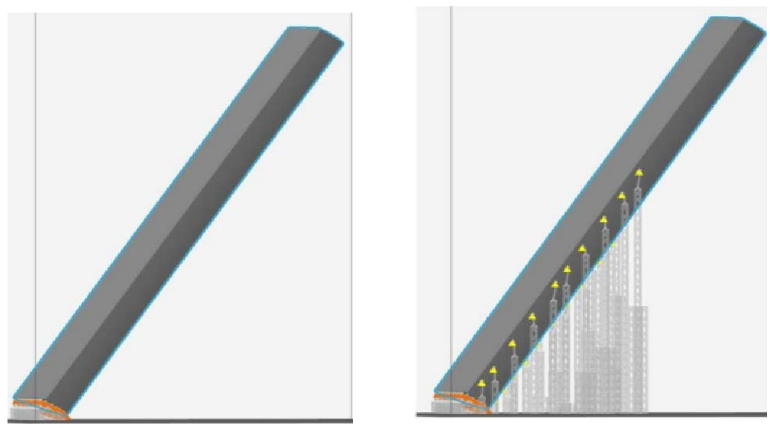
Figure 128: Manually added anchor points (left); improved support structure generated (right)



# Stabilizing High Models

For models that are high above the build platform, add anchor points manually, as described in "Optimizing Anchor Points Manually" (page 22). Anchor points can stabilize a tall structure. This minimizes the likelihood of model quality degradation or failure, at the expense of potential surface damage at the contact points.

Figure 129: Default support generation (left); Manually added anchor points (right)



# Using the Part-to-Part Support Tool

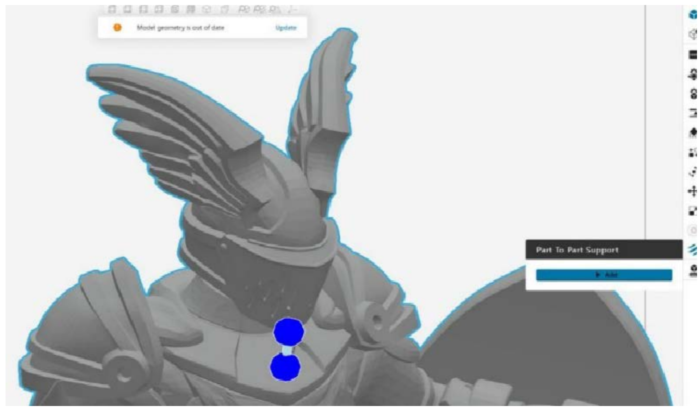
Use the *Part-to-Part Support* tool to manually add struts between faces of a model. The automated process generates a support structure that prioritizes building a support base on the build platform, extending to the anchor points on the model. This usually results in a nicer surface, which may minimize the need for support removal from the model. In some cases, it is preferable to use a part-to-part support strategy, which connects 2 anchor points on 2 faces of the same model.

| |
|---|
| Part-to-part supports cannot connect two separate 3D models. |

1. Select the model.
2. In the right toolbar, click the *Part to Part Support* tab.
3. Click **Add**.
4. Click the part to mark the area that needs support.

5. Click the part to mark where the support should reach.

Figure 130: Part-to-Part Support




6. Click **Update** to create it.

Figure 131: Support strut created by the manual Part to Part Support tool

## Deleting a Part-to-Part Support

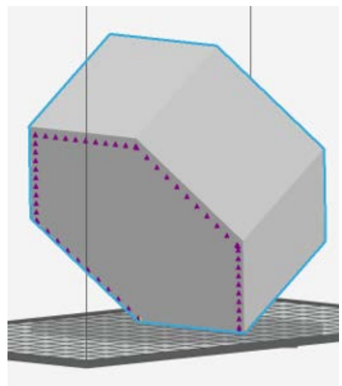
You can delete a part-to-part support, and use another method, if necessary.

1. Select the model.
2. In the right toolbar, click the *Part to Part Support* tab.
3. Mark the manually added support strut you made.
4. Click **Delete**.
5. Click **Update** to delete the support.

# Adding a Line of Anchor Points to Stabilize Models

When printing a large model, additional support structures may be necessary to provide structural strength and stability during the print. After generating automated anchor points, add a line of anchor points manually. This is done by setting the first and last anchor points, as well as the linear spacing. You can make stops and change the line direction, (see "Inline Anchor Points" (page 21)).

Figure 132: Multiple stops—Manually-added anchor points



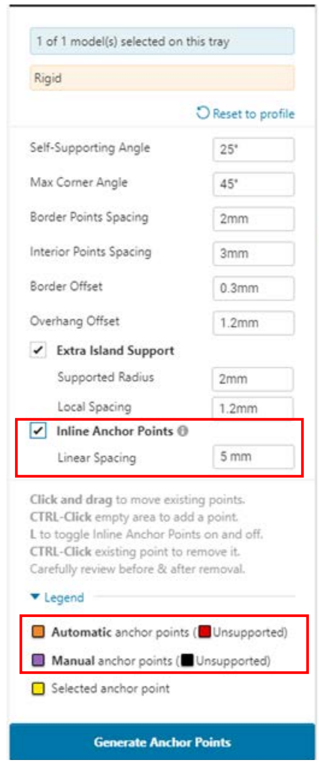
1. Automatically generate anchor point by doing the following:
   a. In the *Anchor Points* tab, set the Anchor Points parameters, (see "Anchor Points Settings" (page 18)).
   b. Click **Generate Anchor Points**.

   The automatically-generated anchor points are displayed in orange.
2. Manually generated anchor points along the line by doing the following:
   a. Enable the inline anchor points by selecting *Inline Anchor Points*.
   b. Set *Linear Spacing*, by entering the enter the gap in mm between the anchor points.
   c. Click the position to place the first anchor point.
   d. Double-click the position to place the last anchor-point.

e. Click **Generate Anchor Points**.

The manually-added anchor points are displayed in purple.

Figure 133: Setting inline anchor points



# Optimizing the Border Offset

To avoid too much support for a model that is perpendicular to the build platform, modify the *Border Offset* setting in the *Anchor Points* tab (see "Border Offset" (page 20)).

In the example in Figure 134, there is a ring-shaped model with a 2.5 mm shell.

Figure 134: Ring-shaped model with a 2.5 mm shell

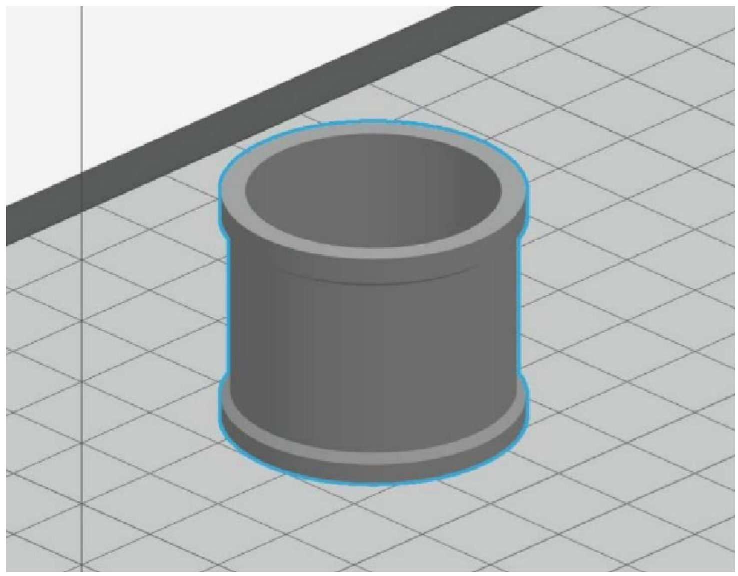
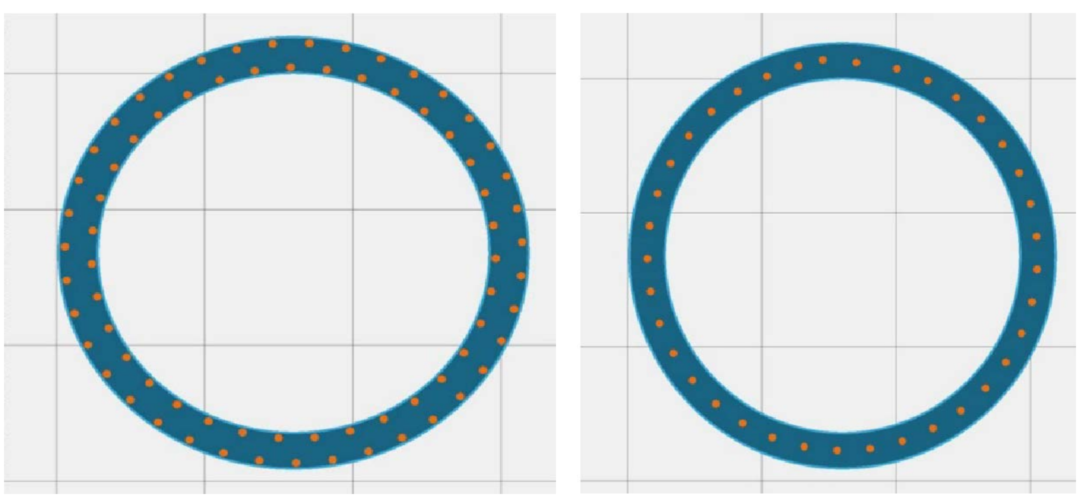


Figure 135 illustrates different *Border Offset* settings. The setting on the right has fewer anchor points and is preferred in this case.

Figure 135: Pre-defined Border Offset 0.25 mm (left); Customized Border Offset 1.0 mm (right)



# Adjusting the Tip Diameter

For each material support profile, there is a default *Tip Diameter* of the support structure that is pre-defined based on the material. An adequate tip size ensures that the support structure remains connected to the model during printing, is easy to remove, and leaves a minimum number of marks on the surface of the model.

You can customize the support profile and change the *Tip Diameter* setting to account for a variety of factors, including geometry (see “Tip Diameter (mm)” (page 26)).

To prevent models from detaching from the support structure that causes print failure, do any of the following:

- Adjust the tip diameter as follows:
  - If the model detaches from the support structure—Increase the tip diameter.

    The increased tip diameter ensures the contact points are thicker and they hold the model successfully during print. It can keep the model from detaching from the support structure during printing to enable successful prints.

    However, the support removal during post-processing may be more difficult and leaves bigger marks on the surface.
  - If the model is difficult to remove from the support structure—Decrease the tip diameter.

    The decreased tip diameter can ease removal from the support structure.
- Add more anchor points as follows:
  - Change the *Anchor Points* settings, such as *Border Point Spacing* and *Internal Point Spacing* (see “Border Points Spacing” (page 20), “Interior Points Spacing” (page 20), etc.).
  - Add anchor points manually by pressing *CTRL-Click* in the position you want to add the anchor point (see “Optimizing Anchor Points Manually” (page 22)).

# Using the Cross-Section View Tool

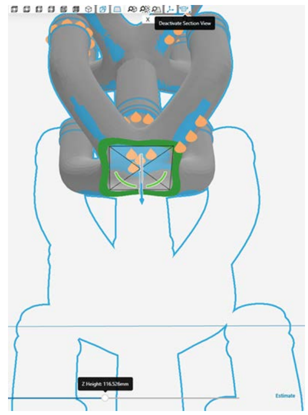
A *Section View* option is available in GrabCAD Print.

1. Toggle the *Activate Section View* button in the viewer.
2. Select a plane to section.

   You may also flip the direction of the plane cut.

3. Use the *Section View* tool to analyze any anchor points that are not otherwise viewable, or in densely supported areas of a model.

Figure 136: Section view



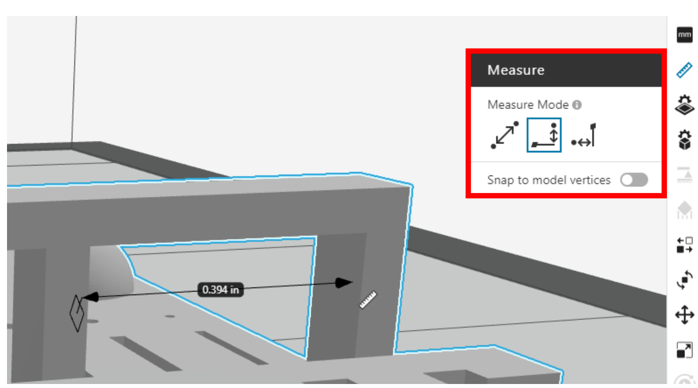
# Using the Measure Tool

It may be helpful to measure feature sizes and distances between features on a model to assist in meeting the requirements in the Material Processing Guide. GrabCAD Print includes a *Measure* tool to aid in this process.

1. Select the ruler in the toolbar to open the *Measure* option.
2. Use the measure tool to do the following:
   a. Select the *Measure Mode*:
      - Distance from a point to another point
      - Perpendicular distance from a plane to a point
      - Perpendicular distance from a point to a plane.

   b. Select *Snap to model vertices*, if relevant.

Figure 137: Measure tool

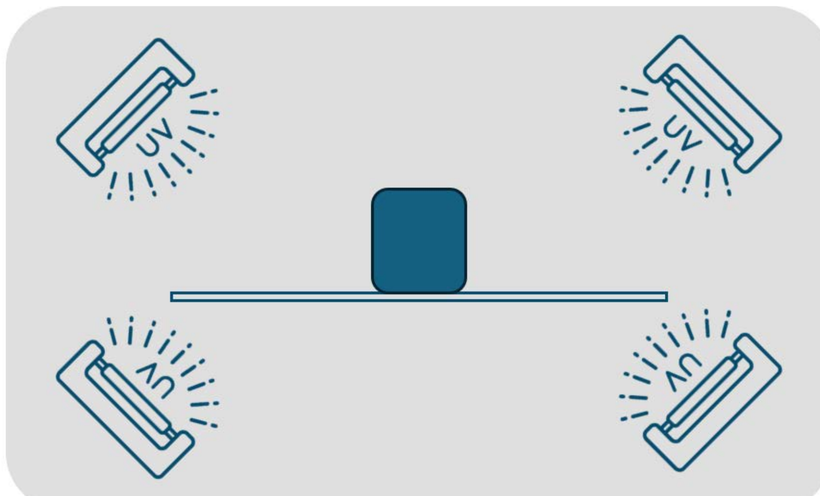



3. Add supports or adjust orientation as needed to achieve adherence to design recommendations.

# Origin Cure™ Guidelines

Origin Cure is a 3D printing post-process LED UV curing chamber designed to be paired with Origin Two and Origin One+ 3D printers. Origin Cure provides 360 degree curing by using mirrored panels with LED UV lamps built into the side walls.

Figure 138: Origin Cure—UV lamp positions




Each material requires a unique UV postcuring process. Some materials require thermal postcuring, as well. You can select a curing program from the 20 programs that are stored on the unit or add new programs, if needed.

The curing programs that you run include varying ramp times and intensity levels for the three active UV wavelengths: 365 nm, 385 nm, and 405 nm.

After the curing program cycle is complete, the model is fully cured and it can be removed from the chamber.

For information regarding Origin materials and the Origin Cure unit, refer to the following content on the Stratasys Support Center:

- Material Processing Guides about the Origin certified resins
- Origin Cure Installation and User Guide about the Origin Cure unit

## Running Curing Programs

Several curing programs are available for the Origin Cure (see "Origin Cure Programs" (page 84))

**To run a program:**

1. Place and orient the parts on the glass tray in the curing chamber.
2. Close the door.
3. Select the program to run.
4. After the program cycle is complete, remove the fully-cured parts from the chamber.

# Part Placement Guidelines

The following part placement guidelines can help ensure quality curing results:

- Ensure all parts are within the hatched frame.
- Spread out the parts uniformly and leave sufficient gaps between parts so they don't touch each other.
- Place tall/big parts at the center of the glass shelf and the smaller ones close to the frame border to minimize shadows on them. In addition, orient big parts at the lowest height possible.

  This minimizes the shadowing of parts that may occur due to big parts next to small ones.

Figure 139: Wrong-Parts too close in addition to high parts shadowing

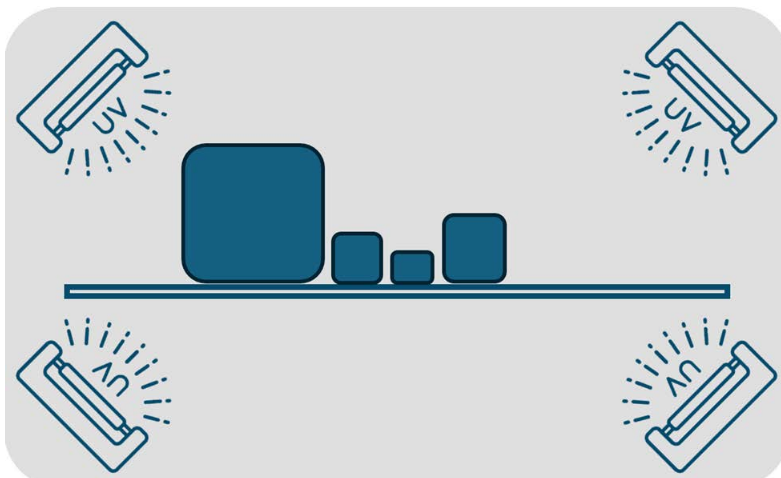
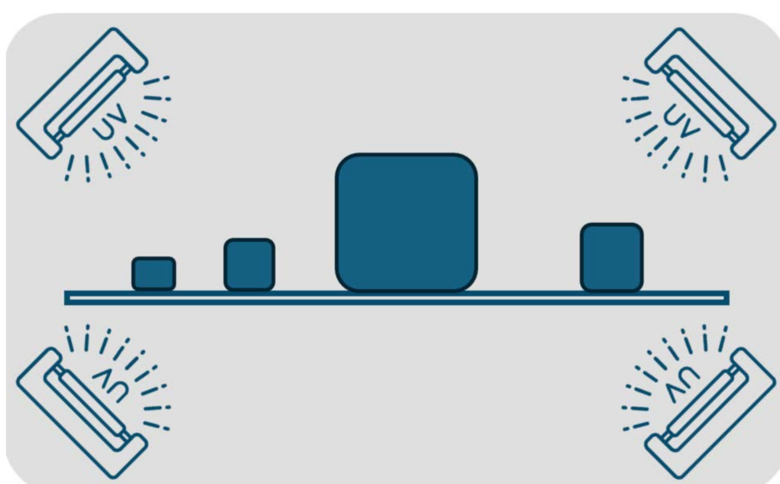



Figure 140: Correct-Parts are spread out and shadowing is minimized



# Orienting Models

The orientation of the model affects curing. During the curing process, parts are heated as part of the chemical process. This heat helps the parts cure properly and achieve the desired mechanical properties. This also means that the parts soften during the process. Softened parts can lead to serious deformation if oriented incorrectly. Parts can also be deformed not only by heat but also from too much polymerization and too little stress relief.

Proper orientation and using the correct curing program are crucial to prevent unexpected issues such as curling.

## Bridge Model

A bridge model can be oriented horizontally or vertically. The geometry influences the optimal orientation.

Figure 141: Bridge model

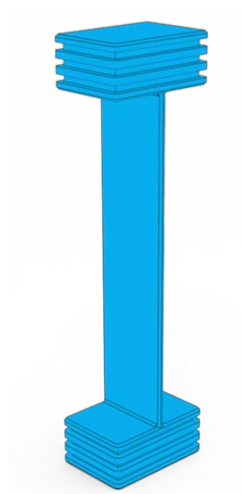


Figure 142: Example 1:Left: Bridge oriented horizontally to the surface; Right: Vertical orientation

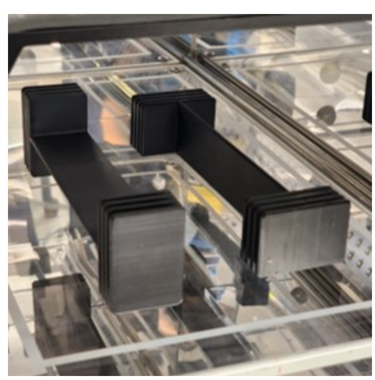
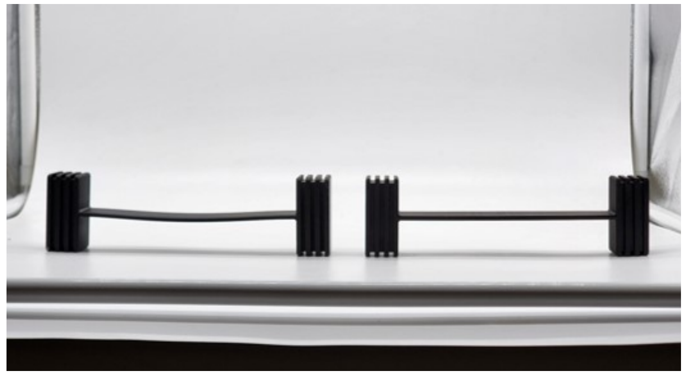


Figure 143: Example 2:Left: Bridge oriented horizontally to the surface; Right: Vertical orientation



## Model with Thin Overhangs

To mitigate the risk of deformation due to heat, do the following:

- Examine significant thin overhangs.
- Check if there are heavy sections supported by relatively thin ones.

If these features exist, make sure the part is oriented in a way that minimizes stress on the thin parts. Usually, the solution will be to lay them down.

Figure 144: Model with thin overhangs

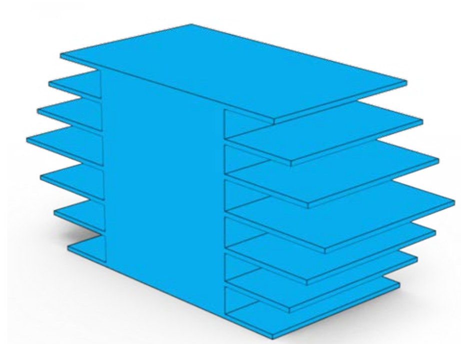


Figure 145: Example 1: Left: overhangs oriented horizontally to the surface; Right: vertical orientation

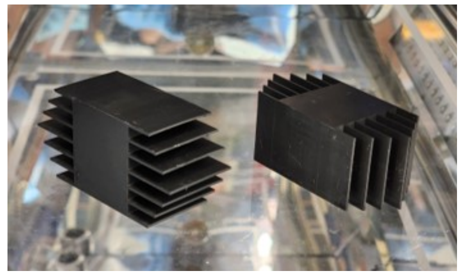
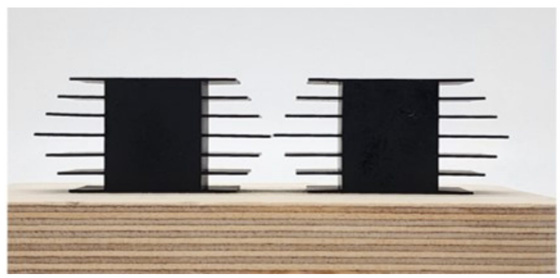


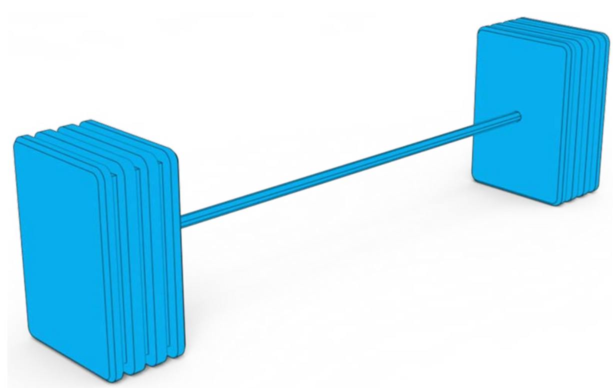
Figure 146: Left: overhangs oriented horizontally to the surface; Right: vertical orientation



## Thin Feature Suspended in the Air

When a thin feature is suspended in the air, it is useful to support it during curing.

Figure 147: Thin feature suspended in the air

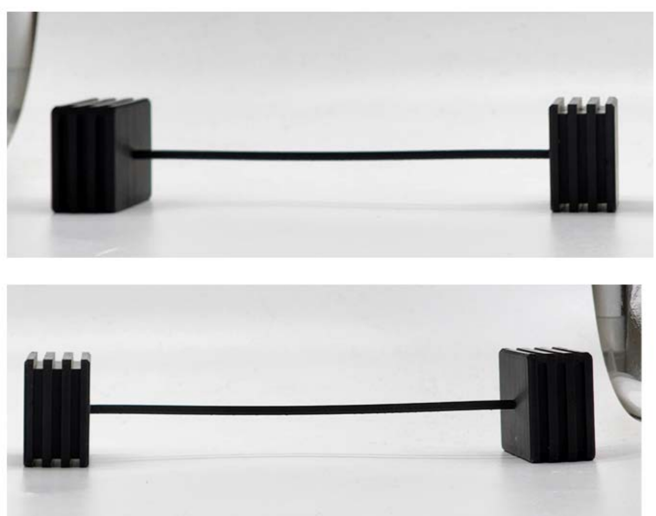


Both models in Figure 148 were printed with the support, but during the curing process one is assisted by the support structure, to keep it from deforming, and the other is cured without the support structure.

Figure 148: Thin feature–Left: Cured without a support; Right: Cured with a support

Both models in Figure 149 were printed with the support, but during the curing process one is assisted by the support structure, to keep it from deforming, and the other is cured without the support structure.

Figure 149: Top: Cured with a support; Bottom: Cured without support



# Origin Cure Programs

The following Origin Cure has 2 programs available by default:

- Standard
- Quick

In addition, you can create new programs or download predefined programs for Stratasys *Validated* or *Preferred* materials from the Origin Cure page on the Stratasys Support Center.

## Standard Program

The *Standard* program can improve the quality of parts as follows:

- Prevent warping of thin-walled parts
- Provide maximum accuracy, for example, for form and fit parts.
- Uses shorter intervals of UV exposure and adds delay times between the intervals.

Selecting the *Standard* program will result in a significantly longer curing time.

## Quick Program

The *Quick* program can have shorter curing times, but it may increase warping of the thin-walled parts or flat parts.

Figure 150: Cured on *Standard* program

Figure 151: Cured on *Quick* program

Figure 152: Left: Cured on *Standard* program; Right: Cured on *Quick* program

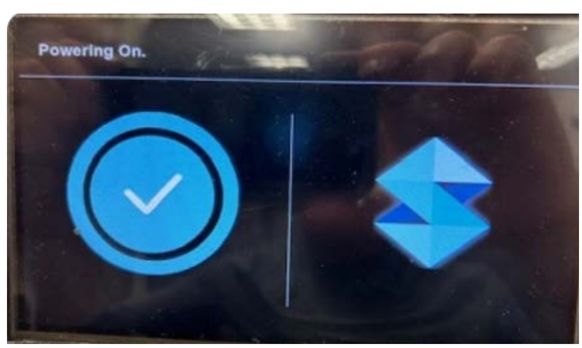
# Creating a New Curing Program

You can create your own programs. This makes it a powerful tool to have in a lab, especially if you have an OpenAM license.

You can access the *Administrator* mode. Operators can make any changes that require Administrator level access, such as modifying the power level, exposure time, program load, and calibration.

1. Power on the Origin Cure.

Figure 153: Powering On




2. From the Touchscreen interface:
   a. Select the green lock icon and type the correct PIN = **0000**.

b. Tap **Enter**.

Figure 154: Entering PIN

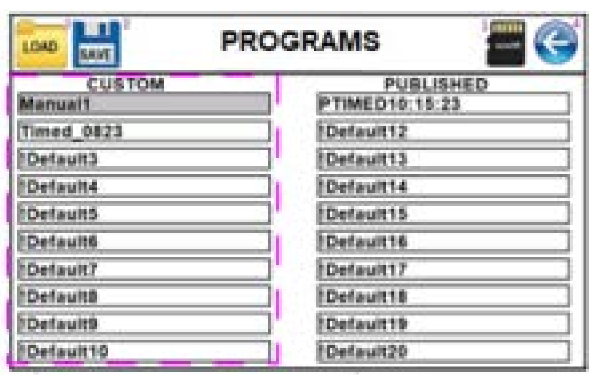

The touchscreen interface displays the red unlocked icon, which indicates you have accessed the *Administrator* mode.

3. While in the *Administrator* mode, you can do any of the following:
   a. “Loading a Custom Program” (page 86)
   b. “Saving a Custom Program” (page 86)

## Loading a Custom Program

1. To enter the *Programs* page, go to the bottom of the *Home* screen and select the current program name.
2. From the *Custom* list, select the desired program.

Figure 155: Custom list programs




3. Tap **Load**.

## Saving a Custom Program

1. From the *Home* page and the *Configuration* page, set the desired operating mode, power level, delay time. and exposure time.
2. To enter the *Programs* page, go to the bottom of the *Home* screen and select the current program name, see Figure 155.
3. Tap **Save**.

4. Enter a filename using the on-screen keyboard and tap **Enter**, see Figure 156.


Figure 156: On-screen keyboard

# Adding a Curing Program

You can use a curing program from the SD card or save it to the Origin Cure unit and then use it:

- "Using a Program from the SD Card without Copying it to the Unit" (page 87)
- "Copying the Selected Program from the SD Card and Saving it to the Unit" (page 88)

Adding a new curing program to the SD card is required when:

- there is a missing or updated program for a specific material that is currently in use.
- adding a newly released material program.

Only 20 curing programs can be loaded on the Origin Cure unit, but more than 20 curing programs can be stored on the SD card.

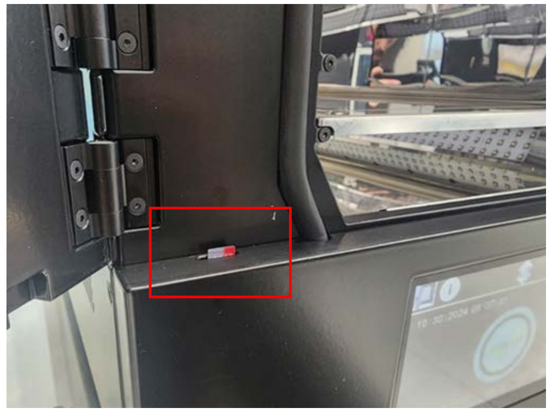
## Using a Program from the SD Card without Copying it to the Unit

1. From the Origin Cure Support Center, scroll down to the curing programs and download the zip file that contains all the available programs.
2. Remove the SD card from the Origin Cure unit:
   a. Open the Origin Cure door.

b. Gently push the SD card until it clicks and is released.

Figure 157: SD card in the Origin Cure unit



c. Remove the SD card.

3. Insert the SD card to the computer,

   An adapter might be needed.

4. Copy the new program to the SD card folder.
5. Insert the SD card into the Origin Cure unit.
6. To view the programs that are stored on the SD card, enter *Admin* mode, as follows:
   a. Tap the **Lock** icon on the upper right corner of the screen.
   b. Tap 0000.
   c. Tap **Enter**.
7. For each program you want to transfer, do the following:
   a. From the *Main* screen, enter the *Programs* list by tapping the currently selected program name.
   b. From the *Programs* list, tap the SD card icon on the upper right corner of the screen.
   c. Tap on the relevant program to transfer and then tap **Load**.

   The selected program is loaded.

## Copying the Selected Program from the SD Card and Saving it to the Unit

1. After loading a selected program to the SD card, go to the *Main* screen and enter the *Programs* list again.
2. Mark the program you would like to replace, and tap **Save**.

   The selected program replaces the program in the slot, if relevant.

3. To complete the process, enter the name of the new program and tap **Save**.

# Stratasys Online Resources

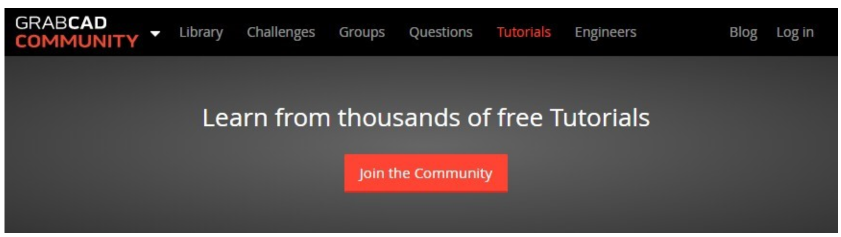
Stratasys encourages you to learn more about additive technologies and your Stratasys printer. A wealth of information is available on our online platforms.

Subscribe to our customer newsletter for quarterly updates on Stratasys knowledge and training. You can learn about the release of new documentation and learning resources.

## GrabCAD Resources

- For information on GrabCAD Print for Origin, refer to the Help at: https://help.grabcad.com/article/283-grabcad-print-for-origin.
- The Tutorials section of the GrabCAD Community portal is a valuable resource for Stratasys-sponsored and user-generated 3D printing tips. You can also ask 3D-related questions on the portal and download free CAD files.

Figure 158: GrabCAD Community

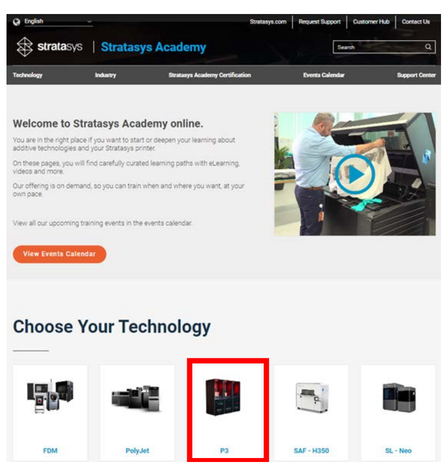



## Stratasys Academy

Stratasys Academy is your online learning platform where you can quickly learn and acquire skills on additive technologies and your Stratasys printer.

To guide you in your learning, our online academy provides a variety of resources such as an extensive library of videos and eLearning modules. We periodically update the site with new content. Start your learning journey today.

Figure 159: Stratasys Academy



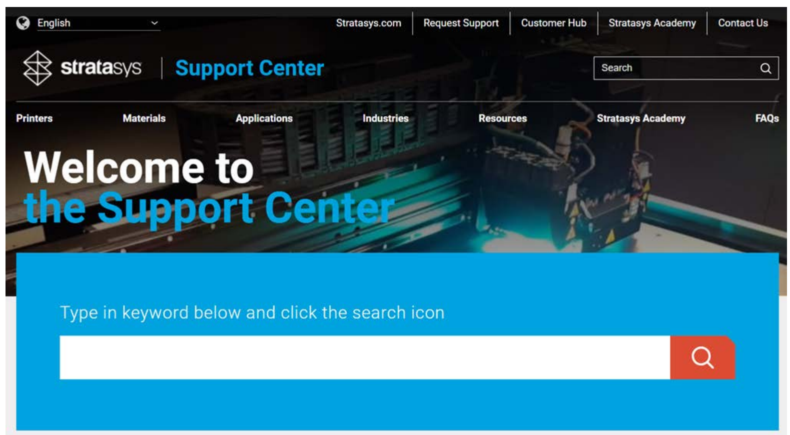
## Stratasys Support Center

The Support Center is a knowledge base that includes information about design, applications, printing material, and links to many other resources.

In addition, you can check the latest revision of the user guide for your Stratasys 3D printer and download documents in different languages.

The Support Center is available in multiple languages. You can change the display language using the language drop-down menu in the top-left corner of the homepage.

Figure 160: Support Center

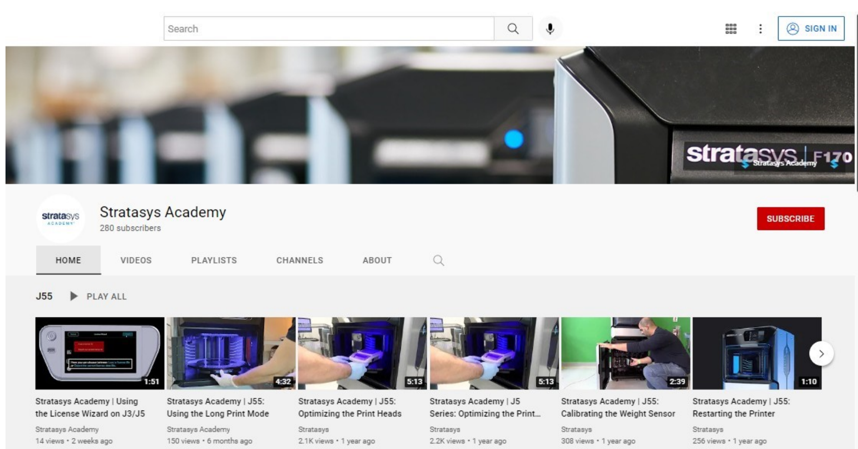



## Stratasys Academy YouTube Channel

The Stratasys Academy YouTube Channel features instructional videos about how to operate and maintain Stratasys printers. The channel includes dedicated playlists for different printers and special topics like post-processing.

Make sure to check out this new Stratasys Academy Channel and remember to subscribe!

Figure 161: Stratasys Academy YouTube Channel



www.stratasys.com

c-support@stratasys.com



DOC-30013 Rev. E