# Alpha 3D

## User Manual

# Alpha 3D



| | |
|---|---|
| **Product Version:** | Alpha 3D 1.8.1 |
| **Document Number:** | TS1M2EP0418 |
| **Release Date:** | October 2019 |
| **Document Version:** | 2.0 |

# Contents

# Chapter 1: Installation

## System Requirements

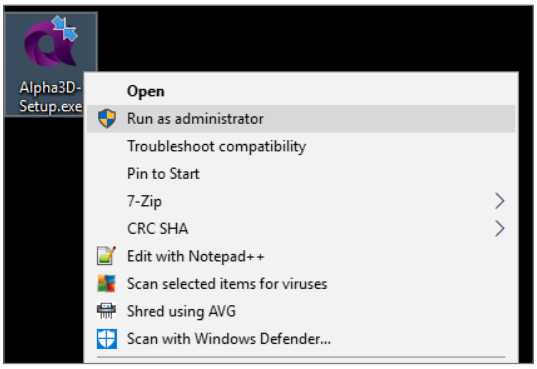
| | Minimum | Recommended |
|---|---|---|
| **CPU** | Intel i3 2.0 GHz dual-core<br>AMD Athalon 2.0 GHz dual-core | Intel i7 2.6 GHz quad-core<br>AMD Phenom II X4/ X6 at 2.6 GHz quad-core |
| **GPU** | Dedicated GPU with 1 GB RAM | NVidia GeForce 830<br>AMD Radeon R7 M340 |
| **Memory (RAM)** | 4 GB | 8 GB |
| **Disk Space** | 1 GB | 2 GB |
| **Operating system** | Windows 7 SP1, 8.1, or 10 | Windows 7 SP1, 8.1, or 10 |
| **Display** | 1600 x 900 | 1920 x 1080 |

## Installation Process

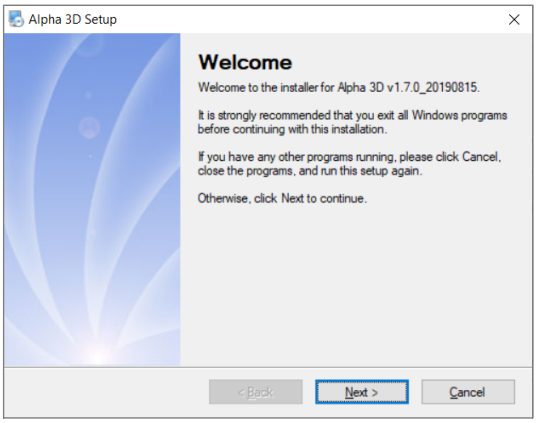
**1.** Download the Alpha 3D software from the WhipMix website: whipmix.com

**2.** When download completes, run the installer.

- Run the installer as the administrator so that Windows does not encounter any privilege warnings.
- Turn off your antivirus and firewall when running the installer to avoid any compatibility issues.



**3.** Follow the on-screen instructions to install.




- The installer may ask you to uninstall an existing version of Alpha 3D. If you have a previous version, Ackuretta recommends uninstalling it.
- You will need to accept the Software License Agreement to complete installation.
- You can change the default installation path. The default installation path is:

  **C:\Alpha 3D**

- Installation takes about 1 minute on a machine that meets minimum requirements. Alpha 3D does not display a complete message, and does not open automatically.

# Chapter 2: Getting Started

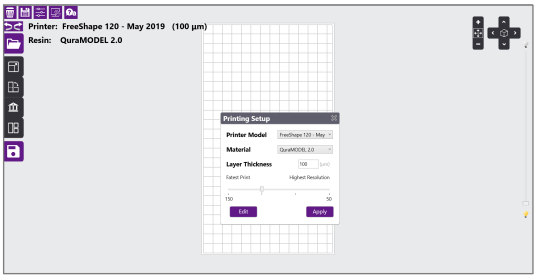
After you install Alpha 3D, the next step is to get your first print up-and-running. This section gives you the general information you need to start up your print, and links you to relevant sections in the rest of the manual that can give you further details.

This section includes the following parts:

- **Printing Setup**
- **System Setting**
- **Print Workflow**
- **Software Overview**

## Printing Setup

**1.** Run the software shortcut.




**A.** Ackuretta recommends running as an administrator.

**B.** When the software opens, the **Printing Setup** screen automatically displays.

**2.** Choose the settings that you intend to use for printing.

**A. Printer Model:** Choose the Ackuretta printer that you purchased.

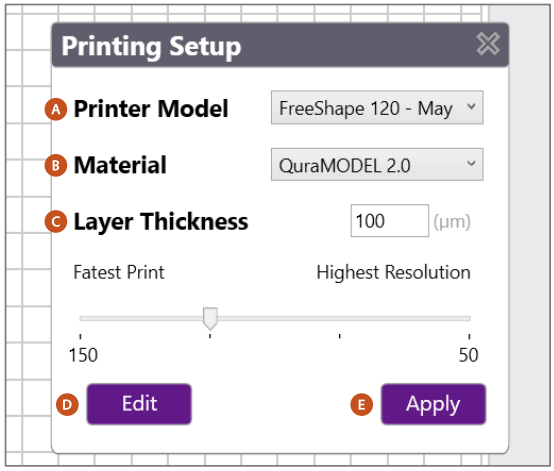
**B. Material:** Ackuretta presets most of the resins compatible with a machine into the Alpha 3D software. Choose your material here.

- If your material isn't shown, you can also do a direct slicing using the setting "Resin Test".

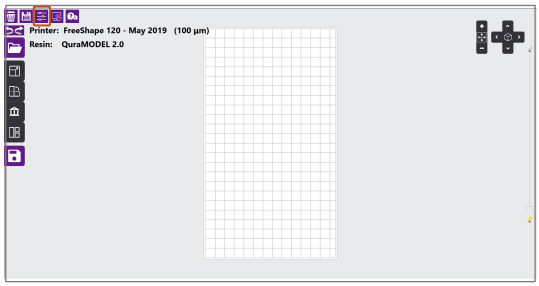
**C. Layer Thickness:** Move the slider to choose your layer thickness. Ackuretta presets layer thicknesses based on the resin.

**D. Edit:** You can modify or add your own resin settings to Alpha 3D. For more information, see **Resin Setting in Chapter 6**.

**E. Apply:** When you've finished, click this button.

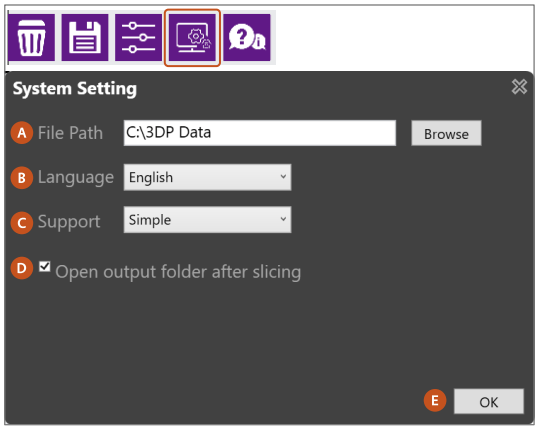



**Tip:** You can return to the Printing Setup screen at any time by clicking the **Printing Setup** icon.



## System Setting

On your first time using the software, click the **System Settings** icon to choose your computer-specific needs.




**A. File Path:** You can change the path to output completed files to. The default output path is:

C:\Ackuretta\Alpha 3D\Bin\Project

**B. Language:** You can toggle which language to use for the software.

- Alpha 3D supports English and Chinese (Traditional).
- Some functions may only show English text.

**C. Support:** Set the types of supports to use. You can also adjust these individually on the Supports screen. For more information, see **Supports**.

**D. Open Output folder after slicing:** Open the slice folder automatically after slicing complete.

- This setting is checked by default.

**E. OK:** When you've finished, click this button.

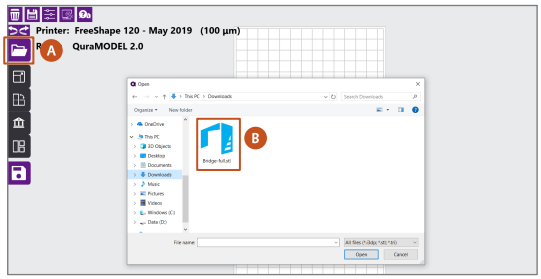
## Print Workflow

**1.** Import your STL or TRI file.

**A.** Click the **Open** icon.

**B.** Select your STL or TRI file and press **Open**.

- You can select and import multiple objects by holding the **Shift** key while selecting.



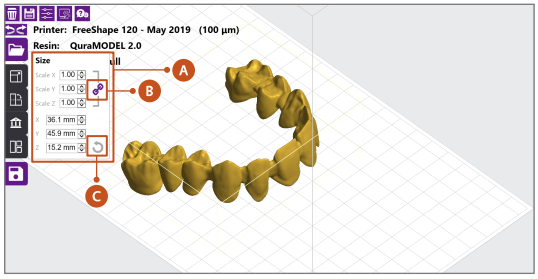

**Note:** After you import your objects, inspect them to make sure that you loaded the right files. For help with moving the camera, see **View and Movement**.

**2.** If necessary, resize your print from the **Size** screen.

**A.** Next to any dimension, use the arrows or type and press Enter to adjust the size.

**B.** Click the icon to link or unlink the dimensions. The Size screen starts with all dimensions linked together.

**C.** Click the icon to return to the original orientation.

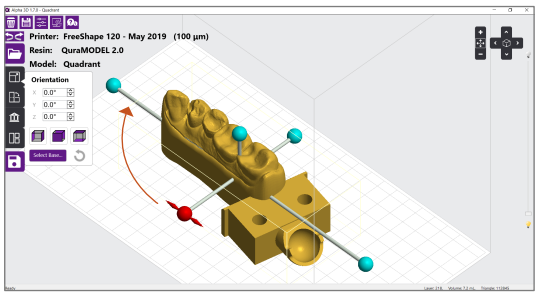


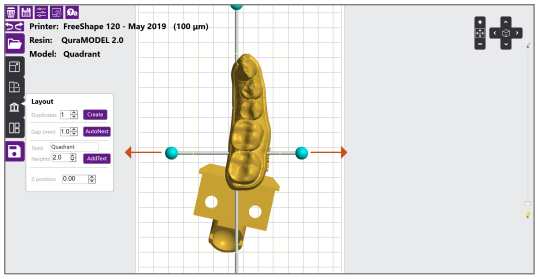

**Note:** Ackuretta presets calibration settings in the Alpha 3D software for the most common print types with a particular resin. As such, the Size function is not necessary for most prints.

**3.** Move and orient your print to your ideal position.

**A.** To rotate, move, or choose the position for adding supports, use the **Orientation** screen.

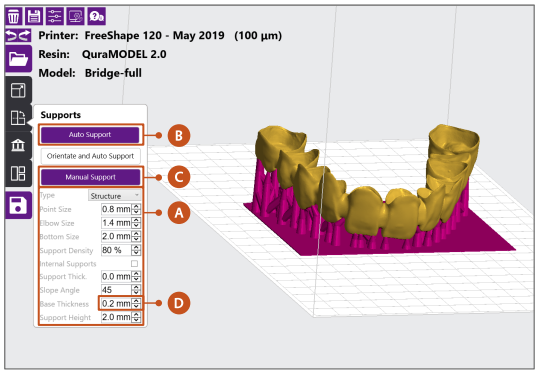
**B.** To move your part or nest multiple objects in the build area, use the **Layout** screen.




**Note:** For details about using the Orientation and Layout screens, see **View and Movement**.

4. Go to the **Supports** screen to add supports to your objects.




**A.** Set up your support parameters.

**B.** Click **Auto Support** to add supports.

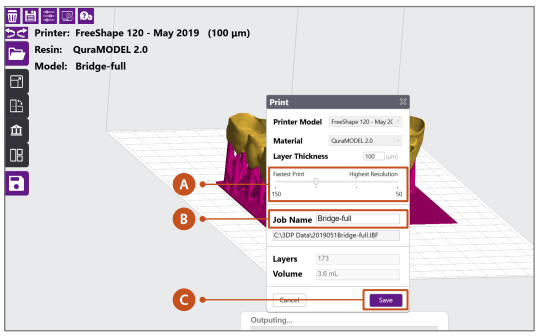
**C.** Adjust your supports with **Manual Support**.

- When you are finished adjusting supports, click **Manual Support** again to see your completed supports.

**D.** Increase the **Base Thickness** to add a base.

**Note:** For details about using the Supports screen, see **Supports**.

5. When your print is ready, click the **Print** button to open up the print dialog.



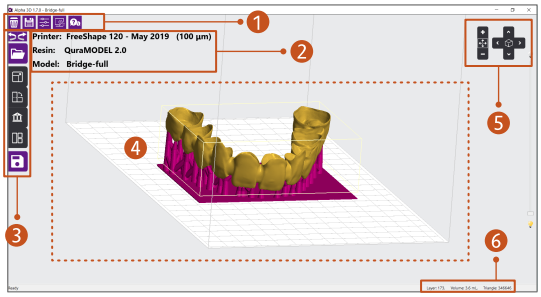

**A.** You can adjust your **Layer Thickness** on this screen.

**B.** Give your print a **Job Name** and press **Save** to begin outputting your print file.

**C.** To adjust your **Printer Model,** or **Material,** press **Cancel**, and then click the icon to go to the **Printing Setup** screen.

## Software Overview

The Alpha 3D software user interface has the following major sections:




1. **Menu bar:** Functions related to saving or restarting your model and configuring your software are here.
2. **Print information:** This area displays the selected **Printer** and **Resin,** and if an object is selected, it shows the **Model** name.
3. **Toolbar:** All of the most important functions of the software can be accessed from these buttons.
4. **Build area:** Your prints appear here along with their supports and base.
5. **View controls:** Choose a preset camera angle, zoom in or out, or check your slices from these controls. For information about these features, see **View and Movement**.
6. **Print statistics:** Some basic information about your print displays at the bottom of the screen, including the layer count and the estimated amount of resin used.

## 1. Menu bar

**Delete All Models** You can clear your plate by clicking this button.

**Warning:** You cannot undo this step. WhipMix recommends saving your project before this action.

**Save Project** Save your models, orientation, and supports. Your project will be saved as a project file with the .i3dp extension.

**Printing Setup** Set your printer, material, and layer thickness.

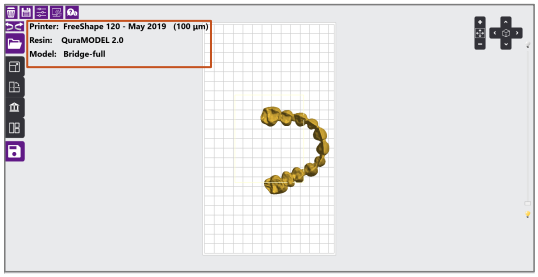
**Note:** For more information, see **Printing Setup**.

**System Setting** Set your output folder and language.

**Note:** For more information, see **System Settings**.

**About Alpha 3D** View the software version and copyright information.

## 2. Print information




**Printer:** This shows the name of the active printer build plate. You can change this setting from **System Setting** .

**Resin:** This shows the name of the active resin. Resin settings affect the default support parameters and the sizing calibration. You can also change this setting from **Printing Setup** .

**Model:** This field displays the name of the selected model. If you have not selected a model, this field does not appear.

## 3. Toolbar

**Undo / Redo** If you make a mistake, you can use undo and redo functions like other programs.

- You can Undo by pressing the icon to go back one step. You can also use Ctrl + Z.
- You can Redo by pressing the icon to go forward one step. You can also use Ctrl + Y.
- Alpha 3D records up to 20 steps of activity.

**Open** Import your STL or TRI files into the build area.

**Size** If necessary, resize your print from the **Size** screen.

**Orientation** Turn or rotate your print to choose the ideal position for adding supports.

**Note:** For details about using the Orientation screen, see **Movement**.

**Supports** Add supports to your objects to ensure that they print successfully.

> **Note:** For details about using the Supports screen, see **Supports**.

**Layout** Move your part or nest multiple objects in the build area.

> **Note:** For details about using the Layout screen, see **Movement**. Additionally, some items shown in this screen are covered in the **Special Features** section.

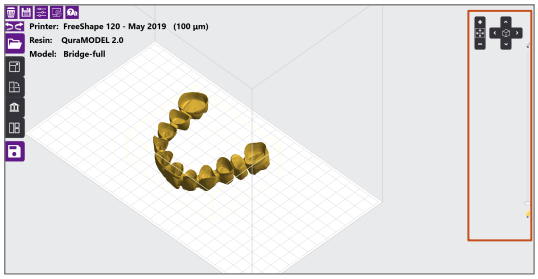
**Print** When your print is ready, slice your print to output an IBF file that you can use on your Veribuild printer.

# Chapter 3: View and Movement

## View

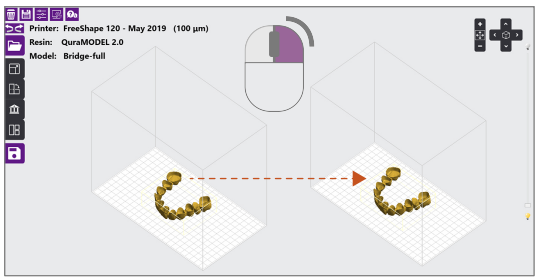
### • View Controls

Alpha 3D contains several view controls located in the top-right of the interface. This section covers those controls, as well as other view functions with the mouse.



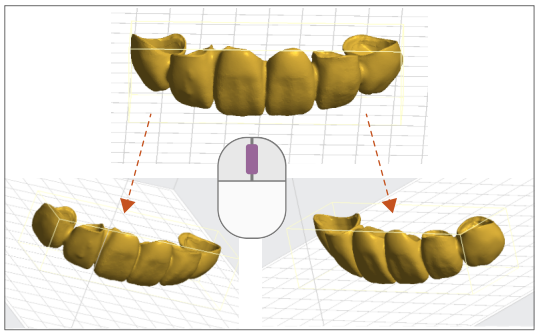

### • Pan Camera

Right click and hold to pan your camera from side to side.




### • Rotate Camera

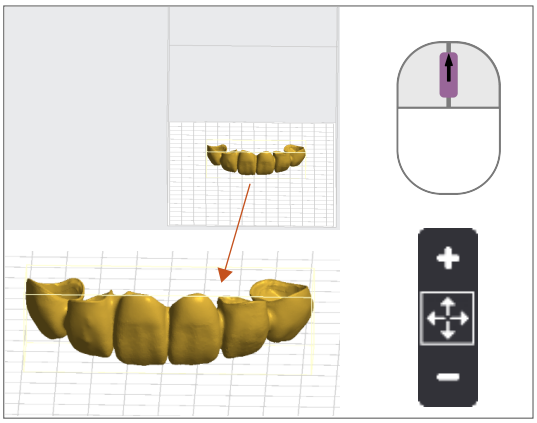
Middle click and hold to rotate and turn your camera. The camera turns more rapidly based on the speed of your mouse motion.



- **Zoom**

  Scroll the middle mouse wheel to zoom in and out.

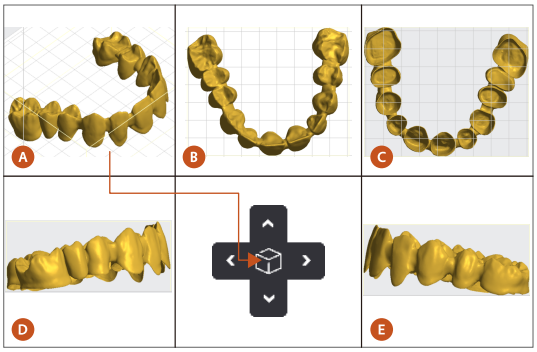
  On the top-right of the screen, press the + or - buttons on the view navigator.



- **View Presets**

  The directional arrow buttons set the view to a specific angle.

  **A. Home** view, which is set to 45° from the top, front, right
  **B. Top** view, or rotate around the print 90° upwards
  **C. Bottom** view, or rotate around the print 90° downwards
  **D. Left** side view, or rotate around print 90° to the left
  **E. Right** side view, or rotate around print 90° to the right

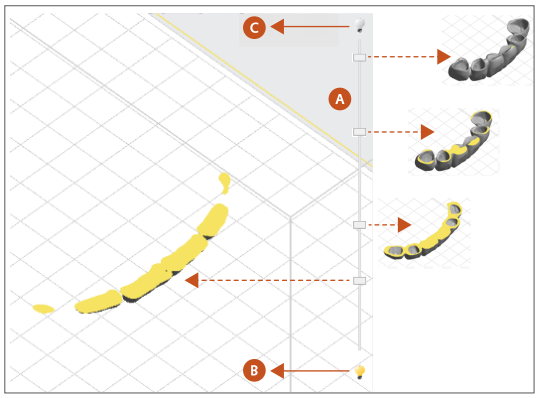



- **Slice Slider**

  Use the right slider to cut a print into slices and hide a print below or above the current slice.

  **A.** Slice slider
  **B.** Click the bottom icon to view slices from the bottom up
  **C.** Click the top icon to view slices from the top down

  **Note:** You can use the ↑ and ↓ arrow keys to raise or lower the current slice one-by-one.



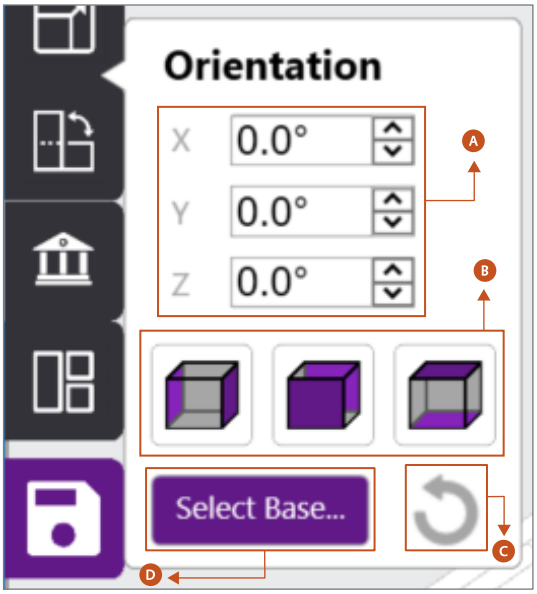

## Movement

You can move objects in Alpha 3D from two different screens:

- **Orientation** : Move and rotate an object to choose the ideal position for adding supports and printing.
  - Changing the Orientation will cause all supports to be broken, and you will need to regenerate them.
  - Set your object orientation before adding supports.
- **Layout** : Move your part or nest multiple objects in the build area.
  - You may move objects around that already have supports, and the supports will be maintained.
  - Movement is the primary function of the Layout screen. The other functions are described in the **Special Features** section.

## Orientation

The **Orientation** screen has the following options:




**A.** Rotate the object to a specific angle.

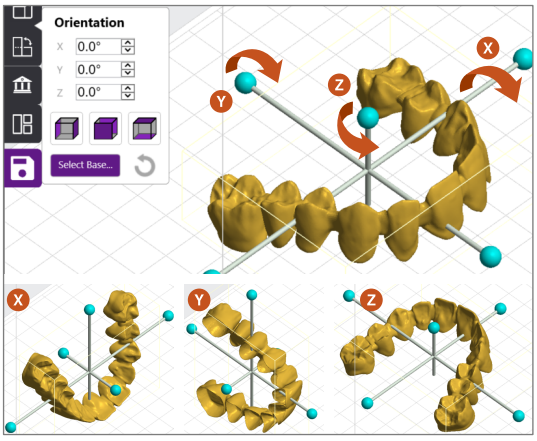
- Use the arrows to adjust the angle of the object in the X, Y, or Z direction.
- Alternatively, type a number in one of the boxes. Press the Enter key and the object will turn the specified angle.

**B.** Click any of the icons to rotate the object in that direction. Clicking the icon a second time will rotate the object in the opposite direction.

**C.** Click the icon to return to the original orientation.

**D.** Click the **Select Base** button to choose a side that will turn to the bottom.

In addition to the functions shown on the **Orientation** screen, move your part manually by dragging the spheres. Spheres are shown for the **X**, **Y**, and **Z** axes.



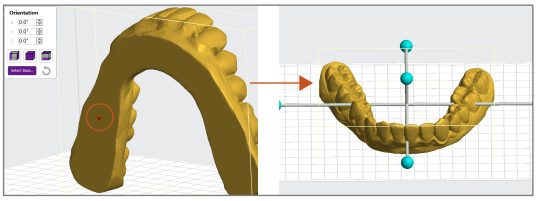

> **Tip:** If a sphere is directly facing you, first turn your camera, and then move that sphere.

Alternatively, if you click on the part and drag it, you can move the part. This can also be done on the **Layout** screen.

If you use the **Select Base** function, your cursor will change to show a red arrow.

Click the part when the red arrow is facing away from the surface you want to be the bottom.

The part turns so that the selected face is on the base.



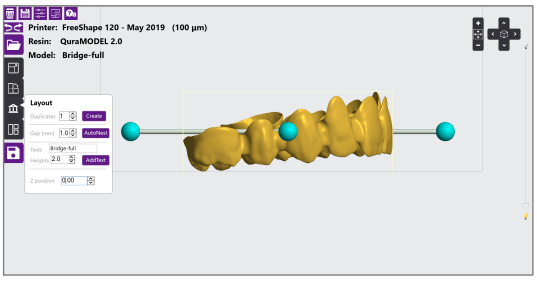
## Layout

The main function of the **Layout** screen is to move objects around the print platform.

- Click-and-drag an object to move it around the platform.
- If you drag an object off of the build platform, Alpha 3D will automatically place it back on the platform.

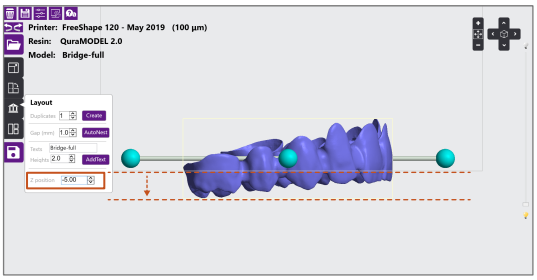
If you do not want to print the whole object, you can decrease the Z-position of the object off of the platform, use the **Z-position** function.

- The object starts directly on the platform.




- Use the arrows to adjust the height, or type in a value and press Enter. The part will fall below the platform.

> **Tip:** This function is useful if you want to reduce the height of your part to make it print faster, or if the bottom of your part has a rough surface, and you want to cut to a flat part of the print.




# Chapter 4: Supports

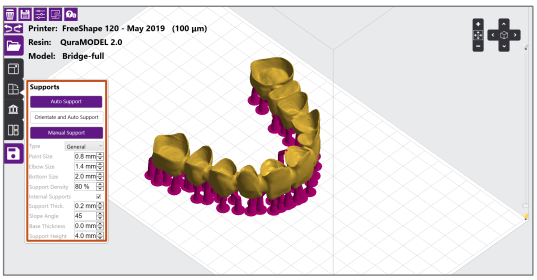
## Purpose

Resin-based printers (DLP, SLA, or LCD) require that every part of the print is properly connected to the build platform during printing. Part that are not connected may drop off of the printer and fail. Alpha 3D allows you to automatically add supports and easily modify them to make sure your prints are successful and hassle-free.

## Support Procedure

**1.** Select your object and go to the **Supports** screen.

**2.** Set up your support parameters.




**Type:** This setting controls the support base and scaffolding.
For more information, see **Support Types**, below.

**Point Size:** The diameter of the top of the support, where it touches the object.
- **Recommended:** 0.6 - 1.2

**Elbow Size:** The diameter of the bend point of the support.
- **Recommended:** 1.1 - 2.2; Larger than the **Point Size** by 0.5 - 1.0 mm

**Bottom Size:** The diameter of the bottom of the support, where it touches the base.
- **Recommended:** 1.6 - 3.2; Larger than the **Elbow Size** by 0.5 - 1.0 mm

**Support Density:** The amount of supports when using Auto Support, with the exception of supporting local lowest points.
- **Recommended:** 70% - 85%, depending on **Point Size**.

**Internal Supports:** Check this to allow supports to touch the model on both the bottom and top.

**Slope Angle:** Supports are only allowed to be placed on the object if their angle to the bottom is less than this number.

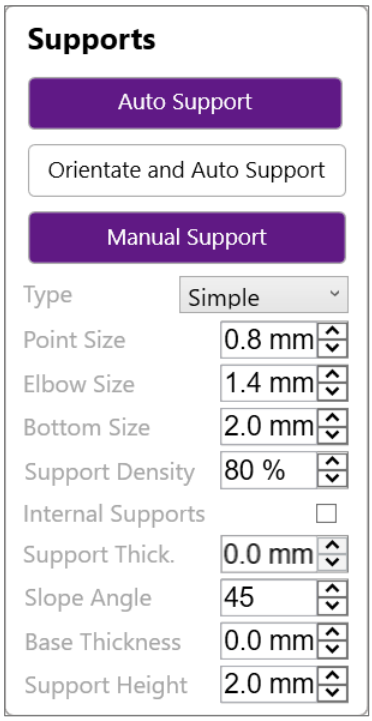
- **Recommended:** 45-55

**Base Thickness:** Increase this number to add a base to all your prints.

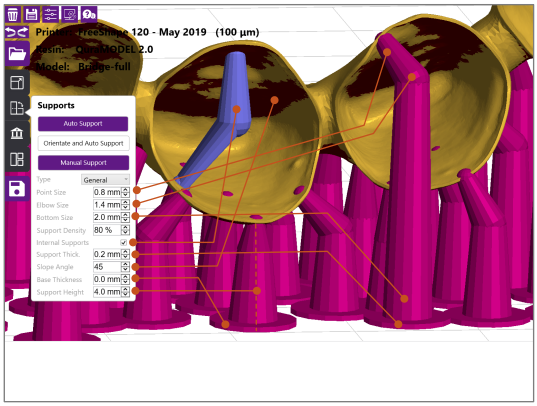
- **Recommended:** 0.2 for Ackuretta printers
- Add this base after supporting your print

**Support Height:** When you add supports, the object will be automatically raised to this height.

- **Recommended:** 1.5 - 2.5

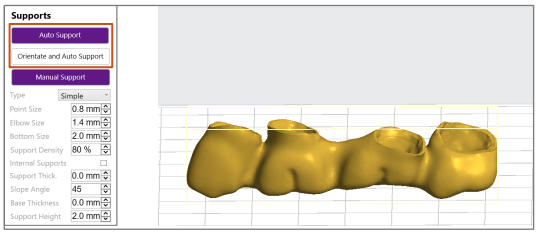


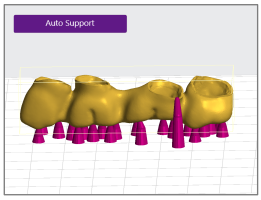

**Note:** The support parameters affect the following areas.

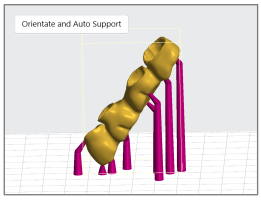



3. Add automatic supports to your print with **Auto Support**.

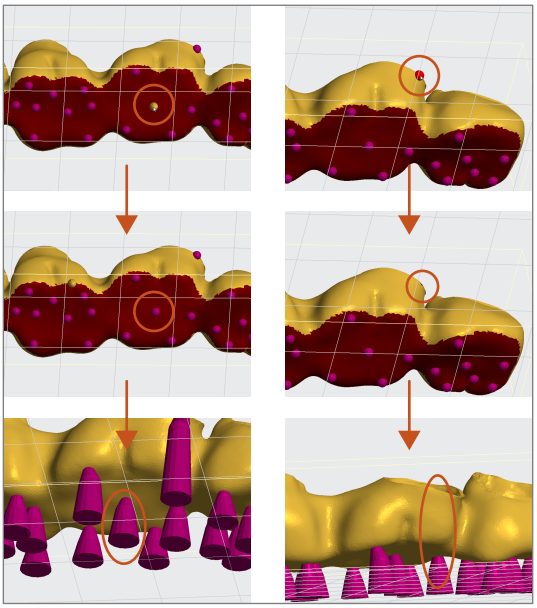
- **Auto Support** maintains your orientation, raises the part to your **Support Height**, and then adds supports directly.
- Alternatively, you can use **Orientate and Auto Support** to allow Alpha 3D to analyze your part and attempt to choose an orientation.



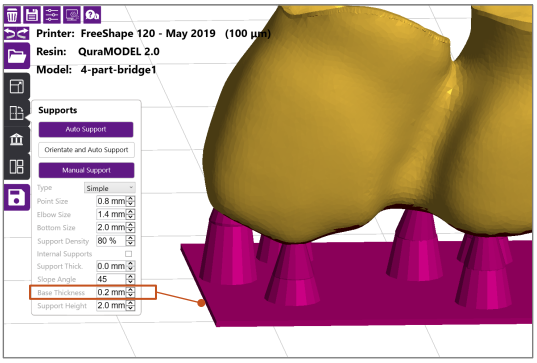
**4.** Click **Manual Support** to adjust your automatic supports.



- This changes the interface to *Manual Support Mode*, which only shows support points.
- Your cursor changes to show a support point. When you click, you will add a support point at that location.
- When you hover over an existing support point, the support point turns red. If you left-click on a red point, you will delete it.
- You can adjust support settings while in *Manual Support Mode*. New supports will use the new settings, but existing supports will be unchanged.
- Ackuretta recommends adding supports all over the red areas marked by the *Slope Angle* parameter. Supports should be spaced so that 1 complete support can fit between any 2 supports.
- Click **Manual Supports** again to exit *Manual Support Mode* and see your completed supports.
- Related information: **Local Minimum Points** and **Editing Individual Supports**.

**5.** Add a base by using the arrows next to **Base Thickness**.

- **Recommended:** 0.2 for Veribuid printers

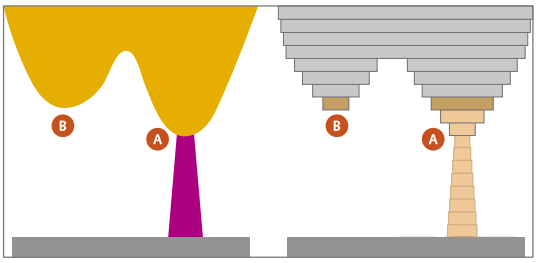



## Local Minimum Points

The most important points to support in a print are the local minimum points (also known as "islands"). These are the parts of a print that would be printed first in a resin-based printer.

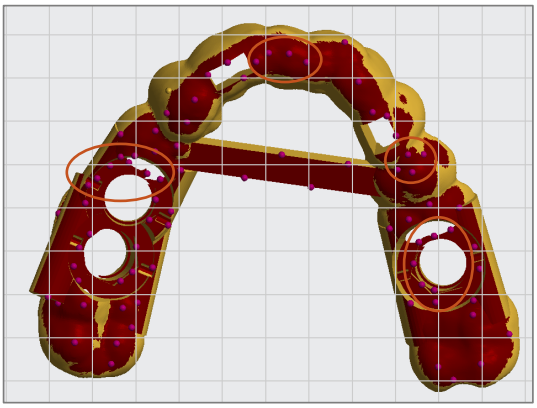
A DLP, SLA, or LCD printer has a moving build platform with a light source that cures resin onto that platform. As the print develops, the build platform moves farther and farther away from the light source, so existing parts of the print must hold new parts to the platform.

In the diagram, the local minimum **Point A** is connected to the platform by a support. The layers of the support hold the first layers of Point A so that the layers afterwards can be held together from that place.



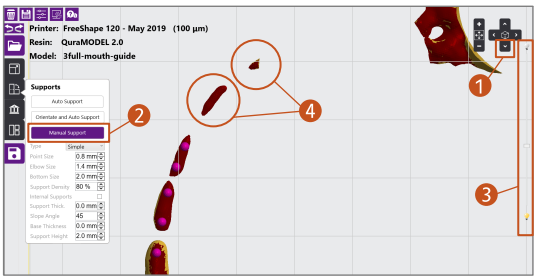

On the other hand, the local minimum **Point B** is not connected by a support. If the light source cures that local minimum, the build platform will be too far away, and the cured layer will stay at the bottom of the vat. All subsequent layers would also have nothing to connect to, and the print would fail. (This is why Point B would be considered an "island" -- it would be solid material in the liquid vat of resin.)

The Alpha 3D **Auto Supports** function will automatically search for local minimum points before adding other supports. If many supports are clustered together closely after using Auto Support, the object may have many local minimum points in that area.



The diagram shows several locations in which many support points are clustered together due to having high densities of local minimum points.

To check your print for local minimum points and add supports on necessary areas, do the following:




1. Set your **View** to the bottom.
2. Go to **Manual Support** Mode.
3. Use the **View Slider**, and raise from the bottom until you see some areas of the print.
4. If you find an area that appears from the bottom with no supports, add a support to the bottommost point you can find.

## Editing Individual Supports

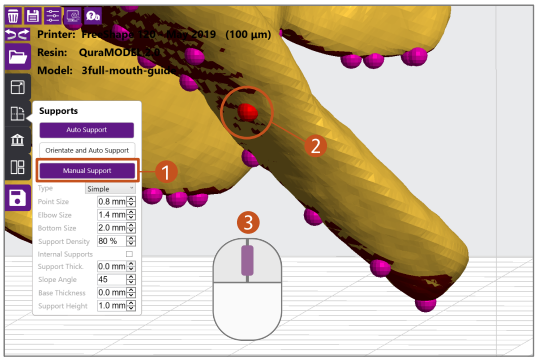
After adding a support, you may modify that support individually to make sure it connects exactly the way you want.

You may want to manually edit a support for a number of reasons:

- The support intersects the object, attaching at an unintended point
- The object may be heavy above the support so the support needs increased size
- The angle of the support elbow is too extreme, which may cause the support to fail

Manually editing individual supports can take a long time. If you want to edit many supports, Ackuretta recommends changing your support parameters and using **Auto Support** again, and modifying only a few problematic supports.

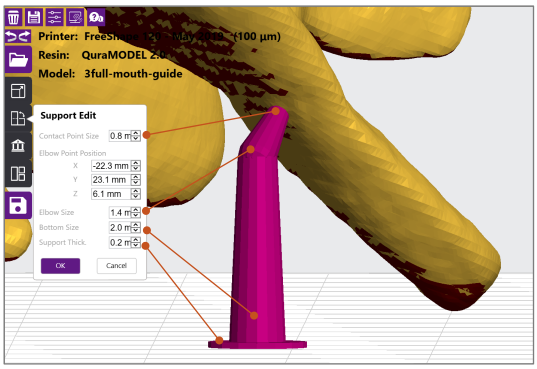
To edit an individual support:




**1.** Go to **Manual Support** Mode.

**2.** Hover over an individual support point so that its color turns red.

**3.** Middle-click that support point.
- Alpha 3D will go into the *Support Edit Mode*.




**4.** Adjust your support parameters.

The parameters **Contact Point Size**, **Elbow Size**, and **Bottom Size** are the same as in the **Supports** screen. You can increase or decrease them and the support will change accordingly.

**Elbow Point Position:** The Elbow Point is the bend point of the support. The Elbow Point Position is based on the center of the object. If you increase the X, Y, and Z positions, the bend point will move accordingly.

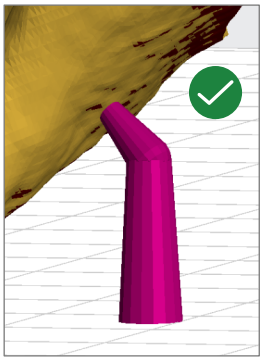
**Support Thickness:** If you increase this value, the support will add a small base with an angled detachment point to the support. Ackuretta printers do not require these types of supports, but if you prefer to use them, they are an option.

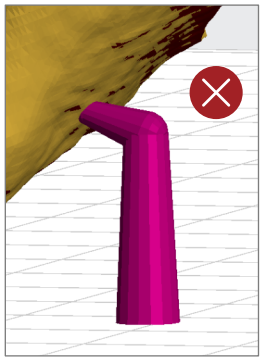
**5.** When you are finished, click **OK**.

**Note:** Keep the angle of the Elbow Point relatively straight so that your support is structurally strong. If your angle is too perpendicular, your support may break at the Elbow Point, which would cause your print to fail.

Completely vertical supports are always preferred over angled supports. If you need an angle to your support do not allow the Elbow Point angle to diverge from completely straight by more than 45°.

To make your support angle more straight, lower the **Z** value under the **Elbow Height Position**.

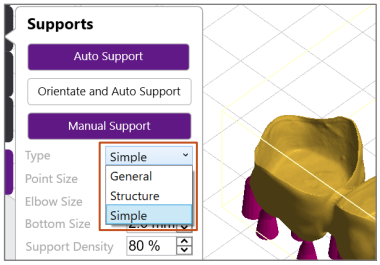




## Support Types

Alpha 3D provides three different types of supports:

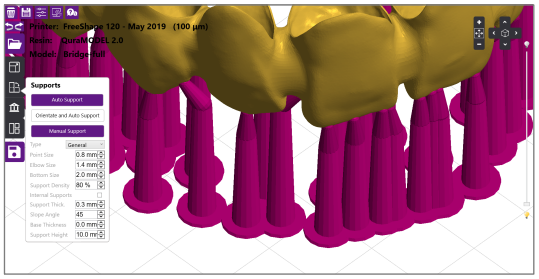
1. General
2. Structure
3. Simple





- **General Supports**

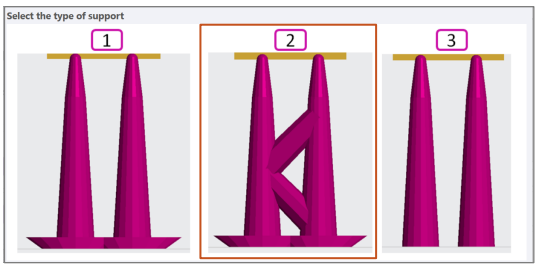
  These supports add a small base to each support. You can increase the **Support Thickness** parameter to adjust the height of that base. This base is separate from the **Base Thickness** parameter.

- **Structured Supports**

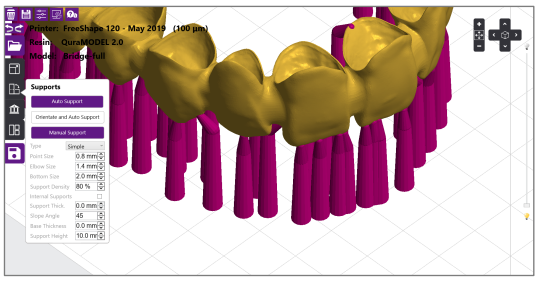
  The main feature of these supports is that they add scaffolding between each support. This amount of scaffolding allows you to use thinner supports while maintaining strength.

  Ackuretta recommends using lower **Point Size** and **Elbow Size** parameters when using Structured Supports. Additionally, you can also use the **Support Thickness** parameter to adjust the height of the base.




- **Simple Supports**

  These support do not add a base nor any structures to your supports. These are best when you are inexperienced with supports, or when you are concerned with using extra material. You can still use the **Base Thickness** parameter to add a base over the entire print area.

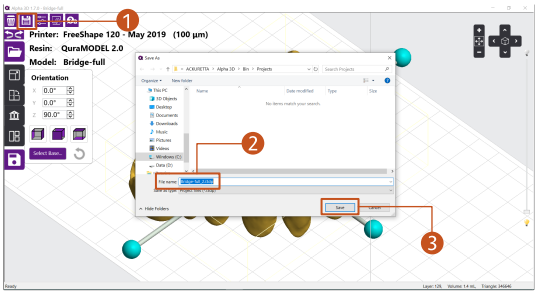

# Chapter 5: Special Features

Alpha 3D includes the following additional features to help you easily create and manage your prints:

- **Save Project**
- **Duplicate**
- **Auto Nesting**
- **Add Text**
- **Delete**

## Save Project

You can save your models, orientation, and supports into a project file by using the **Save As** function. Your project will be saved as a project file with the .i3dp extension. It will be saved to the file path you choose. The default file path is:
C:\Alpha 3D\Bin\Projects

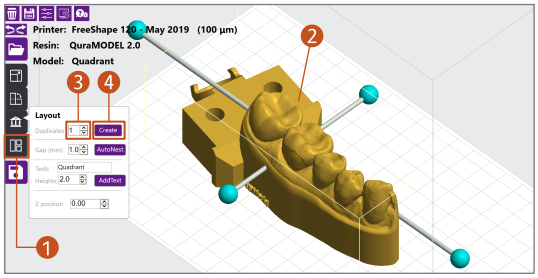



To save your project:

1. Click the  **Save as** icon.
2. Type a file name in the **Save as** window. The default file name is the same as your STL model.
3. Click **Save** to save and exit.

## Duplicate

To make an exact copy or clone of the object, including supports, use the **Duplicate** function.

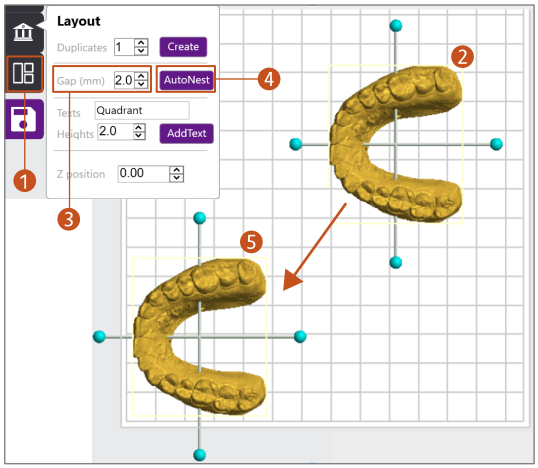



1. Click the **Layout** icon.
2. Select the object.
3. Use the arrows to select a number next to **Duplicates**.
4. Click **Create**.

   A number of copies will be made equal to the number you selected.

## Auto Nesting

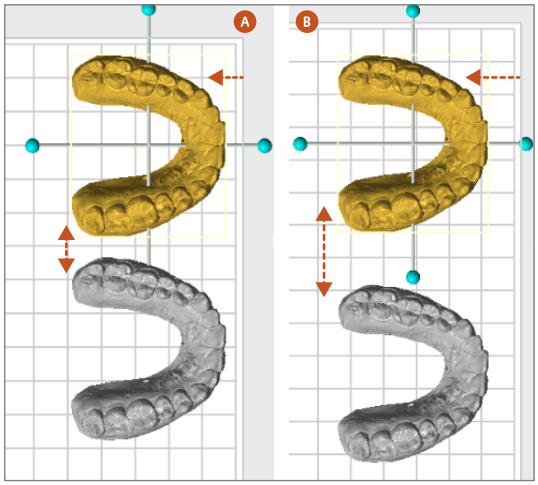
Alpha 3D can move parts automatically onto the build platform. When it does so, Alpha 3D ensures that the parts all have sufficient space between each other for printing.




1. Click the **Layout** icon.
2. You may select an object if you want to only move one object. If you want to move all objects, leave all objects unselected.
3. Choose the amount of space that you want to leave between each object. Use the arrows next to the **Gap**.
   - Ackuretta recommends a minimum **Gap** distance of 1.0 mm, but usually 2.0 mm is preferred.
4. Click **AutoNest**. Alpha 3D will automatically attempt to place the objects.
5. All objects move onto the platform, starting in the bottommost front corner.

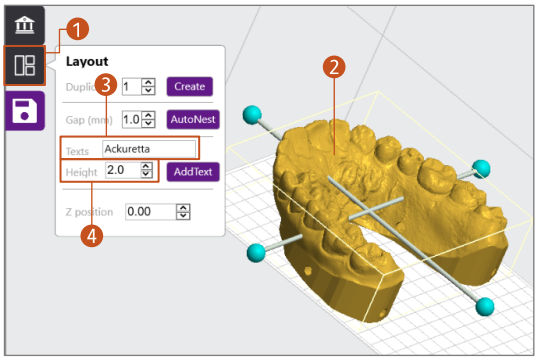
The **Gap** distance affects the distance from the part and from the edge of the build platform. The following images show different Gap distances.

**A.** 3.0mm
**B.** 8.0mm

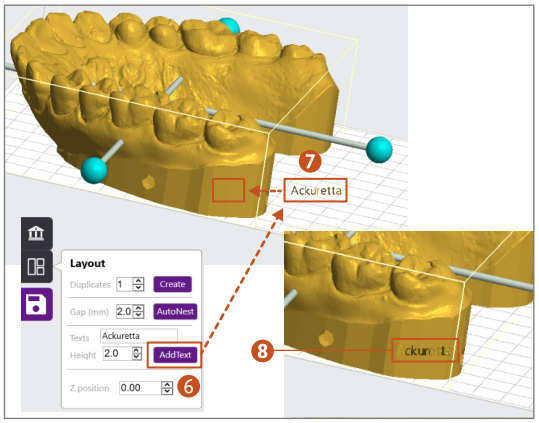


## Add Text

Alpha 3D can also add a physical label to an object. The label will extrude out of the object.




1. Click the **Layout** icon.
2. Select an object.
3. In the **Texts** box, type your text or label.
4. You can change the size of your text by adjusting the **Height** value.
   - Ackuretta recommends a height of 2.0 mm.




5. Choose an angle of the selected part where you want to add your text.
6. Click **Add Text**. Your text will appear in the center of the screen as a 3D model. If your object is in the center of the screen, your text appears on the object.
7. Move your screen so that your text appears where you want it on the object.
8. When you are satisfied, click any location off of the object. Your text becomes part of the object.

## Delete All Models

If you imported incorrect .stl or .i3dp files, and you want to clear your entire build platform, you can use the **Delete All Models** function. This acts as if you started a new project.

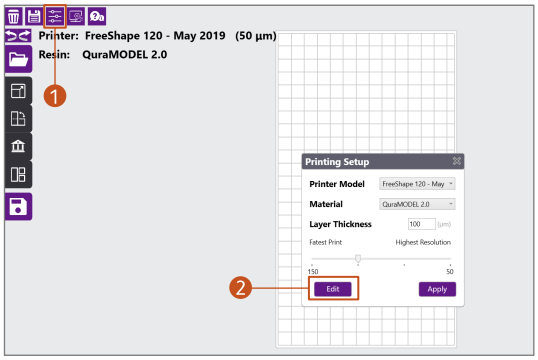
**Warning:** You cannot undo this step. WhipMix recommends saving your project before this action.

# Chapter 6: Configuration

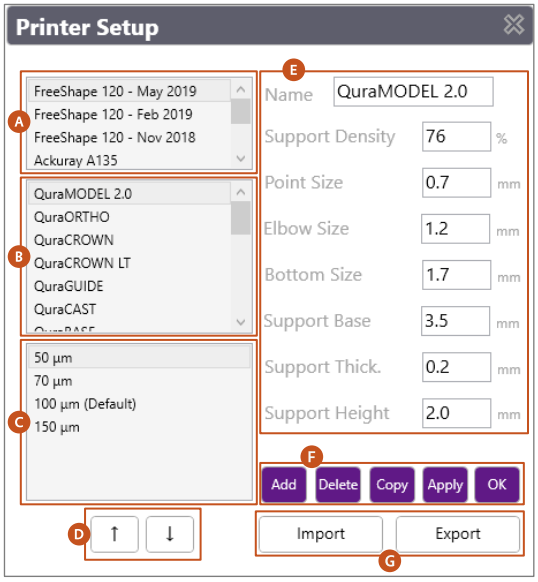
## Configuration Screen

Alpha 3D configuration is performed from the configuration screen, which is labeled **Printer Setup**.

1. To access the Printer Setup screen, start by going to the **Printing Setup** screen. This screen appears when you open Alpha 3D, or you can click the associated button on the top row.
2. Click **Edit**. You will open the configuration screen.




The configuration screen shows the following items:




**A. Printer Settings:**

You can view the size, resolution, and other settings regarding printers.

Every printer has its own set of materials, with settings specific to that printer.

> **Note:** You may not edit settings in this section.

**B. Material Settings:**

Choose a material from the side of this list to view or modify. The settings for that material will appear in the Settings Display.

Every material has its own set of thicknesses, which will change based on the material chosen. The most important settings for each material are found in the **Thickness Settings**.

For more information, see **Adding a New Resin**.

**C. Thickness Settings:**

A number of layer thicknesses are shown here, usually with one being selected as Default. Choose a thickness here to modify.

**D. Order Arrows:**

These arrows adjust the order of the settings, which affect how they will be displayed in the Printing Setup screen.

**E. Settings Display:**

This section shows all the settings available for the given selection. Different settings are shown for the printer, material, or thickness.

**F. Save Options:**

All of these options are available for material and thickness settings:

**Add:** Add a new setting based on the name shown for the setting.

**Delete:** Remove the selected setting.

**Copy:** Duplicate the chosen setting. The new setting will append "_new" after the current name.

> **Note:** You cannot save your settings if two thickness settings have the same layer thickness.

**Apply:** Save your settings.

**OK:** Go back to the **Printing Setup** screen.

> **Note:** Your settings will not be saved if you only click **OK**. You must click **Apply** first.

**G. Import / Export:**

Import a new configuration, or export your configuration to add to a different computer.

For more information, see **Importing and Exporting Configurations**.

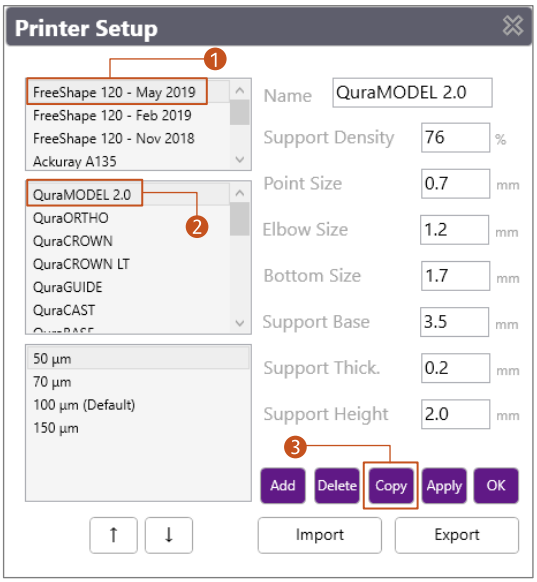
## Adding a New Resin

Alpha 3D is a simple printing software that customers can use to set, slice, and prepare their print files. Ackuretta calibrates resins from trusted partners and adds settings into the Alpha 3D interface.

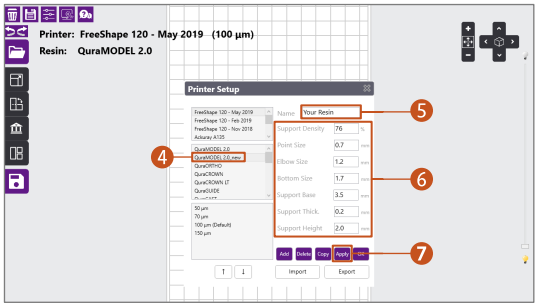
If you want to use third-party resins, you may add the resins in the software directly. This does not add the resins to the FreeShape 120 printer — you must additionally add resin profiles using the same name and layer thickness to the printer afterwards.

From the **Printing Setup** screen, click **Edit** to go to the configuration screen.

1. Choose the printer that you want to add resin settings for. Usually, you will choose the topmost FreeShape 120 version.
2. Choose a resin profile that you would like to use as a template. It is easiest to choose a similar type of resin for your application.
3. Click **Copy**. A duplicate of that resin will be added, with the suffix "_new".

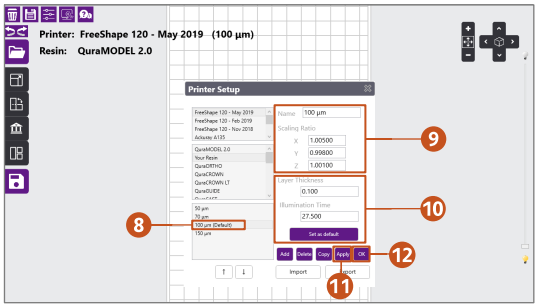


4. Select the new resin profile.
5. Change the name of this profile to the name of your resin.
6. (Optional) You can modify the default support settings for this profile. For more information, see **Supports**.
7. Click **Apply** to save your settings.



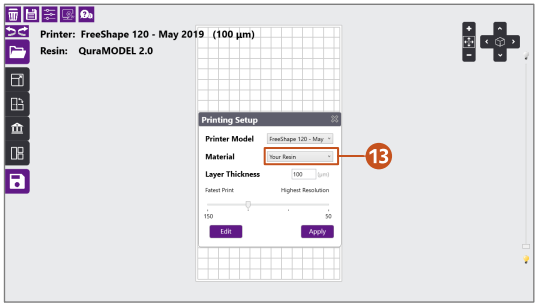

Do the following steps for every layer thickness that you want to use for this resin. If there are thicknesses that you do not want to use or test for, select those thicknesses and click **Delete**.

8. Select the layer thickness that you want to modify.
9. (Optional) Change the name of the thickness, or the scaling ratio.




**Note:** If you are unsure about the scaling ratio, it is best to leave the scaling ratio at 1.00000 for all of the X, Y, and Z dimensions. After you print a few times, you may find that you need to adjust the scaling ratio later.

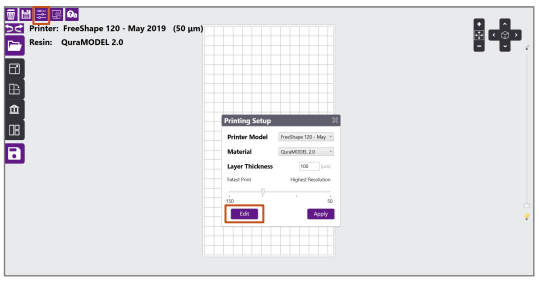
10. Set the **Layer Thickness** and the **Illumination Time** for that thickness.
    - The FreeShape 120 can only move in multiples of 0.01 mm. If you set the **Layer Thickness** to a value different than a multiple of 0.01 mm, the printer will round up 10 microns, distorting your print.
    - The **Illumination Time** is not the same for all thicknesses. You will need to test each layer thickness individually.
    - You can set this thickness to be the default by clicking **Set as default** here.
11. Click **Apply** to save your settings.
12. Click **OK** to confirm all applied settings.
13. When you go to the Printing Setup screen, your new resin appears as an option. All the layer thicknesses associated with that resin are available as well.



## Importing and Exporting Configurations

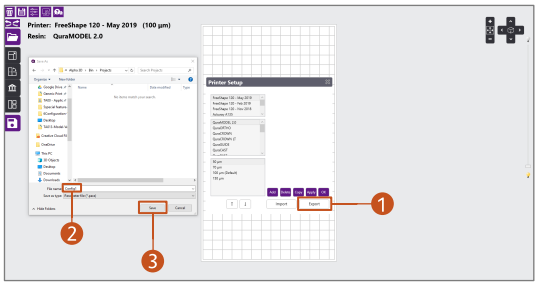
You may receive configuration files in PARA file format from the Ackuretta website or Ackuretta support. Alternatively, you may want to transfer old configuration files to your current software.

To import or export configurations, go to the **Print Setup** screen and click **Edit**.



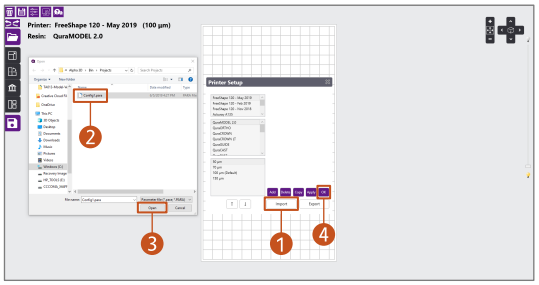

If you want to save your current configuration, use the **Export** function.

1. Click **Export**.
2. Choose a name.
3. Alpha 3D will save your configuration (.para) file in the default folder **C:\ACKURETTA\Alpha 3D\Bin\Projects**.




If you want to load a configuration file, click **Import**. Browse for your file with a .para extension.

1. Click **Import**.
2. Choose the .para file in your drive.
3. Click **Open** to import the .para file.
4. Click **OK** when you are finished.



Whip Mix®