Chapter 1: Installation

System Requirements

<table>
<thead>
<tr>
<th>Minimum</th>
<th>Recommended</th>
</tr>
</thead>
<tbody>
<tr>
<td>32-bit dual core 2 GHz CPU (e.g., Intel i3 or AMD Athlon)</td>
<td>64-bit quad core CPU (e.g., Intel i7 or AMD Phenom II X4/X6)</td>
</tr>
<tr>
<td>4 GB RAM</td>
<td>8 GB RAM</td>
</tr>
<tr>
<td>1600 × 900 display</td>
<td>1920 × 1080 display</td>
</tr>
<tr>
<td>Graphics card with 1 GB RAM</td>
<td>Graphics card with 2 GB RAM (e.g., NVidia GeForce 830 or AMD Radeon R7 M340)</td>
</tr>
<tr>
<td>1 GB of disk space</td>
<td>2 GB of disk space</td>
</tr>
<tr>
<td>Windows 7 SP1, 8.1, or 10</td>
<td>Windows 7 SP1, 8.1, or 10</td>
</tr>
<tr>
<td>Mouse, trackpad, or pen &amp; tablet</td>
<td>3-button mouse, or pen &amp; tablet</td>
</tr>
</tbody>
</table>

Installing Alpha 3D

1. Download the Alpha 3D installer from the FreeShape 120 page on the Ackuretta website: https://ackuretta.com/software/
2. Right-click the installer and select **Run as administrator**.
   - Turn off your antivirus software and firewall when installing Alpha 3D to avoid any compatibility issues.

3. Follow the on-screen instructions to complete the Alpha 3D setup procedure.
   - The installer asks you to uninstall any previous version of Alpha 3D.
   - You must accept the Software License Agreement to complete the installation process.
   - You can change the default installation path. The default installation path is: **C:\Ackuretta\Alpha 3D**

**Note:** Alpha 3D does not show a complete message or open automatically. Installation takes approximately one minute on a machine that meets minimum requirements.
Chapter 2: Getting Started

When Alpha 3D is first installed, the next step is to set up the print environment. This section provides you with the general information that is necessary to complete the setup procedure, and links you to any related information in this manual.

This section includes the following parts:

- Setting up Your Printer
- Software Overview
- System Settings
- Print Workflow

Setting up Your Printer

1. Double-click the Alpha 3D EXE file or shortcut to launch the software.
   
   - Ackuretta recommends running Alpha 3D as an administrator.
   
   - The Printing Setup window automatically appears when the application opens.

   Tip: You can open the Printing Setup window any time by clicking the Printing Setup button.

2. Optimize the settings for your prints and print environment.

<table>
<thead>
<tr>
<th></th>
<th>Printer Model</th>
<th>Select the most recent version of the Ackuretta printer that you are using.</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>Material</td>
<td>Select the material you are using. Most of the resins that are compatible with your machine are available. If your material is not visible, you can perform a direct slicing test via the “Resin Test” setting.</td>
</tr>
<tr>
<td>B</td>
<td>Layer Thickness</td>
<td>Move the slider to set the layer thickness. The layer thickness is automatically set according to the material you select.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Printer Model</th>
<th>FreeShape 120 - May</th>
</tr>
</thead>
<tbody>
<tr>
<td>Material</td>
<td>QuraMODEL 2.0</td>
</tr>
<tr>
<td>Layer Thickness</td>
<td>100 (μm)</td>
</tr>
<tr>
<td>Fastest Print</td>
<td>Highest Resolution</td>
</tr>
</tbody>
</table>

   D Edit  E Apply
You can add/edit/delete resin settings in Alpha 3D. Please refer to the Resin Setting section for more information.

Click the button to apply and save your settings.

### Software Overview

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

<p>| | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>A</strong></td>
<td><strong>Menu bar</strong></td>
<td>Functions related to saving or restarting your project and configuring your software.</td>
</tr>
<tr>
<td><strong>B</strong></td>
<td><strong>Print information</strong></td>
<td><strong>Printer</strong> and <strong>Resin</strong> settings. If an object is selected, it also shows the <strong>Model</strong> name.</td>
</tr>
<tr>
<td><strong>C</strong></td>
<td><strong>Toolbar</strong></td>
<td>The primary functions, such as undo/redo, open, slice and print, and buttons for opening other panels.</td>
</tr>
<tr>
<td><strong>D</strong></td>
<td><strong>Print area</strong></td>
<td>A view of the 3D object to print (including the base and supports).</td>
</tr>
<tr>
<td><strong>E</strong></td>
<td><strong>View controls</strong></td>
<td>Select a preset camera angle, zoom in or out, or check your slices. Refer to the View and Movement section for more information.</td>
</tr>
<tr>
<td><strong>F</strong></td>
<td><strong>Print statistics</strong></td>
<td>Basic print information, such as the layer count and the estimated volume of resin used.</td>
</tr>
</tbody>
</table>
1. Menu bar

<table>
<thead>
<tr>
<th>Action</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Delete All Models</td>
<td>Click to clear the print area.</td>
</tr>
<tr>
<td></td>
<td><strong>Warning:</strong> You cannot undo this step. We strongly recommend that you first save your file before clicking this button.</td>
</tr>
<tr>
<td>Save Project</td>
<td>Click to save your models, orientation, and supports. Your project is saved as a project file with the .i3dp extension. You can set the location by updating the file path in the System Settings.</td>
</tr>
<tr>
<td>Printing Setup</td>
<td>Click to set your printer, material, and layer thickness.</td>
</tr>
<tr>
<td></td>
<td><strong>Note:</strong> For more information, see Printing Setup.</td>
</tr>
<tr>
<td>System Settings</td>
<td>Click to set your output folder and language.</td>
</tr>
<tr>
<td></td>
<td><strong>Note:</strong> For more information, see System Settings.</td>
</tr>
<tr>
<td>About Alpha 3D</td>
<td>Click to view the software version and copyright information.</td>
</tr>
</tbody>
</table>

2. Print information

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

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

**Model:** This field shows the name of the selected model. This field is not visible if you have not selected a model.
### 3. Toolbar

<table>
<thead>
<tr>
<th>Icon</th>
<th>Name</th>
<th>Description</th>
</tr>
</thead>
</table>
| 🔄️ | **Undo / Redo** | If you make a mistake, you can use the undo and redo functions the same way as other programs.  
  - Click the **Undo** button to go back one step. Alternatively, press CTRL+Z.  
  - Click the **Redo** button to go forward one step. Alternatively, press CTRL+Y.  
  - Alpha 3D keeps a record of up to 20 actions. |
| 🗂️ | **Open** | Click to open a window on which you can select and import your STL or TRI files into the build area. |
| 🟢 | **Size** | Click to show the **Size** panel on which you can adjust the size of your model. |
| 🟢 | **Orientation** | Click to show the **Orientation** panel. You can rotate your print and select the best place for adding supports.  
  **Note:** For details about using the Orientation panel, see [Movement](#). |
| 🟢 | **Supports** | Click to show the **Supports** panel. You can add supports to your objects so they print correctly.  
  **Note:** For details about using the Supports panel, see [Supports](#). |
| 🟢 | **Layout** | Click to show the **Layout** panel. You can move your model or nest multiple models in the build area.  
  **Note:** For details about using the Layout panel, see [Movement](#).  
  Additionally, some items shown in this panel are covered in the [Special Features](#) section. |
| 🟢 | **Print** | Click to slice your print and output the result as an IBF file that you can send to your Ackuretta printer. |
# System Settings

Click the **System Settings** button to open a window on which you can set your computer-specific needs.

<p>| | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>A</strong></td>
<td><strong>File Path</strong></td>
<td>The location to where the completed files are added. Click the <strong>Browse</strong> button to navigate to the location in which you want to save files. The default file path is: C:\Ackuretta\Alpha 3D\Bin\Project</td>
</tr>
<tr>
<td><strong>B</strong></td>
<td><strong>Language</strong></td>
<td>Select the language for the software interface. Alpha 3D currently supports English, Chinese (Simplified), and Chinese (Traditional). Some functions still only appear as English text.</td>
</tr>
<tr>
<td><strong>C</strong></td>
<td><strong>Support</strong></td>
<td>Select the type of support to use. The available options are: General; Structure; and Simple. You can also manually make changes to supports on the Supports panel. For more information, see <strong>Supports</strong>.</td>
</tr>
<tr>
<td><strong>D</strong></td>
<td><strong>Open Output folder after slicing</strong></td>
<td>Select the checkbox to open the slice folder automatically when the slicing process is complete. This setting is selected by default.</td>
</tr>
<tr>
<td><strong>E</strong></td>
<td><strong>OK</strong></td>
<td>Click the <strong>OK</strong> button to apply and save your settings.</td>
</tr>
</tbody>
</table>
## Print Workflow

### To print

1. Import your STL, TRI, or i3DP file.
   - **A.** Click the Open button to open the window.
   - **B.** Select your file and click Open.
     - Hold down the *Shift* key to select multiple objects.

   **Note:** Make sure to inspect the objects after you import them to confirm that they are the correct files. For information about moving the camera, see *View and Movement*.

2. If necessary, go to the *Size* panel to make changes to the size of your print.
   - **A.** Key in a number or use the arrows to set a value in the textbox adjacent to any of the size fields. Press ENTER to apply the value.
   - **B.** Click the ![Link](image) button to constrain the dimensions. All dimensions are constrained when you first open to the Size panel.
   - **C.** Click the ![Reset](image) button to put the model back in 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. The values in the Scale X, Y, and Z textboxes adjust the size of the model in relation to the original size. The values in the X, Y, and Z textboxes are the dimension sizes of the printed object. Your model is shaded yellow when it fits within the boundary of the build area/print area. Otherwise, it is purple.
3. Move and orient your print to your ideal position.

   A. Open the Orientation panel to rotate, move, or put the object in a specific position for adding supports.

   B. Open the Layout panel to move your part or nest multiple objects in the build area.

   Note: For information about the Orientation and Layout panels, see View and Movement.

4. Open the Supports panel to add supports to your objects.

   A. Set up your support parameters.

   B. Click Auto Support to add supports.

   C. Click Manual Support to make changes to the supports.

       • When you finish making changes to the supports, click Manual Support again to see your completed supports.

   D. Increase the Base Thickness to add a base.

   Note: For more information about using the Supports panel, see Supports.
5. When your print is ready, click the **Print** button to open the print window. The *Print* window contains the following settings and options:

<p>| | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>A</strong></td>
<td><strong>Layer Thickness</strong></td>
<td>Drag the slider to make changes to the <strong>Layer Thickness</strong>.</td>
</tr>
<tr>
<td><strong>B</strong></td>
<td><strong>Job Name</strong></td>
<td>Key in a <strong>Job Name</strong> for the print and press <strong>Save</strong> to export your print file.</td>
</tr>
<tr>
<td><strong>C</strong></td>
<td><strong>Printing Setup</strong></td>
<td>Click the <strong>Printing Setup</strong> button to open the window on which you can make changes to the <strong>Printer Model</strong> or <strong>Material</strong>.</td>
</tr>
</tbody>
</table>
Chapter 3: View and Movement

View Controls

Alpha 3D contains several view controls located at the top-right of the screen. This section provides necessary information about the view controls, including how to use the mouse to change the view.

Click the Home button to reset the view.

Note: There is a fine grey line that shows the boundaries of the print area and a fine yellow line that shows the boundaries of your model.

Panning the Camera

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

Click the Home button to reset the view.

Rotating the Camera

Middle-click (scroll wheel) and hold to rotate and turn your camera. The change in view is related to the speed at which you move your mouse.
■ Zooming
Use the scroll wheel on your mouse to zoom in and out. You can also press the + or - buttons on the view navigator at the top-right of the screen.

■ Viewing Presets
Click the Direction button at the top-right of the screen to set the view to a specific perspective.

<p>| | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>Home</td>
<td>Change to the home view, which is in an isometric perspective and at 45° to the top, front, and right of the build area.</td>
</tr>
<tr>
<td>B</td>
<td>Top</td>
<td>Click through to rotate to the top view in 90° increments.</td>
</tr>
<tr>
<td>C</td>
<td>Bottom</td>
<td>Click through to rotate to the bottom view in 90° increments.</td>
</tr>
<tr>
<td>D</td>
<td>Left</td>
<td>Change to the left-side view, or click through to continue rotating the view in 90° increments.</td>
</tr>
<tr>
<td>E</td>
<td>Right</td>
<td>Change to the right-side view, or click through to continue rotating the view in 90° increments.</td>
</tr>
</tbody>
</table>
**View Slider**

Use the View Slider on the right-side of the screen to move up and down to a specific level of the model.

A. View Slider

B. Click the bottom Slice View button to view slices between the bottom slice and the View Slider level.

C. Click the top Slice View button to view slices between the top slice and the View Slider level.

**Note:** For better control, you can also press the UP and DOWN arrow keys on your keyboard to move the View Slider one slice at a time.

**Movement**

There are two different panels for moving your model in Alpha 3D: the Orientation panel and the Layout panel. Click either of the buttons in the toolbar to open the panel for each screen.

1. **Orientation** button
2. **Layout** button

**The Orientation panel**

- Move and rotate your object to put it in the best position for adding supports and printing.
- Make sure to set your object orientation before adding supports.
- Changing the orientation causes all supports to break; You must add the supports again if you change orientation.

**The Layout panel**

- Move your object or nest multiple objects in the build area.
- You can move objects that already have supports; The existing supports are maintained.
- Movement is the primary function of the Layout panel. Other functions related to the Layout panel are described in the Special Features section.
**Orientation**

The main function of the Orientation panel is to turn or move objects to be in the best position for printing.

The Orientation panel has the following options:

A. Rotate the object to a specific angle.
   - Click the arrows adjacent to the textbox to adjust the angle of the object in the X, Y, or Z direction.
   - Alternatively, type a number in one of the boxes and press ENTER. The object rotates to the specified angle.

B. Click any of the Object buttons to rotate the object in that direction. Click the same button again to rotate the object in the opposite direction.

C. Click the Reset button to put the object in its original orientation.

D. Click the Select Base button to select a face to be the bottom of the object.

In addition to the functions shown on the Orientation panel, you can move your part by dragging the object handles. Object handles are shown for the X, Y, and Z axes.

Tip: If an object handle is directly facing you, you must first move the camera to change your view to move that handle.

Alternatively, you can click and drag the part to move it. This has the same effect when you are on the Layout panel.
When you use the **Select Base** function, the cursor changes 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 object turns so that the selected face is on the base.

---

### Layout

The main function of the **Layout** panel is to move objects around the build area.

- Click-and-drag an object to move it around the build area.
- If you drag an object out of the build area, Alpha 3D automatically puts it back within the boundary.

If you do not want to print the complete object, use the **Z-position** function to decrease the vertical position of the object so that the unwanted portion is below the platform.

- When you first open a 3D file, the object is always in the middle of the build area and directly on the platform.

- Use the arrows adjacent to the Z-position textbox to adjust the object’s vertical position, or type in a value and press ENTER. You can key in a negative value to move the object below the platform.

**Tip:** This function is useful when 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

To 3D print a model, many consecutive layers are printed and each layer is printed on the previous layer. If your model has an overhang which is not supported by anything below, you must add support structures to make sure that it prints successfully.

Adding Supports

Every part of the print must be securely connected to the print platform when printing on resin-based printers (i.e. DLP, SLA, or LCD). Any part that is not connected to the print platform can be misplaced and cause the print to fail. Alpha 3D can automatically add supports. You still have the option to make changes to the supports.

1. Select your model and click the Supports button to open the Supports panel.

2. Set up your support parameters.

<table>
<thead>
<tr>
<th>Setting</th>
<th>Description</th>
<th>Recommended value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Type</td>
<td>The style of support. The available options are General, Structure, and Simple.</td>
<td></td>
</tr>
<tr>
<td>Point Size</td>
<td>The diameter of the top of the support where it touches the model.</td>
<td>0.6 - 1.2 mm</td>
</tr>
<tr>
<td>Elbow Size</td>
<td>The diameter of the bend point of the support.</td>
<td>1.1 - 2.2 mm; Larger than the Point Size by 0.5 - 1.0 mm</td>
</tr>
</tbody>
</table>
### Setting | Description | Recommended value
--- | --- | ---
**Bottom Size** | The diameter of the bottom of the support where it touches the base. | 1.6 - 3.2 mm; Larger than the Elbow Size by 0.5 - 1.0 mm
**Support Density** | The distance between two adjacent supports when using Auto Support, with the exception of supporting local lowest points. | 5.0 - 7.0 mm, depending on Point Size.
**Internal Supports** | Select this checkbox to allow supports to touch the model on both the bottom and top. | NA
**Sup. Base Thick.** | The thickness of the footing at the bottom of each support. | 0.2 mm
**Slope Angle** | Supports are only added to the model if their angle to the bottom is less than this value. | 45-55
**Base Thickness** | Increase this number to add a base to all your prints. Add this base after adding supports to your print. | 0.2mm for Ackuretta printers.
**Support Height** | The model is automatically raised to this height when you add supports. | 1.5 - 2.5 mm

**Note:** The support parameters affect the following areas.
3. Click **Auto Support** to automatically add supports to your model.

- **Auto Support** uses the same orientation, raises the model to your **Support Height**, and then adds the supports.

- Alternatively, click **Orientate and Auto Support** to allow Alpha 3D to analyze your model and attempt to select the best orientation.

4. Click **Manual Support** to make any necessary changes to the settings that were automatically added.

- The interface changes to **Manual Support Mode**. Only the support points are visible in this mode.

- Your cursor changes to show a support point. Click on a red point to add a support point at that location.

- Existing support points turn red when you move the cursor over them. Left-click a red point to delete it.

- If you are in **Manual Support Mode** and make changes to the support settings, any new supports use the new settings, but existing supports are unchanged.

- We strongly recommend adding supports all over the red areas that are defined by the **Slope Angle** parameter. Supports must be spaced so that one complete support can fit between any two supports.

- Click **Manual Supports** again to exit **Manual Support Mode** and view your completed supports.

- For more information, please refer to **Local Minimum Points** and **Editing Individual Supports**.
5. Key in a value or use the arrows to set the **Base Thickness** and add a base.

- We strongly recommend you set a Base Thickness of 0.2 mm for Ackuretta printers.

### Editing Individual Supports

After adding a support, you make changes to that specific support to make sure that it has the best connection.

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

- The support intersects the model, attaching at an unintended point.
- The model 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 make changes to many supports, we strongly recommend that you change your support settings and apply the **Auto Support** function again. Then, you may only need to make changes to a few supports.

To make changes to an individual support:

1. Click the **Manual Support** button to go into **Manual Support Mode**.
2. Move your cursor over a specific support point so that it changes color to red.
3. Middle-click that support point. Alpha 3D goes into **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 panel. You can increase or decrease these values.

<table>
<thead>
<tr>
<th>Elbow Point Position</th>
<th>Support Thickness</th>
</tr>
</thead>
<tbody>
<tr>
<td>The Elbow Point is the bend point of the support. The Elbow Point Position is based on the center of the model. If you increase the X, Y, and Z positions, the bend point moves accordingly in that specific direction.</td>
<td>If you increase this value, the support adds a small base with an angled detachment point to the support. Ackuretta printers do not require these types of supports, but they are an option if you prefer to use them.</td>
</tr>
</tbody>
</table>

5. Click **OK** when you have made any necessary changes.

**Note:** Make sure that the angle of the Elbow Point is as straight as possible so that the support is structurally strong. If the angle is too small, the support might break at the Elbow Point and cause the print to fail.

Completely vertical supports are best. If you must add an angle to your support, do not allow the Elbow Point angle to be less than 45°.

Decrease the Z value (i.e. vertical axis) for Elbow Height Position to straighten your support.
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 are first printed in a resin-based printer.

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

Local minimum Point A (as shown in Image 1.) is connected to the platform by a support. The layers of the support hold the first layers of Point A so that subsequent layers are held together from that place.

However, there is no support for local minimum Point B. If the light source cures that local minimum point, the Print Platform is too far away, and the cured layer stays 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 is called an “island” — it is solid material surrounded by liquid in the vat of resin.

In Alpha 3D, the Auto Support function automatically searches for local minimum points before adding other supports. If many supports are clustered closely together after using Auto Support, there can be many local minimum points in that place on the model.

Image 2 shows several locations in which many support points are clustered together because there is a high density of local minimum points.

To check your print for local minimum points and to add necessary supports to specific areas:

1. Set your View to the bottom.
2. Click the Manual Support mode button to go into that mode.
3. Use the View Slider to move the model up until you see some areas of the print.
4. If you see an area that appears from the bottom and with no supports, add a support to the bottommost point.
Chapter 5: Special Features

■ Saving Objects

To save an object, including the orientation, supports, and any other settings in Alpha 3D, use the Save function.

1. Click the Save button to open a window.
2. Set the location and file name.
3. Click Save.

■ Duplicating Objects

To make a copy of an object (including supports), use the Duplicate function.

1. Click the Layout button.
2. Select the object.
3. Key in a value or use the arrows to set the number of Duplicates you want to make.
4. Click Create.

■ Nesting

Alpha 3D can automatically move parts onto the build platform. When it does, it makes sure that the parts have enough space to printing without issue.

1. Click the Layout button.
2. Select an object if you want to move only one object. If you want to move all objects, do not select any objects.
3. Key in a value or use the arrows to set the size of the Gap to leave between each object.
   • Ackuretta recommends a minimum Gap distance of 1.0 mm, but 2.0 mm is better.
4. Click Nesting. Alpha 3D automatically optimizes the location of the objects in the print area.
5. All objects move onto the platform, starting in the center of the print area.
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. 3.0 mm
B. 8.0 mm

### Adding Text

Alpha 3D can also add a physical label to an object. The text label either embossed or debossed on the object, depending on your settings.

1. Click the **Layout** button.
2. Select an object.
3. Key in your text or label into the **Texts** field.
4. Select whether you want the text set **Outward** or **Inward** from the model (i.e. embossed or debossed, respectively).
5. Use the arrows to set the **Height** of the text. Ackuretta recommends a height of at least 2.0 mm.

6. Choose an angle of the selected part where you want to add your text.
7. Click **Label**. Your text appears 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 model.
8. Change the view so that your text appears where you want it printed on the model.
9. When the text label is in the correct position, click anywhere outside of the object. The text label is added to the model.
Chapter 6: Configuration

Configuration Screen

All Alpha 3D configuration is performed on the configuration window, which is labeled Printer Setup.

1. Click the Printing Setup button on the menu bar to open the Printing Setup window. The same window is automatically shown when you start Alpha 3D.
2. Click Edit to open the Printer Setup window.

The Printer Setup window shows the following items:

| A | Printer Settings | Every printer has its own set of materials and settings. You select your printer here to update the other sections on this window with the details for that specific printer. |
|   |                 | **Note:** You cannot make changes to any settings in this section. |

| B | Material Settings | Select a material on the list to view or make any necessary changes. The settings for the selected material are shown in the Settings Display. For more information, see Adding a New Resin. The parameters each material are related to its thickness. The values in the Settings Display automatically update when you select a material. The most important settings for each material are shown in the Thickness Settings. |

| C | Material          | 50 μm | 70 μm | 100 μm (Default) | 150 μm |
|   | Settings         |       |       |                  |       |
| D |                  |       |       |                  |       |
| E | Name             | 50 μm |       |                  |       |
| F | Scaling Ratio    | 1.00110 | 0.99956 | 1.00137 |       |
| G | Layer Thickness  | 0.050 |       |                  |       |
|   | Illumination Time| 22,000 |       |                  |       |
### Adding a New Resin

Alpha 3D is a simple printing application that you can use to set, slice, and prepare your print files. Ackuretta calibrates resins with trusted partners and adds optimized settings to the Alpha 3D interface. You can also easily add 3rd-party resins to the application.

**Note:** Adding resins to Alpha 3D does not add the resins to the FreeShape 120 printer; you must also add the resin profiles to the printer, and make sure to use the same name and layer thickness.

---

| C | Thickness Settings | The layer thicknesses for your selected material are shown here. One thickness is usually set as the Default. Click the values to make any necessary changes in the **Settings Display**.
|---|---------------------|
| D | Order Arrows | You can click either of the two arrows to adjust the sequence of the settings. The materials are also shown in the same sequence in the **Material** dropdown list on the **Printing Setup** window.
| E | Settings Display | This section shows all of the settings that are available for the selected printer, material, or thickness.
| F | Save Options | The buttons here allow manage the settings as follows:

- **Add**: Add a new setting based on the name shown for the setting.
- **Delete**: Remove the selected setting.
- **Copy**: Duplicate the selected setting. The new setting has “_new” appended to 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** window.

**Note:** You must first click **Apply** to save settings. Your settings are not saved if you only click **OK**.

| G | Import/Export | Import a new configuration or export the current configuration.
For more information, see **Importing and Exporting Configurations**.
To add a new resin, you must first click **Edit** on the **Printing Setup** window to open the **Printer Setup** window. Then:

1. Select the printer for which you want to add the resin settings. You usually select the most recent FreeShape 120 version.
2. Select the resin profile to use as a template. It is easiest to select a resin with the same qualities.
3. Click **Copy**. A duplicate of that resin is added, with "_new" appended to the original name.

4. Make sure the new resin profile is selected.
5. Change the name of this profile to the name of your resin.
6. (Optional) Make any necessary changes to the default settings for this resin.
7. Click **Apply** to save your settings.
Complete the following procedure for each layer thickness that you want to use for this resin. If there is a thickness that you do not want to use or test, you can select that thickness and click Delete.

8. Select the layer thickness that you want to edit.
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 the X, Y, and Z dimensions. After several prints, you will know whether you need to make changes to the scaling ratio.

10. Set the **Layer Thickness** and the **Illumination Time** for that thickness.
    • FreeShape 120 can only move in multiples of 0.01 mm. If you set the **Layer Thickness** to a value that is not a multiple of 0.01 mm, the printer rounds up 10 micrometres, distorting your print.
    • The **Illumination Time** is not the same for all thicknesses. You must test each layer thickness.
    • You can click **Set as default** to set this thickness as the default thickness.
11. Click **Apply** to save your settings.
12. Click **OK** to confirm all applied settings.
13. When you go to the **Printing Setup** window, your new resin appears as an option. All layer thicknesses associated with that resin are also made available.
Importing and Exporting Configurations

You can get 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 Printing Setup screen and click Edit.

The Export function allows you to save your current configuration.

1. Click Export.
2. Key in a name for the configuration.
3. Click Save. Alpha 3D saves your configuration file (with the PARA extension) in the default folder. The default folder is C:\ACKURETTA\Alpha 3D\Bin\Projects

The Import function allows you to load a different configuration file.

1. Click Import.
2. Choose the PARA file on your drive.
3. Click Open to import the PARA file
4. Click OK when you have completed importing the configuration file.
Chapter 7: Print

Selecting Your Printer and Material

Click the Printing Setup button to open the Printing Setup window where you can set your Printer Model and Material:

A. Select your Printer Model on the dropdown list.
B. Select a Material on the dropdown list.

Setting the Layer Thickness

To adjust your Layer Thickness:

A. Drag the slider to the related position for the Layer Thickness you want. Depending on the material, you may want 75, 100, 150, or 200µm.

Note: You must click and drag the handle on the slider to set the Layer Thickness.
## Print Configuration

Click the **Edit** button to open the **Printer Setup** window on which you add/edit/delete the available options for **Printer Model**, **Material** and **Layer Thickness** settings.

A. **Printer Model**
B. **Material**
C. **Layer Thickness**
D. **Export** your Printer Setup config file (*.para)

**Note:** You must key in a new filename and click the **Save** button to export the config file (*.para)

E. **Import** a config file (*.para) from your computer, a USB flash drive, or over the web.

## Print

When your model is ready to print, you must give the print a **Job Name** and save the file in IBF format. Click the **Print** button to open the **Print** window.

A. Key in a **Job Name** for the print.
B. Click **Cancel** (if necessary) to go back and open the **Printer Setup** window to adjust your **Printer Model**, **Material**, or **Layer Thickness**.
C. Make sure all settings are correct, then click the **Save** button. Alpha 3D shows the progress of the slice and save process, and then opens a window with the location of the new files.

**Note:** The file path is set on the **System Settings** window. The current date and hour are added to the filename.

The number of **Layers** and **Volume** of material is automatically calculated by Alpha 3D.