ALPHA AI

User Manual

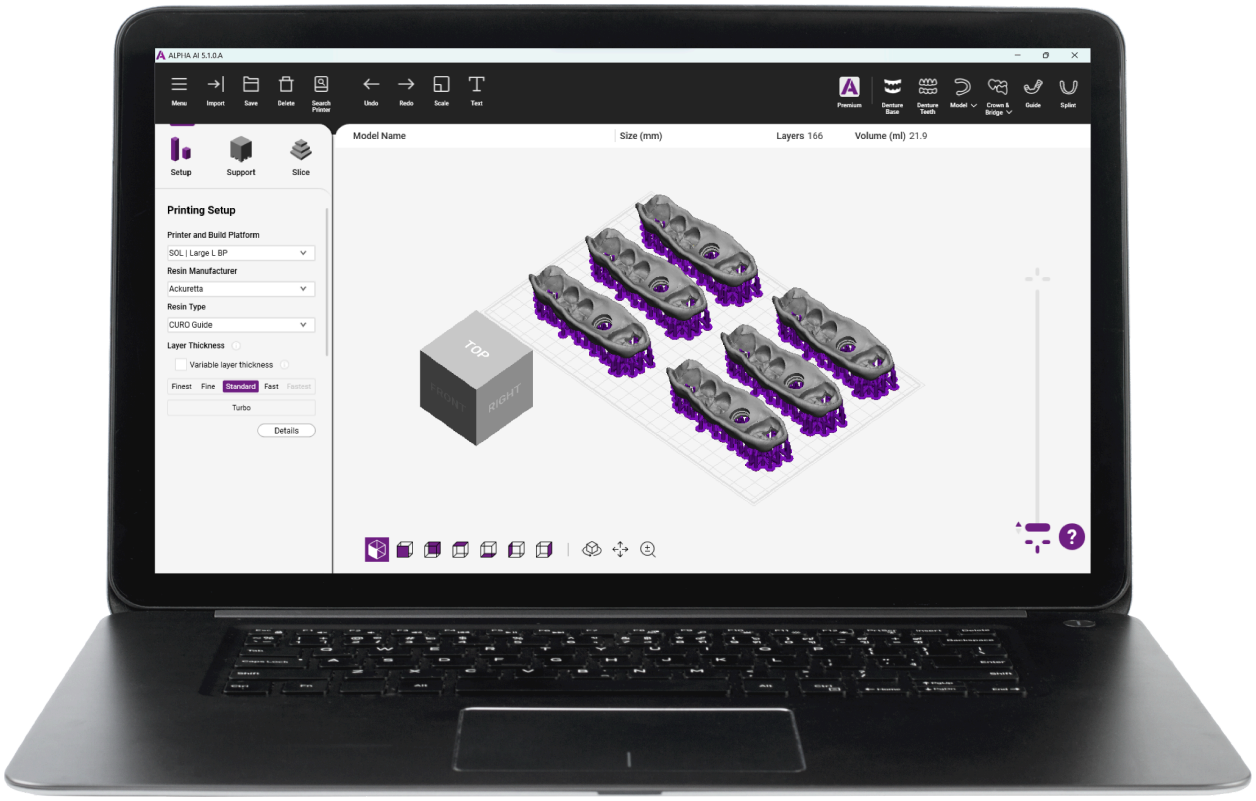
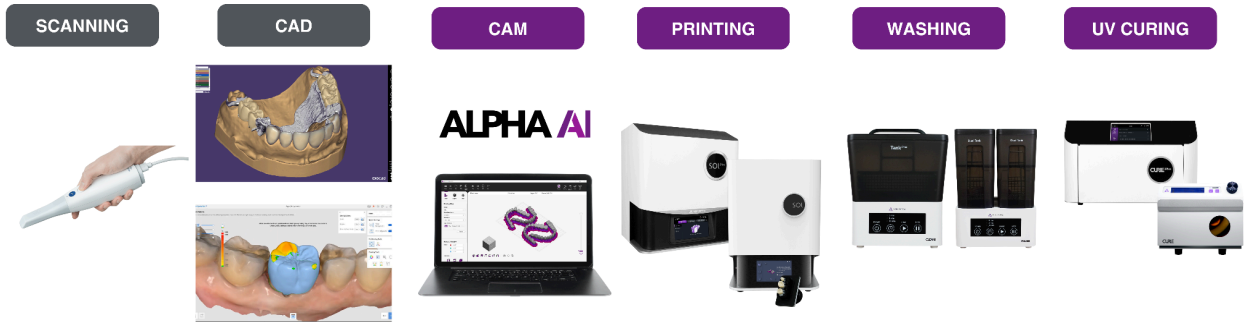


Table of Contents

1. Introduction to ALPHA AI

a) Digital Workflow



A digital workflow is a comprehensive digital system used in modern dentistry to streamline designing and manufacturing of dental restorations such as splints, crowns, bridges, veneers, and more.

Dentists take an impression of patients using an intraoral scanner to capture precise 3D images of the teeth and gum or do a 3D scan of manual impressions. From this acquisition, the model is designed in specialized CAD software (exocad, 3Shape, Medit, Clinux, and DentBird) to turn the scan into a complete geometry and create personalized prosthetics and appliances to send to ALPHA AI.

b) ALPHA AI

ALPHA AI takes the center of the digital workflow to seamlessly connect application design with Ackuretta's 3D Printing Solution, reducing the time and effort required to produce high-quality dental restorations.

ALPHA AI transforms a designed dental appliance into a printable 3D design. The CAM software includes 3D tools necessary to nest, orient, generate support, and slice files. The software is free to install and use. Users can also upgrade to a for-purchase ALPHA AI Premium version; which includes AI modules that generate AI-optimized orientation and support design for prostheses or appliances and an STL export option.

As the final step of the CAM design, applications can be sent via a web browser or USB key to the Ackuretta Printer (SOL, SOL Plus, DENTIQ, FreeShape) to be used.

2. Installation and Requirements

a) System Requirements

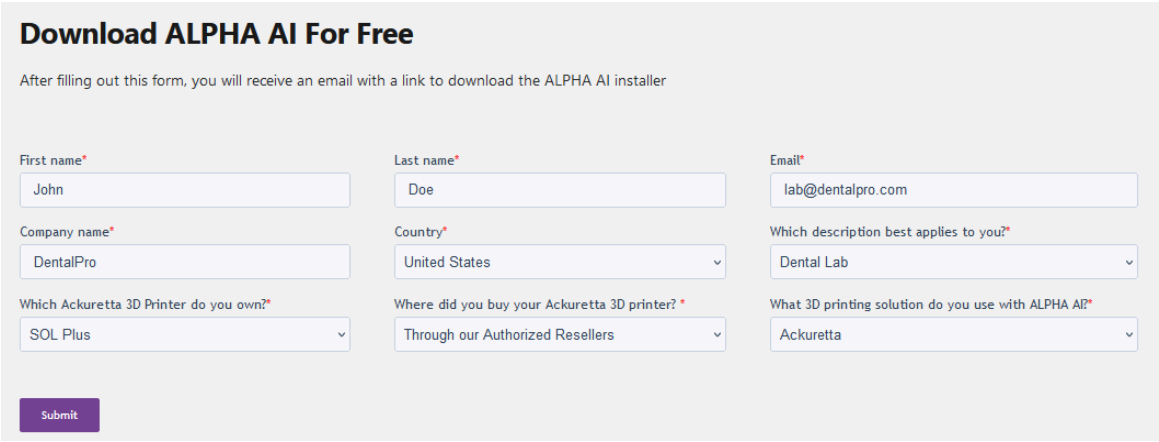
ALPHA AI requires a Windows operating system and a stable connection to the internet due to requests made to our server. Make sure your computer can connect to the following server:

- Server URL: server-n1.ackuretta.com
- Server IP address: 54.159.232.131
- Server port: 9527

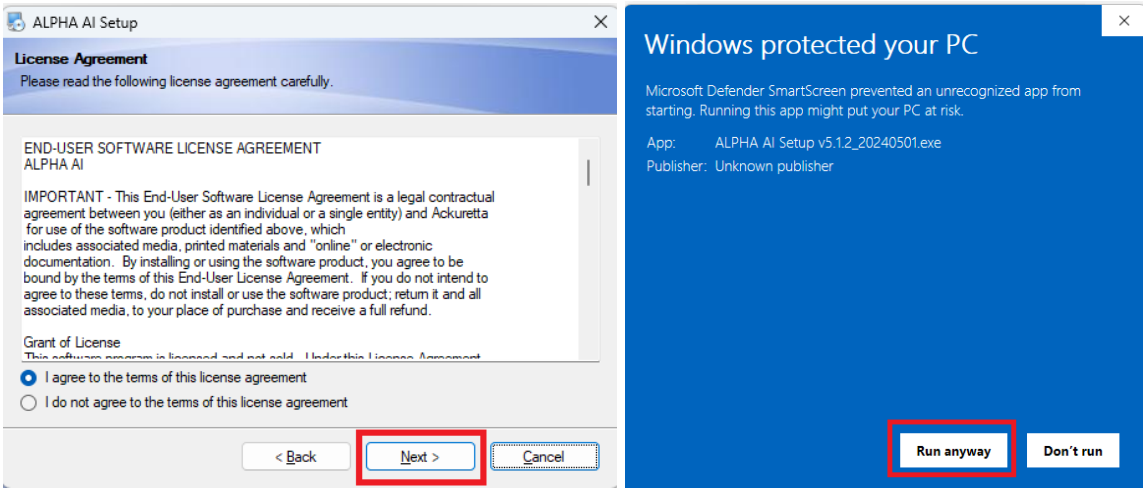
Minimum	Recommended
32-bit dual core 2 GHz CPU (e.g., Intel i3 or AMD Athlon)	64-bit quad core CPU (e.g., Intel i7 or AMD Phenom II X4/X6)
4 GB RAM	8 GB RAM
1600 x 900 display	1920 x 1080 display
Graphics card with 1 GB RAM	Graphics card with 2 GB RAM (e.g., Nvidia GeForce 830 AMD Radeon R7 M340)
1 GB of Disk space	2 GB of Disk space
Windows 11, 10, 8.1 or 7 SP1	Windows 11, 10, 8.1 or 7 SP1
Mouse, trackpad or pen & tablet	3-button mouse, or pen & tablet

b) ALPHA AI Download and Installation

1. Fill out the form from ALPHA AI webpage and fill in your information to receive the download link by email.

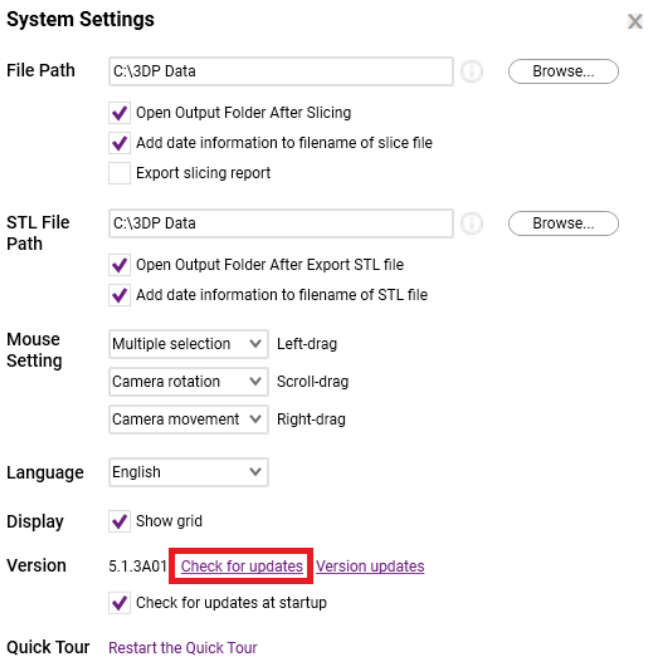


2. Install the file directly from your “Downloads” folder. In case Windows Defender flags the installation, click on “More Info”, then “Run anyway”.



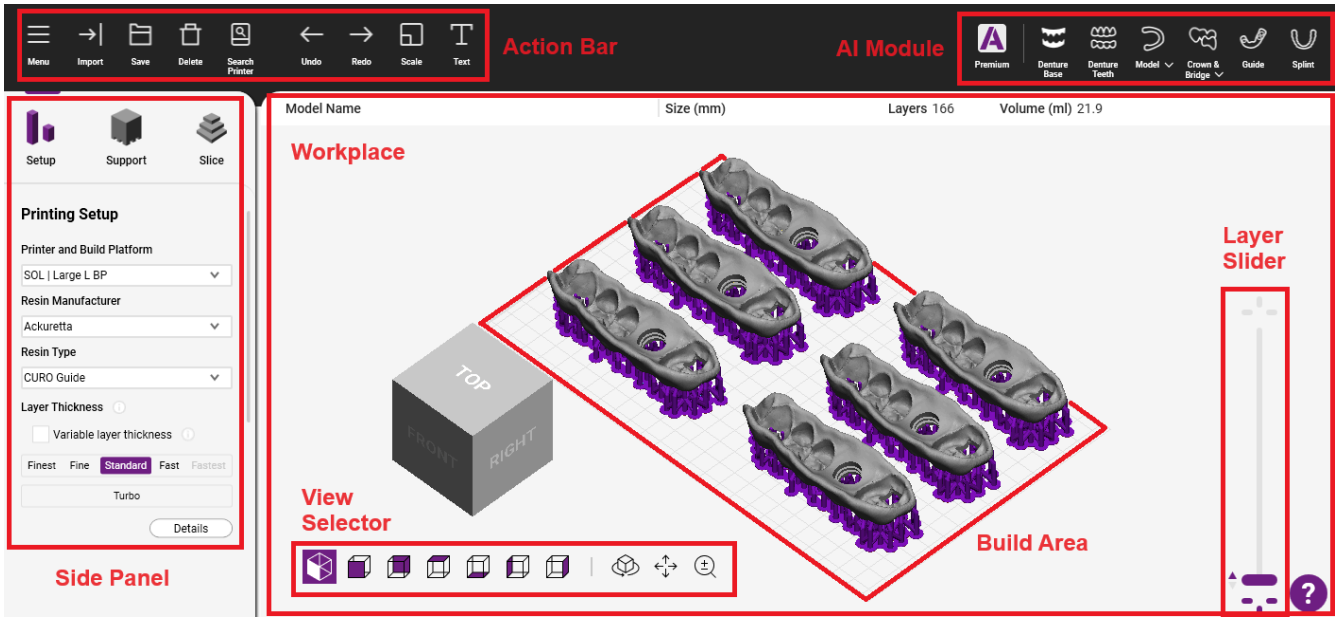
c) Software update

We are continuously updating ALPHA AI, you can manually update the software in **Menu**, **System Settings, and Version Update.** You can consult the list of updates in this page:



3. Interface and Language

a) Interface



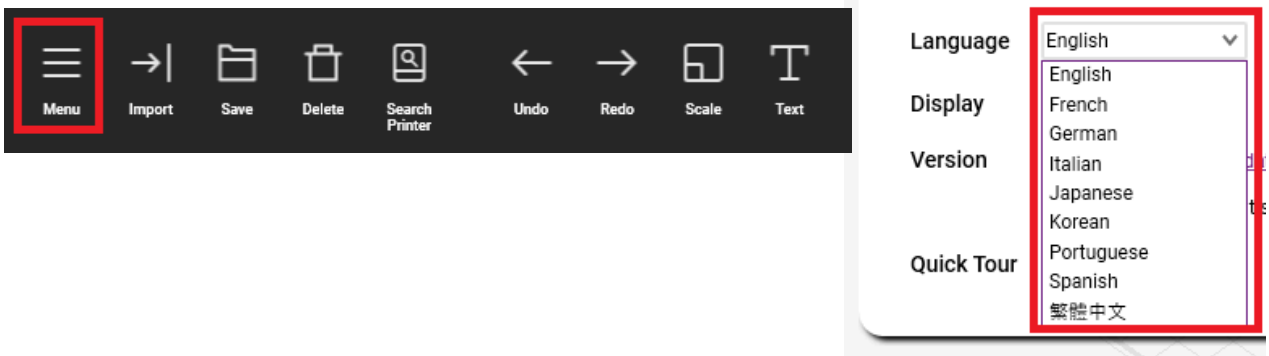
ALPHA AI Software's main interface is divided into the top bar that includes the Action Bar and AI Module, the Workspace, and the Side panel for the Setup Panel, Support Panel, and Slice Panel.

The Side panel can be dragged to fit on the right or left of the Workspace.

b) Language

The language can be changed by following these steps:

1. Go to the Action Bar in the top left corner and select "Menu", then “System settings” to access all the ALPHA AI-related setups.
2. In the Language field select your preferred language. For any other language requests please [Open a Support Ticket](#).

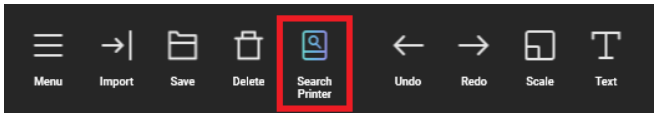


4. Software Setup

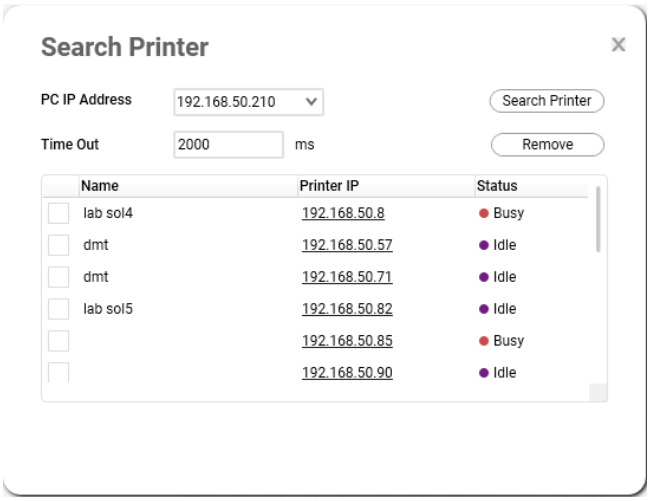
a) Search Printer

To find a specific printer connected to your network, follow these steps:

1. Select "**Search Printer**" in the Action Bar, ALPHA AI will list all printer IP addresses using the same network.



2. Inside the window Select the "**Search Printer**" function if your 3D printer IP address is not listed. If your printer is connected to the same network as your computer, it will appear in the list below.

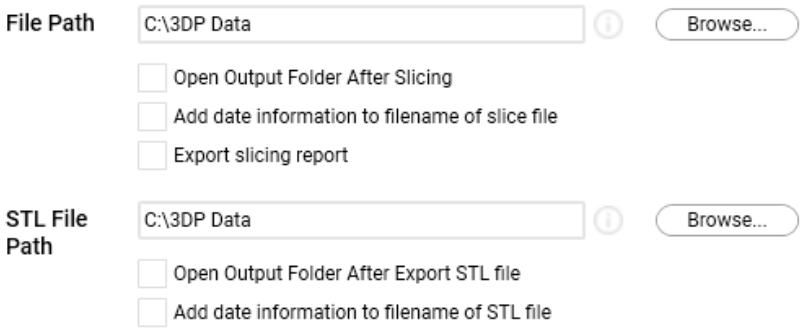


If your printer is not listed after the "**Search Printer**" search is complete, please ensure the WiFi or ethernet is correctly set up for both your printer and computer (Watch this video to see how to set up your printer with WiFi or Ethernet). Ensure the firewall does not prevent a connection.

b) File Path

To change the export path of the file, follow these steps:

1. Go to **System Settings, File Path**
2. Change the export path of the file by entering the desired location. If you have purchased ALPHA AI Premium, you will also be able to change the path of the exported STL file.



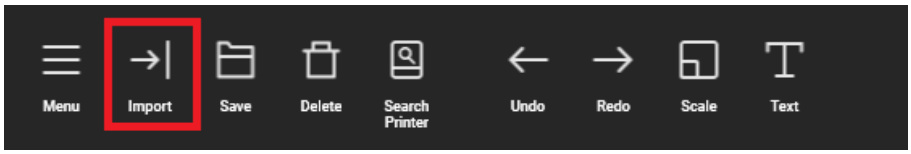
c) Other Settings

Go to the **Menu** then **System Settings** to change the mouse setting, display a grid, verify and update the software version, and check the update changelogs.

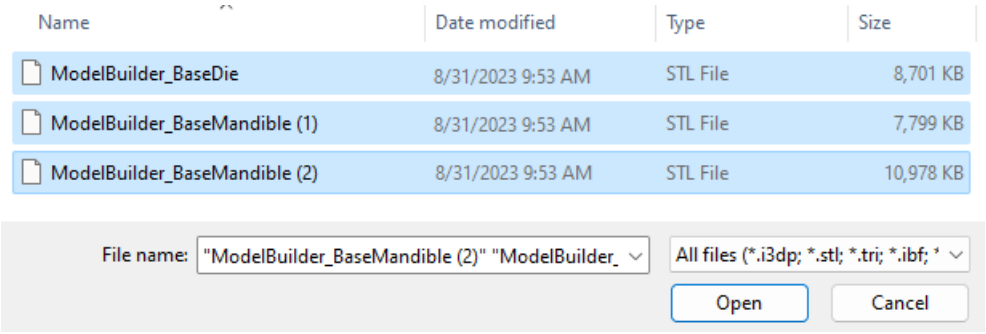
5. Application Setup

a) Import File

1. Reach the Action Bar and select "**Import**" or use CTRL+ O to import your STL file.



2. To import multiple files simultaneously. Press the CTRL key on your keyboard, select all files that you want to print, and drag and drop them into the open ALPHA AI window.

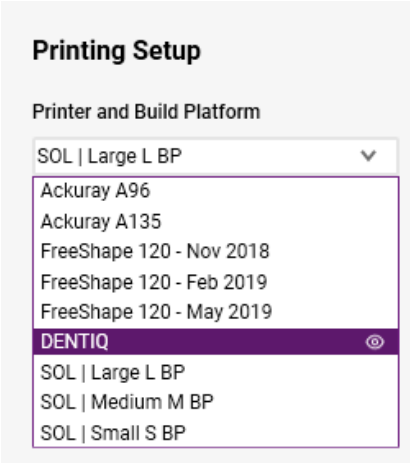


ALPHA AI supports the following file types (STL, TRI, ACKU, I3DP, IBF):

- STL is a file format native to the stereolithography CAD software. It can be generated from a scan or a Computer-aided design (CAD).
- TRI files are 3D mesh files created in the text-based TRI format.
- ACKU is the latest generation slicing file that includes the design with support.
- I3DP is the project file that saves the design with orientation and support.
- IBF was the previous slicing file not used anymore since January 2024 for SOL and DENTIQ. It is still in use for the Ackuretta FreeShape - version November 2018.

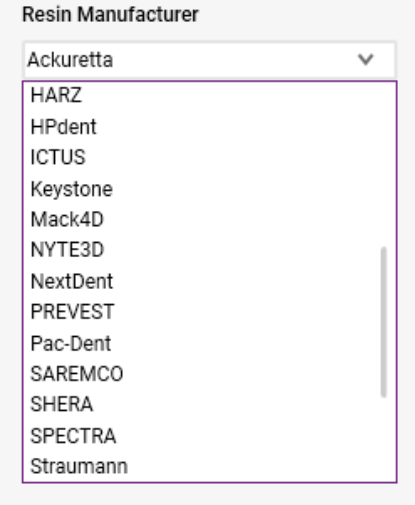
After installation, you will find several STL examples of dental applications under "C:\3DP Data\Sample STL\". You can also find sample files in the USB drive provided with your Concierge Service or Concierge World box.

b) Printer and Build Platform



In the Setup section, select your printer and build platform directly.
You can hide printers that are not relevant by clicking on the eye icon.

c) Resin Manufacturer and Resin Type



Resin Manufacturers are listed alphabetically. Select the desired resin manufacturer to display all the calibrated resins for the selected printer and build platform.

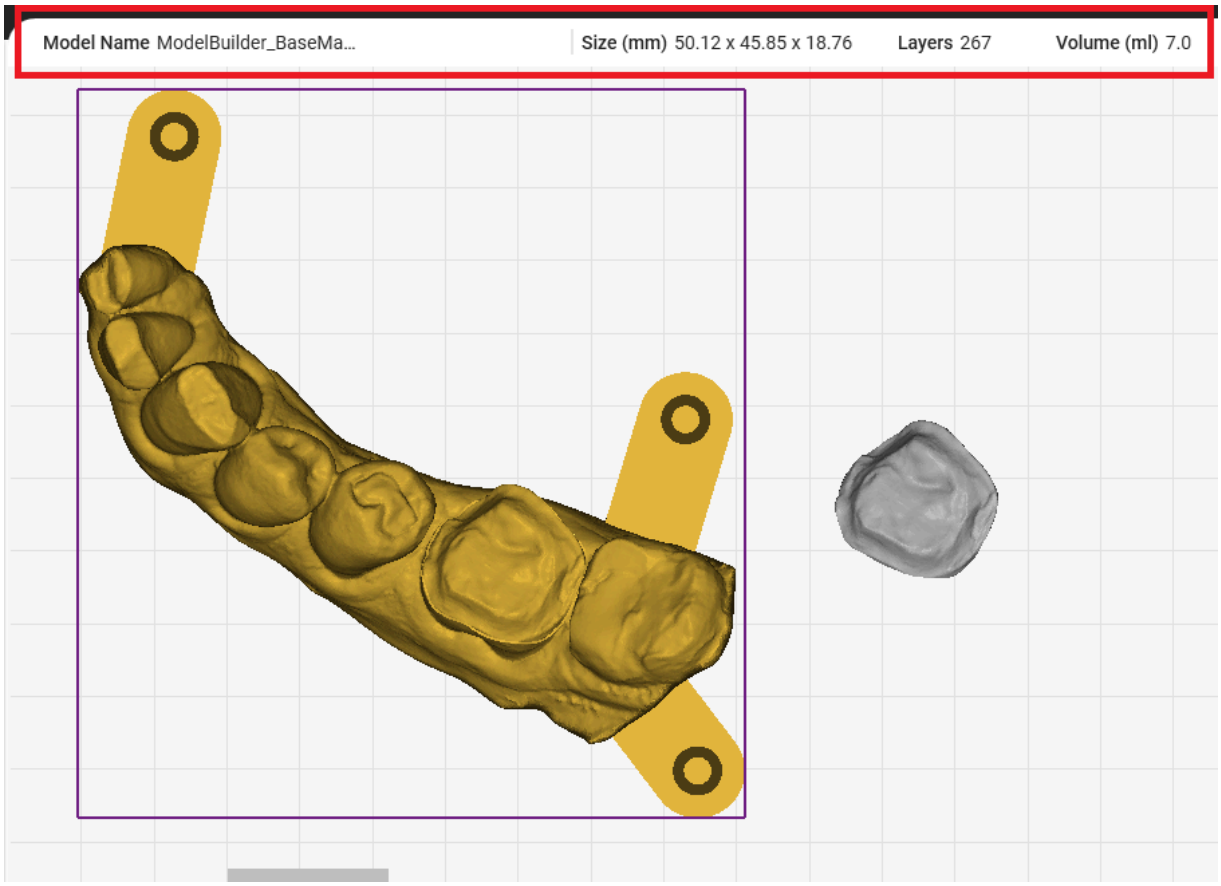
Note:

- If you are not sure about which resin to select, contact us directly and we can recommend a resin for your use case. Find the list of validated resins for each printer here: SOL Plus, SOL, DENTIQ.

- If a resin appears as not validated/under validation the resin is calibrated and awaiting validation by the resin manufacturer. Subscribe to the Ackuretta newsletter for updates on resin validations.

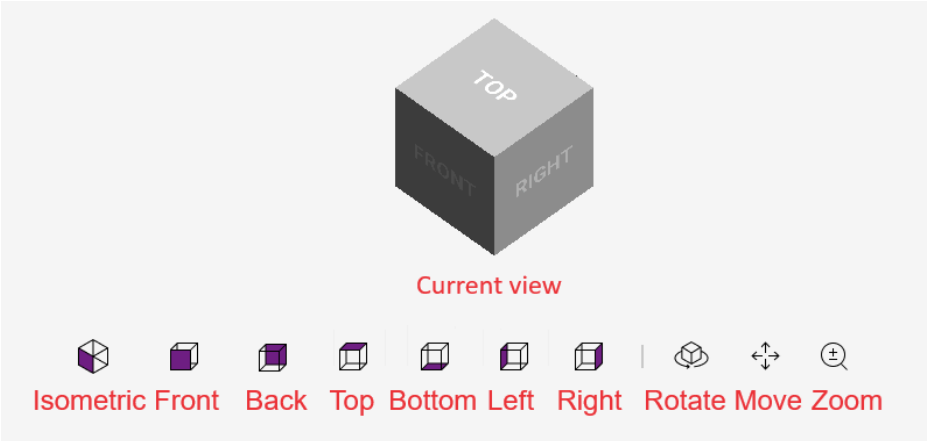
- If a resin doesn't appear on the software, fill out the Resin Calibration Request so that an Ackuretta representative can check the current calibration process.

d) File Details



Imported STL(s) will appear in the Build Area. When selecting an STL file, the file name, size, layers and the minimal volume of resin required will be displayed on the top of the Workspace.

e) View Selection



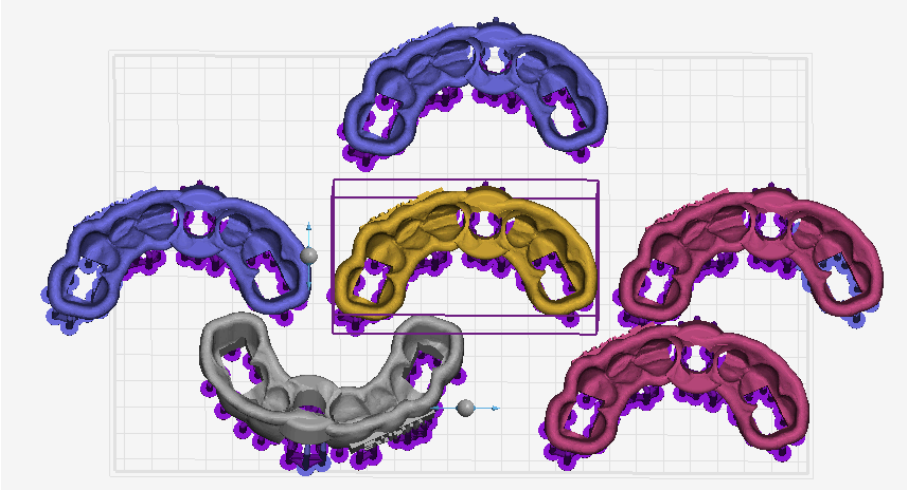
On the bottom right side of the Workspace, the View Selector buttons will let you change the application's point of view n to inspect it from different angles.

- Rotate view: View the application from different angles.
- Move view: Pan the area in view.
- Zoom view: Zoom in or out of the current viewpoint.

d) Application Placement

Once a file is imported, the previously designed application appears in the Build Area. When selecting the application, it will become yellow. From there you can drag the file using your mouse.

For a single print, we recommend placing the application in the middle of the build platform to increase the accuracy of the print.



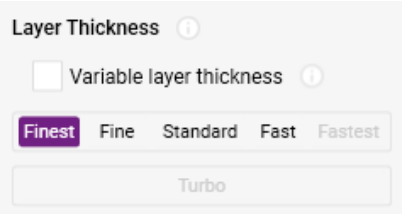
Different colors will help to determine if the file is correctly placed:

- **Gray**: The application is correctly placed on the build platform and unselected.
- **Yellow**: The application is correctly placed on the build platform and selected.
- **Blue**: The application is not fully on the build platform and will not print correctly.
- **Red**: The application is colliding with another application.

e) Layer Thickness and Variable Layer Thickness

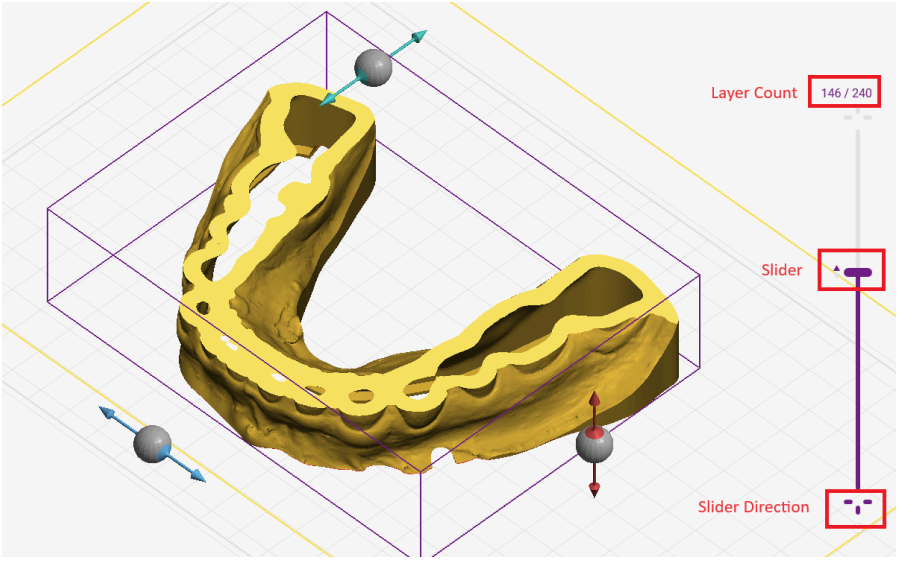
Layer Thickness is often called Z-axis resolution and describes the thickness of each layer. The thinner the layer, the smoother the printed application's surface will be. Conversely the thicker the layer, the printer will print faster. We recommend a low layer thickness for restorative prints to ensure a perfect fit during placement.

There are several layer thicknesses available for the print:
Finest (50µm), Fine (70µm), Standard (100µm), Fast (150µm), Turbo (150µm).
However, not all printers, build platforms and resins can be printed at all layer thicknesses due to technical limitations.

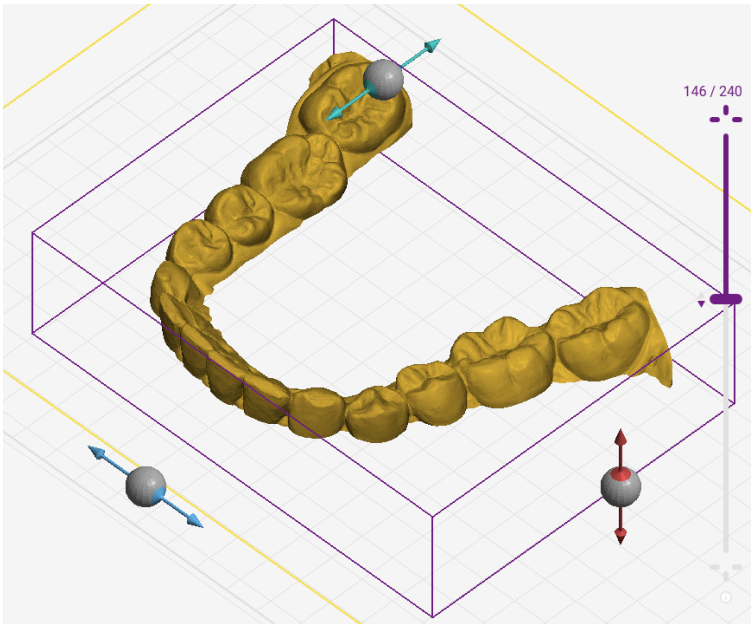


f) Layer Slider

The Layer Slider allows you to check the design layer by layer. The Slider will allow you to navigate through the layer by moving up and down or using the up key and bottom key. The current Layer Count and the total amount of layers are indicated on the top of the Layer Slider.



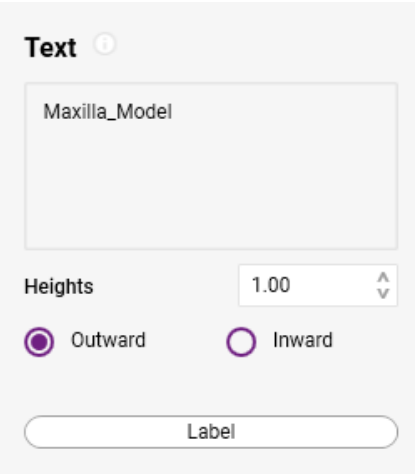
The Slider Direction on the bottom and top of the slider adjusts the direction of the slider from bottom to top or vice versa.

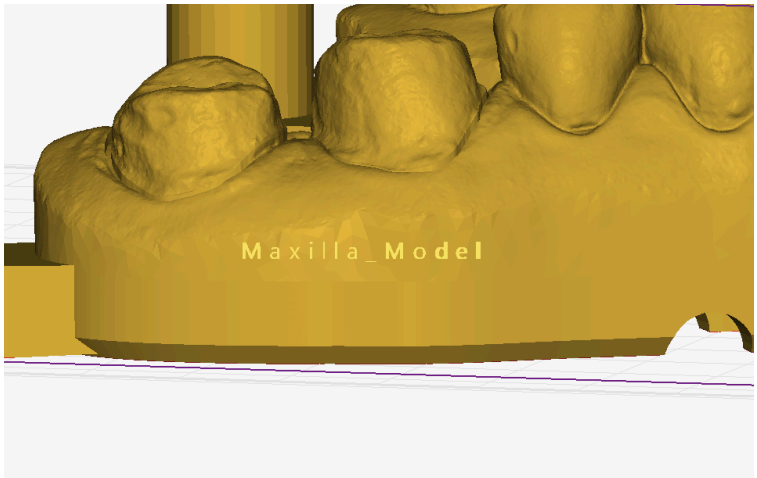


g) Adding Text

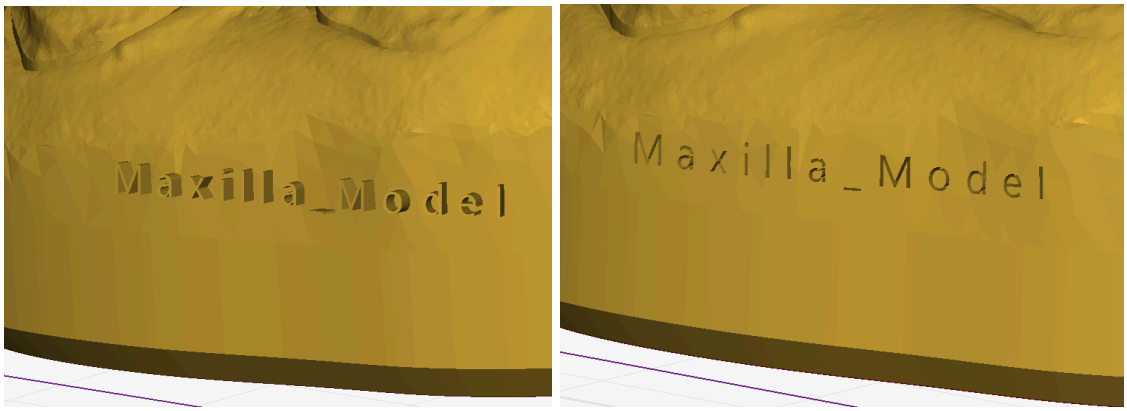
To add text to the application follow these steps:

1. Select the application on which you want to add text, then enter the text in the text area at the bottom of the Setup Section. By default, the text will be the file name, if you modify the text, click “**Label**” for the new text to be updated on the viewport.





2. Orient the design and right-click to place text on the application's surface. You can adjust the text placement by modifying the "heights". Select either "outward" or "inward". An inward text is imprinted, while outward will protrude slightly.

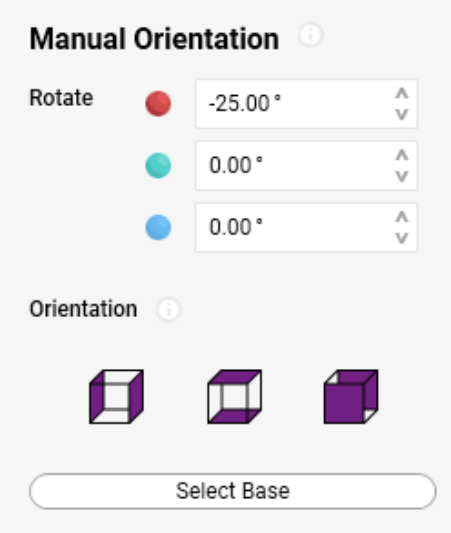


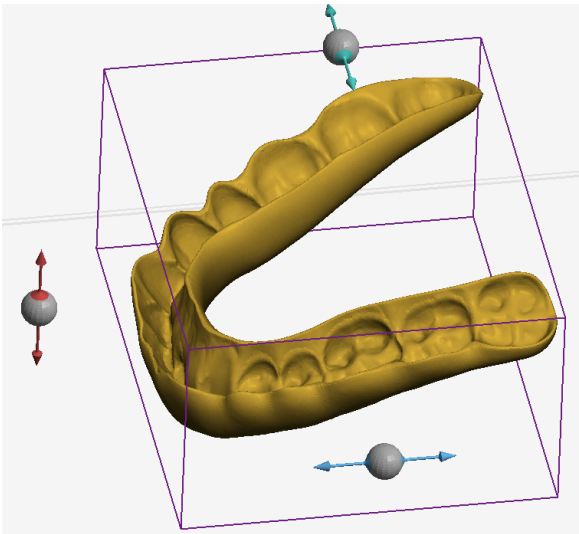
Outward label Inward label

6. Orientation and Support

Here are the recommendations for the orientation and support for the most printed dental applications, case by case: Ackuretta ALPHA AI Orientation & Support Guide. AI Premium module provides automatic orientation and support for the most common dental applications to ensure successful 3D prints.

a) Manual Orientation

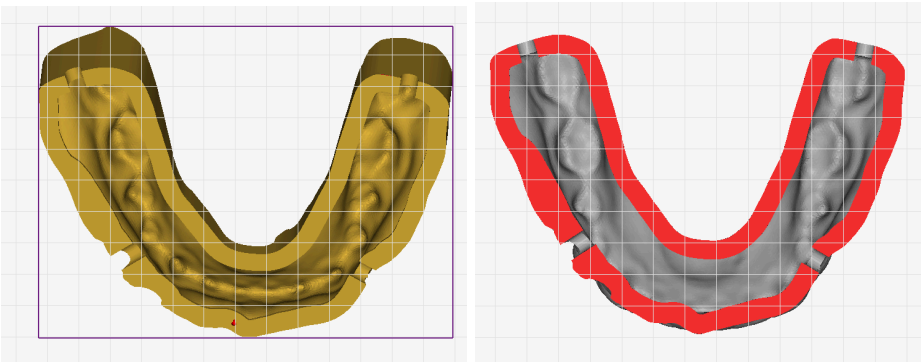




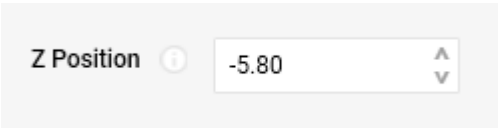
The arrows on each side of the application can be manipulated to orient the application in 3 dimensions.

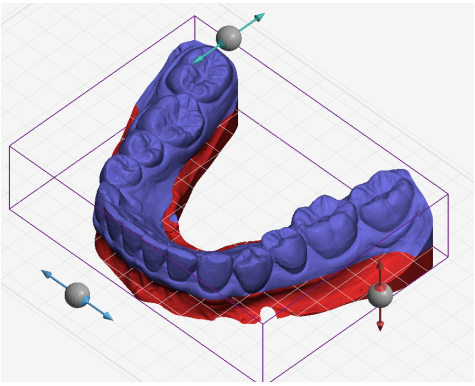
Users can manually modify the orientation of applications, by selecting “Manual Orientation” in the Setup panel.

In the Setup panel, in Manual Orientation, you can modify the application's orientation directly. Each direction is linked to a color code similar to the arrow. You can adjust the orientation angle with a precision of 0.01 degrees.



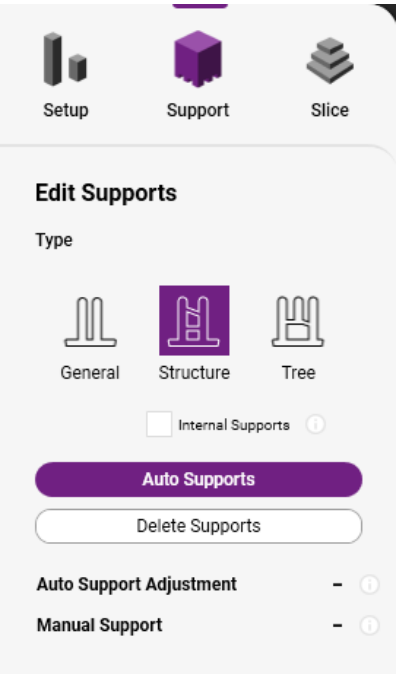
The “Select Base” function will display a vector that indicates the direction. After clicking on the flat surface, this surface will be parallel and in contact with the build platform. The contact area of the dental application with the build platform will be highlighted in red. When clicking on “Select base”, make sure to select the bottom view to verify this surface is in contact with the build platform.





It is possible to trim a dental application. This can be especially useful for dental models with a thick base that is not needed. To trim the base of a dental application, use the “Z Position” option during setup. Reducing the height of a 3D print will save resin and increase printing speed. It is not recommended for intraoral dental applications.

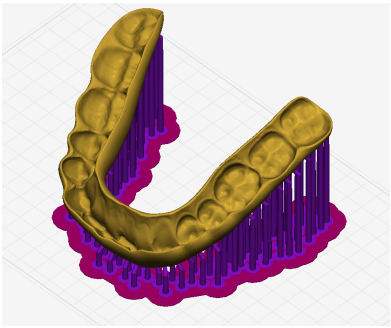
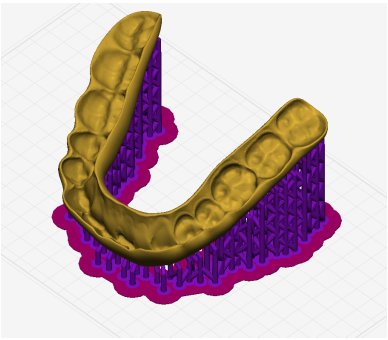
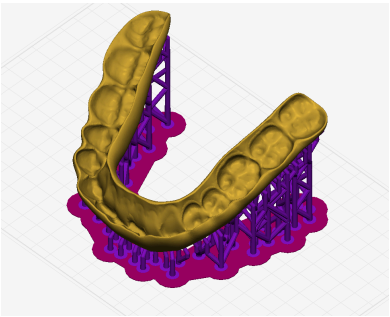
b) Generating and Editing Supports



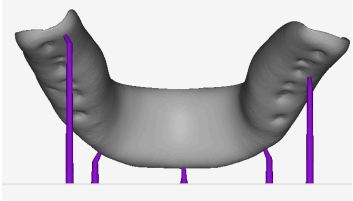
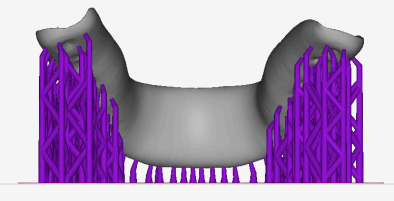
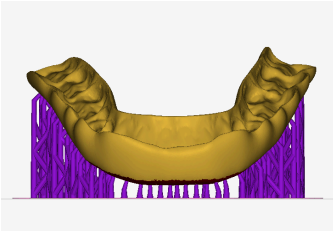
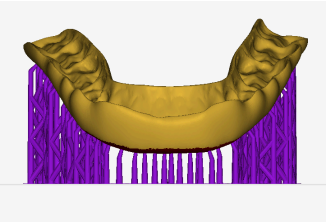
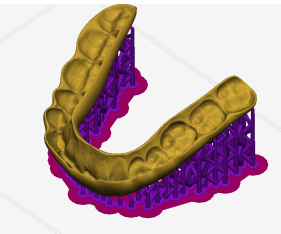
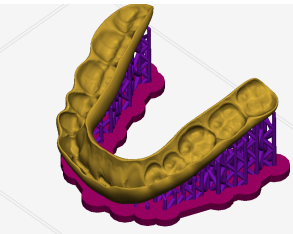
After you have chosen your support type, click on "**Auto Supports**" to add support to your selected application. By default, Auto Supports will apply settings that have been predetermined for the resin you have selected.

If you would like to change the support settings, click on "**Auto Support Adjustment**" or adjust the settings after clicking on "**Manual**". This is only recommended for experienced users.

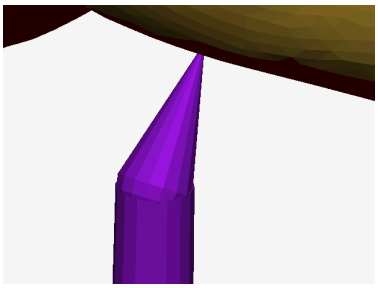
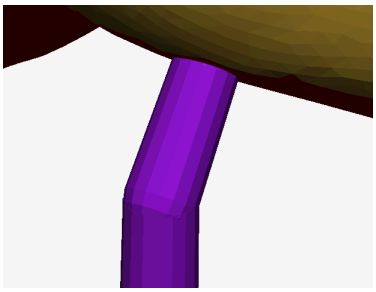
1) Support Type

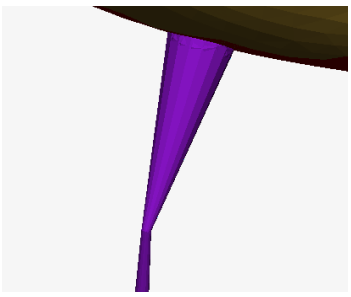
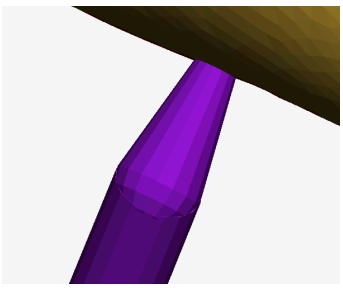
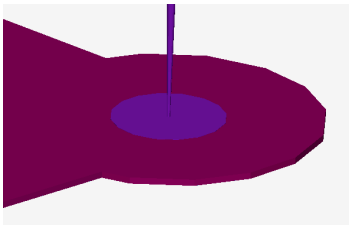
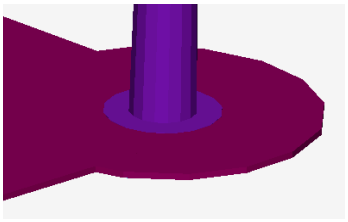
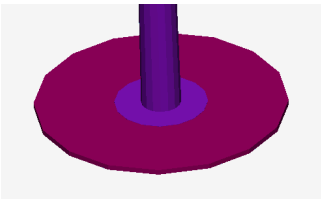
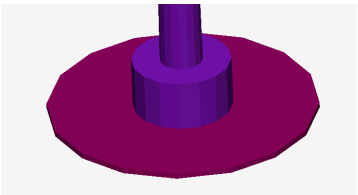
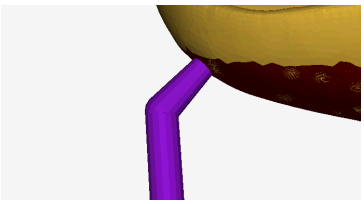
		
General Supports No horizontal beams for stability Uses the least resin *For experienced users	**Structure Supports** Includes horizontal beams for stability Uses the most resin *Recommended for all users	**Tree Supports** Some horizontal beams for stability Uses more resin than General, but less than Structure

2) Main parameters

Support Density The number of supports across the available surface of the application	 Support Density: 10%	 Support Density: 74%
Support Height The length of the support	 Support Height: 2 [mm]	 Support Height: 6 [mm]
Base Thickness The thickness of the added base that connects supports	 Base thickness: 0.2 [mm]	 Base thickness: 2 [mm]

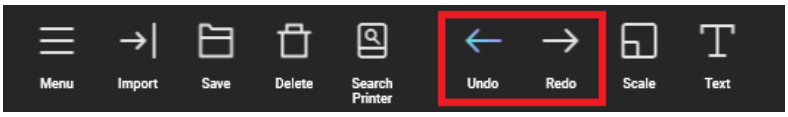
3) Point Parameters Settings

Point Size The width of the support where it touches the application	 Point size: 0.1 [mm]	 Point size: 1 [mm]

Elbow Size The width of the support where it changes thickness	 Elbow size: 0.1 [mm]	 Elbow size: 2 [mm]
Bottom Size The width of the support where it touches the base	 Bottom size: 0.1 [mm]	 Bottom size: 2 [mm]
Support Base Thickness The height of the foot of each support	 Support Bottom size: 0.2 [mm]	 Support Bottom size: 2 [mm]
Slope Angle The maximum angle of corners in the support	 Slope angle: 5%	 Slope angle: 60%

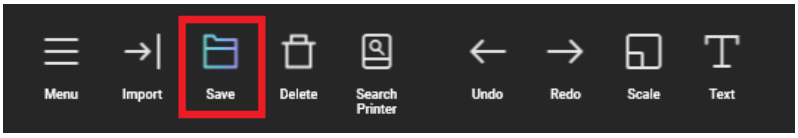
c) Special Functions

1) Undo & Redo



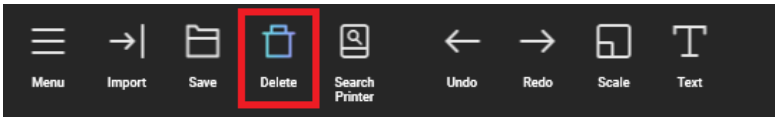
At any step of your slicing process, you can undo and redo any action by clicking the "**Undo**" and "**Redo**" buttons in the Action Bar.

2) Save Project File



You can save your project files at any time. It will save an i3DP file. This file is not the slicing file and cannot be used to print.

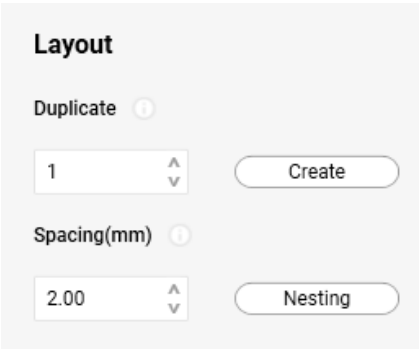
3) Delete Project

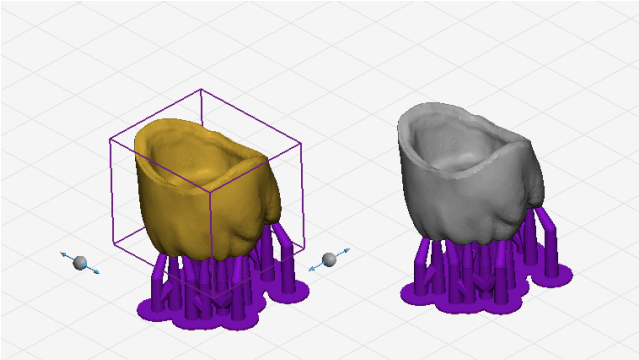


The “**Delete**” button in the Action Bar removes the project file from the Workspace including all the progress.

7. Slicing and Printing

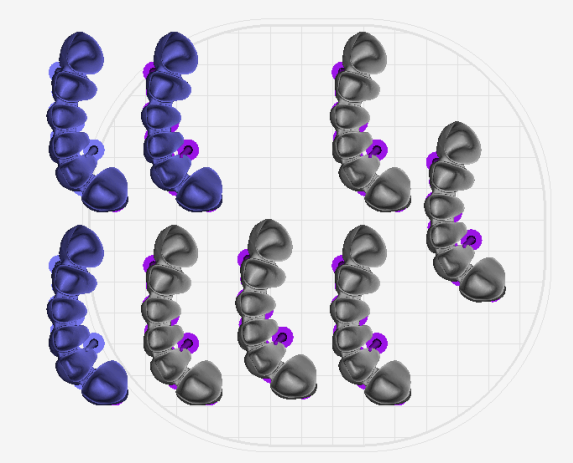
a) Nesting and Spacing

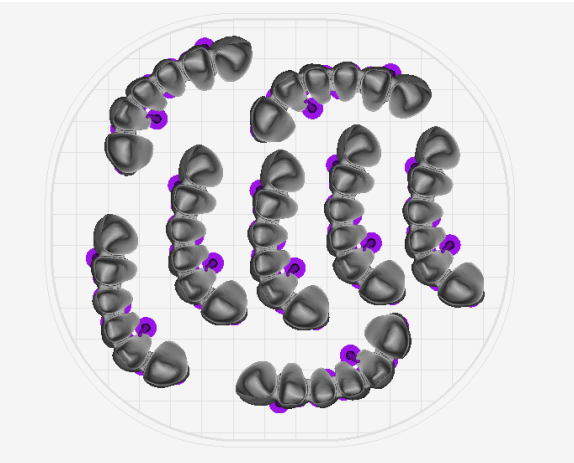




After setting up your application and adding support, you can duplicate applications to print more than one. First, you need to select the application that you want to duplicate then open the Slice panel in the Layout section, select the number of copies, and click on "**Create**".

You can also use keyboard shortcuts CTRL+C and CTRL+V to copy-paste an application.



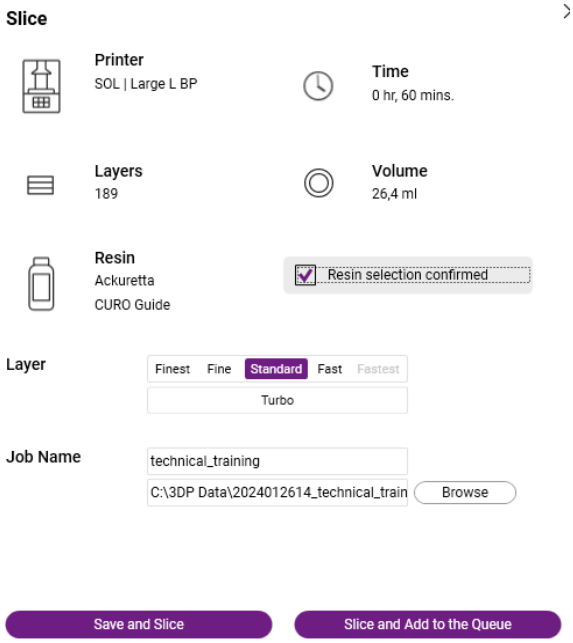


The "**Spacing(mm)**" function automatically arranges your applications on the build platform. The value decides the space between each application. Click on "**Nesting**" to execute the function.

b) Slicing

1) Slicing Window

Under the Layout panel, you will find the Slice section. Clicking on "**Slice**" will open the slicing window.



The Slice window displays the selected printer and build platform size, print duration, number of layers, approximated resin volume, and the selected resin for the print.

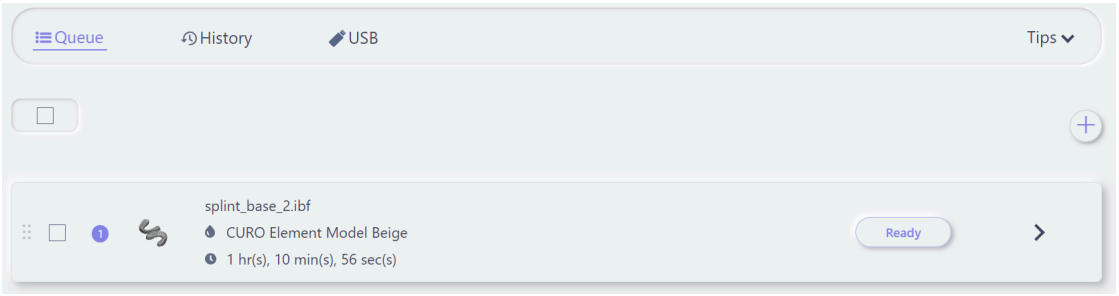
The "Job Name" will be the ACKU filename that will be sent to the printer. When entering the job name do not enter special characters such as [!@#$%^&*()+] as the printer will not recognize it and can cause an error message.

By default, the print job will be saved under C:\3DP DATA\"Your job name" with ACKU file extension.
Select "Browse" to save the print job in your preferred file path.

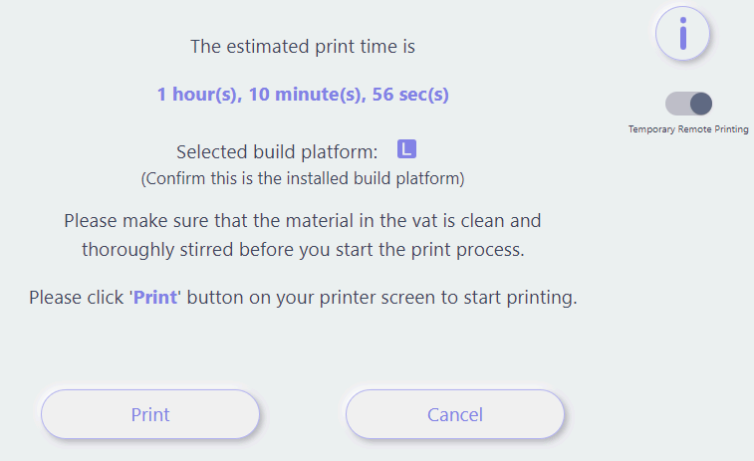
Make sure to double-check the Resin selection then choose "**Save and Slice**" or "**Slice and Add to Queue**"

2) Save and Slice

After clicking "Save and Slice" the file will be turned into an ACKU File format.

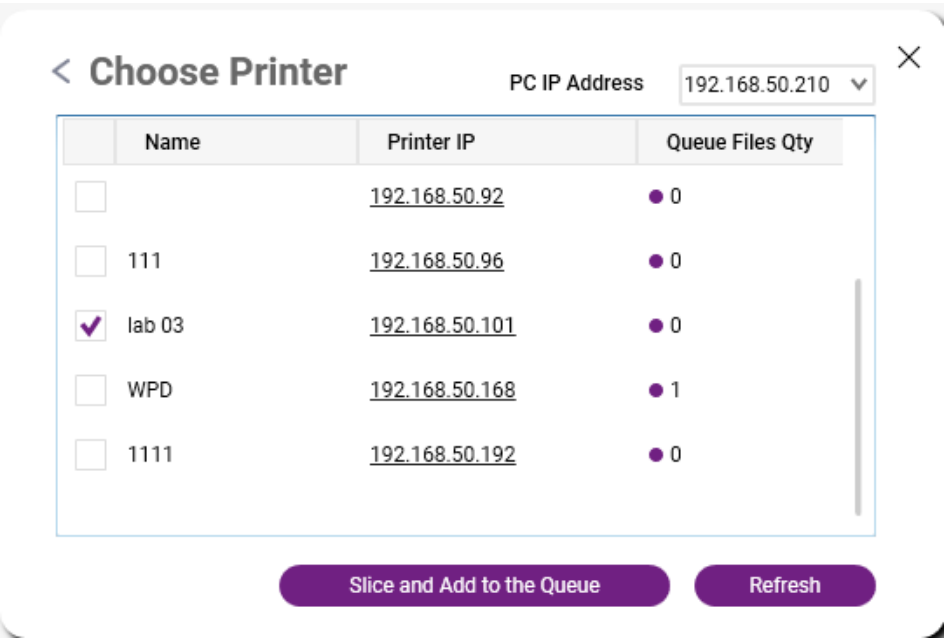


To remotely transfer the ACKU file, copy your printer IP address to the web browser to open your printer console in a web browser then send files. When the file is loaded click the “**Ready**” button to start printing the file.



By activating the “Temporary Remote Printing” function you can start the print directly from your web browser. Make sure to always visually check your printer before printing remotely.

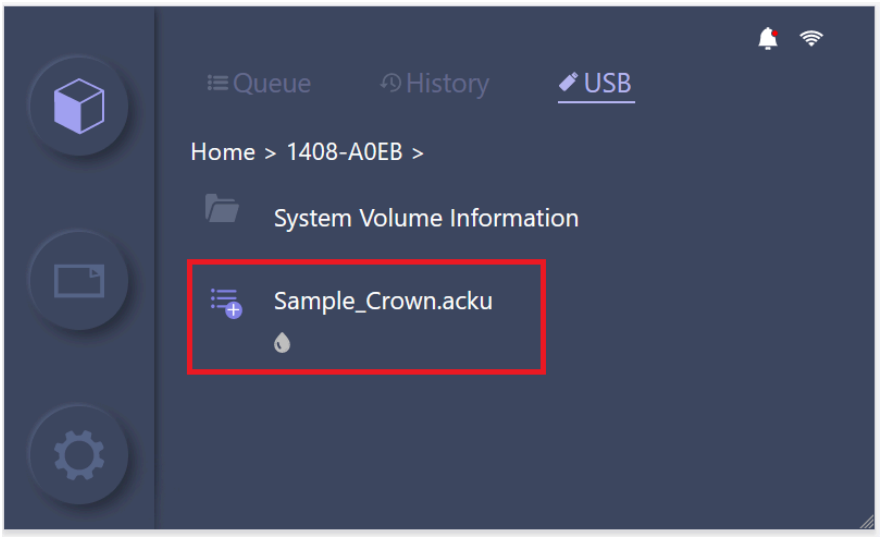
3) Slice and Add to the Queue



The “**Slice and Add to the Queue**” function will open the Printer Selection Window. Please make sure that your printer is connected via WiFi or Ethernet cable and that the computer is on the same network.

At this step, you will see the list of printers connected to your network, select the printer and click “**Slice and Add to the Queue**”. This will slice the file and add it to the printer waiting queue.

4) Open a file from USB

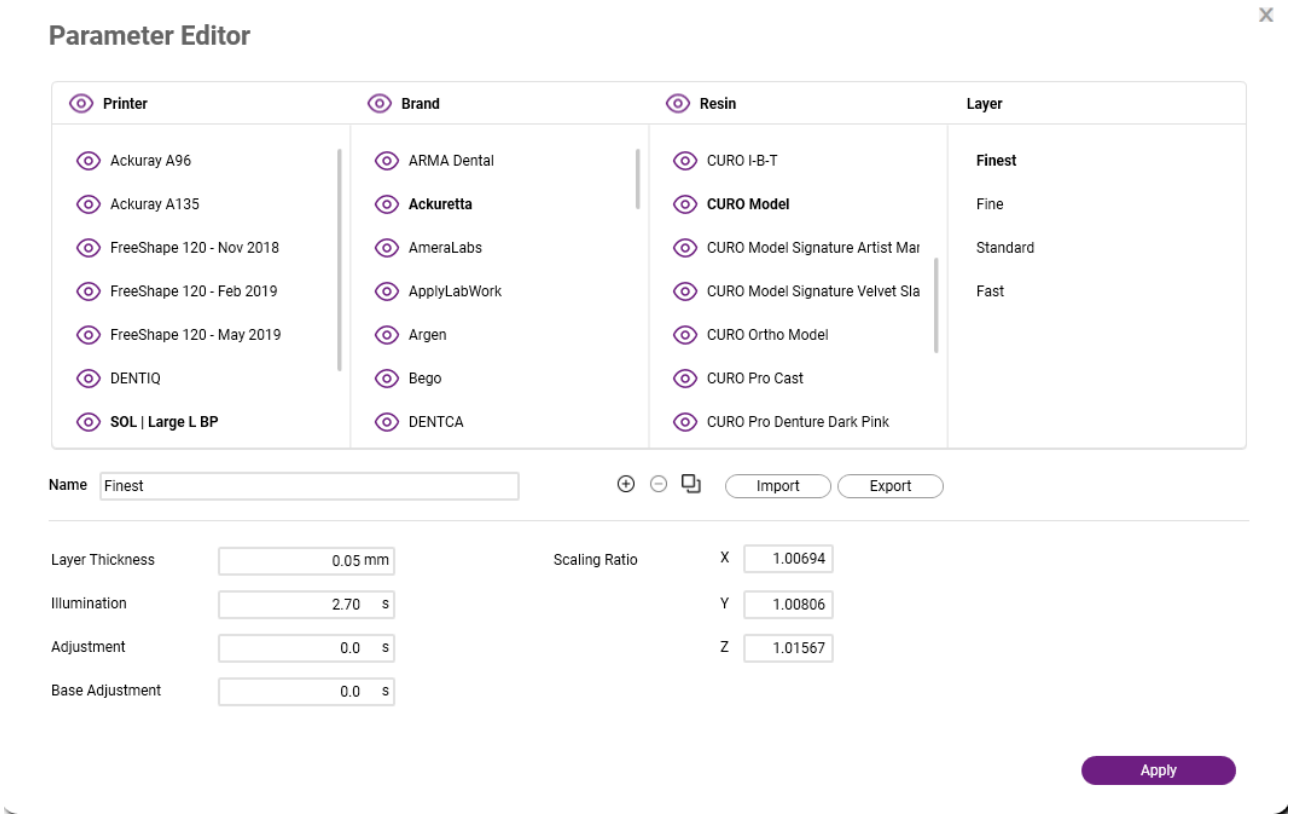


You can also transfer the file from your computer to your printer via a USB key.
Open the folder C:\3DP DATA\“Your job name” and find the file with the ACKU file extension. Copy the file to your USB driver. Plug the driver to your printer. In the USB section select your ACKU file directly and start printing.

8. Advanced features

a) Details

Click on “Details” under the resin selection menu to access the “Details” settings window.



The **“Details”** settings are advised for advanced users and will allow you to change some printing parameters.
Select the Printer / Brand / Resin / Layer to modify the printing parameters:

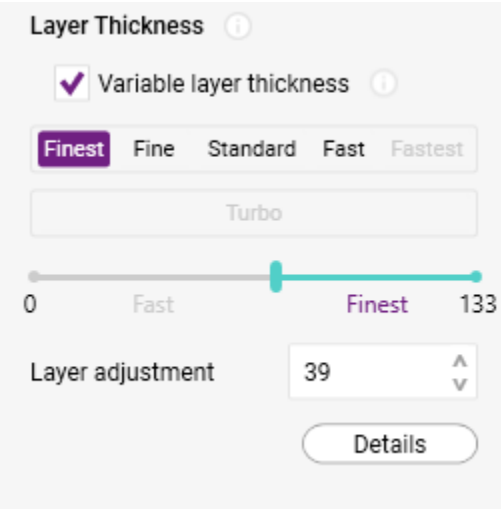
- ***Adjustment time***: Additional illumination time that occurs for each layer. You can change the Adjustment time up to 20% of the illumination time. There is a risk of overturning the print by increasing the illumination time too much.

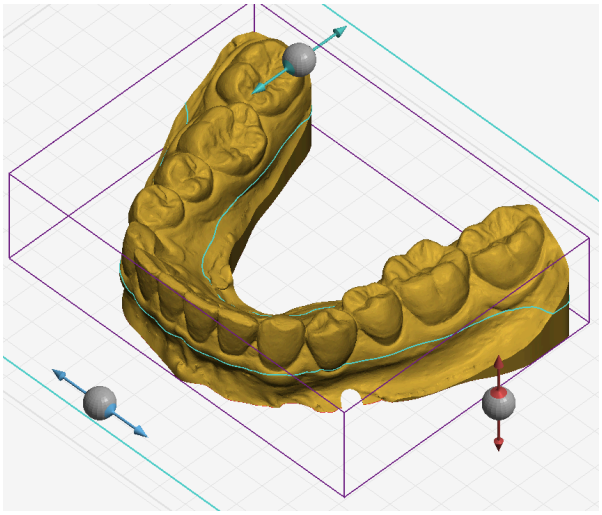
- ***Base adjustment***: Additional illumination time for the base layer. You can change the Base Adjustment time to add 1 or 2 seconds. This can improve the base layer adhesion of the 3D print to the build platform.

For DENTIQ and FreeShape we can import a resin parameter profile (“Para” file). You can load this parameter in ALPHA AI under a new name. This file is provided by a resin manufacturer. **Ackuretta cannot guarantee printing results** as it is out of the calibration process. You can also copy and edit a resin profile. The export function will save your resin parameter profile in a para file that you can share.

If you are looking to use a resin not calibrated on your Ackuretta printer we recommend filling the Resin Calibration Request. Ackuretta does not recommend using resins that are not on the "validated" list for each Ackuretta 3D printer, to view all validated resins for Ackuretta's 3D Printers, click here: SOL Plus, SOL, DENTIQ.

b) Variable Layer Thickness





Variable layer thickness allows you to print your appliance in two different layer thicknesses. This is recommended for models where the base structure is unnecessary for clinical and diagnostic purposes. You can print the first layer of a model with Fast layer thickness and then the latest one with Fine so that will increase accuracy in the model teeth. Once the variable layer thickness is selected, you will see a green section in your design that indicates where the change in the layer thickness occurred.

c) Scale

The Scale function helps to change the scale of the application in the x, y, and z-axis. We do not recommend changing the scale for dental applications.

The lock icon on the right indicates that all dimensions of x, y, and z will be changed proportionally.

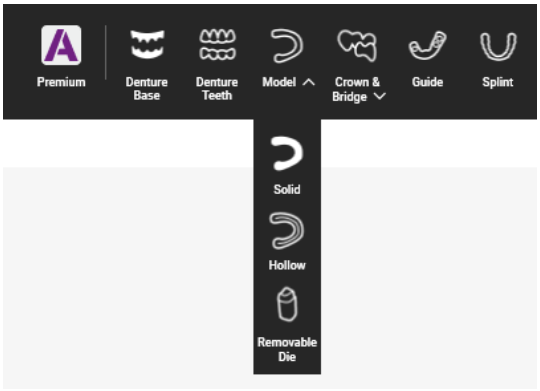
When clicking on the lock, the x, y, and z dimensions will be independent of one another.

9. ALPHA AI Premium

a) Start a Subscription

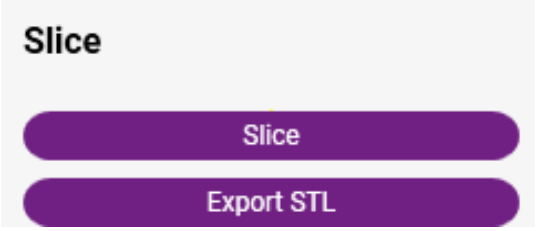
Find the following video to download and subscribe to ALPHA AI Premium: ALPHA AI - How to Download and Upgrade ALPHA AI

b) AI Module



The AI Module is designed to automatically orient dental applications and add the optimal supports required for a successful print. This module currently includes 9 applications, with more under development.

c) Export STL



In the Slice panel, the "**Export STL**" button will save the selected file in STL format. This function is limited to 50 STL exports per month. The STL file created contains your design with orientation and support configuration to be directly sliced and printed using third-party slicing software.

d) Connection to Multiple Devices

You can log in to your ALPHA AI Premium account from up to 4 different computers

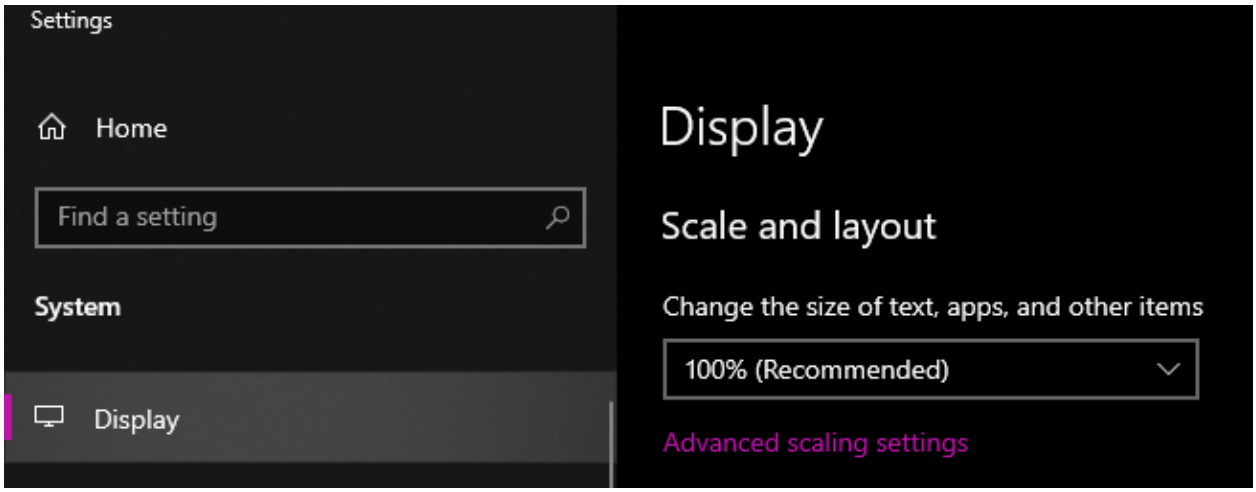
e) Manage Subscription

You can find your current ALPHA AI Premium subscription, edit it, and stop it directly from our Subscription Manager.

You can compare the benefits of an ALPHA AI Premium subscription with ALPHA AI Standard in the following video: ALPHA AI VS ALPHA AI Premium

10. Troubleshoot

a) ALPHA AI Closing at the Start



If ALPHA AI doesn't load or close at the start, reach Windows Settings > Display > Scale and layout then change the scale to 100%.

b) Internet Troubleshoot

ALPHA AI requires an internet connection to load the latest printing setting parameters.

If ALPHA AI displays an error such as “ALPHA AI cannot connect” please make sure to check your internet speed and stability with the following link: Speedtest.

If the internet is stable but you cannot connect, the firewall on your WiFi router may be preventing the connection to the Ackuretta server. Please change the firewall settings on your router to make an exception for ALPHA AI. The process for allowing specific connections on routers is dependent on the brand of router, but you will need the following exception for the Ackuretta server:

Server URL: server-n1.ackuretta.com

Server IP address: 54.159.232.131

Server port: 9527

You can also check the following article ALPHA AI Connection Issue

c) ALPHA AI Premium Connection Issue

Please open a support ticket if you need help to connect or recover your account ALPHA AI Connection Issue – Ackuretta

For sales inquiries, contact
sales@ackuretta.com

For technical support,
Open a Ticket

Find your
Nearest Distributor