INSTRUCTION FOR USE

3D Exacto

Based on the ImplaStation
Instructions for Use 423-3, Rev 12
Software version: 5.5.0
Date of Issue: November 4, 2026

TABLE OF CONTENTS

INTRODUCTION **4**
1. SYSTEM SETTINGS **6**
1.1 System hardware and software requirements for Windows 6
1.2 System hardware and software requirements for macOS 7
2. INSTALLATION AND UPDATE **8**
2.1 Installation on Windows 8
2.2 Installation on macOS 12
3. INTERFACE OVERVIEW **17**
3.1 Start Screen 17
3.2 User Interface 18
3.3 Multi-Planar Reconstruction (MPR) Display 18
3.4 3D-Rendering Window (Volume) 21
3.5 Panoramic Mode 22
3.6 Rotating Slice Window 25
3.7 Menu bar 25
3.8 Workspace control panel 29
3.9 Project & Account control panel 30
3.10 Workflow panel 30
4. INPUT DATA (DICOM) **32**
4.1 Steps for Importing DICOM into a Project 32
4.2 DICOM "Cutting" 34
5. SEGMENTATION **35**
5.1 Segment Cleanup Strategy 35
5.2 Segmentation process 36
5.3 Segment processing 37
6. INPUT DATA (STL) **44**
6.1 Importing STL files / Scan+ 45
6.2 Alignment 47
7. NERVE CANAL TRACING **50**
8. VIRTUAL CROWNS PLACEMENT **52**
9. IMPLANT PLANNING **54**
9.1 Implant Placement 55
9.2 Options of choosing implant elements 58
10. ANCHOR PIN PLANNING **62**
10.1 Pin Placement 62
10.2 Options of choosing pin elements 64
11. SURGICAL GUIDE CREATION **66**
11.1 Surgical Guide based on Surface 66
11.2 Surgical Guide Based on Prosthesis 69

12. MODELING **71**
12.1Model Builder mode 71
12.2 Custom Provisional Crown Modeling 74
12.3 Custom Healing Abutment Modeling 77
SHELF LIFE AND SOFTWARE SUPPORT **80**
WARNINGS AND PRECAUTIONS **81**
ANNEX A How to evaluate the quality and accuracy of alignment? **85**
ANNEX B Dual Scan Technique **86**
ANNEX C How to anonymize your project file **89**
ANNEX D Cloud Service **90**
ANNEX E Cybersecurity Hygiene **93**
ANNEX F Implant Libraries for 3D Exacto **94**
ANNEX G STL Editor **95**
ANNEX H Stackable Surgical Guide **121**
ANNEX I Export **139**

INTRODUCTION

3D Exacto is stand-alone software designed for trained qualified dental practitioners.
The key scientific concept of the 3D Exacto software is the visualization of a patient's medical image data (DICOM file from third-party CT/CBCT scanners) to pre-operative digital implant planning, surgical guide (drill guide) file (output of the pre-operative implant planning) creation.

Note!	The software as medical device has no patient contact.

The following are the major functions of 3D Exacto :

- Patient DICOM dataset loading and visualization;
- Data (DICOM, .stl files) input; Data (.stl file) output for manufacturing; Surgical report (drilling protocol .pdf) output;
- Nerve tracing;
- Virtual crown(s) positioning;
- Virtual implant placement;
- Collision detection;
- Patient treatment plan creation ;
- Surgical guide design and creation;
- Surgical protocol design and creation;
- Modeling of provisional crowns and custom healing abutments;
- Project information management and sharing;
- Designed surgical guide data can be exported to a third-party system for manufacturing**.

Note!	The software is not intended for diagnosis, please use the software offered by the Computed Tomography Scanner manufacturer, on which scanning was performed for the diagnosis and expert opinions. The software does not apply any compression, modifications, or adaptation to the DICOM files and model surface scan STL-files during their upload, alignment, design, and export.

**Paid option

INTENDED USE

3D Exacto is a stand-alone medical device software intended for use by licensed dentists, dental specialists trained in implant dentistry, and licensed dental technicians for the pre-operative planning of dental implant placement in adult patients requiring implant-supported restorations.

The software is intended to import, display, and process CBCT/CT imaging data in DICOM format and surface scan data in STL or OBJ format for the purpose of virtual implant planning and the design of patient-specific surgical guides for guided dental implant surgery.

3D Exacto is intended to support pre-operative treatment planning only. It is not intended for diagnostic use, real-time surgical navigation, intraoperative guidance, robotic control, or automatic implant placement. Final clinical decisions remain the responsibility of the treating clinician.

1. SYSTEM SETTINGS

1.1 SYSTEM HARDWARE AND SOFTWARE REQUIREMENTS FOR WINDOWS

Category	Minimum System Requirements	Recommended System Requirements (Validated Configuration)
System Software / Execution Environment	Windows 11	Windows 11
Core Processing Hardware / System Resource (CPU)	Intel, AMD x86-64 compatible processor, with AVX support	Intel® Core™ i7 class processor (13th generation or newer)
System Hardware Resource / Volatile Storage	8 GB	16 GB
Image Processing Hardware / System Resource	Support for OpenGL 4.1 Core Profile or higher, with 2 GB VRAM or higher	Discrete NVIDIA GPU with ≥4 GB VRAM (RTX-class) OpenGL 4.1 Core Profile or higher
Data Storage Hardware / System Resource	At least 3 GB of free disk space	At least 3 GB free disk space
Display Hardware / Output Device	Full HD (1920×1080) or higher	Full HD (1920×1080) or higher
User Interface Hardware / Input Device	Mouse with scroll wheel	Mouse with scroll wheel
External Network / Data Transfer Resource	Internet connection required for activation, updates, and support	Internet connection required for activation, updates, and support
Security Software	Windows Defender or equivalent operating-system native protection	Windows Defender or equivalent operating-system native protection

Note!	In order to identify user account, import/export order forms, share 3D Exacto projects and communicate with customer partner(s), personal computer must be connected to the Internet.

1.2 SYSTEM HARDWARE AND SOFTWARE REQUIREMENTS FOR MACOS

Category	Minimum System Requirements	Recommended System Requirements (Validated Configuration)
System Software / Execution Environment	macOS 13.0 or higher	macOS 15.3
Core Processing Hardware / System Resource (CPU)	Intel, Apple Silicon M-series	Apple Silicon M4-series processor, or higher
System Hardware Resource / Volatile Storage	8 GB	16 GB
Image Processing Hardware / System Resource	Support for OpenGL 4.1 Core Profile or higher, with 2 GB VRAM or higher	Apple Silicon integrated GPU (8-core or higher, unified memory architecture)
Data Storage Hardware / System Resource	At least 3 GB of free disk space	At least 3 GB free disk space
Display Hardware / Output Device	Full HD (1920×1080) or higher	Full HD (1920×1080) or higher
User Interface Hardware / Input Device	Mouse with scroll wheel	Mouse with scroll wheel
External Network / Data Transfer Resource	Internet connection required for activation, updates, and support	Internet connection required for activation, updates, and support
Security Software	macOS built-in security mechanisms	macOS built-in security mechanisms

Note!	In order to identify user account, import/export order forms, share 3D Exacto projects and communicate with customer partner(s), personal computer must be connected to the Internet.

2. INSTALLATION AND UPDATE

2.1 Installation on Windows

STEP 1 Start installation process manually by executing the loaded **3D Exacto Setup** file on personal computer.



STEP 2 Choose language settings and click **Next** to move to the next step.

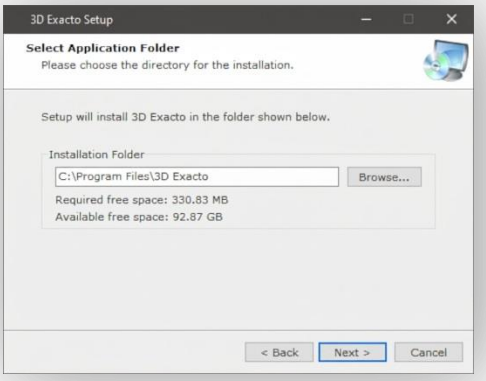


STEP 3 Click **Next** to move to the next step.



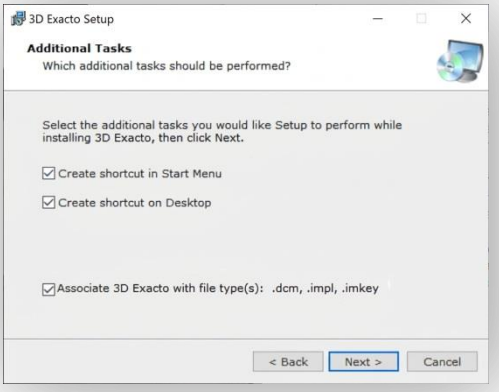
STEP 4 Browse for app location and click **Next** to move to the next step.

*It is recommended to locate main 3D Exacto folder in the "**C:**" drive root*

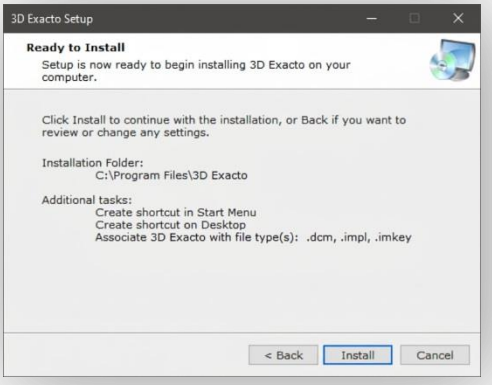


STEP 5 Select the additional tasks you would like setup and click **Next** to move to the next step.

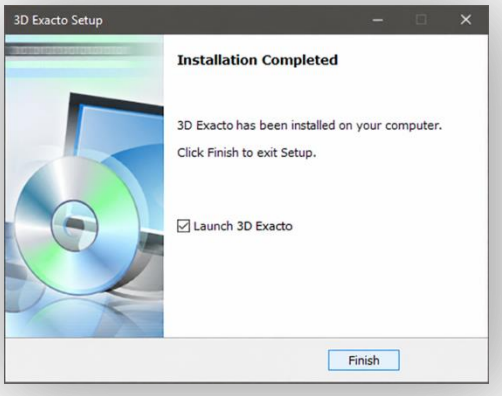
For not experienced user it is recommended to choose all options



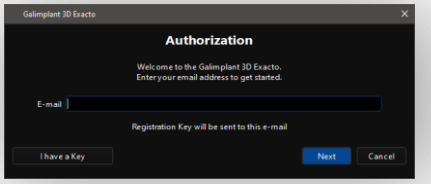
STEP 6 Click **Install** to continue with the installation.



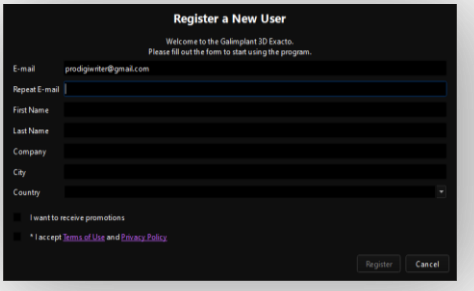
STEP 7 Click **Finish.**



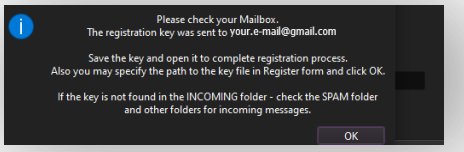
STEP 8 Fill in the E-mail for Authorization and click **Next.**



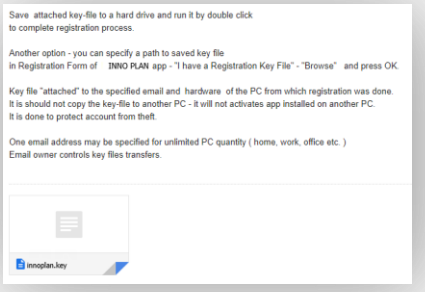
STEP 9 If you are new user fill up the registration form.



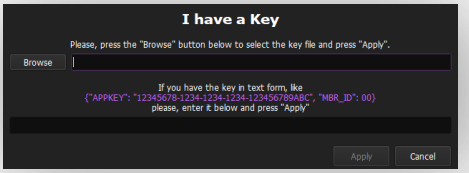
STEP 10 Click **OK**, then software will be automatically closed. Please check your e-mail.



STEP 11 Open your inbox e-mail folder, save attached key-file to a hard drive, open it by double clicking to complete a registration process.



OPTIONALLY you may specify a path to the saved key file using the Registration Form of 3D Exacto app: *open I have a Key tab > Browse tab, choose saved Key file in the folder and click OK;* OR copy-paste info from key-file as a text to the appropriate text-field.



Note!	Key file(s) Library folder and other important information are located in the folder "3D Exacto" (C:\Users\"user name"\AppData\Roaming\3D Exacto) for Windows.

2.2 INSTALLATION ON MACOS

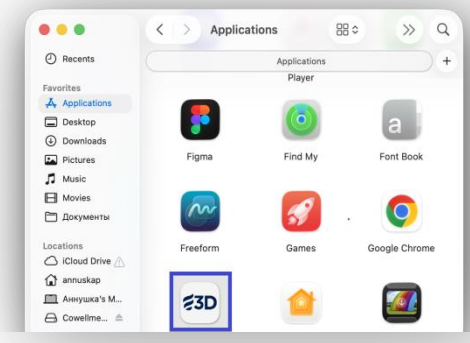
STEP 1 Start installation process manually by executing the loaded **3D Exacto Setup** file on personal computer.



STEP 2 Find the downloaded file in the Downloads folder and double-click it to open. A window will open showing the app icon and a shortcut to the Applications folder. Drag and drop the app icon into the Applications folder.



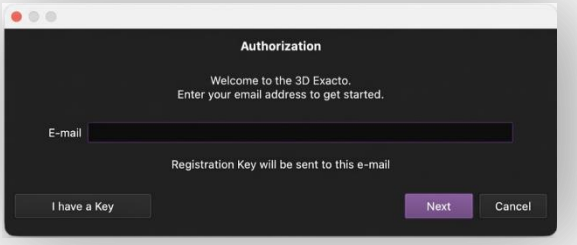
STEP 3 Go to Launchpad or open Finder > Applications. Find and double-click the app to launch it.



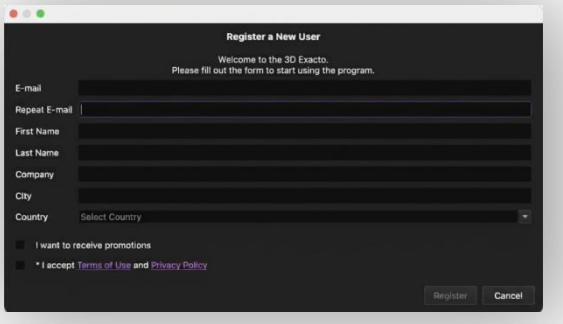
STEP 4 On the first run, macOS might warn that the app was downloaded from the internet. Click **Open**.



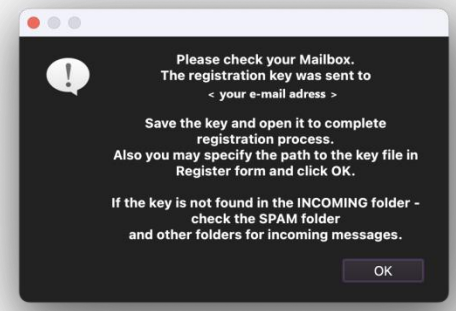
STEP 5 Fill in the E-mail for Authorization and click **Next.**



STEP 6 If you are new user fill up the registration form.

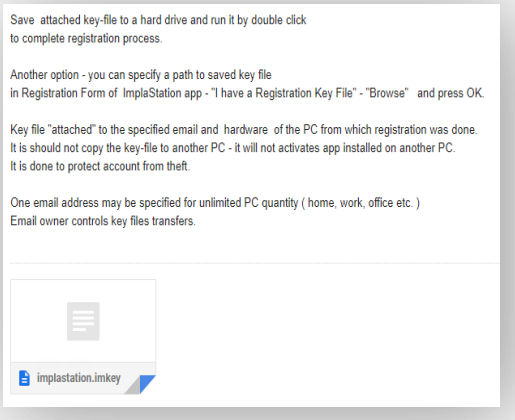


STEP 7 Click **OK**, then, please check your e-mail.

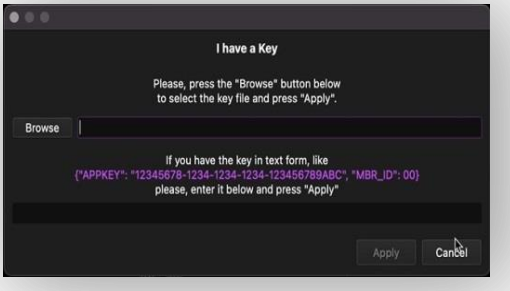


STEP 8 Open your inbox e-mail folder, save attached key-file to a hard drive.





STEP 9 Activate the saved key file using the Registration Form of 3D Exacto app: *open I have a Key tab > Browse tab, choose saved Key file in the folder and click OK;* or copy-paste info from key-file as a text to the appropriate text-field.



If the **3D Exacto License Key** file is not received after registration process, check the SPAM and other folders for incoming messages in your specified email box.
For **technical issues**, email us at support@prodigident.com

Note!	The **3D Exacto License Key** file is "attached" to the specified email address and hardware ID of the PC from which registration was done.

Warning!	The **3D Exacto License Key** file should not be copied and used with another PC. In aim to protect the user's personal information and prevent thefts, the key file will activate app installed only on the PC from which registration was done.

One email address maybe specified for unlimited quantity of PC (home, work, office etc.)
Only an e-mail owner able to control **3D Exacto License Key** files transfers.

The 3D Exacto Settings menu enables user to register several accounts, attached to different e-mail addresses. In case of several accounts are registered on one PC the Select user window appears.
Use **User Manager** button in **Project & Account** controls panel.

If your browser automatically converts key-file to text - copy an appropriate string with the code to bookmark "I have a registration key-file".

UPDATE

To check the current version of the software click the **Help** button on the Menu bar, then select **Check update**.

If an update is available, the **Update Application** menu pops up automatically when the software starts up. Click **OK** to update (recommended), or click **Cancel** to continue to work with the current version.

If the update does not occur, or it gives you an error, reinstall the software by clicking the link in the update dialog window.

Note!	Software installation packages and update packages must be obtained only through Prodigident-approved distribution channels. Windows installation and update artifacts are digitally signed. macOS application and update artifacts are signed using Apple Developer ID and notarized where applicable. Update metadata is retrieved using HTTPS/TLS, and update packages are verified using cryptographic integrity and authenticity controls before installation.

Warning!	Do not install 3D Exacto from unofficial websites, shared links, unknown storage locations, or unverified third-party sources. If the operating system displays a warning about an unknown publisher, invalid signature, failed notarization, or possible tampering, do not continue installation or update. Contact Prodigident support before proceeding.

UNINSTALLATION

To safely uninstall 3D Exacto:

Windows:

Close the software. Open the **Windows system settings**. Go to **Apps** → **Installed Apps** (or **Apps & Features**), find **3D Exacto** , select it, and click **Uninstall**. Confirm the action.

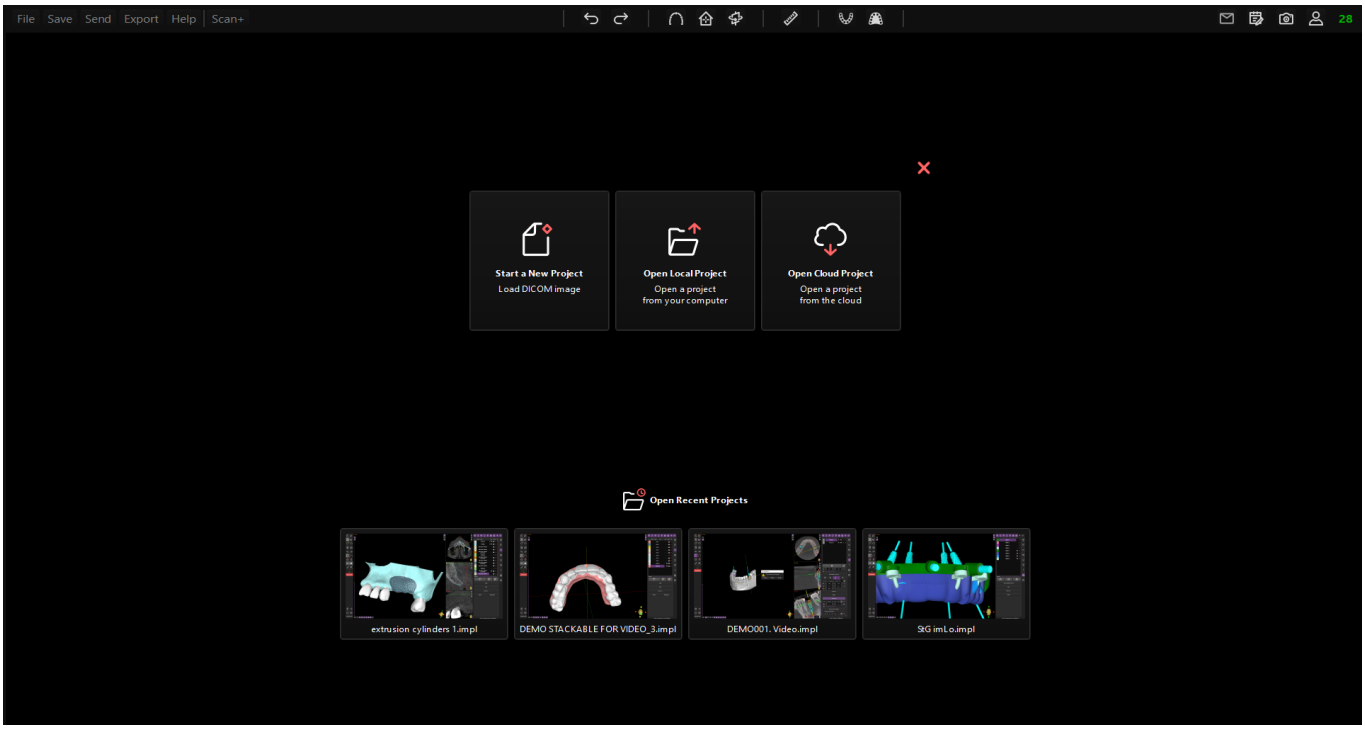
macOS:

Quit the application. Open the **Applications** folder in **Finder**, move **3D Exacto .app** to **Trash**, and empty the Trash.

3.INTERFACE OVERVIEW

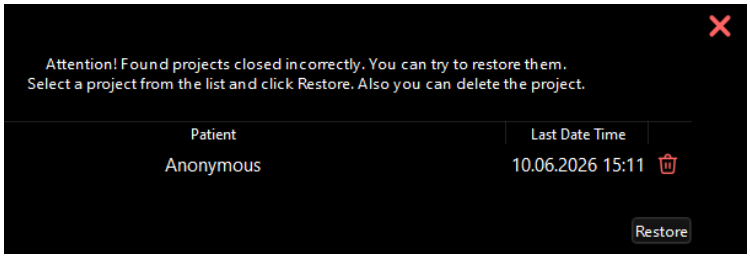
3.1 START SCREEN

When the application launches, the start screen appears, prompting to begin work by creating a new project or opening an existing one. Existing projects can be opened from the computer, from the cloud, or selected from a list of recent projects.



RECOVERY WINDOW

If one or more projects were closed unexpectedly during a previous session, a **recovery window** appears before the regular start screen. The window lists every project that didn't close correctly, with the following options for each item:

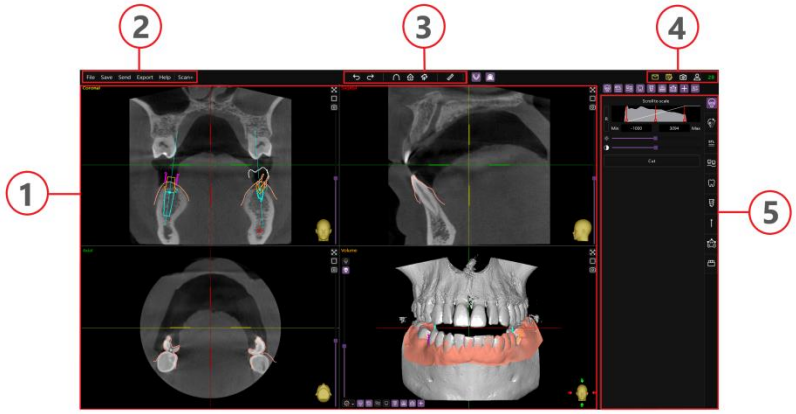


- Restore – reopens the project from its last saved state.
- Delete (trash-can icon) – permanently removes the project from the list.
- Ignore – closing the window with the X button leaves the project in the list without restoring or deleting it.

Hint!	If a crashed project is ignored, the recovery window will appear again the next time the software is launched.

3.2 User Interface

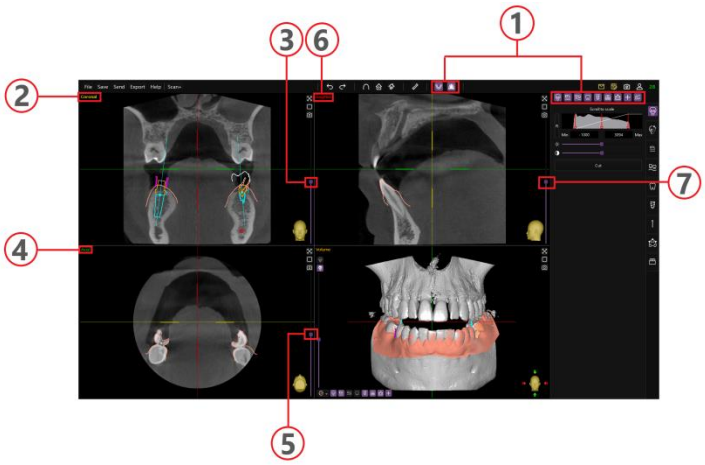
The 3D Exacto user interface enables users to visualize the patient's dataset using the Coronal, Sagittal, Axial, Panoramic, Rotating Slice and 3D view in corresponding windows.
MPR and Panoramic display can be used to generate interactive slices in free, oblique planes.
In addition to the visualization windows, the interface includes several menus that provide access to project actions, libraries, settings, and other tools. These menus are described in detail in the following sections.



1	Viewports
2	Menu bar
3	Workspace controls
4	Project & Account controls
5	Workflow panel

3.3 Multi-Planar Reconstruction (MPR) Display

The software automatically provides multi-planar windows (coronal, sagittal, and axial) and 3D view. This Multi-Planar Reconstruction can be used to work on any spatial plane to obtain different types of high-quality diagnostic images obtained from 3rd parties medical CT or CBCT scanners.



1	Element visibility buttons
2	Coronal window
3	Slider to view the whole sequence of coronal plane
4	Axial window
5	Slider to view the whole sequence of cross-sectional images in axial plane
6	Saggital window
7	Slider to view the whole sequence of cross-sectional images in saggital view

MPR VISUALIZATION PANEL


Visibility of lower jaw-related objects


Visibility of upper jaw-related objects


Visibility of DICOM



Vsibility of STL surface


Visibility of Nerve Tracing


Visibility of Crown


Visibility of Digital Implant

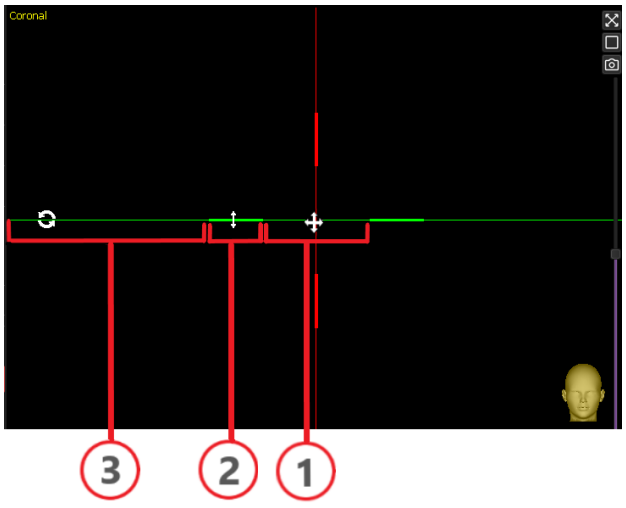

Visibility of Measurements


Visibility of Surgical Guide


Visibility of Axis


Visibility of Patient Data

PLANE-AXIS NAVIGATION



1 Center of the axes intersection for moving

2 Bulky part of axis for parallel movement of the plane

3 Behind bulky part of axes for 2D view rotation

HOW TO ADJUST THE PLANES

Free adjustment of the view by moving the center of the crossed planes
To move the center of the view axes intersection, put the cursor on it, then left-click and drag. The intersection of the view axes will move within the chosen plane. This movement will be synchronized with the change in slice depth in other MPR windows.

Hint!	Use the Head icon in the corresponding window to see the orientation of scans.

Quick adjustment of the view by clicking the desired point
To move the center of the crossed planes, put the cursor on the desired position and double left-click on it. The center of the axes intersection will move to the selected point immediately.

Quick adjustment of the view by clicking the implant image or name
The double left-click on the implant image or implant name in the implants list enables users to set up the coronal axis in the same position with the implant axis and move the center of axes intersection to the implant reference point.

Adjustment of the oblique view (plane rotation)
To rotate the plane, put the cursor on the plane line of the corresponding plane, behind bulky part, left-click, hold and rotate it. The plane will rotate around the center of the crossed planes. By adjusting the only one plane, the other two views in oblique multi-planar will be generated automatically.

Zoom in/out
Zoom in/out the views by holding down the right mouse button and moving the mouse forward or backward.

2D view rotation
To rotate the 2D view, put the cursor on any point of the desired window, press and hold down the left mouse button and rotate the object.

Volume view rotation
To rotate the Volume view, put the cursor on Volume visualisation window and use Scroll wheel.

MOVEMENTS OF THE PLANES

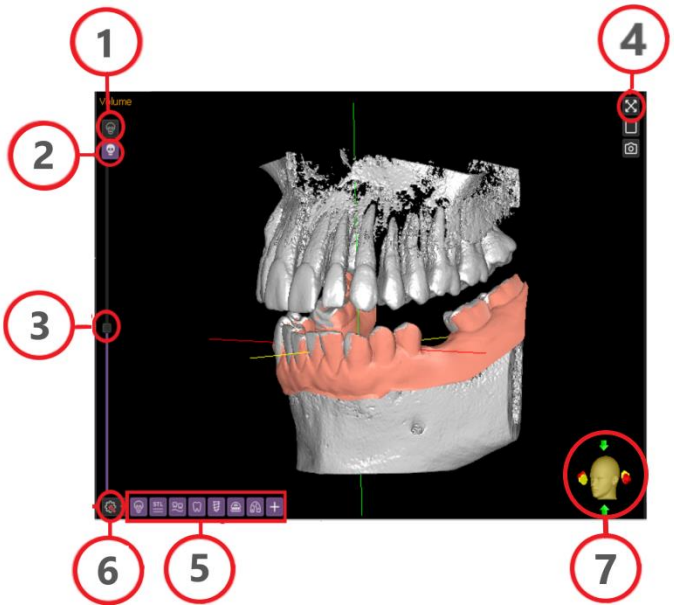
Parallel movement of the planes
To provide the parallel movement of the plane, put and hold the left mouse button on the bulky part of the indicator line of the plane and drag it.

Move the plane by scrolling the mouse wheel
Put the cursor on any point of the MPR window, and then scroll the mouse wheel.

Moving/dragging views in windows
To move a view in a 2D or 3D Window, press and hold down the mouse wheel, then move the object within the selected window. **Without a mouse** - press and hold the Space button on a keyboard, and then move an object within a selected window.

3.4 3D-Rendering Window (Volume)

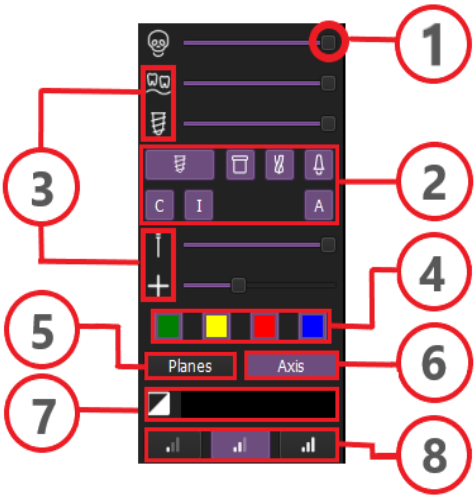
The 3D volume rendering view can be used to visualize 3D data generated by X-Ray and optical scanners in three-dimensional space in aim to simplify the spatial orientation and object placement control.



1	"MIP mode" (Maximum Intensity Projection) button
2	"ISO mode" (Isosurface)
3	Slider to adjust the DICOM rendering optical density threshold.
4	"Maximize or minimize 3D Volume Rendering Window" button
5	Visualization panel
6	Settings button of the 3D Volume Rendering Window
7	"Quick-control" buttons for switching the 3D view

Hint! Click appropriate arrow near 3D head icon for quick set of 3D view.

SETTINGS MENU OF THE 3D VOLUME RENDERING WINDOW



1	"DICOM opacity" slider
2	Buttons of toggling the visualization of the implant, sleeve, drill or abutment
3	Objects opacity threshold settings
4	Axis planes visualization settings
5	"Planes" button
6	"Axis" button
7	Buttons of switching between low, medium and high-performance graphics settings, depending on the customer PC graphics card processor capacity
8	Color indicator to change the background colour of 3D window

OBJECT MANIPULATION IN THE 3D VOLUME RENDERING WINDOW

In the Volume Rendering window, surface scans, crowns, implants, and pins can be moved and rotated using **modifier keys.**

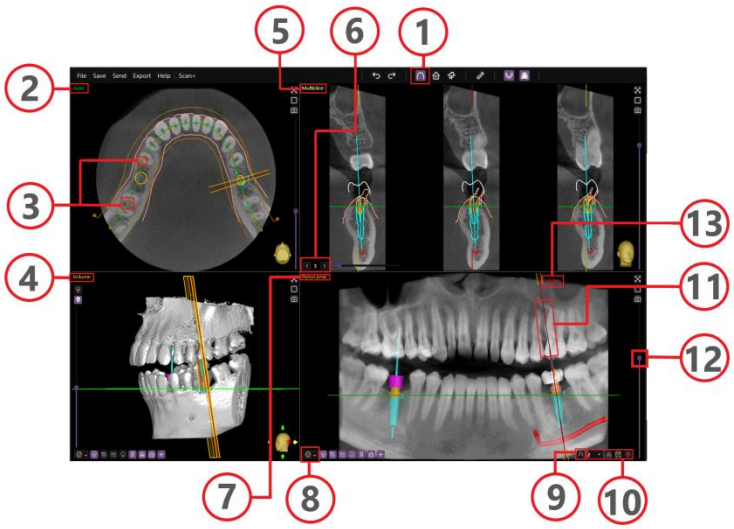
Note!	The object must be unlocked and fully opaque for these actions to work.

Holding **Shift + Left mouse button**	Moves the object within the screen plane.
Holding **Shift + Right mouse button**	Rotates the object around an axis perpendicular to the screen plane, passing through the object's reference point.
Holding **Ctrl + Left mouse button**	Rotates the object in 3D.
Holding **Ctrl + Right mouse button**	Scales the object (available for crowns).
Shift + Left-mouse button click	Opens the Editor menu for any object provided the object has an Editor menu available.

Note!	Shift + left-click for opening the Editor menu is not limited to the Volume window – it also works in other views.

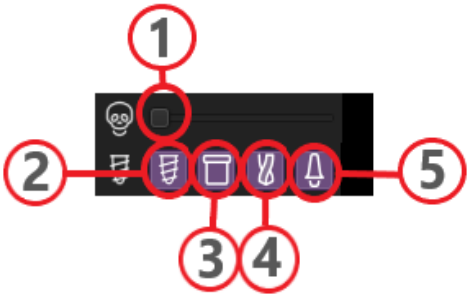
3.5 Panoramic Mode

Serves for general review of the maxillofacial region and allows users to make expanded evaluation of the implant(s) position.



1	"Switch to PANO mode" button
2	Axial viewport
3	Yellow dots for editing existing panoramic curve
4	Volume viewport
5	Multislice viewport
6	Number of slices of Multislice viewport
7	Panoramic viewport
8	Settings menu of Panoramic viewport
9	"Add new Panoramic curve" button
10	Control panel of Panoramic Window
11	Indicator Line of the panoramic slice
12	Slider in Panoramic Window to change the panoramic curve thickness
13	Numerical value of the angle of inclination

SETTINGS MENU OF THE PANORAMIC WINDOW



1	Image opacity threshold
2	"Implant visualization" button
3	"Sleeve visualization" button
4	"Drill visualization" button
5	"Abutment visualization" button

CONTROL PANEL OF THE PANORAMIC WINDOW



"Add Panoramic Curve" button



Click on the Panoramic View list button to choose initial or created panoramic view(s)



Click on the button to switch between Narrow or Wide mode



Click on the button to add the panoramic image to Surgical Protocol



"Delete Panoramic Curve" button

PANORAMIC CURVE

The **Panoramic Curve** identifies the dental arch position.

Hint!	The best area to place the curve is in half of the length of teeth roots, where the canals are going to be good markers for the curve tracing.

Put the mouse cursor over the image and draw new panoramic curve using the left mouse button by placing points one after another on the arch, then double-click to finish tracing. In case of misplacement, move the dots by clicking with the left mouse button on each yellow dot and dragging it into the right position.

The **Indicator Line** of the panoramic slice appears in the axial and in panoramic window. Put the cursor on the **Multislice Window**, scroll the mouse wheel to move the indicator line along the panoramic curve to see the desired cross-sectional slice(s).

Hint!	Quick adjustment of the view. To move the indicator line, put the cursor on the desired area in axial or in panoramic view and double left-click on it. The indicator line of the panoramic slice will move to the selected point immediately.

To adjust the inclination of the cross-sections in the panoramic view click on indicator line and tilt it. The numerical value of the **angle of inclination** will appear near the indicator line.

Click and drag the **slider** in Panoramic Window to change the panoramic curve thickness.

The default settings for **Multislice window** assume three separate cross-section images with distance increment of one millimeter. The number of slices can be increased to five and reduced to one.

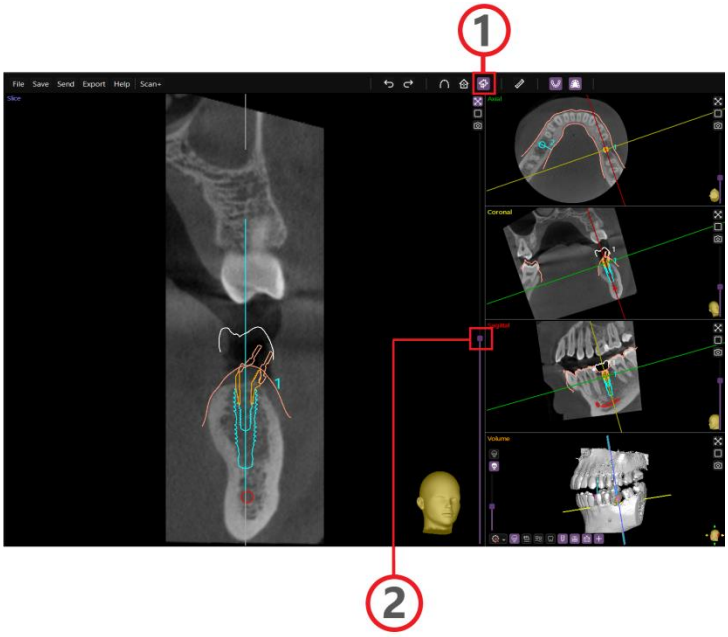
Note!	The angle of inclination will be indicated correctly only if the occlusal plane of the chosen jaw is parallel to the axial plan.

3.6 Rotating Slice Window

The main purpose of the Rotating Slice View is visualization of tissues around the implant for final control and precise correction of the implant and sleeve position.

Double click on an implant in 2D, 3D or implant list to adjust rotational slice axis along implant axis.

To rotate the image around the Implant axis, put the cursor on any point of the Rotating Slice window and scroll the mouse wheel or click and drag on the right part of the window.



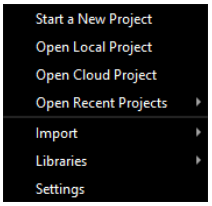
1 Open Slice button to open the Rotating Slice Window

2 Slider to rotate the image around the Implant axis

3.7 Menu bar

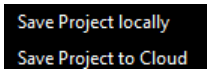
File

Main Menu
Opens the main application menu:



Save

Save
Saves the current project locally or to the cloud.



Send

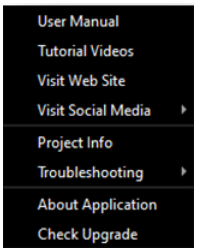
Send
Sends the project to another user.

Export

Export
Opens the Export Menu for exporting project. (see **Annex I** for details).

Help

Help
Opens the help and support section.



Scan+

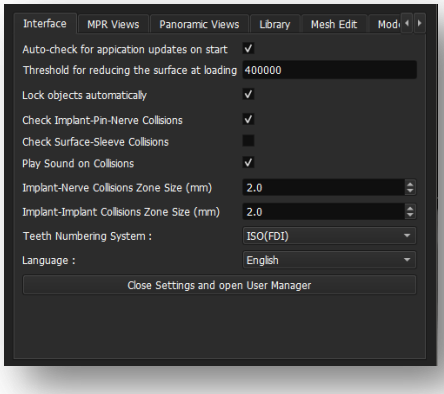
Scan+
Opens the Scan File Manager – a tool for loading and managing optical scan files.

SETTINGS MENU

This menu allows setting the preferences for the software.
The following settings can be set from this page:

1.INTERFACE TAB

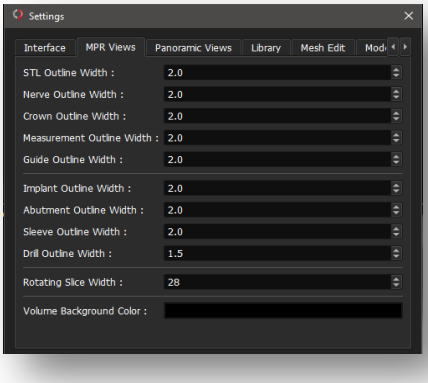
- auto-updates mode;
- max weight of imported STL;
- auto-block of objects movement;
- objects collision detection settings;
- language;
- Teeth Numbering system;
- register a new user or delete current user.



Note! When collision function is selected, the software checks the collisions of the objects automatically. If collision is detected, the triangular sign "Attention" will appear.

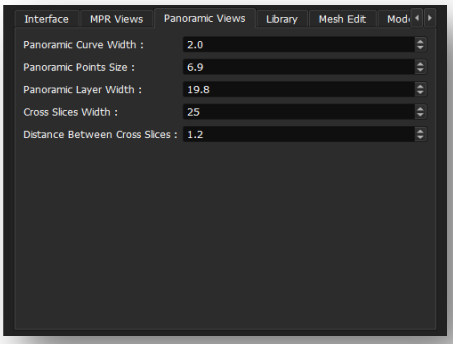
2.MPR VIEW

- outline thickness of elements in MPR mode;
- slice width adjustment in Rotation Slice Window mode;
- default background color of 3D window.



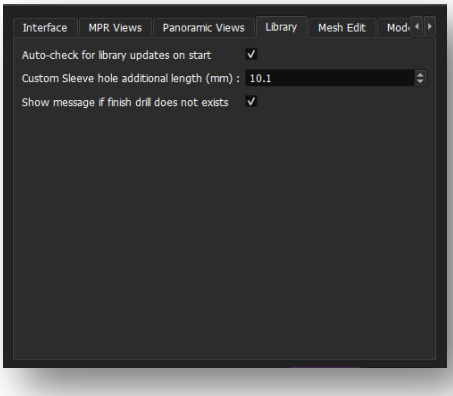
3.PANORAMIC VIEWS

- size of panoramic points;
- outline thickness of Pano mode elements;
- distance between cross slices adjustment.



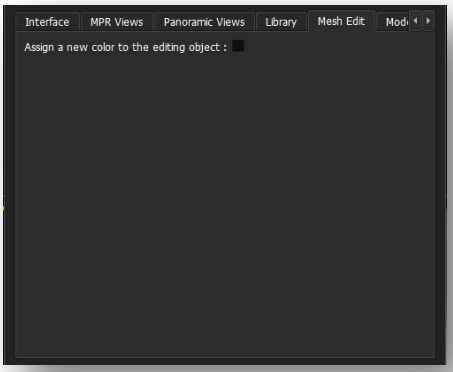
4. LIBRARY

- Library auto-update mode;
- Initial V-factor settings;
- Custom sleeve hole deepness;
- Notification "original finishing drill is not loaded in the local library" settings.



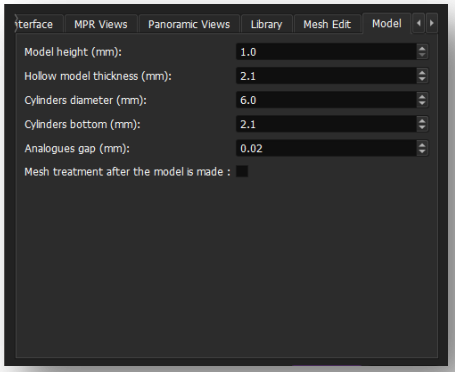
5. MESH EDIT

Mesh Edit setting has the checkbox "Assign a new color to edit object". When it is selected, it determines the color of the newly edited STL file. If the checkbox is not selected, the color of the original STL file is duplicated in the edited version.



6. MODEL

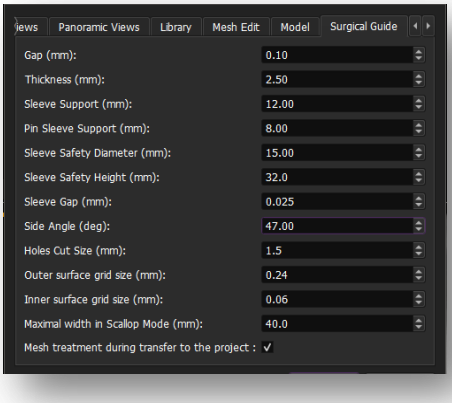
Set default values of the model that will be automatically applied when the Model Builder is accessed.



7. SURGICAL GUIDE

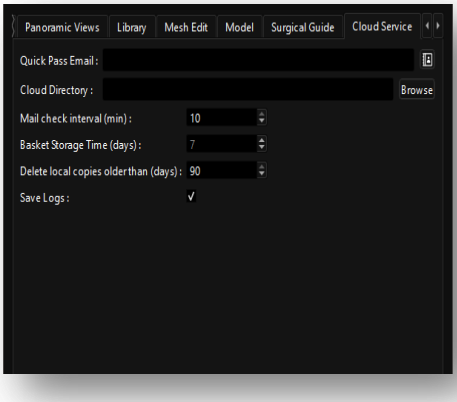
- Surgical guide default settings;
- Default settings of guide surface mesh grid size.

See chapter "**Surgical Guide**" for a more detailed explanation.



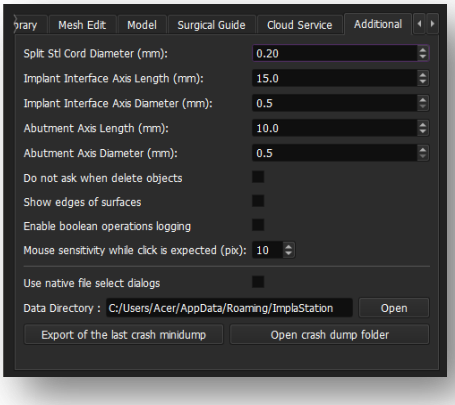
8. CLOUD SERVICE

- e-mail for Quick Pass Email;
- Cloud directory;
- Duration for Cloud Basket Storage, mail check and local copied files;
- auto save logs.



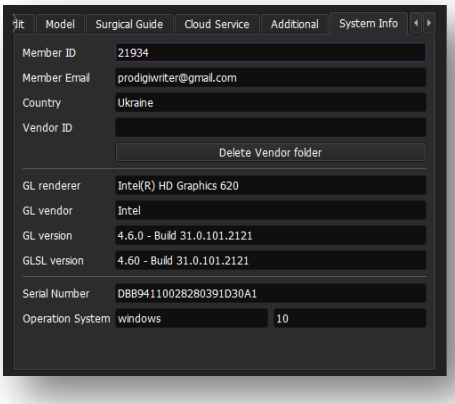
9. ADDITIONAL

- Implant and Abutment Axis parameters;
- Split cord default diameter;
- Data directory;
- Additional parameters checkboxes.



10. SYSTEM INFO

Shows account details, such as email, country, and vendor ID, along with device specifications.



3.8 WORKSPACE CONTROL PANEL



Undo button - Cancels the last action



Redo button - Reverses the last Undo. Used only after Undo.



Switch MPR/ PANO mode button - Switches between view modes



Open Slice button - Opens the Rotating Slice Window



Reset Views button - Resets view settings to default



Measurement button - Performs distance measurement

MEASUREMENT

The **Measurements** tool allows distance and angle measurements directly on the patient's data.
Click the **Measurement** button on the workspace control panel to enter measurement mode and select the measurement type from the menu that appears:

- Distance
- Angle

To **measure distance**: once the mode is active, the ruler is highlighted. Place a start point and an end point to complete one distance measurement. To **measure angle**: place three points – the first two define the first side of the angle, the third defines the second side. Press **Space** key after completing a measurement to start the next one.

Hint!	Pressing Space is required to begin each additional measurement; this is not indicated visually beyond the ruler highlight.

To finish measuring, click the **Measure Distance** button again to release it. Once released, the Space key no longer starts new measurements. Click the **Delete** option to remove all measurements at once using the dedicated button.

3.9 PROJECT & ACCOUNT CONTROL PANEL


Click to open **Cloud**


Click the **Order Form** button to note an information or fill out the digital order form.


Click the **Take Snapshot** button to take a snapshot


Click the **Account** button to open User Manager or order and purchase the export packages.



Regular exports balance

3.10 WORKFLOW PANEL

Tab panel helps users navigate through the patient's case


DICOM management


Segmentation



STL surface management



Nerve tracing



Crown design



Digital Implant, Sleeve, Drill, Abutment management



Anchor Pin management



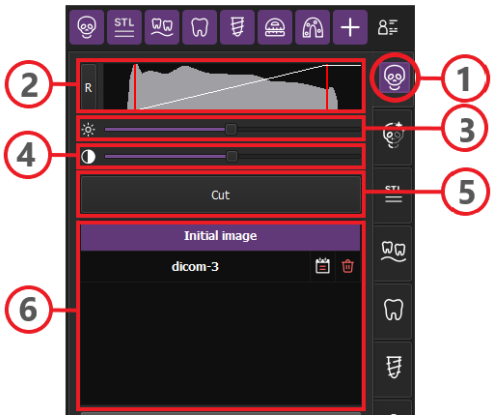
Surgical Guide design



Modeling

4.INPUT DATA (DICOM)

DICOM TAB MENU



1	DICOM tab
2	DICOM Image Histogram
3	Brightness slider
4	Contrast slider
5	“Cut tool” button
6	DICOM list

4.1 Steps for Importing DICOM into a Project

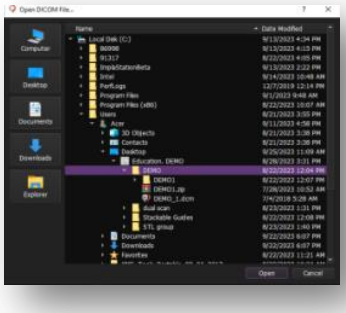
Step 1

Click the Start a **New Project** button via the Start Screen or Main Menu of Menu bar which is a way either to import a dataset from a selected source.

Step 2

In the appeared Data Explorer window, select any object from the following:

- folder with DICOM data;
- multi-frame DICOM file;
- single-frame DICOM files set, one from the set;
- DICOMDIR file;
- Zipped DICOM file.



Step 3

Click **Open**

Hint!	To see DICOM-files without extension choose "All files" option. Ensure to select the correct study and series by comparing them with the patient name in the CT/CBCT data selection dialog. After you have uploaded the relevant dataset, it is displayed in the processing window.

Step 4

In Processing window set the uploaded CT/CBCT dataset:

- adjust the **image opacity threshold** by dragging the corresponding slider;
- with the cropping tool, **remove disturbing or unrelated parts** of the CBCT/CT scan by left-clicking, holding and dragging the purple cropping box inside the windows and area outside the box will be deleted.
- In case of DICOM is displayed with an incorrect orientation upside down, turn it so that the Maxilla is on top and the Mandible is on the bottom using rotation slider.

Click **OK**.

Step 5 (optionally)

After importing DICOM files, image visualization can be optimized using brightness and contrast sliders or the DICOM Image histogram.

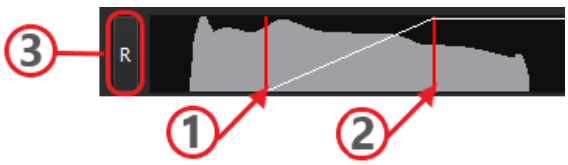
PROCESSING WINDOW SETTINGS



1. Image opacity threshold slider
2. **MIP** mode (Maximum Intensity Projection) and **ISO** mode (Isosurface) switching buttons
3. Cropping box
4. Rotation slider

DICOM IMAGE HISTOGRAM

The **DICOM Image Histogram** provides a graphical representation of grayscale values within the loaded image. It allows for adjustment of image display parameters to enhance visual perception of anatomical structures.



1. **Left Threshold Marker** defines minimum displayed intensity.
2. **Right Threshold Marker** defines the maximum displayed intensity
3. **Reset** button

To optimize the visual representation of a DICOM image, the histogram allows manual adjustment of the visible intensity range using two threshold markers. Adjust the threshold markers by dragging them horizontally:

– Move the **Left Threshold Marker** to the right to increase the minimum brightness threshold, making darker areas less visible.

– Move the **Right Threshold Marker** to the left to decrease the maximum brightness threshold, suppressing overly bright regions.

Use the **R** (Reset) button to restore the default threshold range at any time.

Scrolling the mouse wheel over the histogram adjusts the scale, expanding or compressing the histogram curve.

Note!	DICOM histogram adjustment affects only image visualization and does not modify the original DICOM data.

4.2 DICOM "CUTTING"

The 3D and 2D visualization of the CT/CBCT scan of the patient may be "cut off" by an overlay of the dark mask on a selected area of the DICOM. Created masks can be turned on and off to hide and display fragments of CT/CBCT. This feature is most in demand for working with 3D rendering.

Step 1

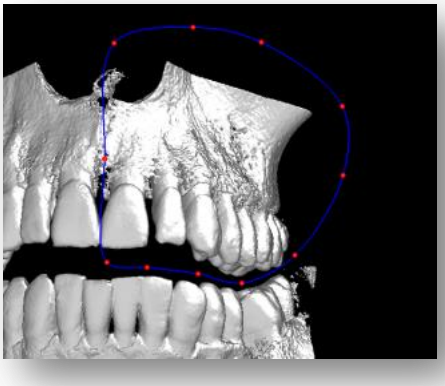
Setup the 3D image position.

Step 2

Click the **Cut** button in DICOM directory in the right part of the screen.

Step 3

Identify the area on which the DICOM mask is going to be cut. Draw the borderline by placing points one after another around the cutting area. Continue to draw the curve that goes back to the starting points and then double-click to finish selection.



Step 4

Either click the **Cut inside selected area** button to cut inside or **Cut outside selected area** button to cut outside.



Step 5

Click the **Save** button to save the created DICOM mask and mask name will appear in the list in DICOM directory on the right part of the screen. Optionally, it is possible to change the name of the saved DICOM mask by clicking the **Edit Title** button. During the design process choose any saved DICOM mask by left-clicking on it.

5.SEGMENTATION

The Segmentation tool extracts and edits 3D information from CT/CBCT data, creating detailed 3D surfaces of various tissues.

It allows for the creation of editable 3D textures from CT data or STL objects, which can then be converted to STL format to model both hard and soft tissues.

5.1 SEGMENT CLEANUP STRATEGY

Texture is a method of representing 3D data as sets of voxels (3D pixels).

Segment is a delineated region of a three-dimensional medical image corresponding to a specific anatomical structure or tissue, used for creating accurate 3D models in navigational dentistry.

SEGMENTATION ALGORITHM:

1. Data extraction from CT/CBCT images (Auto, Custom, from STL).
2. Creation textures from the Data (creates automatically).
3. Texture processing and cleanup (Edit).
4. Conversion to STL format.

SEGMENT CLEANUP STRATEGY

To obtain a high-quality result, follow these steps:

1. Remove external shells (number buttons in 3D window).
2. Seal internal cavities (Edit, 3D brush; Enter button in 2D windows).
3. Remove internal shells (Enter button in 2D and 3D windows).

SEGMENTATION TAB MENU



1 "Segmentation tab" button

2 STL list

3 Segment list

4 Color indicator

5 Visibility button

6 "Delete" button

7 "Lock On/Off "button

8 "Auto Segmentation" button

9 "Add Custom Segment" button

10 "Add Segment from STL" button

11 "Edit" button

12 "Combine" button

13 "Save to STL" button

14 "Exit" button

5.2 SEGMENTATION PROCESS

Step 1

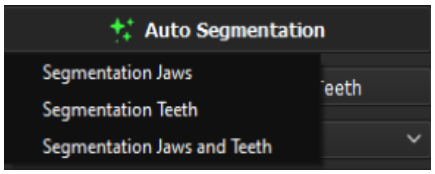
Navigate to the segmentation menu by clicking on the corresponding tab on the workspace panel (**1**).

Step 2

Create a segment using the provided methods:

Auto Segmentation

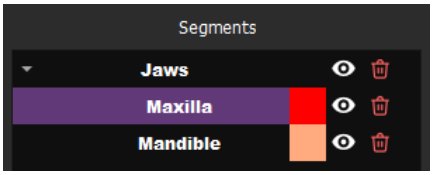
Automatic segmentation uses math algorithms to accurately segment jaws and teeth from noisy CT data.



1. Click the **Automatic Segmentation** button (**8**).
2. Select the object for segmentation: jaws, teeth, or both, via clicking the corresponding button.
   When DICOM files exceed 100x100x100mm: Clicking Auto Segmentation > Segmentation Teeth/Jaws/Jaws and Teeth will prompt a choice for an Area of Interest. Segmentation will only occur within the selected area.
   The automatic segmentation process runs in the background, allowing you to continue working in the program. Monitor the status via the color indicators on the segmentation tab icon or the automatic segmentation button. The colors indicate: White (process not started), Yellow (process in progress), Green (process completed).



3. Once segmentation is complete, add the object to the segment list (**3**) via clicking the appropriate button and specifying the object type. A segment obtained through auto-segmentation can include multiple elements displayed in a cascading list. You can set the entire segment or its individual parts.



Warning!

The data obtained with the Automatic Segmentation Tool must be reviewed, edited if necessary, and approved by the operator before being used for implant planning, guide design, export, or clinical decision-making.

Add Custom Segment

This method is used to segment a specific area.

1. Activate this method by clicking the **Add custom Segment** button (**9**).
2. Adjust the region of interest by moving, setting borders, or rotating the cube boundaries in the MPR or Rendering windows. Modify the size and position of the region of interest by left-clicking, holding, and dragging the boundaries of the purple box. To rotate, left-click, hold, and drag the yellow and green marks in the MPR windows. To move the entire working area, left-click, hold, and drag the square dot located in the center of the purple box.
3. Click the **Apply** button to add the segment to the list. If needed, adjust the optical density thresholds using the sliders that appear.

Add Segment from STL

This method allows you to convert an STL surface into a segment.

1. Click the button to activate this segmentation method (**10**).
2. Select the desired STL surface from the list (**2**) and click **Apply**.

Hint!	An alternative way to add a segment from an STL surface is to drag`n`drop the object from the STL list to the segment list.

Note!	No more than four segments can be added to the segment list.

Step 3(optional)

Edit the added segments using the provided tools:

- Combine (**12**);
- Edit (**11**);

The function menu will be activated after clicking the corresponding button.

Step 4

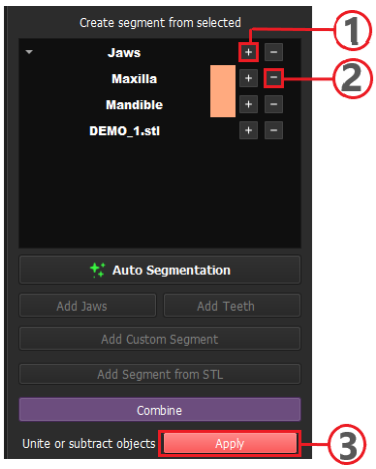
Save segment as STL and complete the segmentation process by clicking the **Done** button (**14**).

5.3 SEGMENT PROCESSING

COMBINE

The **Combine** function merges segments by uniting or subtracting them to create a new texture.

To activate the "Combine" menu, click the **Combine** button. Once activated, control elements will appear in the segment list:



1 "Plus" (+) button to unite segments

2 "Minus" (-) button to subtract segments

3 "Apply" button

Step 1

Click **Combine** button to initiate the process of creation of new segment.

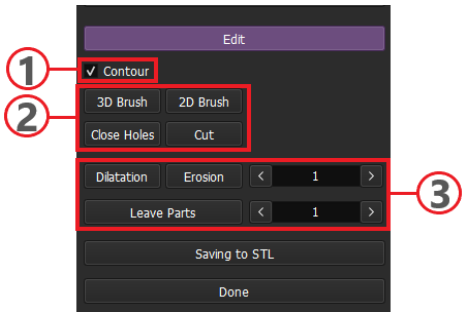
Step 2

In the Segment list click the **plus** (+) next to the desired segment (**1**) in the segment list to unite a segment to the new one or click the **minus** (-)next to the desired segment (**2**) in the segment list to subtract a segment

Step 3

Click **Apply** (**3**) to complete the process and the resulting segment will be shown in the segment list as a new line.

EDIT



The Segment Editor includes groups of tools for local editing (**2**), general editing (**3**), and texture display modes (**1**). Each tool is activated by clicking its corresponding button.

Step 1

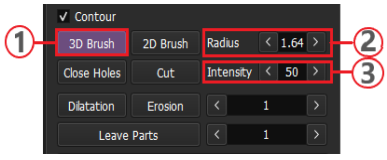
Select the necessary texture from the segment list.

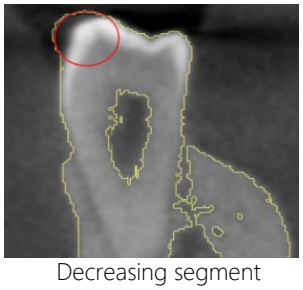
Step 2

Configure the display mode of the segment in slices by using the **Contour** checkbox (**1**). When activated, the selected segment will appear as a contour. When deactivated, the segment will be filled with a background color, and its opacity can be adjusted in the menu.

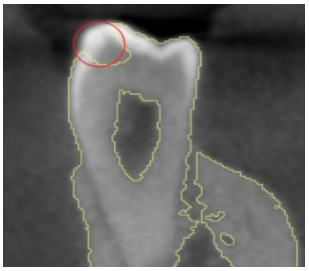
Step 3

Use the provided editing tools to modify the segment.





Decreasing segment



Increasing segment

3D Brush

The 3D Brush tool is designed for localized volumetric (by sphere) adjustments to the ROI threshold level (**1**).

Once activated, configure the brush's radius by the arrows in the menu (**2**) or by scrolling while holding the right mouse button in visualization windows. and intensity (**3**) in the menu.

Place the circle-shaped cursor (2D visualization of the sphere) on the segment. In the MPR windows: *Left-click* to **decrease** the local threshold within the spherical area or *Right-click* to **increase** the threshold level. The 3D Brush tool is used in the MPR windows but can also be used in the rendering window by holding the Shift key.

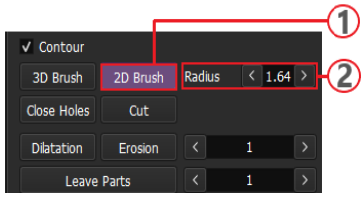
Hold the appropriate mouse button and move the cursor to change the threshold of a bigger area.

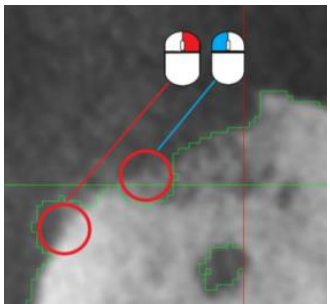
2D Brush

The 2D Brush tool allows for localized editing of the texture within a single layer (**1**).

After activating the tool, adjust the brush radius in the menu (**2**).

Place the circular cursor on the desired segment. Use the *left mouse button* to **add** volume. Or u se the *right mouse button* to **reduce** volume.



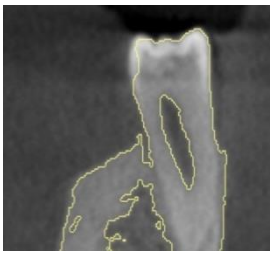


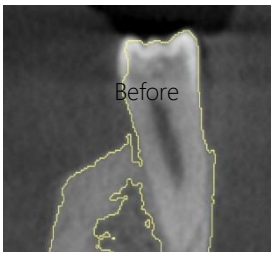
Note! Rotation and scaling of objects in MPR windows with 2D brush and 3D brush are achieved by using the Shift key. Clicking the left mouse button while holding Shift rotates the object, and clicking the right mouse button scales it.

Close Holes / Internal Shells Removing

Activating the Close Holes tool detects and closes all internal holes in the active texture.

Alternatively, internal holes can be closed by hovering over the visualization window (rendering or MPR) and pressing the Enter key. Results depends on the position of cursor: is curor located on on of the MPR windows pressing Enter close internal shells of current slice, In the rendering window cfor all active segment.



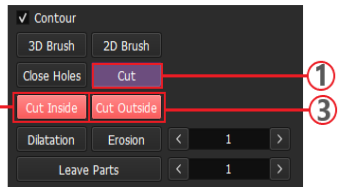


After using Close Hole tool

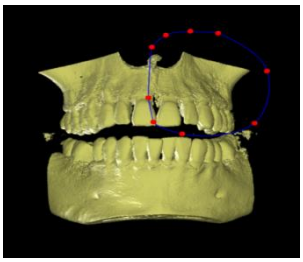
Cut

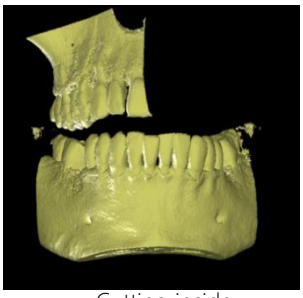
The **Cut** function is designed to remove a specific area from a segment (**1**)

After activating, identify the area of cutting: draw the borderline by placing points one after another around the cutting area. Continue to draw the curve that goes back to the starting points and then double-click to finish selection.

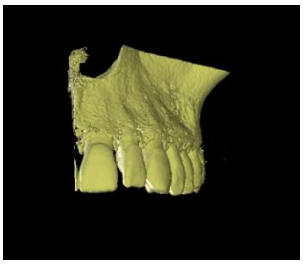


To complete the cutting, go to the editing menu and either click **Cut inside** button (**2**) or **Cut outside** button to cut outside the selected area (**3**).





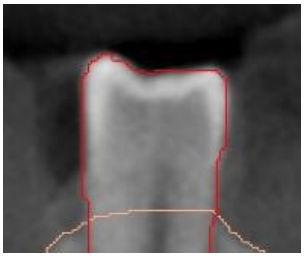
Cutting inside



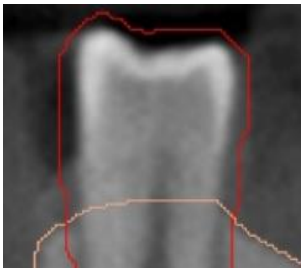
Cutting outside

Dilatation and Erosion

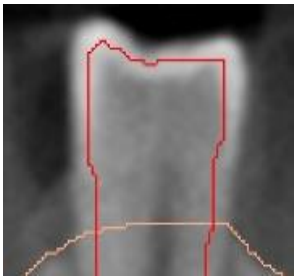
These tools enable the increase or decrease of the entire active segment by a specified number of voxels.
Enter the value into the input field and click:
Dilatation to add voxel layers of the selected edge.
Erosion to reduce voxel layers of the selected edge.



Before



After Dilatation

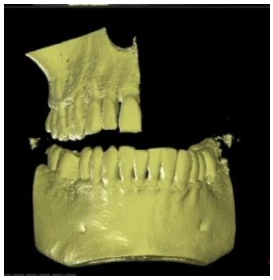


After Erosion

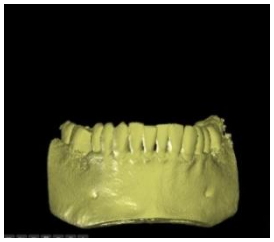
Leave Parts / External Shells Removing

Leave Parts Tool removes the smallest parts of the segment, leaving only the specified unconnected volumes of the active segment.

Enter the number of texture parts you want to keep and click the **Leave Parts** button.



Before



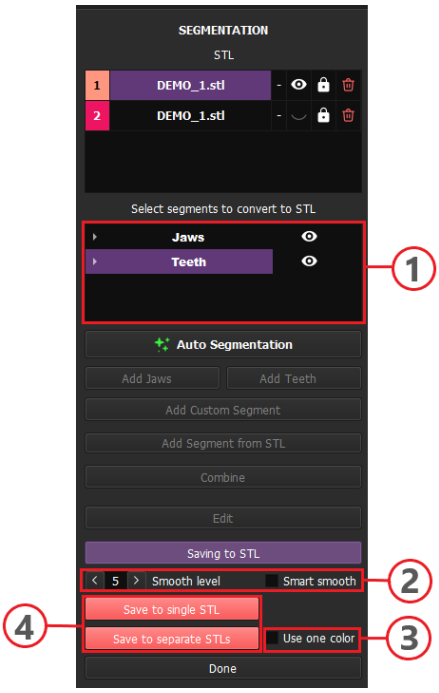
After

Hint!

To remove unnecessary segment obtained through auto-segmentation hover over the segment in the rendering window and right-click while holding down the Shift key.

SEGMENT TO STL CONVERSION TOOL

Saving to STL tool is designed to convert segment to STL. Activating the tool displays a settings menu for configuring the conversion process.



1 Select segments to convert to STL

2 Adjust the Smoothing Level by entering a value into the numeric input field.
Check the "Smart Smooth" checkbox to prevent loss of segment shape during smoothing sharp edge.

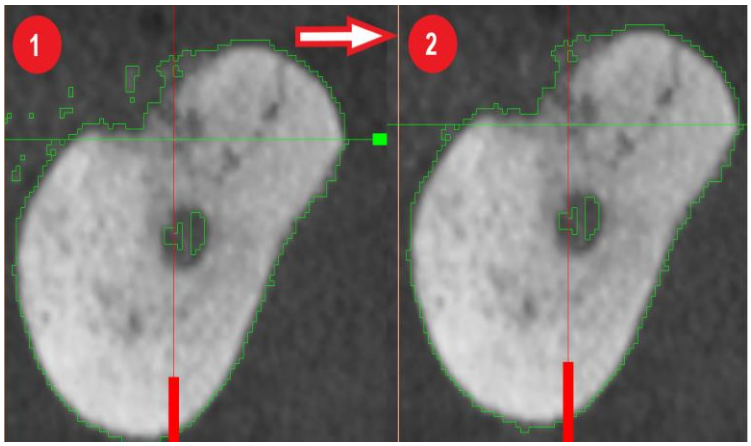
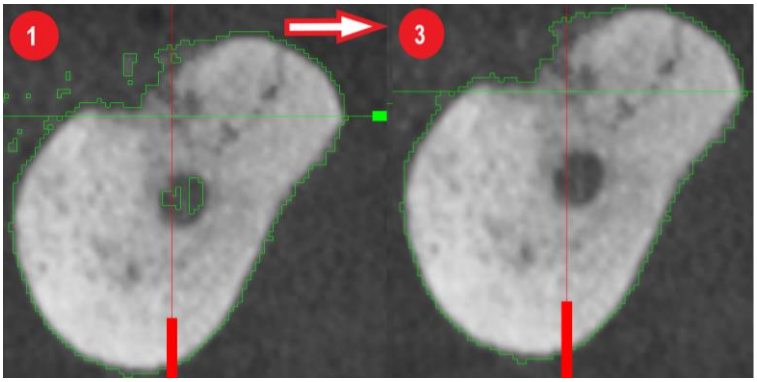
3 To unify the color of teeth generated through auto segmentation, activate the "Use One Color" checkbox.

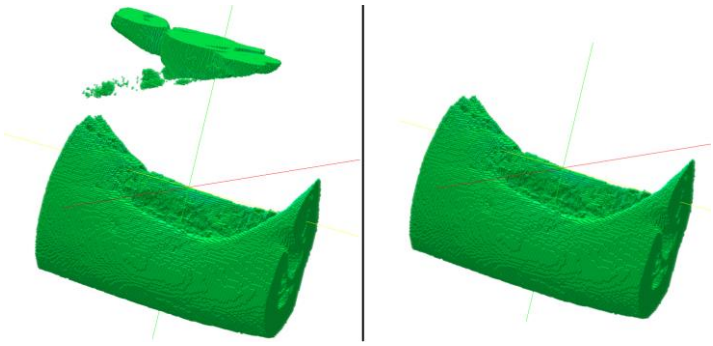
4 Complete the conversion process by choosing the method for saving multiple segments.

Hint!

Convert button has been removed in current release. To convert DICOM to STL after selecting optical density in rendering window add Custom Segment, select ROI and proceed to Save to STL.

HOTKEYS

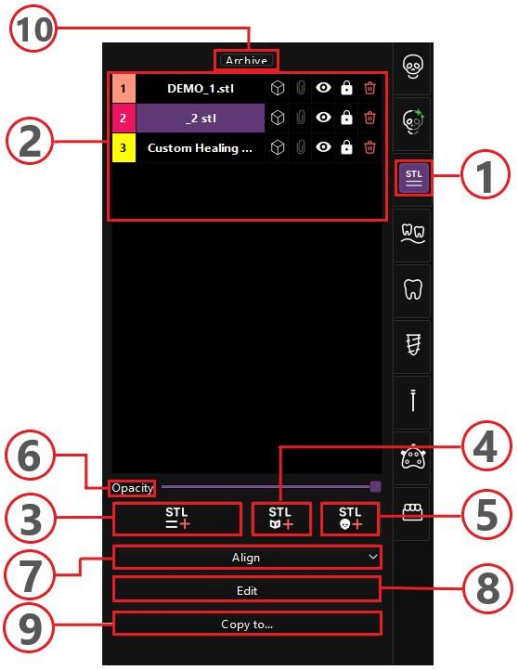
Shortcuts	Functions
When the cursor is in the slice window	
Number Button 1-9	1 – removes all external shells, leaving one largest object. 2 – removes all external shells, keeping the two largest objects 3-9 - the 3-9 largest objects are left. 
Enter button	in chosen 2D window - removes internal holes in 3D window - removes all internal shells 

When the cursor is in the rendering window	
Number Button 1-9	1 – removes all external shells, leaving one largest object. 2 - removes all external shells, keeping the two largest objects 3-9 - the 3-9 largest objects are left. 
Enter button	Remove all internal shells
Modifying operations	
Backspace	Cancellation the operation
Dragging the necessary STL line to the empty part of the texture table	Conversion the STL to the texture
Slice numbering	The number of MPR slices on each axis depends on the size of the ROI box and the selected Grid Size dispersion.

6. INPUT DATA (STL)

Surface files in **.stl, .obj** format can be imported and implemented into the project. Jaw surface scans can be obtained using third-party intraoral or laboratory optical or laser scanners.

STL TAB MENU



1 "STL tab" button

2 "STL surfaces" list

3 "Load surface from file" button

4 "Load surface from library" button

5 "Load surface from DICOM image" button

6 STL Opacity threshold

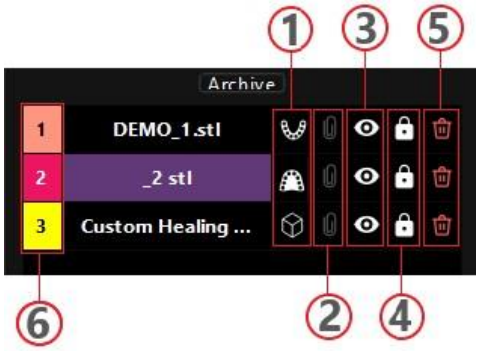
7 "Align" button

8 "Edit" button for opening Editor menu

9 "Copy to" button

10 "Archive" button

STL LIST MENU



1 "Jaw type" button

2 "Link" button

3 "Show on(off) / Hide" button

4 "Lock on/off" button

5 "Delete STL surface" button

6 Color indicator: click on it to change color of STL surface

Hint!

Archive hides STL files from the list without deleting them.
To **archive** a surface: hold **Shift** and click its **Show(on)/Hide** icon (**3**). To **restore** it: click the Archive button to reveal hidden files, then hold **Shift** and click the icon (**3**) again.

6.1 IMPORTING STL FILES / SCAN+

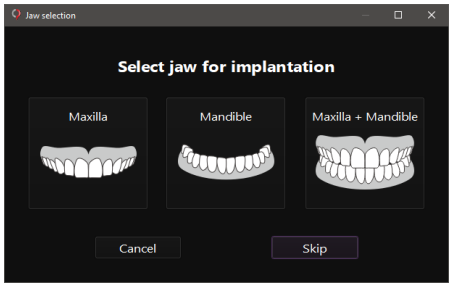
Step 1

Click the **Scan+** button in the Menu bar to open the Scan Manager when several scans need to be imported at once.

Hint!	Scan can also be imported directly using the **STL+** button on the STL tab control panel.

Step 2

Select the jaw for surgical guide planning.

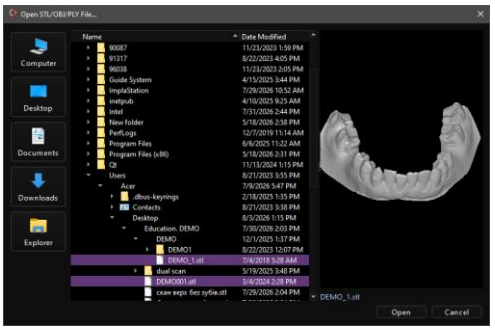


Note!	This window appears only the first time the Scan+ button is clicked within a project.

Step 3

Select the appropriate STL file/files on your computer and click **Open.**

To select several scans at once, hold **Ctrl** and click each file to add it to the selection individually, or hold **Shift** and click the first and last file in a range to select all files between them.



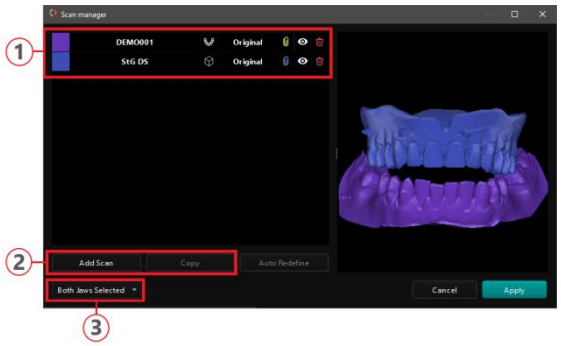
Note!	The software's built-in file explorer is convenient for selecting scans located in different folders; switching between the built-in explorer and the OS file explorer is available in **Settings**.

Step 4

Configure scans in the **Scan Manager**.

For each scan, the following can be **adjusted** (**1**):

- File Name
- Color
- Jaw Type
- Import Position (Align, Original, Global Origin, Axes Origin)



Note!	Scan Manager **automatically** detects and proposes a Jaw Type, Import Position, and grouping for each scan; these suggestions should be reviewed and, if needed, adjusted.

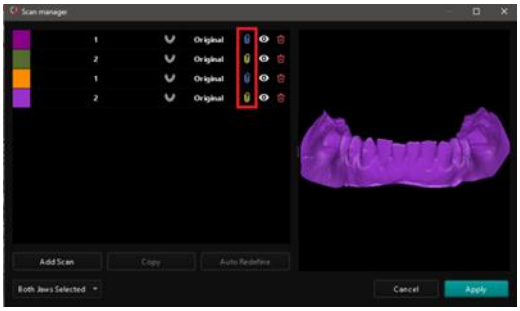
Add additional scans with (**2**):
- Copy Button
- Add Scan Button.

Correct the jaw selection for surgical guide planning, if needed, using the dropdown at the bottom of the Scan Manager window (**3**).

Step 5 (optional)

Group the scans, if needed, using the link icon next to each scan in the list. The main purpose of this feature is an ability to move selected and grouped up STL objects together. The mutual position of the grouped up STL surfaces remains unchanged.

Clicking the link icon assigns the scan to a color group; clicking it again cycles to the next color. Scans sharing the same link color form a group.



Note!	If only one jaw was selected for planning, only one group can be created; if both jaws were selected, up to two independent groups can be created.

Within a group, one scan is selected as the reference surface – it is aligned, automatically or by points – and the other scans are manually assigned to the group; they are positioned in their original coordinates relative to the reference surface.

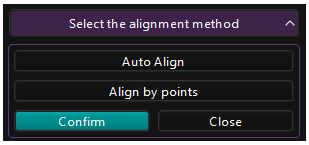
Step 6

Click **Apply** to confirm the configuration and transfer the scans into the project.
Alignment is then performed according to the Import Position selected in Step 4. Once the alignment is complete, click **Confirm** to accept the result. All loaded surfaces appear in the **STL list**.

Note!	If the amount of triangles in an STL file is exceeding a threshold (400000 triangles on default, may be adjusted in the settings menu), the software would propose to reduce the number of triangles in the file.
Note!	The scan can also be repositioned, rotated, or scaled in the Volume window using modifier keys (**3, 4**).

6.2 ALIGNMENT

Scan files can be aligned to DICOM data automatically or manually.



AUTO ALIGNMENT

3D Exacto automatically matches each STL file to the corresponding surface.

Step 1

Choose Auto Align mode in Scan manager or via Align button in STL tab panel.

Step 2

Select the surface with which you want to align. The alignment can be used to DICOM or to another STL file.

Step 3

Check and adjust the result of alignment manually:
On MPR mode click on STL surface center (the square point which is marked in color of corresponding STL surface) and drag it or click on the STL surface borderline and tilt it. (**see Annex A**)

Step 4 (optional)

After the scan has been aligned with the DICOM data, the **Fine Tuning** option becomes available in the alignment panel. The **Deeper** and **Shallower** buttons make incremental adjustments to the scan position to achieve optimal alignment. Clicking Reset restores the scan to its original aligned position..



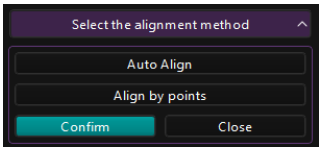
Click **Confirm** to apply the result.

To realign STL files or STL and DICOM files click on **Align** button on control panel of STL tab.

POINT ALIGNMENT

Step 1

Choose **Point Align** mode in Scan manager or via Align button in STL tab panel.

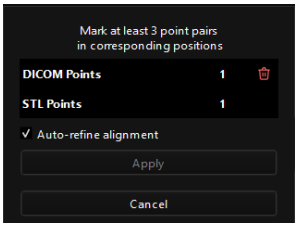


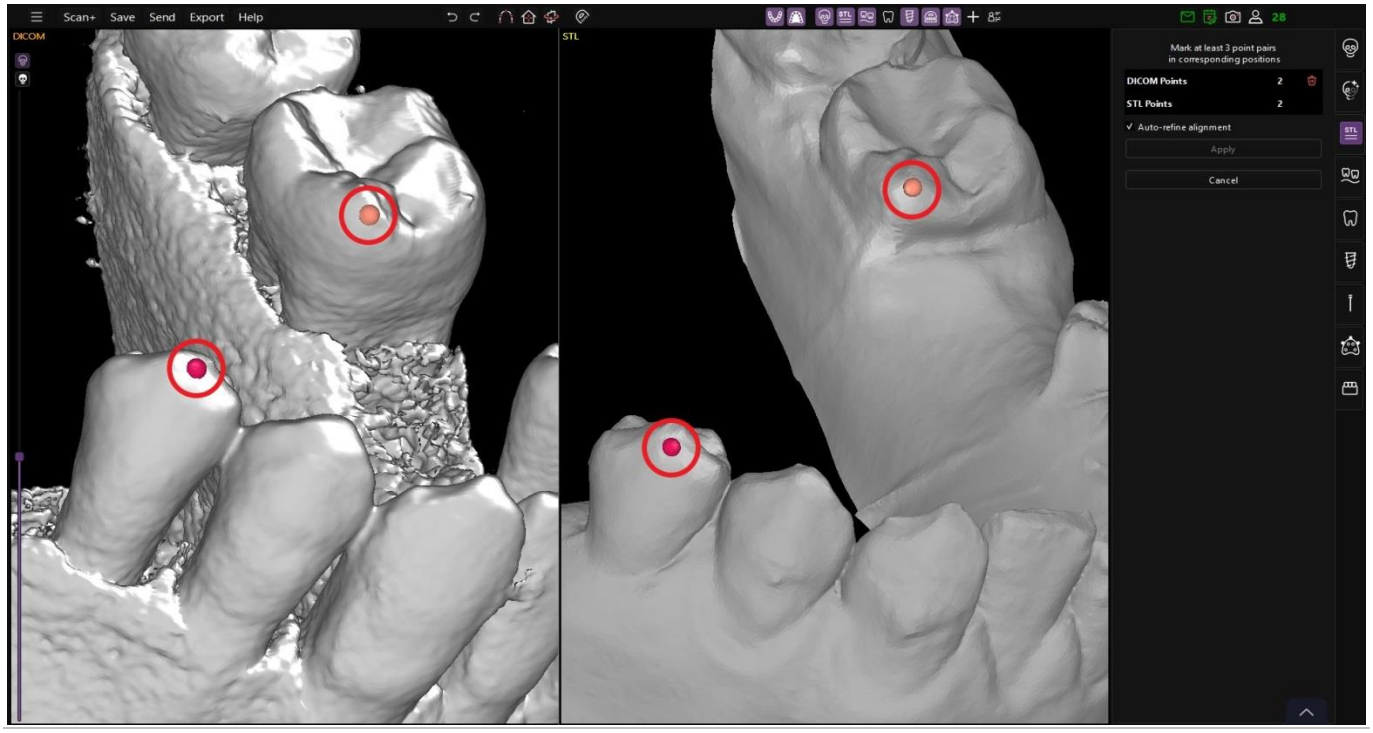
Step 2

Select the surface with which you want to align. The alignment can be used to DICOM or to another STL file.

Step 3

In the **Alignment window**, align the objects by selecting matching landmark points on both surfaces:
Left-click a point on one surface to mark it as a landmark. Click the corresponding point on the other surface. To remove a point, right-click it. If the **Auto-Refine Alignmen**t checkbox is selected, 3D Exacto automatically fine-tunes the alignment after the points are applied, improving the match beyond the manually placed landmarks.
Select at least three corresponding point pairs, then click **Apply**.





Step 4

Check and adjust the result of alignment manually:
On MPR mode click on STL surface center (the square point which is marked in color of corresponding STL surface) and drag it or click on the STL surface borderline and tilt it (**see Annex A**).

Step 5 (optional)

After the scan has been aligned with the DICOM data, the **Fine Tuning** option becomes available in the alignment panel. The **Deeper** and **Shallower** buttons make incremental adjustments to the scan position to achieve optimal alignment. Clicking Reset restores the scan to its original aligned position..

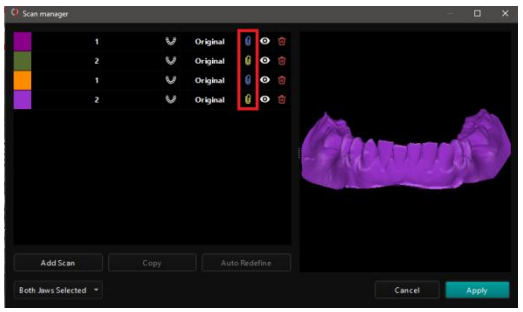


Click **Confirm** to apply the result.

To realign STL files or STL and DICOM files click on **Align** button on control panel of STL tab.

ALIGNING MULTIPLE SURFACES

When DICOM and surface files are loaded, 3D Exacto matches each surface to its corresponding jaw (upper or lower), based on the jaw selection made at project start.



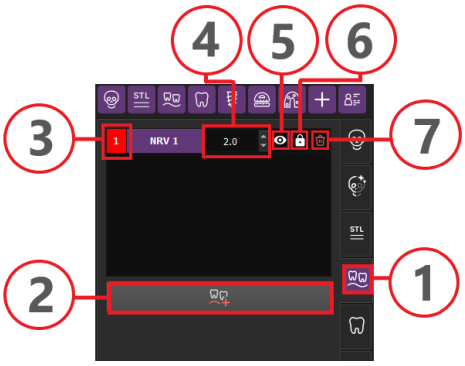
Surfaces can be grouped using the Link tool. In a group, one surface aligns to the DICOM – automatically or by points. The other surfaces in the group simply follow it. This way, all surfaces in the group move and align together, instead of being positioned one by one.

This makes it easier to manage projects where several surfaces are loaded for the same jaw, keeping them aligned and moved as a single unit instead of positioning each one individually.

Note!	Multiple link groups can only be created if both jaws were selected at project start. If only one jaw was selected, all linked surfaces belong to a single group.

7. NERVE CANAL TRACING

NERVE MENU



1 "Nerve tab" button

2 "Draw new nerve" button

3 Color indicator. Click on it to change color of nerve

4 Nerve diameter window

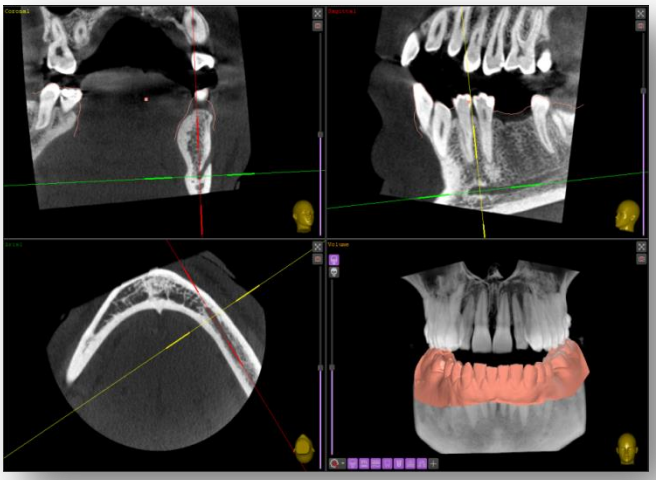
5 "Show On/Off" button

6 "Lock On/Off" button

7 "Delete" button

Step 1

Select MPR mode. Setup the slice planes to visualize nerve canal clearly in axial, cross-sectional and sagittal view.

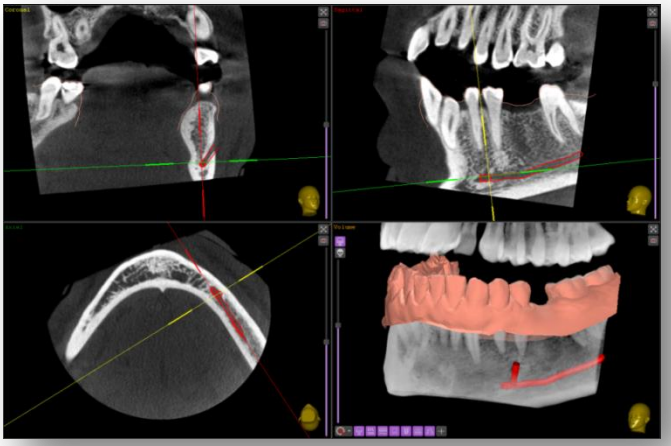


Step 2

Click **Nerve** button on tab panel on the right part of the screen and then **Draw new Nerve** button on the workflow panel.

Step 3

Trace nerve canal by using the left button of the mouse points one after another from frontal part to distal part in the *Sagittal* window or by clicking and scrolling through the Cross-sectional view.



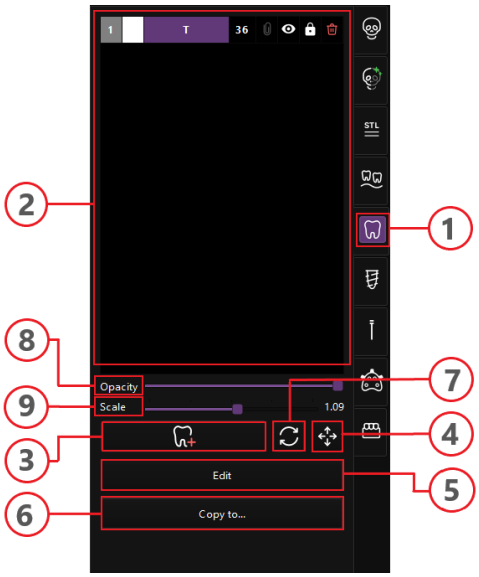
Step 4

Complete the operation by double-clicking the left mouse button or by clicking the "**Lock**" button in the nerve tracing menu panel.

Warning!	Make sure that the nerve is correctly traced. Always maintain an appropriate safety distance to the nerve canal.
Warning!	The pathway of imaged nerves is for display only, location accuracy of the traced nerve is not tested, and pathways of imaged nerves can not be used as sole information for the clinician to make clinical decisions.

8. VIRTUAL CROWNS PLACEMENT

WORKFLOW PANEL OF CROWN MODE



1 Crown tab

2 Crowns list

3 “Place new Crown or Tooth” button

4 “Move Crown or Tooth Mode” button

5 “Edit” button

6 “Copy to” button

7 “Replace” button

8 Opacity threshold

9 Scale slider

Step 1

Select MPR mode. Set up the slice planes to visualize the tooth/teeth position to be planning in axial, cross-sectional and sagittal view.

Step 2

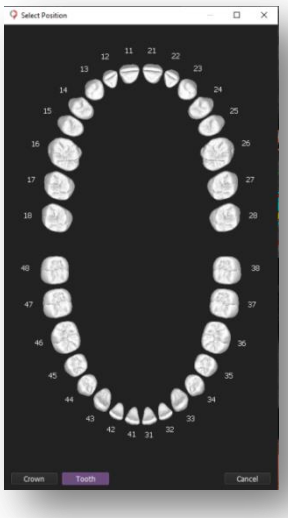
Click **Crown** button in the tab panel on the right part of the screen then **Place new Crown or Tooth** button on the workflowpanel.

Hint!	Click the **Export** button to export the crown or teeth as stl-file to specified folder on the PC.

Step 3

In appeared **Select Position** window select:

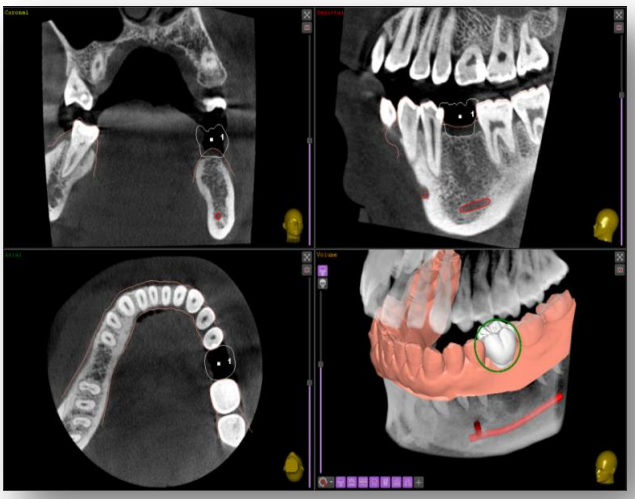
- the crown or tooth mode;
- click on the desired tooth in the virtual schem, and the virtual crown or tooth is going to appear in the pre-set area.



Step 4

Manage the selected crown/tooth:

- Change the tooth position in axial, cross-sectional and sagittal view by clicking on and holding central square and dragging it or clicking on and holding the line tilt it.
- change the tooth position in 3D mode, by clicking on the tooth. Left-click, hold and drag this green line to tilt the tooth. Right-click, hold and move the mouse forward to increase the tooth size or move the mouse back to decrease tooth size.
- Change the size of the tooth/crown using **Scale** slider.
- Replace the tooth/crown using the **Place new crown/tooth** button.
- Reposition the tooth/crown using the **Move Crown/Tooth Mode** button.
- Edit a crown/ tooth with the Editor, by clicking the **Edit** button (see **Annex G**).



Note!	The crown can also be repositioned, rotated, or scaled in the Volume window using modifier keys (**3, 4**)

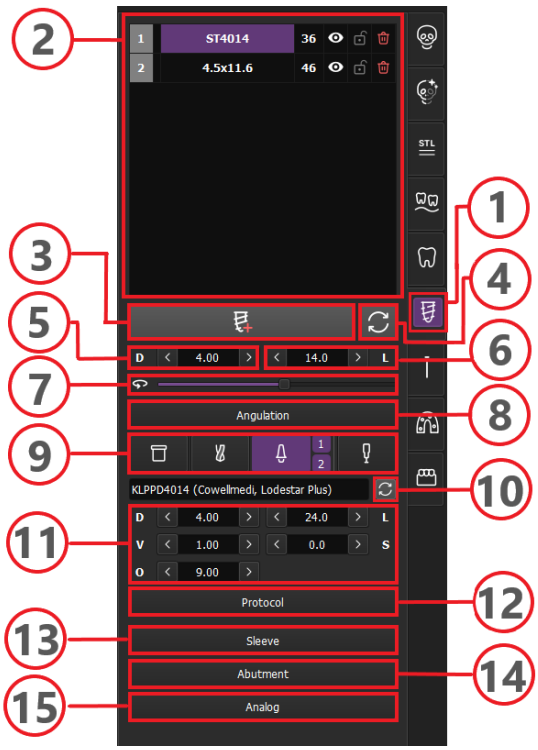
Step 5

Click **Lock On/Off** button in the Crown tab panel to lock in the required settings.

9. IMPLANT PLANNING

The software allows users to perform implant planning using Panoramic, MPR or Rotation Slice mode.
The general idea of the planning concept is: - the implant is the primary object; - the drill tip is connected to the implant apex (with adjustable v-factor shift); - occlusal surface of the sleeve is connected with a drill stopper surface (with adjustable Spacer).
If the drill length is changed, the sleeve position will be changed equally, increasing and decreasing sleeve offset.

WORKFLOW PANEL OF IMPLANT TAB



1 Implant tab
2 Implant list
3 “Place new Implant” button
4 “Replace implant” button
5 Occlusal diameter
6 Length
7 Rotation implant slider
8 “Angulation submenu” button
9 “Visualisation of Sleeve, Drill, Abutment, Analog” buttons
10 “Replace Drill” button
11 “Drill” submenu
12 “Protocol menu” button
13 “Sleeve submenu” button
14 “Abutment submenu” button
15 “Analog submenu” button

9.1 Implant Placement

Step 1

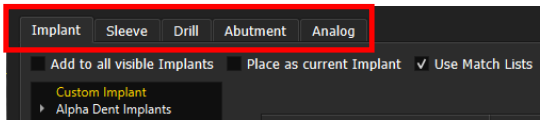
Select MPR or Pano mode. Set up the planes-axis to visualize the further implant position to be planning in axial, cross-sectional and sagittal view.

Step 2

Click **Implant** button in the tab panel on the right part of the screen. Click **Place new Implant** button.

Step 3

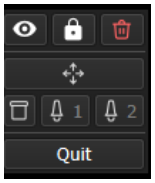
At the appeared window, choose the "implant", "sleeve", "drill" and, if necessary, "abutment" corresponding submenu by left-clicking. Options choosing implants:



- Custom implants;
- Implants from library.

Step 4

Manage implant position:



- in axial, cross-sectional and sagittal view just click on and hold square mark at the occlusal line of the implant and drag it, or click on and hold the implant contour line and tilt it.
- in 3D window right-click on the implant. In the modal window, click the **Move** button. The green and blue round lines are going to appear. Left-click into green circle, hold and move implant, or pull green circle to tilt the implant. Left-click, hold and move the blue circle to rotate the implant.

Note!	The Implant can also be repositioned, rotated in the Volume window using modifier keys (**3, 4**).

- Click the **Set Parallels to Current** button on the workflow panel (**8**) to align the parallels of the implants to the selected implant. First, choose the implant to be used as the reference, then click **Set Parallels to Current**, and select the implants you wish to align. Fill in the value of the Max deflection angle (deg) and confirm the selection by clicking **OK**.

Step 5

Manage implant parameters:

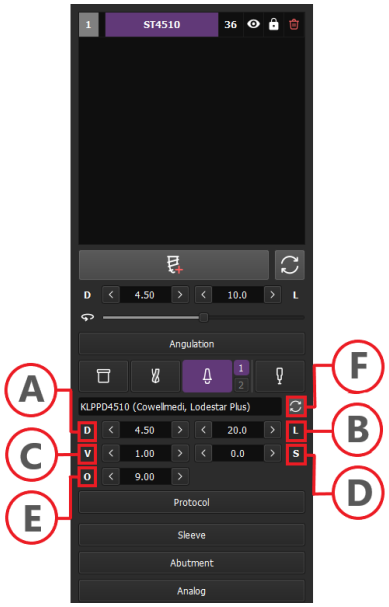
- Adjust the implant size by modifying the **D (Diameter)** (**5**) and **L (Length)** (**6**) fields on the workflow panel. To do this, either click on the field and scroll the mouse wheel, manually enter the desired value, or use the arrows.
- Configure the Implant, Drill, Sleeve, and Abutment settings by clicking the **Replace Implant** button (**4**).

Click **Lock On/Off** button in the Implant tab panel.

Step 6

In **Drill** submenu set up the drill diameter "D" (**A**), drill length "L" (**B**), drill V-factor (**C**), and offset (**E**). The drill spacer "S" (**D**) is an option to add the regulated space between stopper of the drill and contacted surface of the sleeve to change a position of the sleeve and decrease the sleeve offset during digital implant planning.

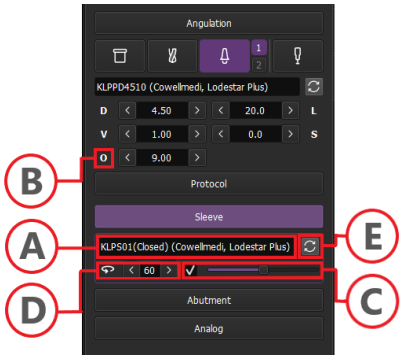
To reset / replace drill, click **Replace Drill** button (**F**).



Step 7

Click the **Sleeve** submenu and check the model of the sleeve (**A**), offset height "O" (**B**).

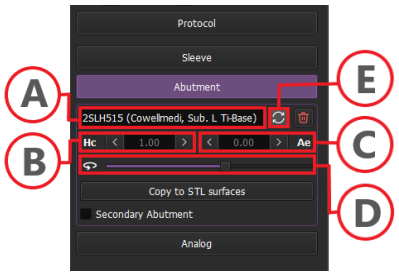
To rotate a sleeve separately from implant, click on the slider and drag it (**C**) to rotate a sleeve by an angle multiple of 15, 30, 60, 90 degrees, choose the rotation angle checkbox "R" and select one of the four options: 15, 30, 60, 90 (**D**).
To change the current sleeve, click on the **Replace Sleeve** button (**E**).



Note! The initial rotational position of the sleeve is related to the rotational position of the implant.

Step 8

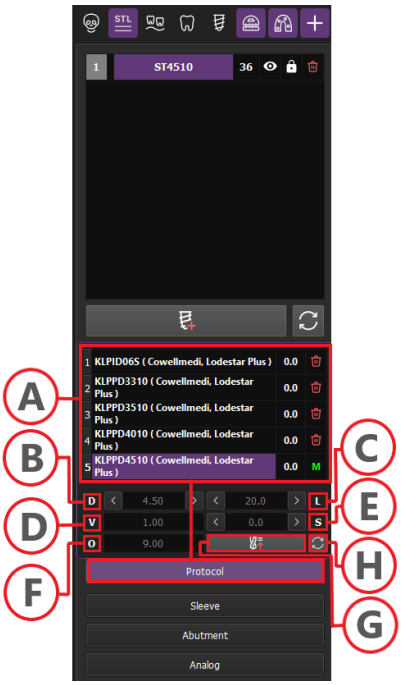
Click the **Abutment** submenu in the right part of the screen and check the model of the abutment (**A**), collar height "Hc" (**B**) and extension angle "Ae" (**C**).
The initial rotational position of the abutment is related to the rotational position of the implant.
To make a rotation of the abutment, click on the slider and drag it (**D**). If library abutment has specified information about rotation grid, its rotation will available with this grid only. Fine rotational adjustment will be available together with the implant.
To change the current abutment, click on the Replace Abutment button (**E**).



Step 9

Click **Lock On/Off** button in the Implant tab panel.

PROTOCOL



The **Protocol** submenu (**A**) allows to review and control the drilling sequence.
Clicking the Protocol button with the left mouse button:

1) the sleeve positions blocked for current drill. That drill will be marked as a M – master drill. In depending from preferable drilling protocol, the Master drill may be the Final drill in the drilling sequence or intermediate.
2) displays a submenu with drills sequence tab and additional buttons and parameters including drill diameter "D" (**B**), drill length "L" (**C**), drill V-factor "V" (**D**), drill spacer "S" (**E**) and offset "O" (**F**). A new drill bit can be added by left-clicking the (**G**) button. The current drill bit can be replaced with the (**H**) button.

After adding a drill, its parameters can be adjusted in the value windows. The sequence of drills can be easily arranged by dragging them to the corresponding tabs.

The drilling sequence can either be pre-set within the drilling library and automatically logged in the protocol or can be manually set.

Edit the parameters of each drill in the protocol by selecting the respective drill and setting the values for drill diameter "D," drill length "L," drill V-factor "V," and drill spacer "S."

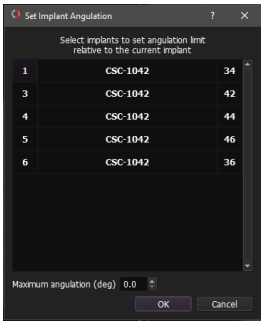
Use drag`n`drop to change the sequence of drills in the protocol, including the Master-drill.

ANGULATION CONTROL



The **Angulation** submenu is used to evaluate and control of the angle of deflection between implants or abutments.
After clicking the **Angulation** button **(1)**, the corresponding menu opens. The system automatically selects the pair of implants with the greatest deflection angle**.**
The type of element used for angle calculation – implant or abutment – can be selected.
When the position of components is adjusted, the deflection angle is recalculated and displayed in real time **(5)**. If needed, result can be updated manually by clicking the **Reload** button **(3)**.
To manually select element for angle evaluation, click on the item display box **(4)** and choose positioned element from the dropdown list. Value of angle is calculated based on the selected pair.

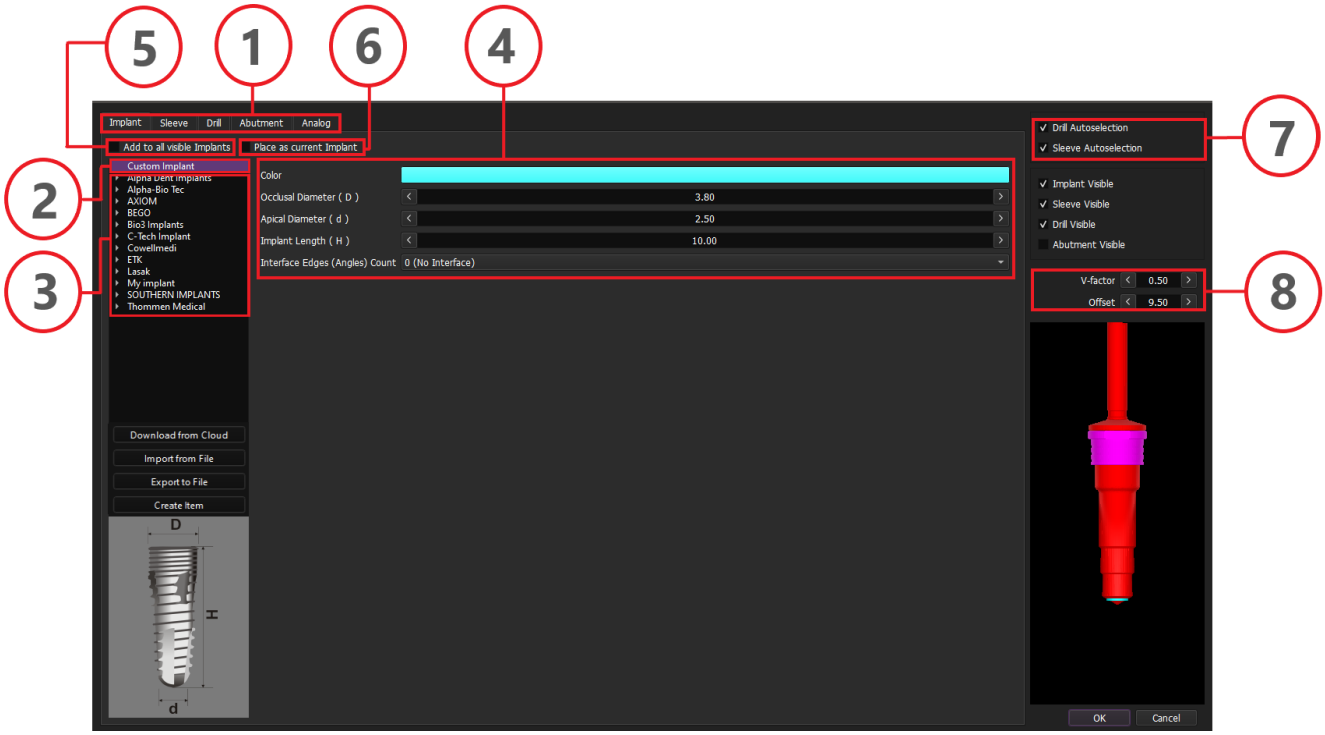
SET IMPLANT ANGULATION



The **Implant Angulation Limit** feature is used to align the axes of selected implants relative to reference implant.
Select the necessary implant from the list. Then click **Set Implant Angulation** button **(6)** and choose the implants to be aligned.
Set the maximum allowable deflection angle (in degrees), and confirm by clicking **OK**.

9.2 OPTIONS OF CHOOSING IMPLANT ELEMENTS

Placing an simplified implant from the constructor of Custom Implants



1	Implant, Sleeve, Drill, Abutment, Analog tabs
2	Custom implant
3	List of downloaded libraries
4	Parameters of Implant
5	"Add to all visible implant" checkbox
6	Place as current Implant checkbox
7	Autoselection checkboxes
8	V-factor and Offset inputs

CUSTOM ELEMENTS

Step 1

Click the **Implant** submenu and select **Custom Implant** option on the list.

Step 2

Choose the implant color and enter relevant dimensions for "Occlusal Diameter" (D), "Apical Diameter" (d), "Implant Length" (H) and "Interface Edges (Angles) Count".

Step 3

Mark the **Place as a last Implant** checkbox to save settings for the next implant(s) to be placed. To restore settings for the next start of the software, mark the appropriate checkbox in the lower right of the submenu window.

Step 4

Add the V-factor depth by scrolling the mouse wheel, or writing the number, or by clicking on the arrows. Increasing or decreasing the value of the V-factor leads to the drill tip shifts down or shifts up relative to the implant apex.

Step 5

The same can be performed for custom sleeves. Click the **Sleeve** submenu, select **Custom Sleeve** option, choose the sleeve color and enter required dimensions for "Sleeve Inner Diameter" (d), Sleeve Outer Diameter (D), Sleeve Height (H), Sleeve Edge Diameter (De), Sleeve Edge Height (He).

Step 6

Custom Drills. Click the **Drill** submenu, select **Custom Drill** option, choose the drill color and enter required dimensions for "Drill Length" (L), "Drill Diameter".

Step 7

Click the **Abutment** submenu, select **Custom Abutment** option, choose the abutment color and enter required dimensions for "Collar Height" (Hc), "Abutment Angle" (A).

Step 8

Click **OK** and select the tooth position.

PRELOADED ELEMENTS

To upload implant libraries from the server

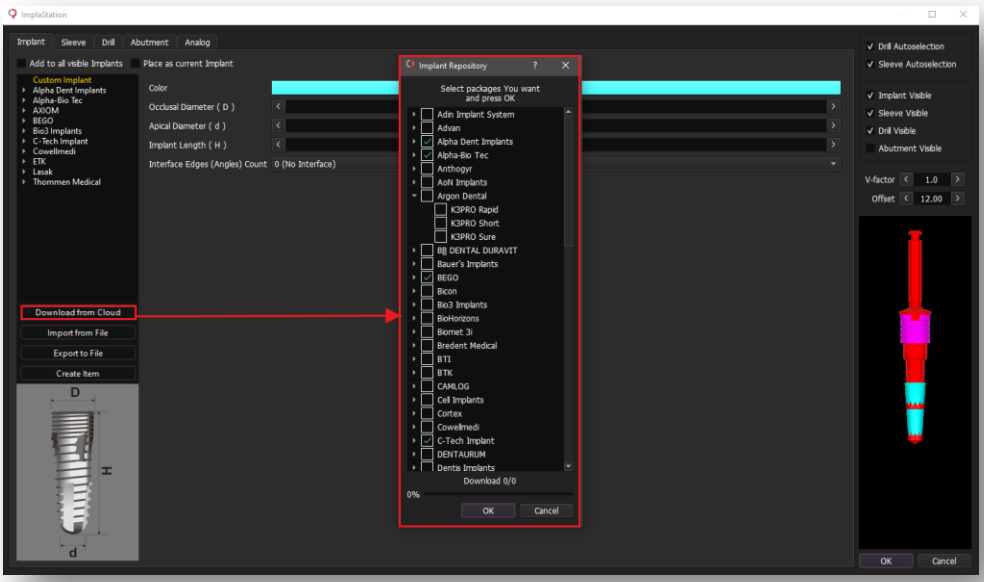
Step 1

Click **Replace Implant** button and then click the **Download From Cloud** button.

Step 2

In appeared window select the implant line by marking checkbox and click **OK** and the chosen library will be downloaded into the software Implant list.

The same downloading procedure can be performed for drills, sleeves, and abutments libraries.



Warning!	Libraries identified as reverse-engineered, non-original, or testing libraries are intended for software testing only. They must not be used for clinical patient planning, surgical guide design, surgical protocol generation, patient-treatment export, or guide manufacturing.

Hint	There are pre-set protocol with appropriate drill kit and sleeves for most of implant lines in the library. It is recommended to download appropriate drill kits and sleeves with implant lines.

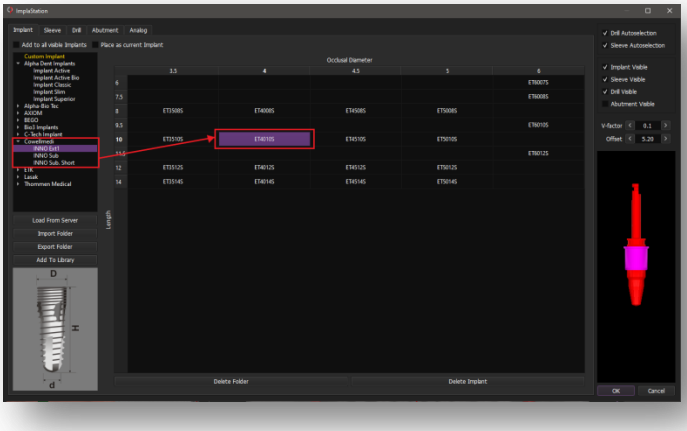
To place preloaded elements

Step 1

Click the **Place new Implant** button or the **Replace Implant** button.

Step 2

In the appeared menu choose the implant manufacturer name, implant line name, and implant size.



Step 3

Enable the **Drill Autoselection** and **Sleeve Autoselection** checkboxes (**7**) to automatically select the optimal master-drill and sleeve for dental implant placement.

When this feature is on, the program will select the original drill and sleeve, if available on the server.

If you choose a non-original drill or sleeve while the checkboxes are enabled, they will automatically be turned off for the next implant. If disabled, the system will continue to use the same drill and sleeve from the previous procedure, keeping all settings, including the V-factor.

Step 4

Click the **Sleeve** submenu, choose the sleeve manufacturer name and sleeve line name.

Step 5

Click the **Drill** submenu, choose the drill manufacturer name, drill line name, and drill size. Set up "V-factor" depth, and then click **OK**.

Step 6

Click the **Abutment** submenu, choose the abutment manufacturer name, abutment line name, and abutment size and type.

Step 7

Click the **Analog** submenu, choose the analog manufacturer name, analog line name, and analog size and type.

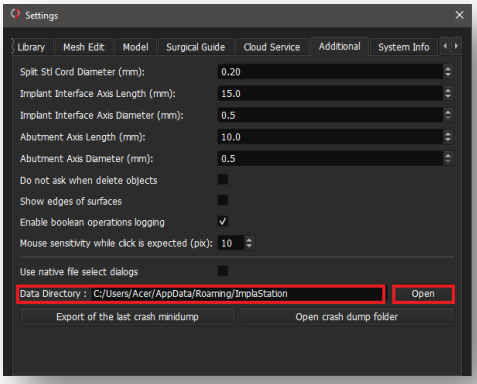
Step 8

Check the implant position and click **Add Drill** to add a final drill to the drilling list.

To delete preloaded elements

Step 1

Move to Settings and navigate to the Additional tab.
Locate the Data Directory field: it displays the path to the local folder where the software stores its data.

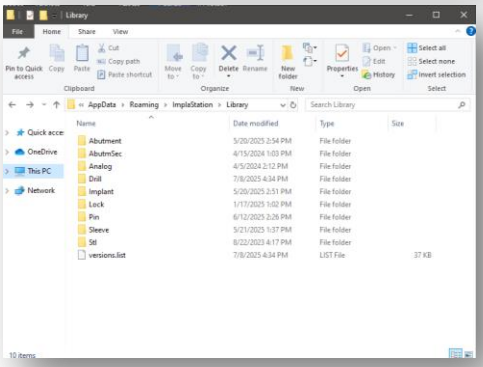


Step 2

Click **Open** button next to the Data Directory field. The software will open the specified folder in the system file explorer.

Step 3

Find and open folder named "Library". This folder contains all previously downloaded element libraries.
Inside the element folder, navigate through subfolders sorted by manufacturer, and inside those – product line folders. Open necessary one.



Step 4

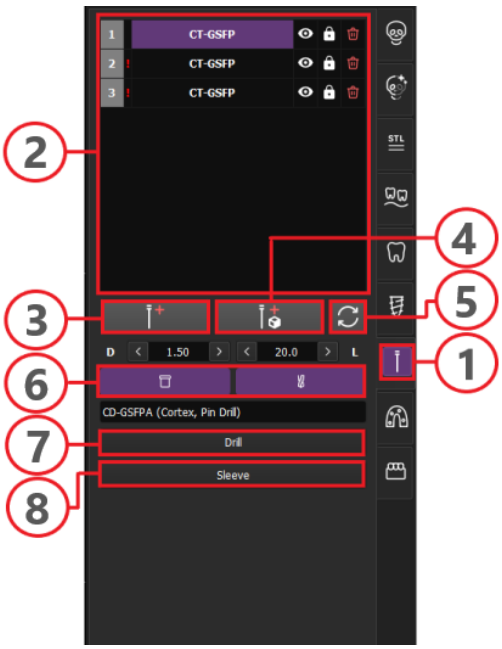
Remove selected element(s): you can delete either necessary element or the entire folder(s).

Note! Ensure the selected items are not in use within an open project before deleting.

10. ANCHOR PIN PLANNING

The pin planning using 3D Exacto has the same concept as an implant positioning.

WORKFLOW PANEL OF PIN MODE

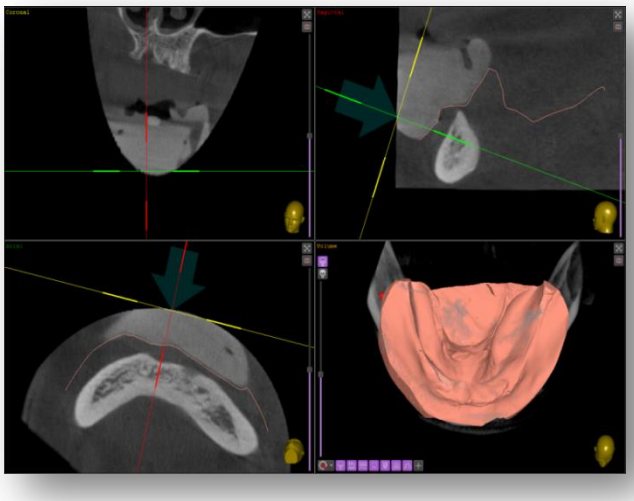


1	Pin tab
2	Pins list
3	“Place new Pin” button
4	“Place new Pin in Volume” button
5	“Replace Pin” button
6	“Visualisation of Sleeve, Drill” buttons
7	“Drill submenu” button
8	“Sleeve submenu” button

10.1 Pin Placement

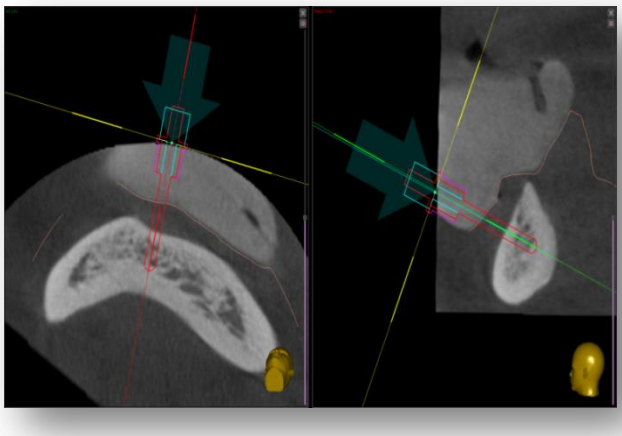
Step 1

Select **MPR** mode. Setup the slice planes to visualize the further pin position to be planning in axial, cross-sectional and sagittal view.



Step 2

To add a virtual pin click **Pin** button in the workflow panel on the right part of the screen. Green transparent arrow will appear in Axial and Sagittal windows in an aim to show the position and direction of the pin.
The pin can also be repositioned and rotated in the Volume window using modifier keys (**see 3.4**)

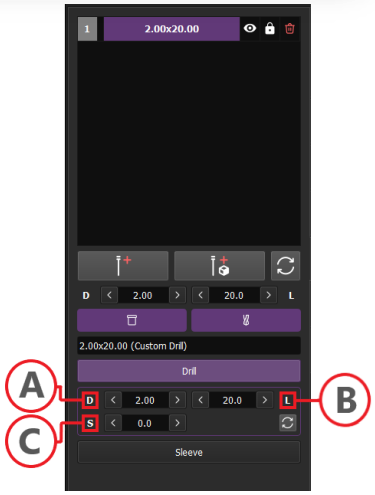


Step 3

Click **Place new Pin** or **Place new Pin in Volume** button and at the appeared window, choose the "pin", "sleeve", and "drill" corresponding submenu by left-clicking.

Step 4

Click the **Drill** submenu in the right part of the screen and set up the drill diameter "D" (**A**), drill length "L" (**B**), the drill spacer "S" as an option to add the regulated space between stopper of the drill and contacted surface of the sleeve to change a position of the sleeve and decrease the sleeve offset during digital pin planning (**C**).

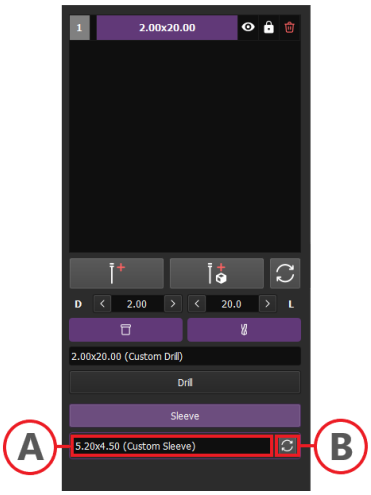


Hint!

There are pre-set protocol with appropriate drill and sleeve for most of pins in the library. It is recommended to download appropriate drill and sleeve with pins.

Step 5

Click the **Sleeve** submenu in the left part of the screen and check the name of the sleeve (**A**). To change the current sleeve, click the **Replace Sleeve** button (**B**).



10.2 OPTIONS OF CHOOSING PIN ELEMENTS

CUSTOM PIN

To place a pin that does not available on the pin library list

Step 1

Select **Custom Implant** option.

Step 2

Choose the pin color and enter relevant dimensions for "Diameter" (d), and Length" (H). To restore settings for the next start of the software, mark appropriate checkbox in the lower right of the submenu window.

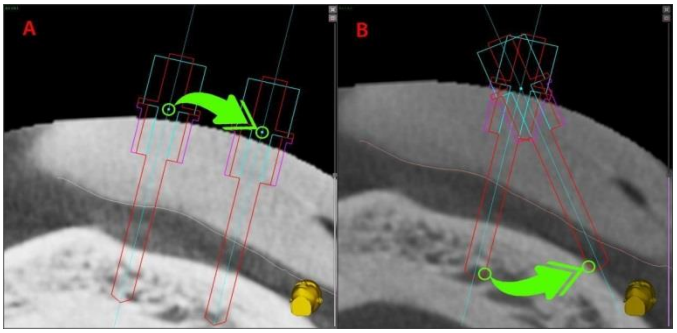


Step 3

The same can be performed for custom sleeves and custom drills (see Implant settings part). Then click the **OK** button.

Step 4

To change the pin position in axial, cross-sectional and sagittal view just left-click on and hold square mark at the stopper-contact surface area of the pin and drag it (**A**), or click on and hold the contour line and tilt it (**B**).



Step 5

Click **Lock On/Off** button in the **Pin** tab panel or go to the next step.

PLACE PRELOADED PINS

To load Pin libraries from the server:

Step 1

Click the **Replace Pin** button and then click the **Load From Implant Server** button.

Step 2

In appeared window select the system of the pin by marking checkbox/es and click **OK** and the chosen library will be downloaded into the software Pin list.

The same downloading procedure can be performed for drills and sleeves libraries.

Now there is a possibility to use the pin, drill, and sleeve provided by a manufacturer.

To place preloaded elements:

Step 1

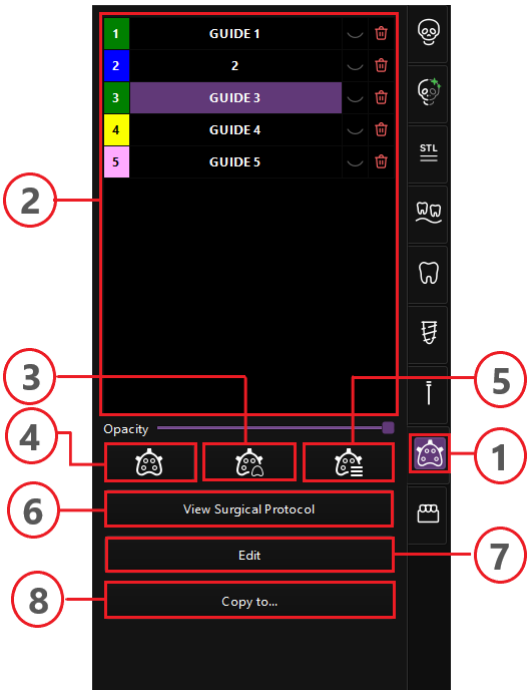
Click the **Place new Pin** button or the **Replace Pin** button.

Step 2

In the appeared menu choose the pin manufacturer name, pin line name, and pin size.

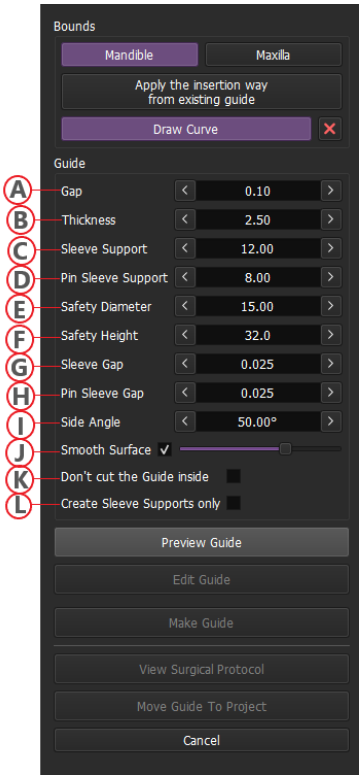
11. SURGICAL GUIDE CREATION

WORKFLOW PANEL OF SURGICAL GUIDE MODE



1. Surgical Guides tab
2. Guides list
3. "Make Surgical Guide based on Prosthesis" button
4. "Make Surgical Guide based on Surface" button
5. "Make Stackable Surgical Guide based on Surface" button
6. "Surgical protocol" button
7. "Edit" button
8. "Copy to ".. button

11.1 SURGICAL GUIDE BASED ON SURFACE



Step 1

To start a virtual surgical guide creation click **Surgical Guides** button in the workflow panel on the right part of the screen. And click **Make Surgical Guide based on Surface** button

Step 2

At the appeared window check the STL surface position – Maxilla or Mandible.

If direction of the surgical guide suggested by the program incorrectly, select the appropriate Surgical Guide orientation - "Maxilla" or "Mandible" manually.

Set up the Surgical Guide settings for printing:

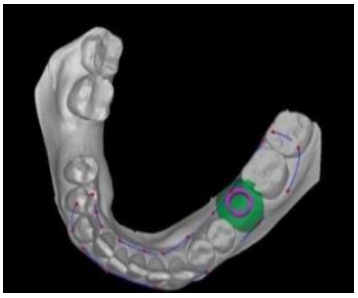
A Gap (mm) between surgical guide and supporting surface (set up an additional compensation value for the expansion or shrinkage of the guide material during the printing process. The settings of this parameter avoid difficulties during the process of placing and

removing the surgical guide, and are necessary to compensate for scanning errors or other aggravating factors.);
B Thickness (mm) (surgical guide wall thickness settings);
C Sleeve Support (mm) (set the diameter of the sleeve support);
D Pin Sleeve Support (mm) (set the diameter of the pin sleeve support);
E Safety Diameter (mm) (set the diameter of the dental handpiece head safety zone around the sleeve);
F Safety Height (mm) (set the height of the dental handpiece head safety zone above and around the sleeve);
G Sleeve Gap (mm) (set the positive radial offset between a sleeve and surgical guide sleeve support);
H Pin sleeve gap (mm) (set the positive radial offset between a pin sleeve and surgical guide);
I Side Angle (degrees) (set the value of the angle of the slope of the guide edge);
J Smooth Surface Slider (set the level of guide smoothing);
K Don`t cut inside checkbox (turn on to avoid cutting inside the guide);
L Create sleeve supports only checkbox (turn on to enable to generate only the sleeve supports guide).

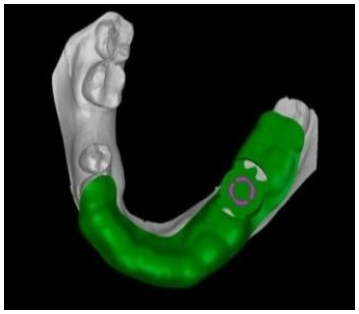
Hint!	Default settings can be changed in the Settings Menu.

Step 3

To identify the area on which the Surgical Guide is going to be supported, draw the guide borderline by placing points one after another around the Surgical Guide area.
Continue to draw the curve that goes back to the starting points and then double-click on this line or click the **Draw Curve** button.



Note!	Pull placed dots by left mouse button for position editing.



Note!	Before generating the guide, set up the guide insertion path! The path of insertion is a perpendicular to the screen! So, place the STL surface in the appropriate position when you see minimum undercuts.

Click the **Preview Guide** button and software will go ahead and create the preview model of the surgical guide.

Step 4 (optional)

If necessary, click the **Edit Guide** button to open the Surgical Guide Edit Menu.

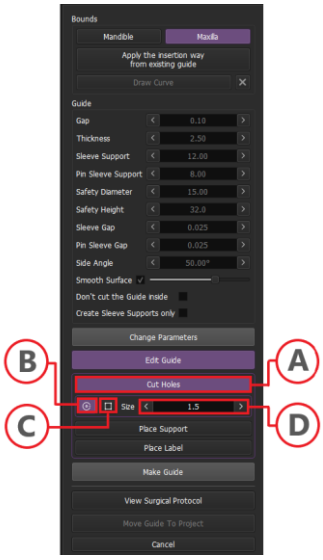
Note!	Editing also available after guide creation finishing.

Step 4.1

Click **Cut Holes** button (**A**).

To cut viewing windows either click on the round window button or click on the square window button, set up the size of the window by scrolling with the mouse wheel (**C-D**).

The cursor is going to be round or square shape cutter, move the cursor over the STL surface of the generated surgical guide then left-click to make a hole.



Step 4.2

Click **Place Support** button (**A**).

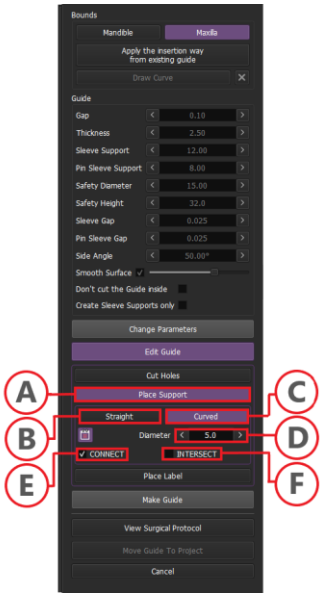
To add a straight/curved support bar from the menu either click on appropriative buttons (**B-C**) or click on the square button, set up the diameter of the support bar (**D**).

Choose one option of constructing a support bar (**E-F**):

CONNECT option is used to construct a bar when it does not fully intersect one or more intermediate parts of the object in its middle section. This option is also suitable for building a support bar when one or both parts of the object have thin walls.

INTERSECT option is used when constructing a bar in a way that its middle section fully intersects one or more intermediate parts of the object.

Place a support bar on the guide surface and modify it by using the points for rotation and size adjustment.



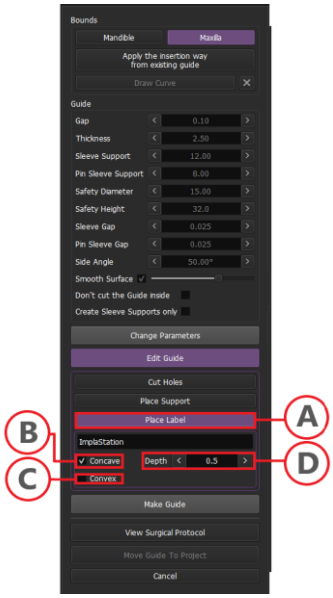
Step 4.3

Select the **Place Label** button (**A**).

Type in the desired text in the label field.

Modify the desired label text depth (**D**) by scrolling with the mouse wheel.

To add a label to surgical guide move cursor over the guide and left-click where you wish to place tag. Click the **Concave** (**B**) or **Convex** (**C**) checkbox to modify the way a Label imprints on the surgical guide surface.-



Step 5

Check the surgical protocol by clicking the **Surgical Protocol** button in the Surgical Guide menu on the right part of the screen.

Step 6

Check Surgical guide settings and modifications then click **Make Guide** button to achieve the final view of the generated guide.

Step 7

To move created Surgical Guide to the project press **Move Guide to Project**, then in the appeared window, click **Yes**.
Click **Export** button to save guide as STL-file on your computer.
To get file of the Surgical Protocol saved or printed, click **View Surgical Protocol** and chose appropriate option.

Note!	Paid option. One export credit will be charged from your account.

11.2 Surgical Guide Based on Prosthesis

Step 1

To start the creation of the surgical guide based on the denture click the **Surgical Guides** button in the workflow panel on the right part of the screen. Click on the **Make Surgical Guide based on Prosthesis** button.

Step 2

At the appeared window setup the STL surface position.

- Select the Surgical Guide orientation - **Maxilla** or **Mandible**.
- Set up the Surgical Guide settings for STL printing.

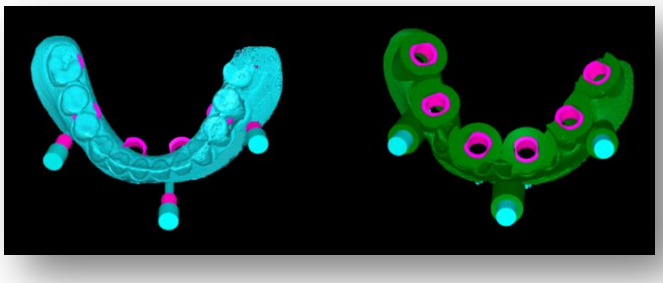
Note!	The pass of insertion is doesn't matter for guides created from dentures.

Step 3

Click the **Generate Guide** button and software will go ahead and create the surgical guide.

Step 4

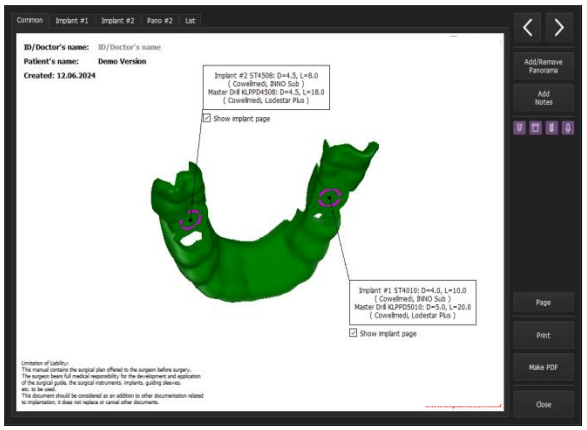
To move created Surgical Guide to the project click **Move Guide to Project**, then in the appeared window, click **Yes**.
Click on **Export** button to save guide as STL-file on your computer.
To get file of the Surgical Protocol saved or printed, click **View Surgical Protocol** and chose appropriate option.



Note!	Paid option. One export credit will be charged from your account.

SURGICAL PROTOCOL

The 3D Exacto creates a surgical protocol together with the surgical guide based on virtual implant planning, sleeve(s) position and selected surgical drill sequence.
Chose appropriate guide in the list of surgical guides. Click the **View Surgical Protocol** button in the right part of the screen to open the Surgical Protocol Window.
The following image shows an example of a surgical protocol. Positions of guide and cards with implant info are adjustable by pulling them with left mouse button.
The surgical protocol is available per implant providing detailed information together with the images of the planning view. Adjust the guide position as you need. To get more info select the option **Show implant page**.
To add a panoramic image to the Protocol select the option **Add to surgical protocol**.
To get file of the Surgical Protocol saved or printed, click **Make PDF** button and save it.

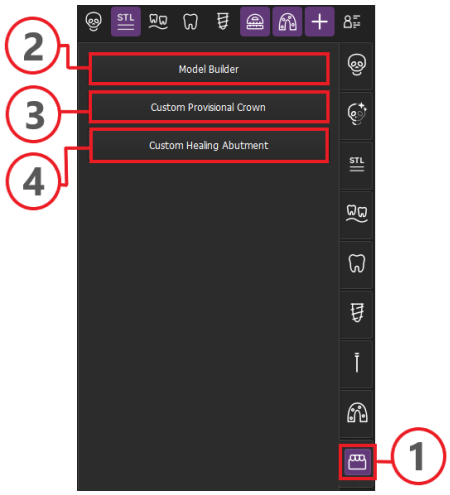




12. Modeling

The model design feature within the 3D Exacto software is instrumental in creating models of objects within the field of dental implantation. It plays a crucial role in optimizing the dental implantation process and comprises the following functional tools: the Model Builder, the Custom Provisional Crown, and the Custom Healing Abutment.

WORKFLOW PANEL OF MODELING MODE



1 Modeling tab

2 Model Builder button

3 Custom Provisional Crown button

4 Custom Healing Abutment button

12.1 MODEL BUILDER MODE



1 Edit button

2 Mandible or Maxilla buttons

3 Add Base button

4 Base boundaries button

5 Gridsize input

6 Inverted checkbox

7 Gingival mask submenu

8 Hollow model submenu

9 Add Analogs submenu

10 Suprastructures Cut submenu

11 Make model button

12 Move to the Project button

Step 1

Select the desired STL file.

Note!	Model Builder exclusively operates with unclosed surfaces.

Step 2

Click the **Modeling** section within the workflow panel.

Step 3

Activate the STL editor by clicking the **Edit (1)** button. If necessary, prepare and correct any defects in the STL surface for modeling.

Step 4

The **Mandible or Maxilla (2)** buttons are automatically highlighted based on the software's identification of the jaw. You can manually switch them if needed.

Step 5

Rotate the STL surface in the 3D window with respect to the base part's boundaries, represented by the horizontal purple lines:

- Move the cursor over the STL surface while holding the left mouse button to perform arbitrary 3D rotation of the STL.
- Moving the cursor outside the STL surface while holding down the left mouse button allows you to rotate the STL around the vertical axis.
- Use the scrolling wheel anywhere in the rendering window to rotate the STL around an axis perpendicular to the screen plane.

Step 6

Position the Model Base:

- Drag the green circle. Between the purple lines while holding the left mouse button to position the base part relative to the STL surface.
- Dragging the purple lines (this is in addition to dragging the green circle) allows you to change the height of the base part of the model's base.

Step 7

Adjust the size of the base triangles by filling in the Grid size **(5)** in the digital window next to the Add Base button.

Step 8

Click the **Add Base (3)** button to create the model's base.
If necessary, you can reverse the orientation of all triangles on a surface by clicking the Invert button.

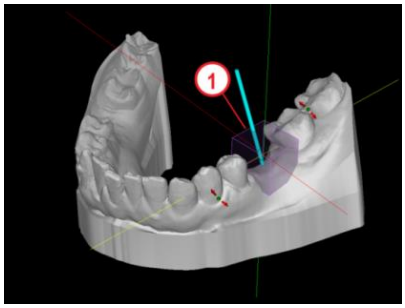
Note!	Clicking the Add Base button after correcting the STL position and base boundaries allows you to reshape the model base.

Step 9

Once the model is created, the model design buttons will become available:

Gingival Mask:

Clicking the **Gingival Mask** button generates separate STL files for gingival masks, represented as purple boxes in the 3D rendering window (**1**). Select the box's position relative to the STL surface by checking the relevant checkbox. You can manually resize the box's sides by dragging and adjusting the box's boundaries in the 3D rendering window or by entering the size value in the digital windows. Click the **Add New Mask** button to create additional gingival masks.



Hollow Model:

Clicking the **Hollow Model** button creates internal voids in the model with specified wall thickness parameters and optional internal surface smoothing.

Add Analogs:

Click the **Add Analogs** button to add support for analog implants. You can fill in adjustable parameters on the control panel, such as Diameter, Bottom, and Gap.

Suprastructures:

To remove the model volume from the simplified gingival mask, click the **Suprastructures Cuts** button. To view the cut teeth area, select the Cut teeth option.

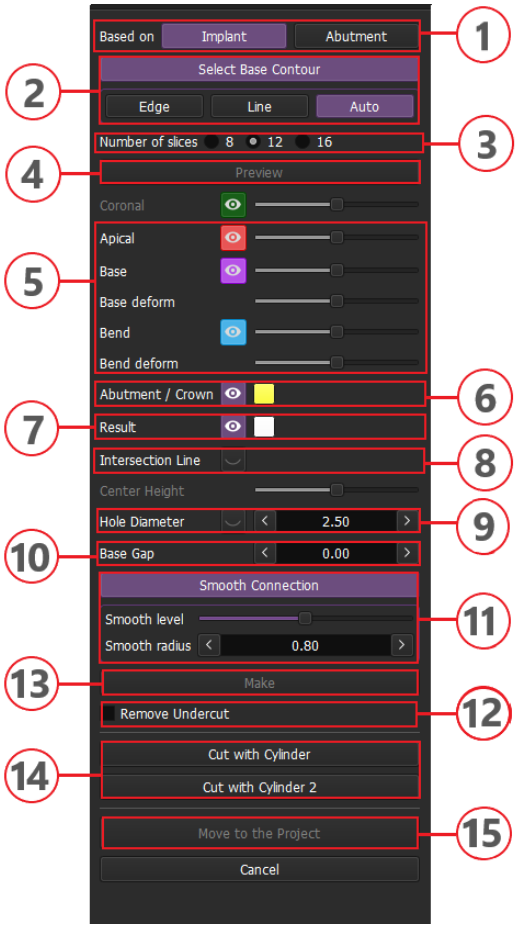
Step 10

Finalize the model creation by clicking the **Make Model (11)** button.

Step 11

To transfer the model file and gingival mask files (if available) to the STL list, click the **Move to the Project (12)** button.

12.2 Custom Provisional Crown Modeling



1 "Based on" controls
2 "Base Contour" tools
3 Number of slices
4 "Preview" button
5 Shape contours control sliders
6 "Abutment/Crown visualisation" button
7 "Result visualisation" button
8 "Intersection line visualisation" button
9 Hole Diameter setting
10 Base Gap setting
11 Smooth Connection submenu
12 "Remove Undercut" checkbox
13 "Make" button
14 " Cut with Cylinder" buttons
15 "Move to the Project" button

Step 1

Choose the necessary elements:

- relevant STL surface.
- implant.
- suprastructure to be used in conjunction with the chosen implant.
- crown or tooth.

Note!	When opting for a tooth-shaped crown, it must intersect with the selected STL surface. In contrast, if you select an unclosed crown, it doesn't necessarily need to intersect with the STL.

Step 2

Click the **Custom Provisional Crown** button in the Modeling section.

Step 3

Begin by selecting the location where the provisional crown will be based, either at the Abutment or Implant levels **(1)**.

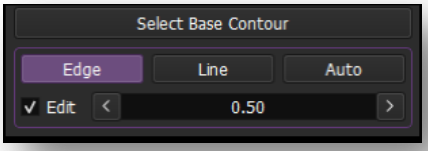
Step 4

Choose the **Base Contour (2)** using the necessary instrument for shaping the border: **Edge**, **Line**, or **Auto**.

- The “Edge” tool: The Base Contour will be automatically generated based on the nearest surface edge to the set point. To set the point, left-click at the desired location, and the base contour will be created. It will be displayed as a red line. Double-click the left mouse button to lock the line.
- The “Line” tool: This tool allows manual drawing of the base contour, which will also appear as a red line. Set contour points by left-clicking at the necessary positions in the rendering window. Right-click can be used to delete points. Complete the contour drawing by double-clicking the left mouse button.
- The “Auto” tool is designed for the automatic placement of the base contour at any specified point location. To set the point, left-click at the necessary location and double-click to lock the base contour.

Step 5

To adjust the contour, check the **Edit** checkbox. The line will be represented by a set of points that can be moved using the left mouse button or deleted using the right mouse button. The density of points on the line can be changed in the digital window.



Step 6

Choose the **Number of slices (3)** by selecting one of the available positions. Slices represent meridian segments used for editing individual parts. The default value is 12. Increasing this value allows for more precise editing.

Step 7

Click the **Preview (4)** button to create a prototype of the future custom part for further editing. It will be visualized as Meridian slices, divided into three parts:

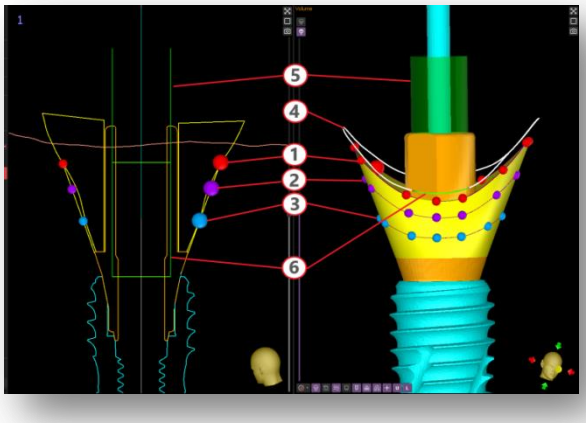
- Apical
- Base
- Bend.

Each part forms the corresponding contour with specific color-coded points. The Apical contour follows the crown surface, while the Base and Bend contour points are positioned in intermediate locations between the Apical contour and the boundary line.

Step 8

Edit contours of the prototype of the model by moving and changing the curvature of the contours.

The Apical, Base, or Bend contour can be moved in two ways: by adjusting individual points or by modifying the entire contour as a whole.



- Apical Individual contour adjustment: To change the contour individually, move individual points using the left mouse button in the rendering window or in rotating slice window (**1**).
- Contour adjustment as a whole: Use the respective control in the control panel to modify the entire contour (**5**).

Changing the curvature of the Base and Bend contours can also be done both collectively and individually.

- Perform individual editing of the curvature of Base and Bend contours in the rendering window by scrolling the mouse while holding the right mouse button or by dragging points with the left mouse button pressed in the rotation slice window (**2, 3**).
- Group editing of Base and Bend contour curvature: You can collectively edit the curvature of Base and Bend contours using the corresponding "Base Deform" and "Bend Deform" controls on the control panel (**5**).

Use the specified buttons on the control panel to toggle specific elements on or off, such as:

- Apical, Base, or Bend contours **(5)**

- Crown/Abutment **(6)**
- Result (the intermediate model result before project export) **(7)**
- Intersection line (the line where the crown intersects with the STL surface) **(8)**
- Hole diameter (the screw head shaft) **(9)**

Hint!	It is recommended to edit the Apical contour first and then the Base and Bend contours to avoid losing the editing results of the latter when attempting to edit the Apical contour.

Hint!	To determine the part of the edit you're working on, pay attention to the size of the dots. They will be larger in the active slide.

Step 9

Set:

- the diameter value of the screw head shaft In the **Hole Diameter** digital window **(9)**.
- the **Base Gap size** to determines the gap size between the cylindrical portion of the superstructure's base and the crown **(10).**

Step 10

Adjust the **Smooth connection (11)** by scrolling the mouse wheel or clicking and dragging the slider on the control panel, and set the Smooth radius value

The Smooth Connection button is used to smoothen the transition between the apical and coronal surfaces.

Step 11

Enable the **Remove Undercut (12)** option to eliminate any undercuts found in the cylindrical section of the superstructure.

Step 12

Activate the **Cut with Cylinder/Cut with Cylinder 2 (14)** buttons to create and cut additional objects of different shapes, which will be subtracted from the crown in the final step. Adjust the desired parameters by either filling in the digital windows or using the scroll**.**

Step 13

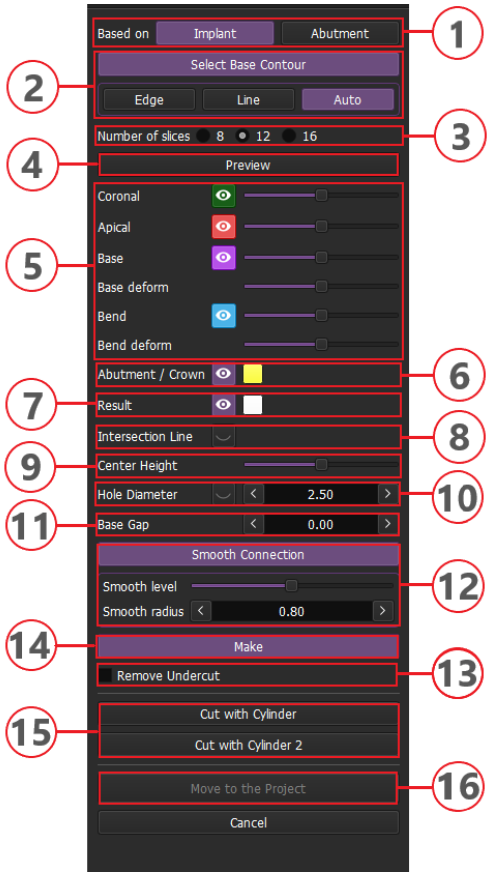
Complete the modeling creation by clicking the **Make (13)** button to finalize the model creation.

Hint!	After creating the model, it is possible to make additional modifications; use the 'Make' button again to refresh the results.

Step 14

Click the **Move to the Project (15)** button to transfer the model file to the project.

12.3 Custom Healing Abutment Modeling



1. "Based on" controls
2. "Base Contour" tools
3. Number of slices
4. "Preview" button
5. Shape contours control sliders
6. Abutment/Crown visualisation button
7. "Result visualisation" button
8. "Intersection line visualisation" button
9. Center Height control slider
10. Hole Diameter setting
11. Base Gap setting
12. Smooth Connection submenu
13. "Remove Undercut" checkbox
14. "Make" button
15. "Cut with Cylinder" buttons
16. "Move to the Project" button

Step 1

Choose the necessary elements:

- relevant STL surface;
- Implant;
- suprastructure to be used in conjunction with the chosen implant;
- crown or tooth.

Note!	When opting for a tooth-shaped crown, it must intersect with the selected STL surface. In contrast, if you select a unclosed crown, it doesn't necessarily need to intersect with the STL.

Step 2

Click the **Custom Provisional Crown** button in the Modeling section.

Step 3

Begin by selecting the location where the provisional crown will be based, either at the Abutment or Implant levels **(1)**.

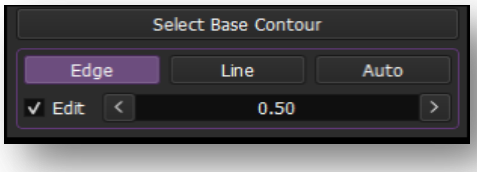
Step 4

Choose the **Base Contour (2)** using the necessary instrument for shaping the border: **Edge**, **Line**, or **Auto**.

- The “Edge” tool: The Base Contour will be automatically generated based on the nearest surface edge to the set point. To set the point, left-click at the desired location, and the base contour will be created. It will be displayed as a red line. Double-click the left mouse button to lock the line.
- The “Line” tool: This tool allows manual drawing of the base contour, which will also appear as a red line. Set contour points by left-clicking at the necessary positions in the rendering window. Right-click can be used to delete points. Complete the contour drawing by double-clicking the left mouse button.
- The “Auto” tool is designed for the automatic placement of the base contour at any specified point location. To set the point, left-click at the necessary location and double-click to lock the base contour.

Step 5

To adjust the contour, check the **Edit** checkbox. The line will be represented by a set of points that can be moved using the left mouse button or deleted using the right mouse button. The density of points on the line can be changed in the digital window.



Step 6

Choose the **Number of slices (3)** by selecting one of the available positions. Slices represent meridian segments used for editing individual parts. The default value is 12. Increasing this value allows for more precise editing.

Step 7

Click the **Preview (4)** button to create a prototype of the future custom part for further editing. This prototype consists of meridian slices, divided into four parts:

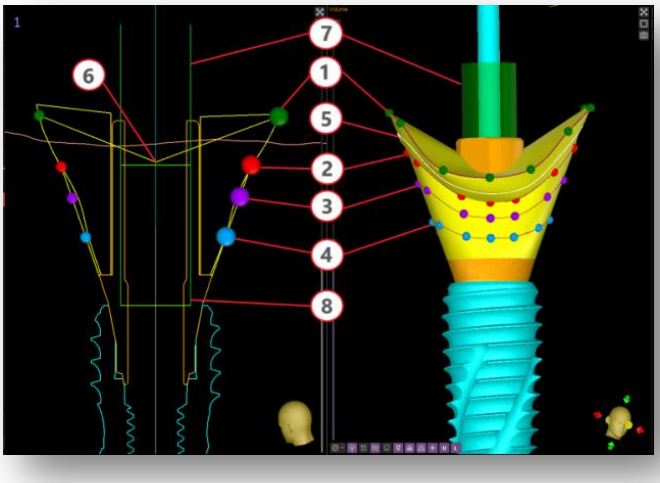
- Coronal
- Apical
- Base
- Bend

Each part corresponds to a specific contour with colored reference points.
The coronal contour, similar to the apical contour, moves along the crown surface

Step 8

Edit contours of the prototype of the model by moving and changing the curvature of the contours.

- The Coronal, Apical, Base, or Bend contours can be modified in two ways: by adjusting individual points or by modifying the entire contour collectively.
    1. Coronal and Apical Individual Contour Adjustment: to individually change the contour, move individual points using the left mouse button in the rendering window or the rotating slice window (**1**,**2**).



Hint!	It is recommended to edit the Apical contour first and then the Base and Bend contours to avoid losing the editing results of the latter when attempting to edit the Apical contour.

2. Contour Adjustment as a Whole: You can modify the entire contour by using the respective controls in the control panel (**5**).

- Changing the curvature of the Base and Bend contours can also be performed both collectively and individually.

Individual Editing of Base and Bend Contours Curvature: This can be done by scrolling the mouse while holding the right mouse button (**3, 4**).

Group Editing of Base and Bend Contours Curvature: Group editing is possible using the corresponding "Base Deform" and "Bend Deform" controls on the control panel or by dragging points with the left mouse button pressed in the rotation slice window **(4)**.

Hint!	To determine the part of the edit you're working on, pay attention to the size of the dots. They will be larger in the active slide.

Use the specified buttons on the control panel to toggle specific elements on or off, such as:

- Coronal, Apical, Base, or Bend contours **(5)**
- Crown/Abutment **(6)**
- Result (the intermediate model result before project export) **(7)**
- Intersection line (the line where the crown intersects with the STL surface) **(8)**
- Hole diameter (the screw head shaft) **(10)**

Step 9

Set:

- diameter value of the screw head shaft In the **Hole Diameter** digital window **(10)**
- The **Base Gap size** to determine the gap size between the cylindrical portion of the superstructure's base and the crown **(11)**

Step 10

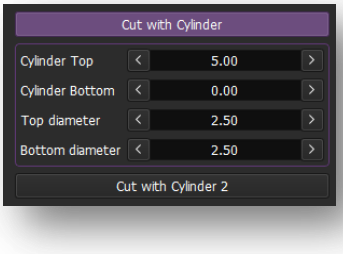
Adjust the **Smooth connection (12)** by scrolling the mouse wheel or clicking and dragging the slider on the control panel, and set the Smooth radius value.
The Smooth Connection button is used to smoothen the transition between the apical and coronal surfaces.**)**.

Step 11

Enable the **Remove Undercut (13)** option to eliminate any undercuts found in the cylindrical section of the superstructure.

Step 12

Activate the **Cut with Cylinder/Cut with Cylinder 2 (15)** buttons to create and cut additional objects of different shapes, which will be subtracted from the crown in the final step. Adjust the desired parameters by either filling in the digital windows or using the scroll**.**



Step 13

Complete the modeling creation by clicking the **Make (14)** button to finalize the model creation.

Hint!	After creating the model, it is possible to make additional modifications; use the “Make” button again to refresh the results.

Step 14

Click the **Move to the Project (16)** button to transfer the model file to the project.

SHELF LIFE AND SOFTWARE SUPPORT

SHELF LIFE

3D Exacto has an unlimited shelf life under standard storage and usage conditions.

SOFTWARE SUPPORT

Each released version of 3D Exacto is supported by the manufacturer for a minimum of 5 years from its release date. Support includes technical assistance and documentation management.

OBSOLESCENCE MANAGEMENT

Users are notified in advance before the end of support for a particular version. To ensure optimal safety, performance, and cybersecurity, users are advised to upgrade to the latest supported version.

WARNINGS AND PRECAUTIONS

There are no chemical, physical, electrical, mechanical, electromagnetic and biological hazards to the 3D Exacto software.
However, there are several warning functions designed to remind the user of his legal responsibility to verify implant planning stages and results.

INDICATIONS FOR USE

Warning!	The 3D Exacto software must be used in accordance with this Instruction for Use.
Warning!	This software is not tested on the pediatric patient population.
Warning!	Prior to working with software, please make sure to have received appropriate training and instructions in software operation. Prodigident offers regular online webinars for 3D Exacto which is open for all users.
Warning!	Correct design of the surgical guide lies within the sole responsibility of the user.
Warning!	3D manufacturing is out of 3D Exacto software control, depends on many external factors and lie within the sole responsibility of the user.
Warning!	Manufactured surgical guides for implant placement are classified as medical devices in the USA by the FDA (under 21 CFR 892.2050), in the European Union (under the Medical Device Regulation (MDR) 2017/745), and in Australia (under the Therapeutic Goods (Medical Devices) Regulations 2002). Surgical guides are subject to legal requirements such as registration and listing as a manufacturer of medical devices, validation of production equipment, processes, compliance with ISO 13485.
Warning!	In the US. 3D printed surgical guide is a medical device to be manufactured at an FDA registered and listed manufacturing location.
Warning!	The accuracy of treatment planning depends on the quality of the input imaging and scan data. The user must verify that CBCT/DICOM and STL datasets are of sufficient diagnostic quality before use.
Warning!	The software is not intended to determine patient eligibility for implant therapy. The final decision on patient suitability for implant surgery remains solely with the treating clinician, based on medical conditions, contraindications, and clinical judgment.

SYSTEM REQUIREMENTS

Warning!	3D Exacto must be installed only on systems meeting the minimum technical requirements specified by Prodigident Inc. Use on unsupported operating systems or hardware may lead to software instability, data loss, or inaccurate performance.

Warning!	Prodigident releases software updates to correct known issues and improve safety. Users are responsible for ensuring 3D Exacto is kept up to date. Failure to install updates may result in the use of outdated algorithms or libraries.
Warning!	To avoid performance issues during complex rendering, ensure that your system meets the recommended hardware specifications. System lag or freezing may occur if used on underpowered or unsupported systems. Always save progress frequently.
Warning!	3D Exacto must be used with a monitor that supports at least 1920×1080 resolution and accurate grayscale rendering. Use of low-resolution or low-quality displays may impair visualization of critical anatomical structures and affect planning accuracy.

MAINTENANCE

Warning!	Prodigident validates each release of 3D Exacto through structured testing. Users are responsible for verifying that the updated version is appropriate for their clinical environment before use. Refer to the release notes for version-specific updates.

DATA SECURITY

Warning!	Ensure that patient data is protected in accordance with applicable privacy regulations. 3D Exacto stores patient data locally. The user is responsible for securing the PC against unauthorized access and ensuring safe data transmission when sharing files.
Warning!	To avoid data loss due to malware or system failure, users must ensure up-to-date antivirus software is installed on the planning PC and regular backups of patient data are performed to secure, external storage. Prodigident Inc. is not responsible for data loss resulting from compromised or unprotected local systems.
Warning!	3D Exacto includes a crash recovery feature for restoring open planning sessions. However, users are responsible for backing up completed case data, exported STL files, and reports. Prodigident is not liable for data loss due to lack of user-initiated backups.
Warning!	Please ensure that finalized planning data and exported surgical guides are backed up to prevent accidental deletion or data loss. Deleted cases cannot be recovered unless backed up by the user.

DICOM

Warning!	The production of CT/CBCT scans lies within the full responsibility of the clinicians or appropriately qualified personnel. The CT/CBCT scanner should be maintained within original manufacturer specifications.

NERVE CANAL

Warning!	Make sure that the nerve is correctly traced. Always maintain an appropriate safety distance to the nerve canal.
Warning!	The pathway of imaged nerves is for display only, location accuracy of the traced nerve is not tested, and pathways of imaged nerves cannot be used as sole information for the clinician to make clinical decisions.

IMPLANT PLACEMENT

Warning!	During implant placement, please assure that an implant/pin does not collide with an existing implant/pin, tooth root(s) or nerve canal. A collision of the implant/pin with another implant, nerve, or any other main anatomical structure can cause severe damage.
Warning!	The user must be able to recognize the triangular sign "Attention" to detect warnings such as “Collision between implant/pin and implant/pin”, "Collision between sleeve and scan STL surface", and "Collision between implant/pin and nerve canal".
Warning!	Reverse-engineered, non-original, or testing implant/drill/sleeve libraries must not be used for clinical patient planning, surgical guide design, surgical protocol generation, patient-treatment export, or guide manufacturing.
Warning!	Before exporting the surgical guide, the clinician must verify that the implant plan is complete, accurate, and appropriate for the patient. 3D Exacto provides guidance tools but does not substitute for professional judgment.
Warning!	Only verified implant, drill, sleeve, abutment, analog, and pin libraries provided by Prodigident’s official partners or otherwise approved by Prodigident must be used for clinical patient planning and surgical guide design. Use of unapproved, altered, reverse-engineered, or testing libraries may result in inaccurate planning outputs or guide design and is outside the intended clinical use of 3D Exacto .

SURGICAL GUIDE

Warning!	Make sure that created STL file of the surgical guide or produced surgical guide is intended to be used only by trained qualified dental practitioners.
Warning!	Ensure the exported STL file is reviewed and validated using approved 3D printing software before guide production. Corrupted or altered files may lead to inaccurate guide fabrication and surgical complications.
Warning!	Ensure that the guide sleeve used in clinical practice matches the specified dimensions in the 3D Exacto surgical guide design. Use only verified sleeves from approved suppliers. Inaccurate sleeve matching may result in drill deviation and surgical error.
Warning!	Ensure you confirm the export file path and naming before exiting the export workflow. Failure to locate exported files may lead to treatment delays or fabrication errors.

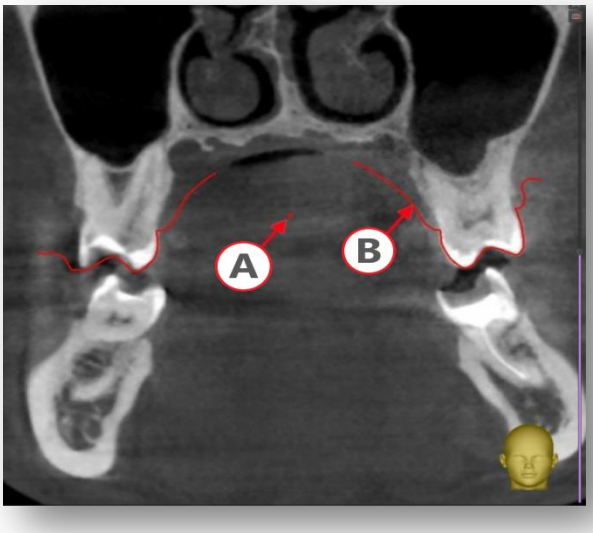
ALIGNMENT

Warning! Make sure that CBCT/CT and STL surface scans are well aligned in all relevant areas, particularly in the implant placement area.

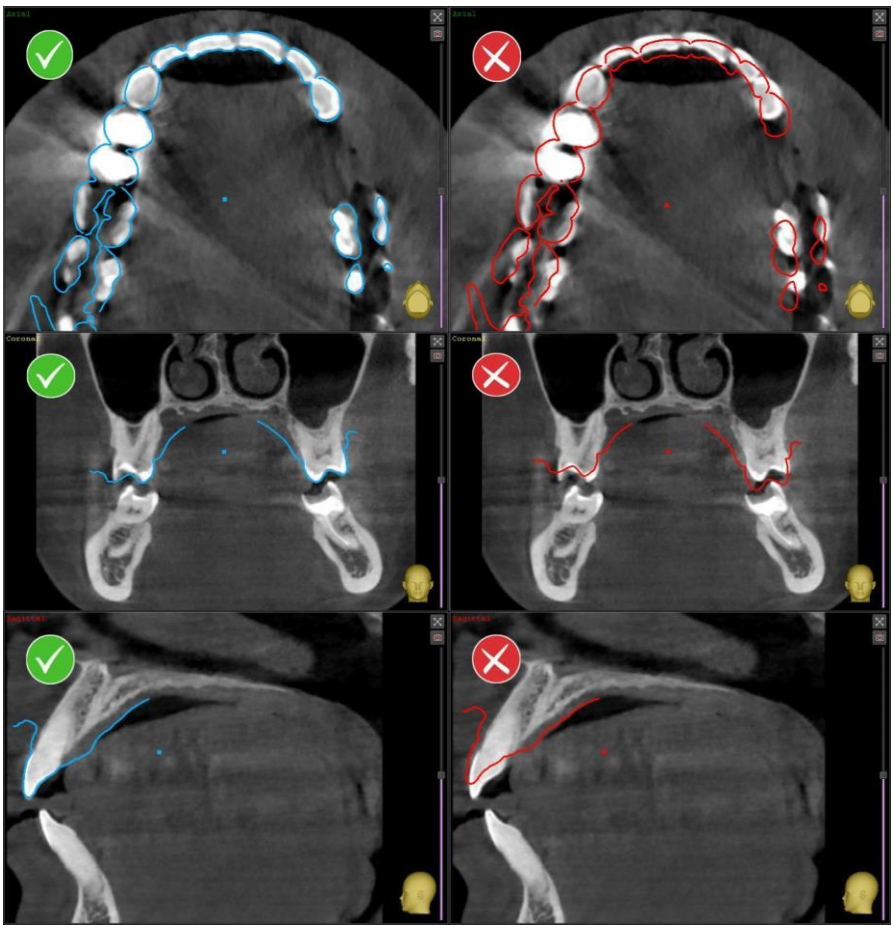
ANNEX A

How to evaluate the quality and accuracy of alignment?

The software enables users to overlay and align DICOM visualization to the STL surface by picking points on both surfaces as a landmark.
There is a tool to correct an alignment manually in MPR or Panoramic mode. Dragging the scan with the square point (**A**) or rotating the scan using the control point on contour line (**B**) of the scan provides your movement to one dimension in Axial, Coronal, and Sagittal views (see picture).
The aim is to get the contour of the STL surface scan to coincide with the corresponding object on the CT scan. Teeth are a good landmark for alignment.



Examples of the alignment results



ANNEX B

Dual Scan Technique

Full Digital Workflow for the Treatment Planning of an Edentulous Patient with Guided Surgery using Dual CBCT/CT Scanning Technique.

Hint!

Definitions:
Dual Scan Technique (DST). The dual scan (dual CBCT scanning technique) is the term used when a dental appliance, such as a set of dentures, is superimposed over the patient CBCT/CT scan.
Scan Appliance is a denture with temporary radiopaque markers that are applied directly to the inside and outside of the denture.
DST Patient CBCT/CT Scan is a head CBCT/CT scan of the patient wearing the denture prepared with temporary radiopaque markers. The patient's dentures should be in occlusion when the scan is taken.

3D EXACTO DUAL SCAN TECHNIQUE WORKFLOW

Step 1

Import an existing project (see section 4.6.2 - Tool Panel) or create a new project.

Step 2

Import Patient's CBCT/CT scan data:

Click the **Start New Project** button jn the Menu bar to upload the patient's DICOM data (see Chapter 5. Input Data (DICOM)).

Step 3

Import Scan Appliance DICOM data:

Click the **STL Surfaces** button in the tab panel on the right part of the screen.

Click on 

button and select the Scan Appliance DICOM file on your computer.
Click **Open** and wait for it to finish loading.

Note!

To use the Dual Scan Technique, you need to import DICOM data of the Scan Appliance(s) with radiopaque markers.

Step 4

Convert the dental appliance to STL

Converted STL is shown in STL list and named "Converted from DICOM".

Hint!

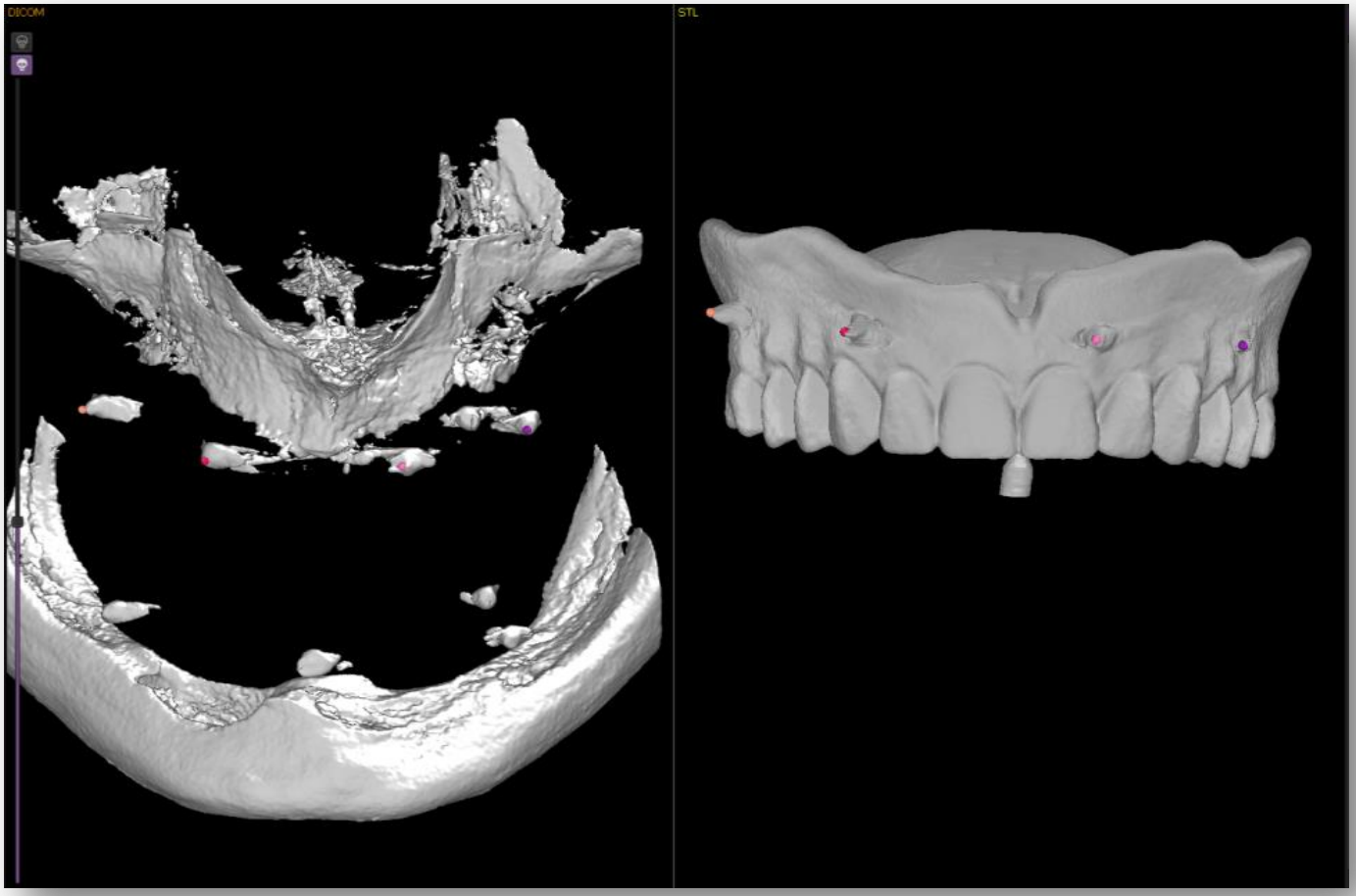
To convert Dental Appliance scan data, follow the instruction described in Chapter 5.2 "DICOM to STL Conversion" of this User Manual.

Step 5

Align converted STL:

Click **Align** button on STL workflow panel.

Place points on the radiopaque markers scan in the left 3D pane and corresponding points on the Surface Scan in the right 3D pane. Then click the **Align Surface to selected object** button.



CREATE A SOFT TISSUE SURFACE (SPLIT AND INVERT)

Once the CBCT/CT scans are aligned, you can proceed to the soft tissue surface detection step.

Step 1

Open the STL tab and choose necessary STL.

Step 2

Click the **Edit** button, then open the **SELECT** tool.

Step 3

Click **Line** then click **Dotted** or **Projected**.

Set up the 3D object position, draw the borderline by placing points one after another around the area of interest.

Step 4

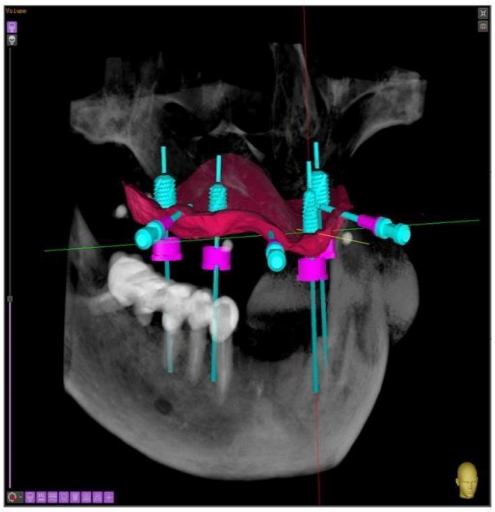
Click **Split** to split an object into two objects with open boundaries along the selection line.

Step 5

Then invert the object you want to use as a guide planning surface (see **Annex G – 1.1.4 Invert**).

PERFORM IMPLANT PLANNING

Once the soft tissue surface is created, you can proceed to the Implant planning, follow the instruction described in Chapter 9 – "IMPLANT PLANNING" of this User Manual.

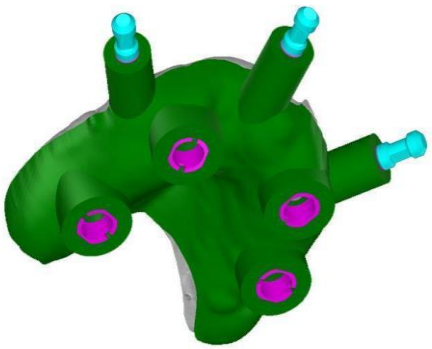


CREATE THE SURGICAL GUIDE

Follow the instruction described in Chapter 11. SURGICAL GUIDE CREATION.

CREATE THE SURGICAL GUIDE BASED ON PROSTHESIS

Optionally you can proceed to the surgical guide creation exactly after the CBCT/CT scans alignment step. (see Section 11.1 Surgical Guide Based on Prosthesis).



ANNEX C

How to anonymize your project file

Before sending, sharing, or saving a project file using the **Cloud Service**, the user should anonymize the project file unless transfer of identifiable patient information is necessary, permitted by applicable law, and authorized by the user's organization. The user must verify that the project file name, patient name, patient ID, description field, screenshots, reports, and exported files do not contain unnecessary patient identifiers before upload or sharing.

Step 1

Click the **Save** button located on tool panel.

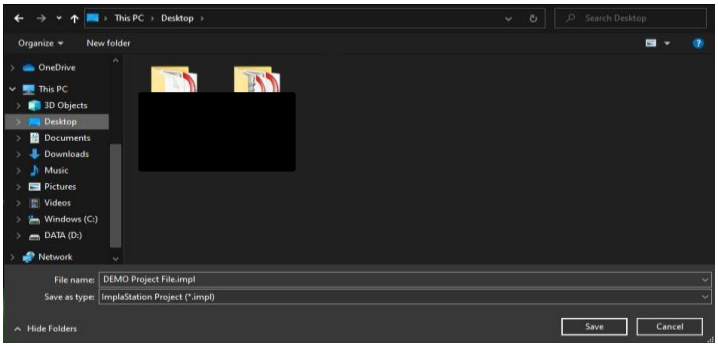
Step 2

Under the Save File submenu, decide on your save location.

Step 3

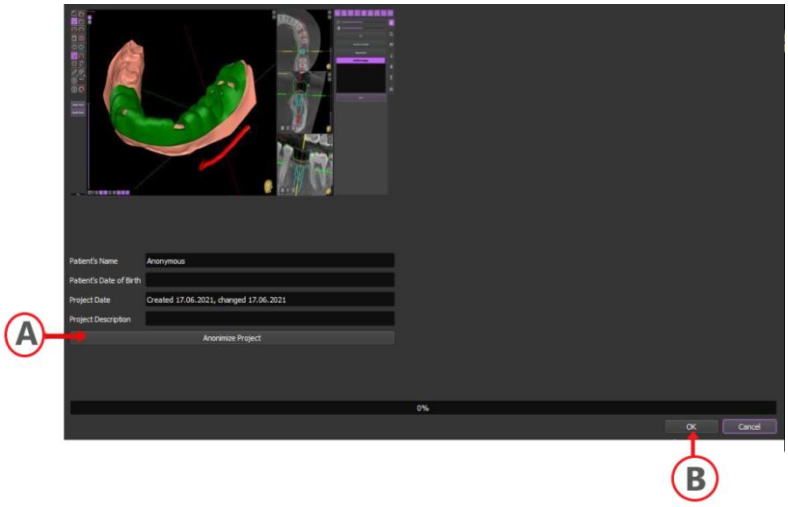
In the **File Name** field, type in your preferred Project file name (do not use patient name in project file name).

Click **Save** to save your file.



Step 4

Under the next "Save Project" submenu, click **Anonymize Project** (**A**) .
Click **OK** (**B**).



Step 5

Before uploading, sending, or saving the project file to the Cloud Service, **review** the anonymized project information and confirm that unnecessary patient identifiers have been removed.

ANNEX D
Cloud Service

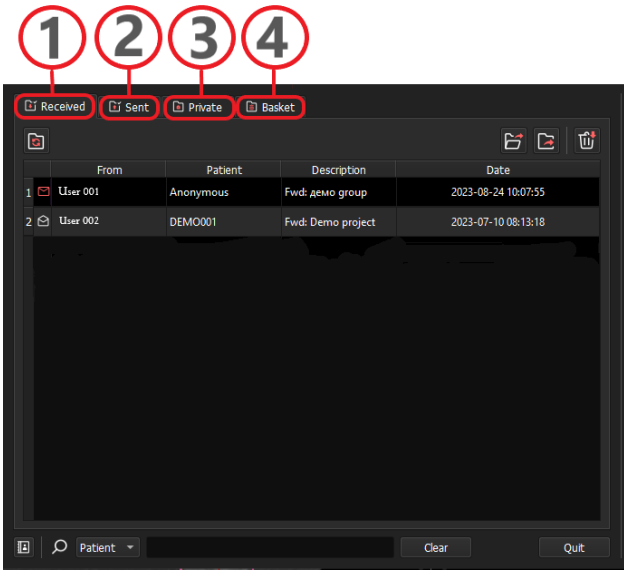
To open a project stored in the **cloud**, do one of the following:

- Click **Open Cloud Project** in the Main Menu of Menu bar.
- Click the **Envelope icon** in Project & Account control panel. The icon displays a notification badge for received, unread projects. Clicking it also opens the cloud menu.

The **Cloud Service** is an optional user-initiated function for project transfer, sharing, or retrieval. It is not required for local clinical planning. Patient/project data are stored locally by default unless the user actively chooses to send, open, or save a project using the Cloud Service.

Warning!	Before using the Cloud Service to send, open, or save project files, anonymize the project file according to Annex C whenever possible. Do not upload unnecessary patient identifiers unless identifiable transfer is necessary, permitted by applicable law, and authorized by your organization.

CLOUD SERVICE TABS



1 **Received** tab. Double-click a received project file to open the project or select received project file

2 **Sent** tab. Click a “sent” project file to open the project, you sent before.

3 **Private** tab. Click an “Anonymous” project file to open the anonymous project.

4 **Basket** tab. Deleting project files from your inbox doesn't permanently delete them immediately - they'll move to the "Basket" folder where they'll remain for 90 days before being deleted automatically.

BUTTONS

	Refresh
	Open project
	Forward project file
	Delete file
	Restore deleted file
	Contacts
	Add new contact

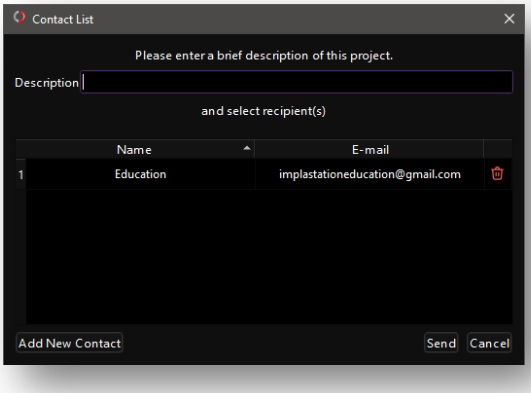
HOW TO SEND A CASE

Warning! Before sending a case using the Cloud Service, anonymize the project file according to Annex C whenever possible. Verify that the recipient is correct and that the project description does not contain unnecessary patient identifiers.

Step 1

Click on **Send** button on Menu Bar. Doing so prompts a **Contact List** window.

In the Contact List window, enter a brief description of the project in the Description field.



Step 2

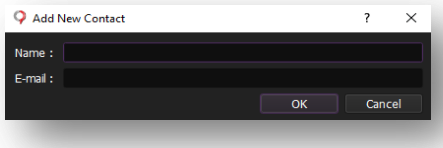
Choose how to send the project:

To send it to the **Quick Pass** address, leave the recipient list as is and click **Send**.

To send it to a different recipient, select contact from the list, then click Send.

Note! The default Quick Pass address is configured in Setting/Cloud Service. If no recipient is selected, the project is sent to this address automatically.

In the **Add New Contacts** text box, type in the email address or select the existing recipient's email in the list.



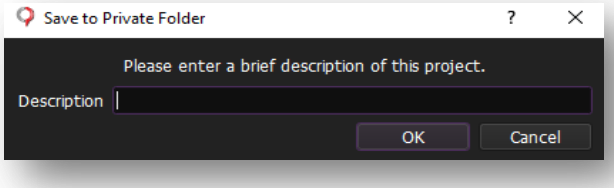
HOW TO SAVE IN THE CLOUD

Warning!

Before saving a project file to the Cloud Service, anonymize the project file according to Annex C whenever possible. The Cloud Service should not be used as the primary medical-record archive. Users remain responsible for maintaining required clinical records, local backups, and compliance with applicable privacy requirements.

Saving your Project File in the cloud lets you access them from anywhere.

To save Project File online with Prodigident Cloud, click **Save** button in **Menu Bar**. Choose **Save Project to Cloud** option, make a brief description of the case and click OK.



Warning!

If the user actively uses the Cloud Service, encrypted project files may be transmitted to and stored using Prodigident's AWS-based cloud services for case sharing or retrieval. Users are responsible for anonymizing or minimizing patient identifiers before cloud upload or sharing, unless identifiable transfer is necessary and permitted by applicable privacy requirements. Prodigident will notify users before cloud-stored project files are removed according to the applicable retention period.

ANNEX E

Cybersecurity Hygiene

3D Exacto stores patient/project data locally by default. Patient/project data are not transmitted to Prodigident backend, payment, licensing, or distribution services during normal local planning use. If the user actively uses the Cloud Service, Send function, or Save Project to Cloud function, encrypted project files may be transmitted and stored for case sharing or retrieval. The user is responsible for anonymizing or minimizing patient identifiers before cloud upload or sharing, unless identifiable transfer is necessary and permitted by applicable privacy requirements.

To protect data against loss or unauthorized use, several security mechanisms have been implemented in 3D Exacto :

Restrict user access to the storage media, operating system by setting up a strong user password;

Create a unified way of working. Installation / Upgrade of product security patches and software packages by an authorized user and/or possibly authorized HDO or Dental Lab staff.

Configure the operating system to prevent further access to the system by initiating a session lock after 10 min of inactivity or upon receiving a request from a user;

Configure the operating system to restrict the access to security features of the PC;

Use data encryption to secure Patient data, Project Files, Drilling protocols on your computer system and storage media;

Use the anonymization function to protect patient personal data if required;

Backup your data regularly and always backup before updating or uninstalling the software;

Use anti-virus software, firewall.

ANNEX F
Implant Libraries for 3D Exacto

3D Exacto contains original libraries of implants and other elements of Galimplant and of the following manufacturers:

Bio3 Implants Gmbh; Adin Dental Implant solutions; Advan s.r.l.; Alpha Dent Implants GmbH; Alpha Bio Tec. LTD.; AoN Implants S.r.l.; Bauer's Implants; Bicon LLC; B&B Dental; B.T.I. Biotechnology Institute; BEGO Implant Systems; BioHorizons Implant Systems; Bredent GmbH & Co; BTK (Biotec Srl); CAMLOG Biotechnologies; CELL Implants; Cortex Dental Implants Industries Ltd.; COWELLMEDI IMPLANT; C-TECH Implant; Dentsply Sirona: Ankylos; Astra Tech; DS Omni Taper; DS Prime Taper; XiVE; Dentis Implants; Dentaurum; Dynamic Abutment Solutions; Geass s.r.l.; Global D; Hiossen; IML SA; Implant Direct; Innobiosurg CO; Iterum – Dental Implants & Equipment Ltd.; Intraoss; iRES; IZEN; Kentech Co.; Lasak; Leone spa; LYRA ETK group; BrainBase Corporation; MODE Implant; MIS; Multysystem s.r.l.; Neoss Ltd; Noris Medical; NDI Medical; NucleOSS; OSSTEM Implant; Oxy Implant; PSM Medical Solutions; REX Implants; Ritter Implants GmbH&Co.KG; Straumann; SIC invent AG; SGS Dental; Southern Implants; STECO; Sweden & Martina; T.A.G. Medical Products; Thommen Medical; TICARE; ROOTT Implant system; TRI; VulKan; ZimVie Dental.

The following implant and other element libraries were created by reverse engineering or other non-original methods and are intended for software testing purposes only. These libraries are not intended for clinical patient planning, surgical guide design, surgical protocol generation, patient-treatment export, or guide manufacturing:

Anthogyr; Argon Dental; Dentium; DIO; JDentalCare srl; NeoBiotech; NOBEL; ICX-Magellan; Megagen; Neodent; TruGuide.

Prodigident Inc. disclaims and makes no representation or warranty with respect to the Implant and other elements libraries created by reverse engineering or otherwise, or any portion thereof, and notwithstanding anything contained to the contrary herein assumes no liability for any claim that may arise with respect to such Implant and other elements libraries or Customer's use or inability to use the same.

ANNEX G
STL Editor

STL EDITOR (FUNCTIONAL)

The Edit tools within 3D Exacto software is a comprehensive feature that helps to effectively modify STL surfaces. It grants access to a diverse array of powerful tools and functions, enabling efficient manipulation of STL geometry.

Step 1

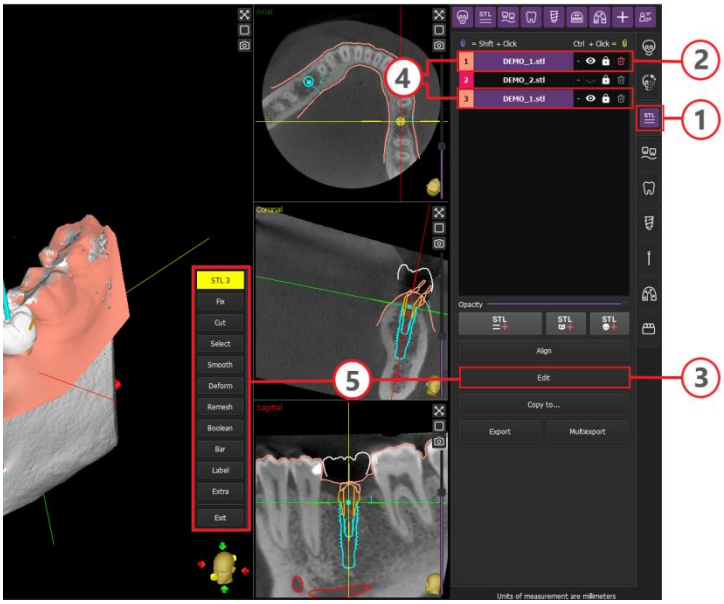
Click the **STL** button on the **Tab panel** (**1**).

Step 2

Pick the necessary **STL** object from the list of available STL (**2**). The selected object will be highlighted in the list.

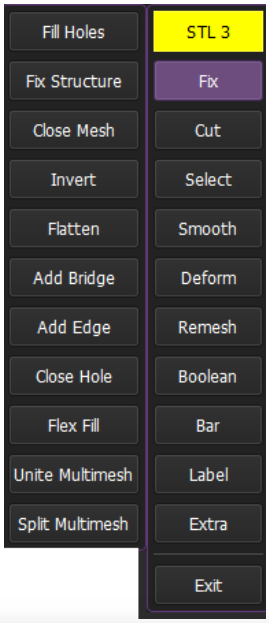
Step 3

Left-clicking on the **Edit** button (**3**) beneath the list will initiate editing. After clicking the Edit button, the editing toolbar will open (**5**).
Once the Edit button is clicked, the 3D Exacto will create a **duplicate STL** object with the same colour (**4**).



Hint!	The Editor menu can also be opened with Shift + left-click on the desired object in any visualization window.
Hint!	All changes made during editing will be applied to this copy, and it is always possible to revert to the original object.

FIX TOOL PANEL



Step 1

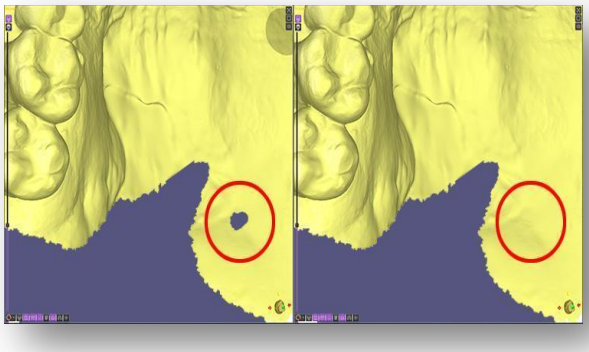
Click **Fix** (**1**) to open the tool panel.

Step 2

Choose necessary instrument:

Fill Holes

Use **Fill Holes** to fill all holes in the surface mesh and remove all small unrelated to the main object parts.



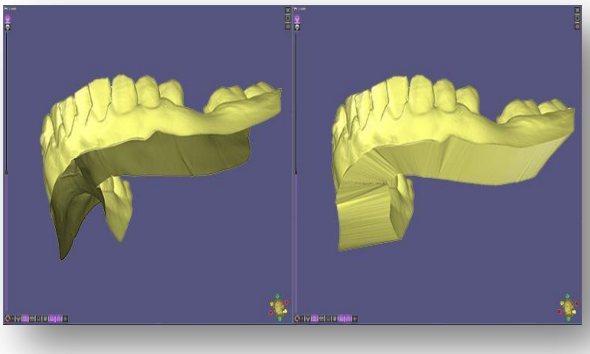
Fix Structure

The STL file errors could be intersections between triangles and non-manifold edges.
Use **Fix Structure** to fix the intersections between triangles, and non-manifold edges.
This tool as well removes all small unrelated to the main object parts



Close Mesh

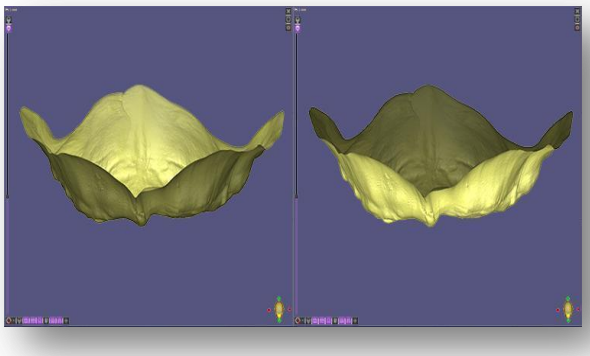
To close an open boundaries use **Close Mesh** and the software will make a new face.
Non-closed mesh is detected if some edges of the STL file are not connected to only one face, this essentially means that the model has a hole and does not represent a closed surface.



Invert

Use **Invert** to reverse the orientation of all triangles on a surface.
This tool flips any triangles with inverted normals on a surface. This is helpful if an entire portion of the model has inverted normals.

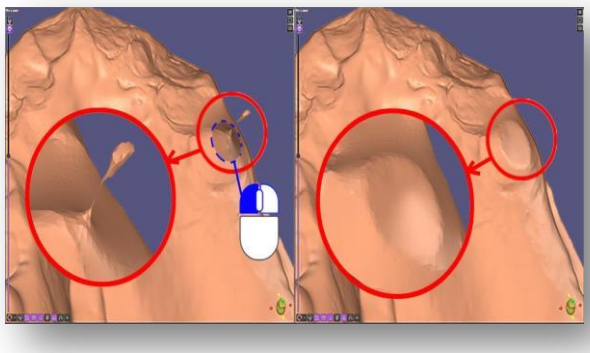
Hint!	This tool is more often used when creating a surface from a prosthesis file obtained using the double scan technique.



Flatten

There are thin areas on an STL that has actual errors in its mesh that are hard or impossible to fix using **Fix Structure** or "**Smooth** tools.
Point the cursor (brush) at the error area on the mesh. Change the brush size with the mouse wheel.
Click the left mouse button to blend the improved area completely into the model.

Hint!	This action includes selecting, deleting and closing selected open boundaries.



Add Bridge



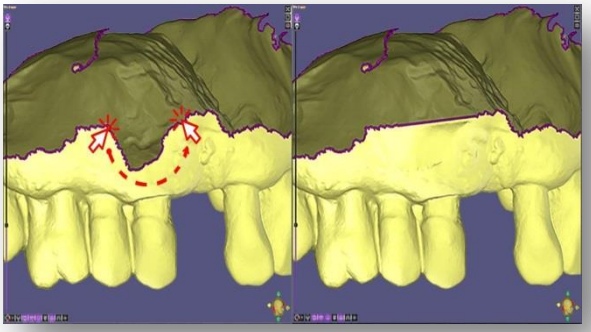
In case, there is a need to manually connect the gap (hole) at the surface, the **Add Bridge** tool comes in.

Use **Add Bridge** and gap (hole) boundaries will be automatically highlighted and visualized in the 3D Rendering Window.
Point the cursor and click the left mouse button at the open boundaries of the hole.

Click a position at the desired open boundaries to manually add bridging triangles.
Repeat the **Add Bridge** feature for a few areas around the perimeter of the gap (hole), and then use the **Fill Holes** tool to fill the remaining holes.

Hint!	A good error prevention practice is to use the **Fix Structure** tool before repairs in order to increase and homogenize the triangulation in the area concerned.

Add Edge

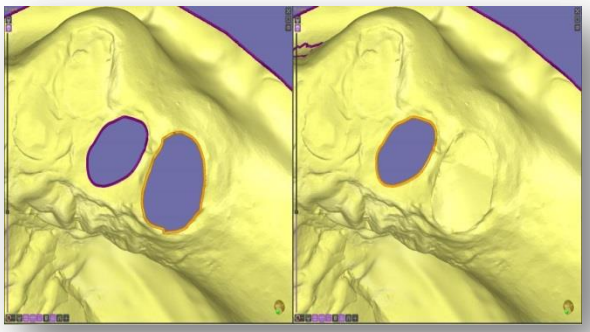


This tool is very similar to the "Bridge" tool, with the only difference being that it works with open edges.
Point the cursor and click the left mouse button at the open edge of the surface
Click a position at the desired open edge to manually add triangles and make a new face.

Hint!	A good error prevention practice is to use the **Fix Structure** and **Fill Holes** tools before repairs in order to increase and homogenize the triangulation in the area concerned.

Close Hole

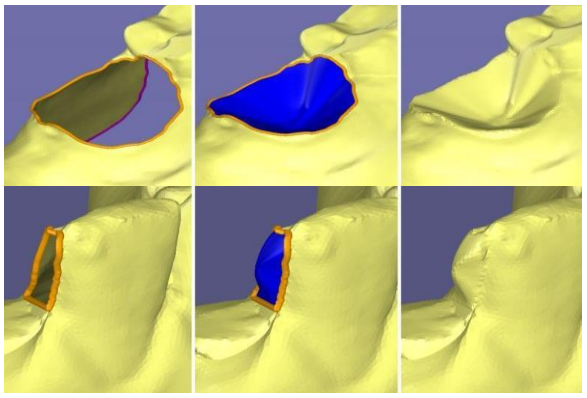
This tool is very similar to the Fill Holes tool, with the only difference you can choose which hole to close.
Use **Close Hole**. Gap (hole) boundaries will be automatically highlighted and visualized in the 3D Rendering Window.
Point the cursor at the highlighted boundaries of the hole and double-click on it to close the mesh.



Note!	If a triangle edge does not have a valid neighbour, that edge will be highlighted to indicate a problem.

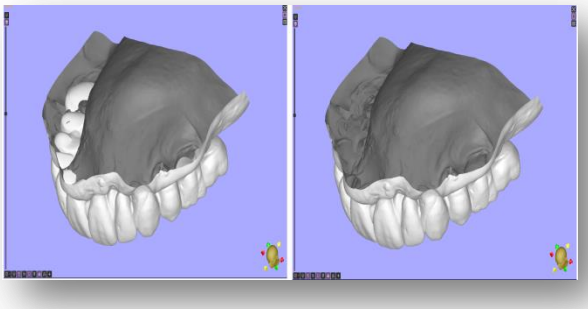
Flex Fill

Point the cursor at the highlighted boundaries of the hole and double-click on it.
Then the software should create a blue-colored patch, and you can change the curvature of the patch by scrolling the mouse wheel, making surface concave or convex. To close the mesh, just double-click on the patch again.



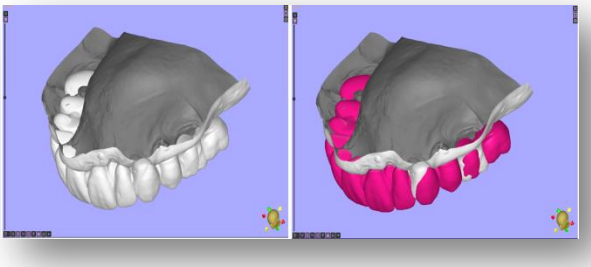
Unite Multimesh

Use **Unite MultiMesh** tool when an STL surface contains multiple meshes. It merges all meshes into a single, unified mesh structure within the STL surface.
This is commonly used when the model appears as a single object but is composed of separate elements, such as gingiva and teeth, that need to be processed as one continuous geometry.

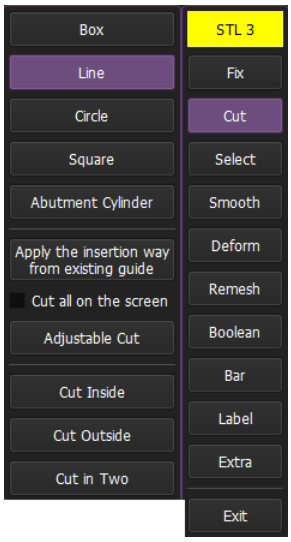


Split Multimesh

Use **Split MultiMesh** tool to separate an STL surface into multiple parts based on the meshes it contains.
Each mesh is extracted and saved as an individual STL surface/file.



CUT TOOL PANEL



The segment of the object can be cut and split in different ways. You can select segment using Box, Line, Circle, Square, and Abutment Cylinders tools.

Step 1

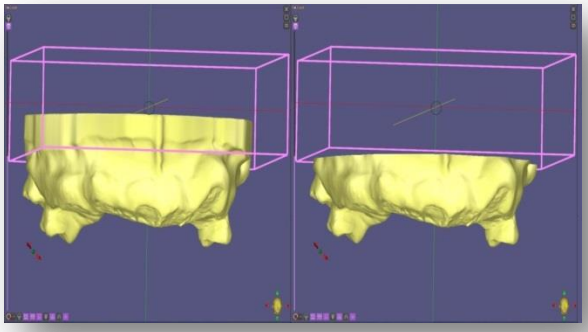
Click **Cut** to open the tool panel.

Step 2

Choose cutting tool:

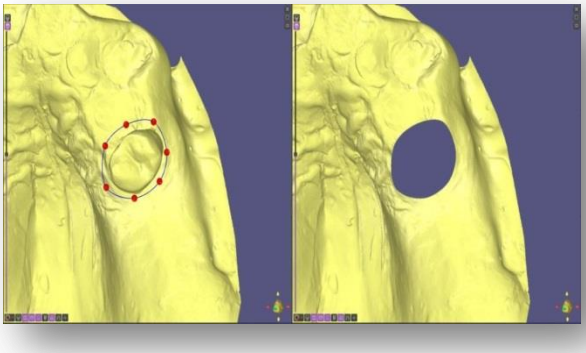
Box

Set up the box position by choosing the next options: **Below the Axial**, **In the Center**, **Above the Axial** to move the box along the X-axis.
Adjust the box size. The connection between the box and coordinate axes allows adjusting the orientation of the cutting plane either in 2D windows by dragging the center of the intersection of the axes with the left mouse button or in a 3D window by using the red arrows to rotate and adjust the cutting box.



Line

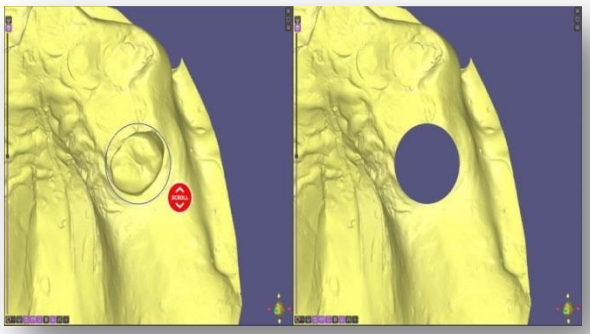
Point a freeform shape around the area you want to select with Left Mouse Button.



Note!	Holding the Shift key while placing the first point switches the line to a segmented (angular) mode, resulting in a broken line instead of a smooth contour.

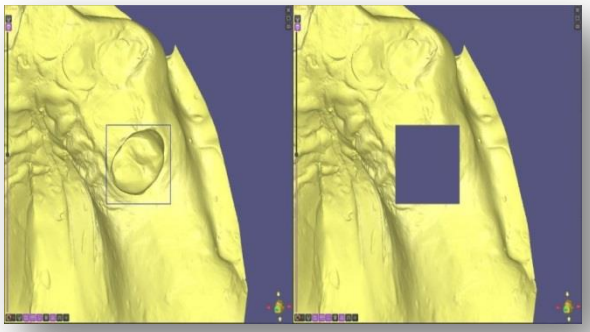
Circle

Drag a circle shape to the area you want to select with Left Mouse Button. Adjust the circle size with Scroll Wheel.
Optionally you can now either click arrows up and down or double-click the number and input a specific number adjusting the size of the shape.



Square

Drag a Square shape to the area you want to select with Left Mouse Button. Adjust the square size with Scroll Wheel.
Optionally you can now either click arrows up and down or double-click the number and input a specific number adjusting the size of the shape.



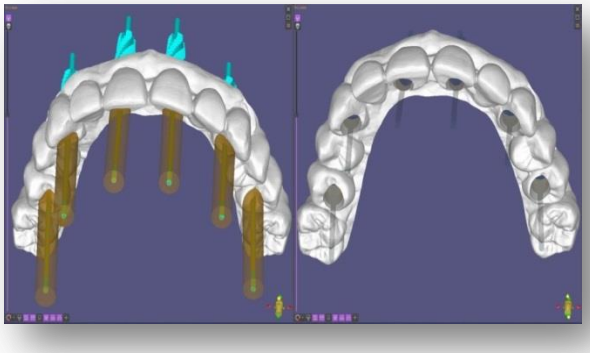
Abutment Cylinder

The Abutment Cylinder cutting tool can be used to create circular holes along the axes of the abutments.

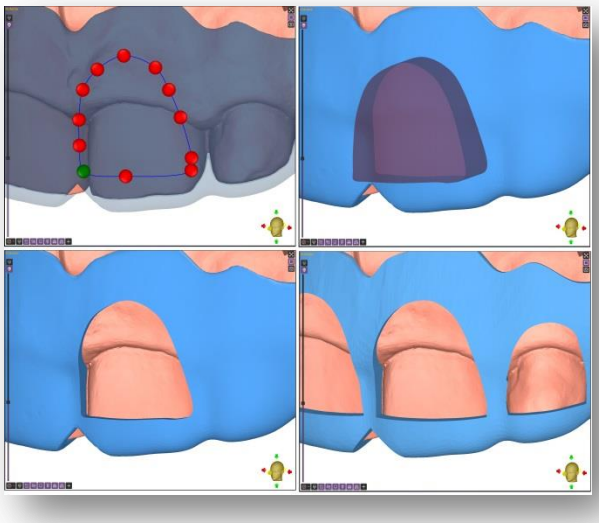
Click **Abutment Cylinder** to visualize the axes of the implant abutments in the 3D rendering window.

Adjust the diameter and position of all cylinders.
Optionally set up each cylinder separately - move the cursor to the desired axis and change the diameter by scrolling the Mouse Wheel.

Adjust the position of the bottom of the cylinders in the Cut tool panel.



Step 3(optional)



Use The **Adjustable Cut** mode to control the depth of the cutting area. This option is available for the **Line**, **Circle**, and **Square** cutting tools.
Once activated, define the cutting area by placing points and finalizing it with double-click.
Set the parameters of **Front** and **Rear** depth boundaries.
If the surface is closed, you can choose how the cutting area is formed: **Filled** or **Unfilled**.

- Use "Filled" option to create a closed cutting volume, even if the boundaries do not intersect the outer surface.
- Use "Unfilled" option to keep the cutting area open and avoid generating internal volumes.

The result can be previewed in the Volume visualization window to fine-tune the depth before finalizing the cut.

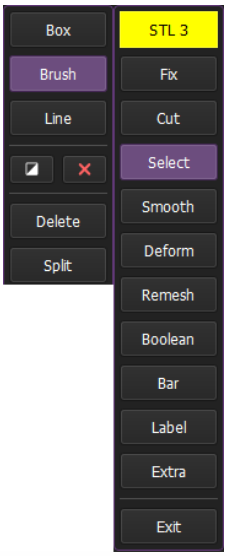
Step 4

Click either

Cut Inside to cut a selected segment into the box,
Cut Outside to cut an unselected segment of the surface,
Cut in Two to split the surface.

The **Cut all on the screen** checkbox applies the cutting shape to all visible objects on the screen.

SELECT TOOL PANEL



In the **Select** section, 3D Exacto offers several tools that can be applied directly to the mesh to select the desired area.

Step 1

Click **Select** to open the tool panel.

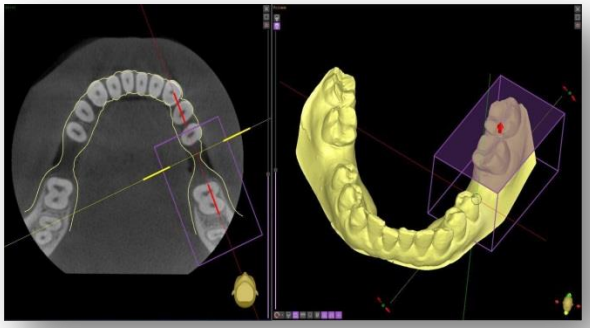
Step 2

Choose necessary instrument:

Box

Set up the box position by choosing the next options:
- **Below the Axial**,
- **In the Center**,
- **Above the Axial** to move the box along the X-axis.

Adjust the box size. The connection between the box and coordinate axes allows adjusting the orientation of the selecting plane in 2D windows by dragging the center of the intersection of the axes with the left mouse button. The red arrows are used to adjust and rotate the box in a 3D window.

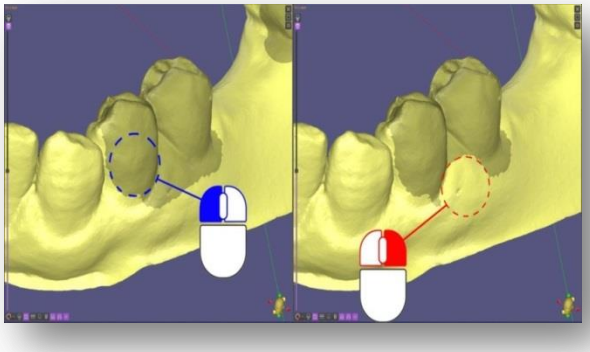


Brush

The Brush tool can be used to select an area on the STL surface.

To adjust the diameter of the Brush tool, simply scroll the mouse wheel.

To use the tool, move the cursor in the **3D** rendering window, click and hold down the left mouse button while moving the brush to select the desired area. To remove the selection click and hold down the right mouse button while moving the brush.



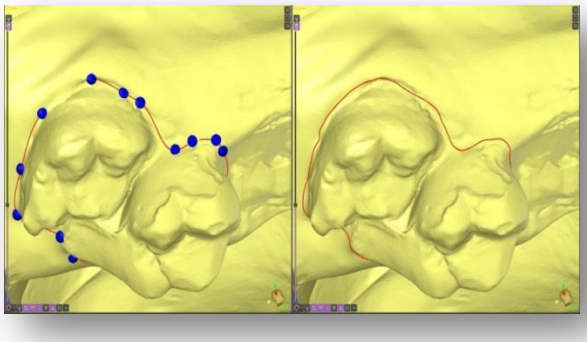
Line. Dotted/Projected

Click **Line** then click **Dotted** or **Projected**.

Point the cursor at the desired area on the mesh and click the left mouse button to place the first point, and then draw a line by dragging the mouse and clicking to place more points.

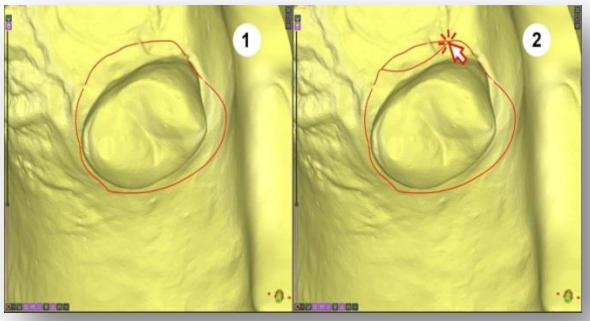
Double-click to close the line.

Select the **Edit** mode to adjust the shape of the outline, move the cursor over the line and use the available control points.
In **Projected** mode - the line is projected onto the STL surface perpendicular to the screen immediately at the moment of point positioning. For acceptable line drawing, ensure that the surface is parallel to the screen when the point is positioned. The next point is not set if the previous one, due to the rotation of the STL, becomes invisible.



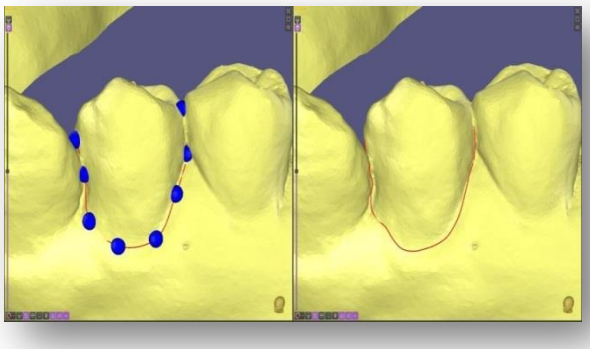
Line. Lasso

The **Lasso** tool is a freehand selection tool, it won't try to predict what you want to select.
Click **Line**, then click **Lasso**.
Point the cursor at the desired area on the mesh and click, hold the left mouse and drag to outline the shape of your selection on the STL surface.
Select the **Edit** mode to adjust the shape of the outline, move the cursor over the line and use the available control points.
Or move a mouse cursor over the line and draw an additional line outside or inside the contour to redraw and adjust the shape of the outline.



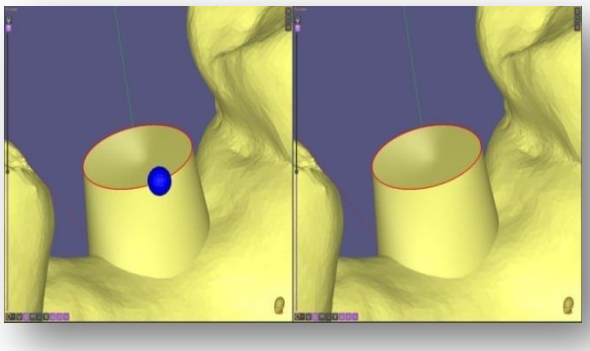
Line. Adaptive

This tool is very similar to the **Dotted** tool, with the only difference being that the software adapts the line to the edges of the triangles.
Click **Line** then click **Adaptive**.
Point the cursor at the desired area on the mesh and click the left mouse button to place the first point, and then draw a line by dragging the mouse and clicking to place more points.
Double-click to close the line.
Select the **Edit** mode. Now you can move the cursor over the line and use the available control points to adjust the shape of the outline.



Line. Edge

The **Edge** tool is an automated selection tool, which means it would automatically snap to the edge of objects and try to predict what you want to select. 3D Exacto will help complete your shape by connecting small gaps between your starting point and endpoint.



Click **Line** then click **Edge**.
Point the cursor at the desired edge on the mesh and click the left mouse button to place the first point, and the software automatically complete the shape.
Double-click to close the line.
Select the **Edit** mode. Now you can move the cursor over the line and use the available control points to adjust the shape of the outline.

Step 3

After finishing the selection, the next options are available:

- invert selected area >



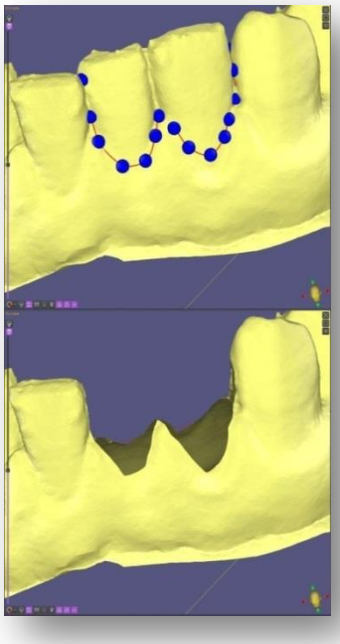
- cancel the selection >



Step 4

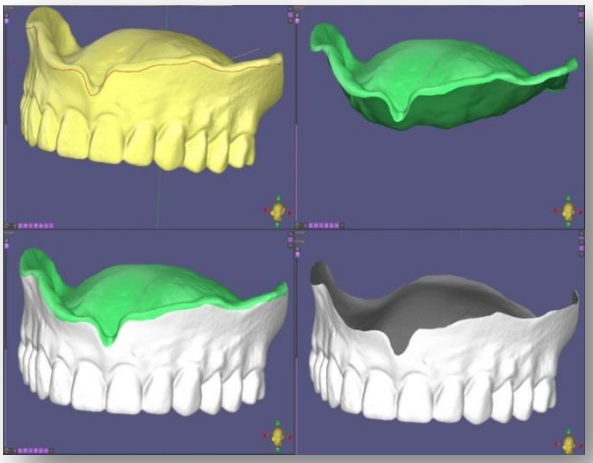
Select. Delete

Click **Delete** to cut off a selected segment of a surface into the selected area.



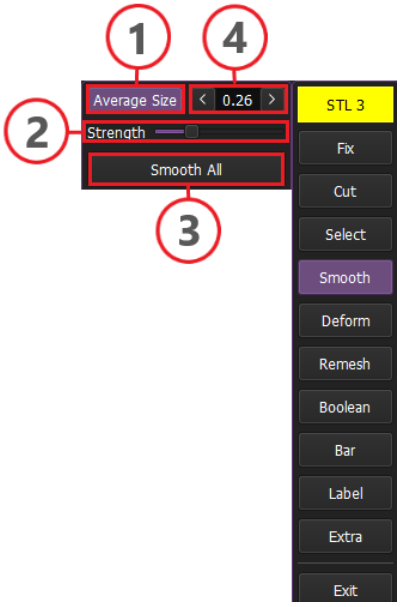
Select. Split

Click **Split** to split an object into two objects with open boundaries along the selection line.



Hint!	This tool is more often used when creating a surface from a prosthesis file obtained using the double scan technique.

SMOOTH



The Smooth brush matches the selected area to the curvature of the surrounding area and makes it smooth.

Step 1

Click **Smooth** in the **Edit** menu.

Step 2

Set

Average Size (**1**) to get the average size of the triangle at the smoothed surface. Adjust the size yourself by entering numerical values (**4**) in the corresponding window.

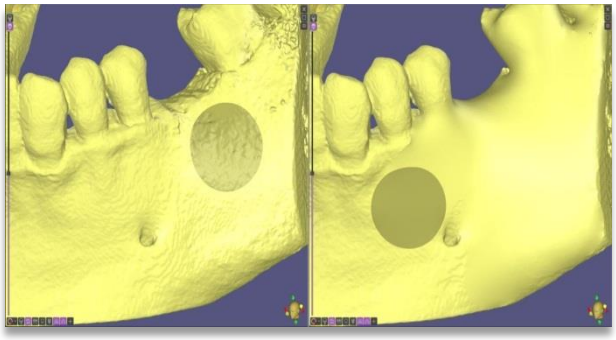
Strength (**2**) to increase or decrease the smoothing intensity.

Step 3

Point the cursor and click or click and hold the left mouse button on the area you want to smooth.

OR

Click to **Smooth all** (**3**) to remesh, optimize triangle count and smooth the whole surface of the object.



DEFORM TOOL PANEL

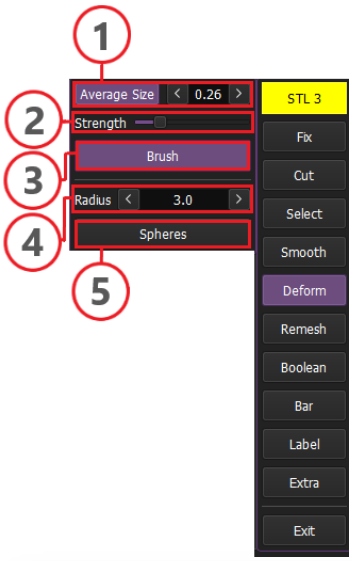
Use this function if you want to change the mesh curvature locally.

Step 1

Click **Deform** in the **Edit** menu.

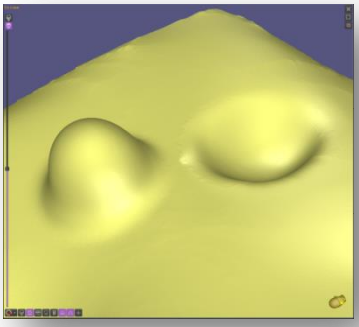
Step 2

Choose one of deform tool:



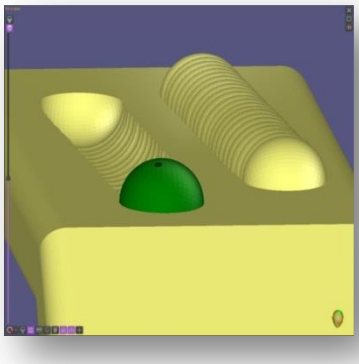
Brush (3)

Set **Average Size** (1) to get the average size of the triangle of the surface. Adjust the size of brush by entering numerical values in the corresponding window or adjust the brush size (4) with scroll wheel.
Use the **Strength** (2) slider to increase or decrease the smoothing intensity.



Spheres (5)

The sphere is a very specific brush. The center of the Sphere is always on the surface of the mesh.
Adjust the size of sphere by entering numerical values in the corresponding window or adjust the sphere size with scroll wheel.

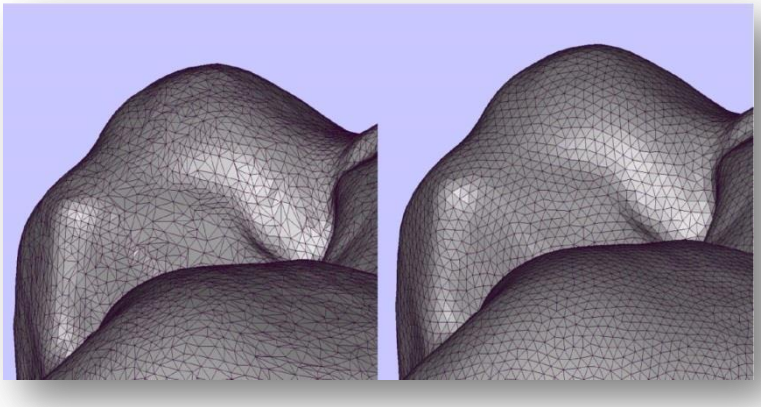


Step 3

Point the cursor at the desired area, click and hold the left mouse button on the area making it convex, or click and hold the right mouse button on the area making it concave.

REMESH

Split triangles – is an edge split and it would insert a new vertex which decreases the length of the original long edge if it's longer than high.
The key aim for the Isotropic remeshing is to set all the triangle edge lengths to be the same and all the vertices have perfect valence.
The main benefit of this feature is the simplification of subsequent model editing.



Step 1

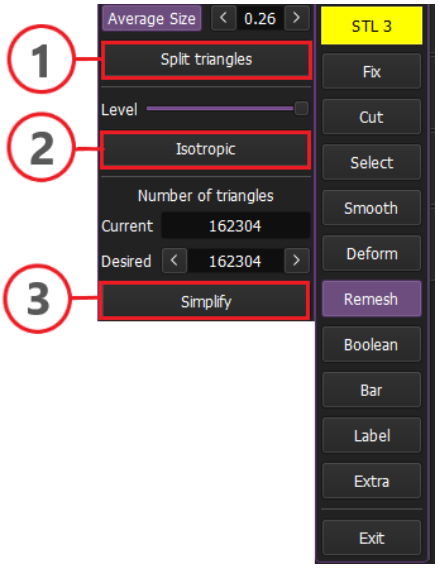
Click "**Remesh**" in the Edit tab.

Step 2

Click **Average Size** to get the average size of the triangle of the surface. Adjust the size of triangles by entering numerical values in the corresponding window.

Step 3

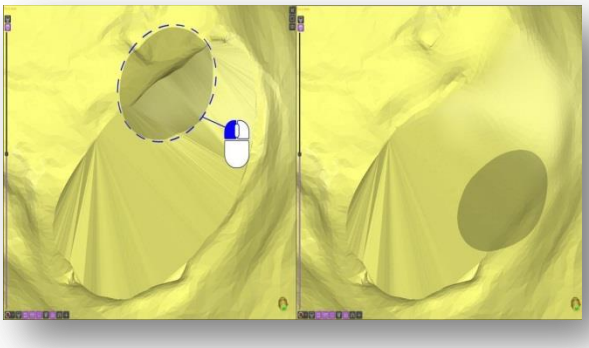
Click
Spit Triangles (**1**)
or
Isotropic (**2**) to start remeshing.



Step 4

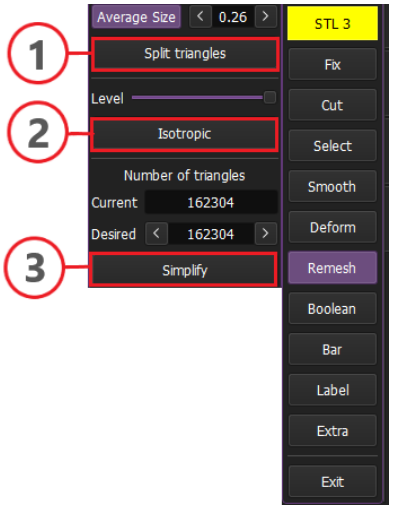
If there is a need for local remeshing, point the cursor and click and hold the left mouse button on the area you want to remesh.

Adjust the brush size with scroll wheel.



Remesh: Simplify

The more triangles in a scanned or converted STL model, the smoother the shapes of the surface will be, but also the larger the file size, the longer the editing time. Conversely, fewer triangles in an STL model means it will fix and edit faster.



Step 1

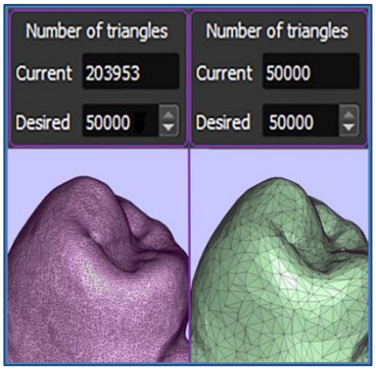
Click **Simplify** (3) to reduce the number of triangles.

Step 2

You can now either click arrows up and down or double-click the number and input a specific number.

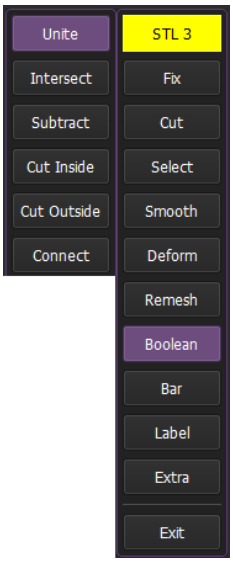
Note!

Look what happens when you drop the triangle number down from 203953 to 50000. You can still make out the basic shape of the STL object, but its smooth curves have been lost underneath a sharply-faceted surface.



BOOLEAN OPERATIONS

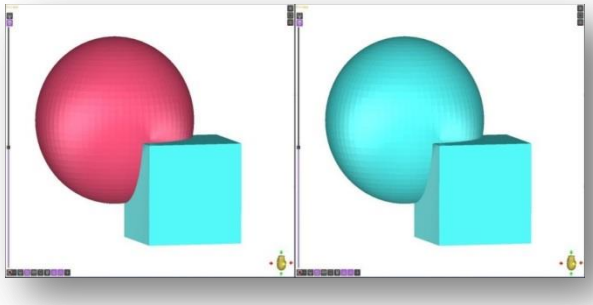
In STL Editing, by Boolean operations, we mean creating intersections and unions of objects, as well as subtracting objects from each other.



In 3D Exacto the following Boolean operations are available:

Unite

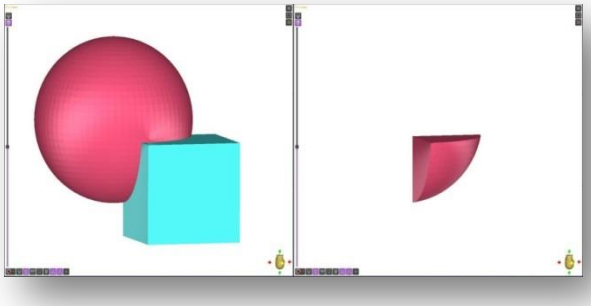
Unite is a complex algorithm that merges two objects into a single mesh.



Click **Boolean** in the **Edit** menu. Choose **Unite** in Boolean Tool Tab.
And then either click directly on the object you want to attach in the 3D visualization window, or on the object in the list of the STL files.

Intersect

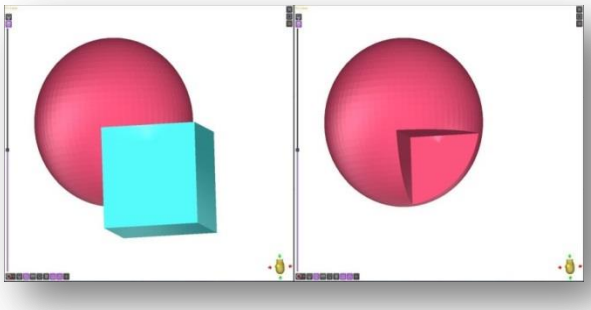
The **Intersect** tool creates an STL object from the intersection area between two STL objects



Click **Boolean** in the Edit tab. Choose **Intersect** in Boolean submenu. And then either click directly on the object you want to intersect in the 3D visualization window, or on the object in the list of the STL files.

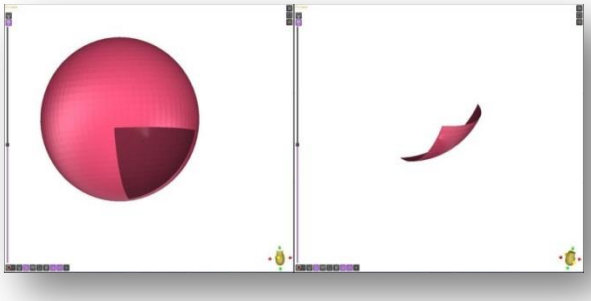
Subtract

The subtraction occurs between two objects - the chosen object is subtracted from the main initial object.
Click **Boolean** in the Edit tab. Choose **Subtract** in Boolean submenu. And then either click directly on the object you want to subtract in the 3D visualization window, or on the object in the list of the STL files.



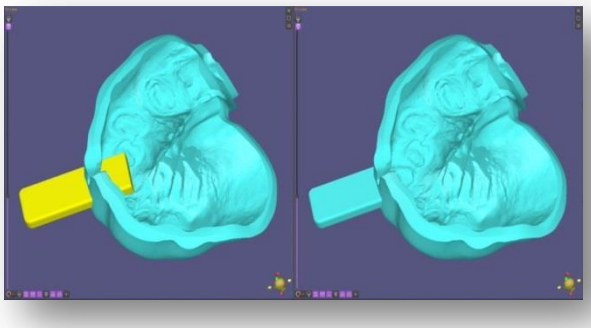
Cut inside/Cut outside

The cutting tool is used to cut the surfaces of STL objects, resulting in an opened STL surface.
Click **Boolean** in the Edit tab. Choose **Cut Inside** or **Cut Outside** in Boolean Tool Tab.
And then either click directly on the object you want to attach in the 3D visualization window, or on the object in the list of the STL files.

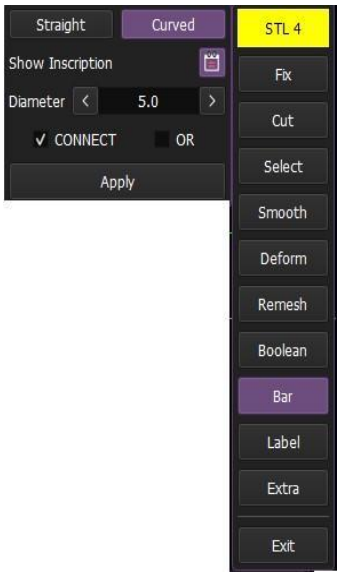


Connect

This tool is very similar to the **Unite** tool, with the only difference being that it connects only the major part of the chosen object (yellow), all intersection and minor parts would disappear.
Click **Boolean** in the Edit tab. Choose **Connect** in Boolean Tool Tab.
And then either click directly on the object you want to connect in the 3D visualization window, or on the object in the list of the STL files.



BAR



Step 1

Click **Bar** in the Edit tab.

Step 2

Set:

- Type of Bar

Straight
OR
Curved

- Adjust the **Diameter** of bars by entering numerical values in the corresponding window.

Choose one option of constructing a bar:

CONNECT option is used to construct a bar when it does not fully intersect one or more intermediate parts of the object in its middle section. This option is also suitable for building a support bar when one or both parts of the object have thin walls.
OR
INTERSECT option is used when constructing a bar in a way that its middle section fully intersects one or more intermediate parts of the object.

Step 3

Point the cursor at the desired area, click the left mouse button and when you will move your mouse away from this point, a purple solid bar will be displayed between the initial point and the mouse cursor position, then click the left mouse button again and the software will set the bar.



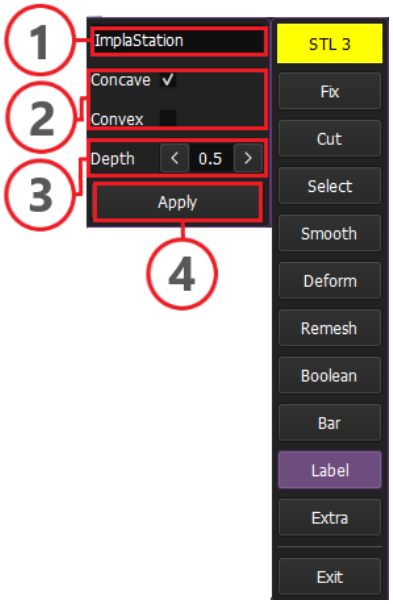
Step 4

Now you can change the shape and diameter of the bar by dragging active points and scrolling the mouse wheel.

Step 5

Click **Apply**.

LABEL



Step 1

Click **Label** in the Edit tab.

Step 2

Type the desired text into the text field (**1**).

Step 3

Set:

- label design:

concave

OR

convex (**2**).

- label depth.

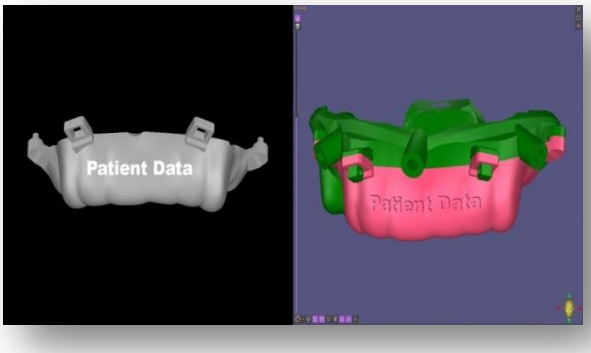
Step 4

Move the text to the desired position on the STL surface and click on it to set the label position.

Step 5

Click on **Apply**.

Hint!	Delete a label by right-clicking on it.



EXTRA

Offset (1)



The **Offset** tool expands or contracts a selected STL object by a specified amount of millimeters.

Step 1

Click **Extra** in the Edit menu..

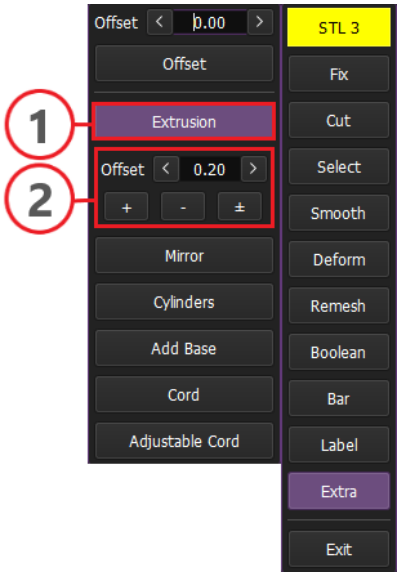
Step 2

Set the numerical values in the corresponding window.

Step 3

Click **Offset**.

Extrusion (1)



The **Extrusion** tool extrudes selected surface locally.

Step 1

Select desired area on the STL surface using the **Select** submenu.

Step 2

Click **Extra** in the Edit menu.

Step 3

Set the numerical values in the corresponding window.

Step 4

Click either (2)



The software creates the extrusion outside of the initial surface

OR

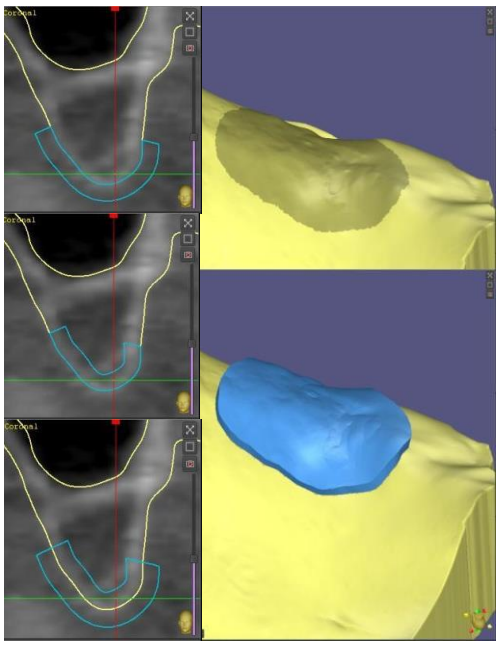


The software creates the extrusion inside of the initial surface

OR



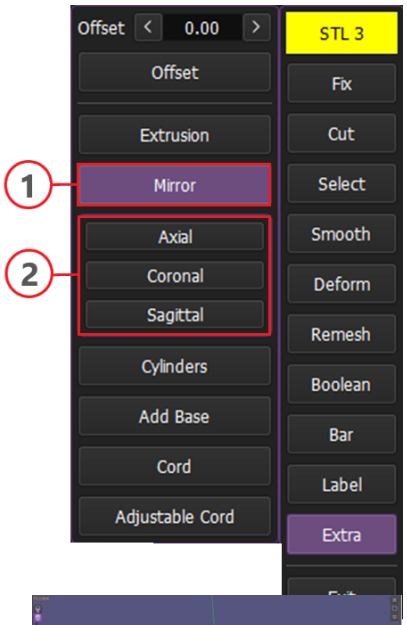
The software creates the extrusion in the middle of the initial surface



The new object will be appeared in STL list.

Mirror (1)

The **Mirror** tool creates a mirrored duplicate of a selected object.



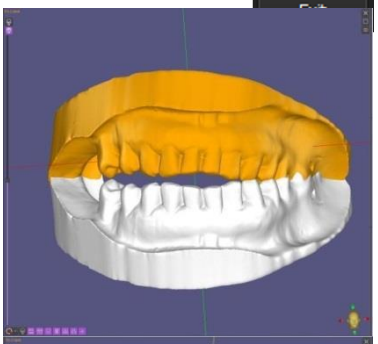
Step 1

Click **Extra** in the Edit menu then click **Mirror** to mirror the object along the axial, coronal, or sagittal axis.

Step 2

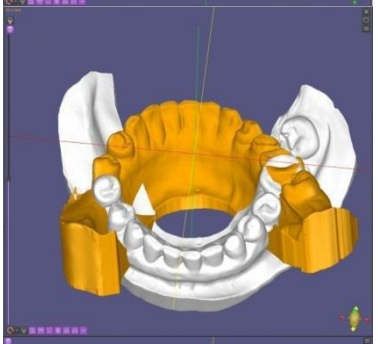
Choose one of plane you want to mirror relative to (**2**):

The **axial** (or transverse plane) is a horizontal plane dividing the object into superior (upper) and inferior (lower) sections.



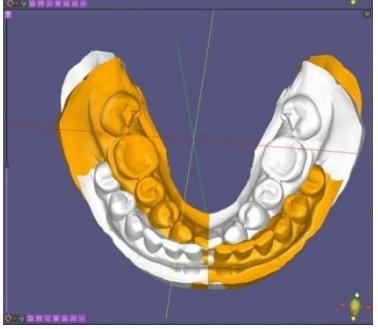
OR

The **coronal** plane is a longitudinal plane, dividing the object into anterior (front) and posterior (back) sections.



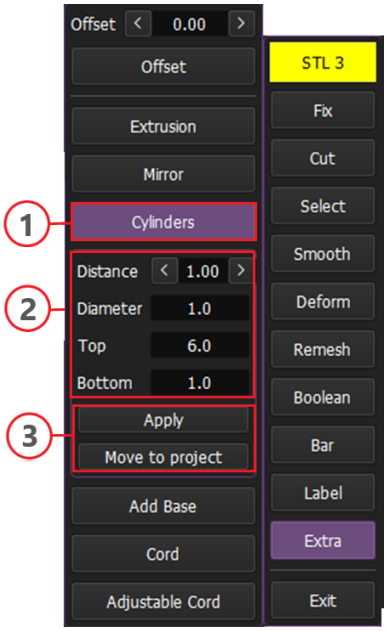
OR

The **sagittal** plane refers to a vertical plane that divides an object into two parts: the right and left sections.



Cylinders (1)

The **Cylinders** tool creates a single cylinder or a group of cylinders located perpendicular to the surface plane.



Step 1

Click **Extra** in the Edit menu.

Step 2

Click **Cylinder** and set up the settings (**2**):

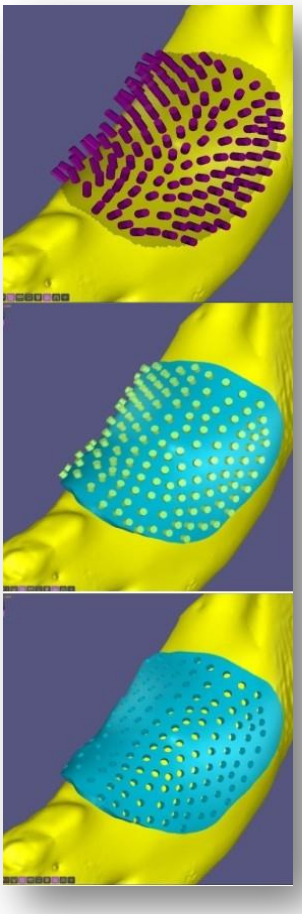
- Distance between the cylinder centers (mm),
- Diameter of the cylinder (mm),
- Top height of the cylinder above the STL surface (mm),
- Bottom height of the cylinder below the STL surface (mm).

Step 3

In the 3D rendering window, point the cursor at the desired area and then click on the STL surface to create a cylinder.

Step 4

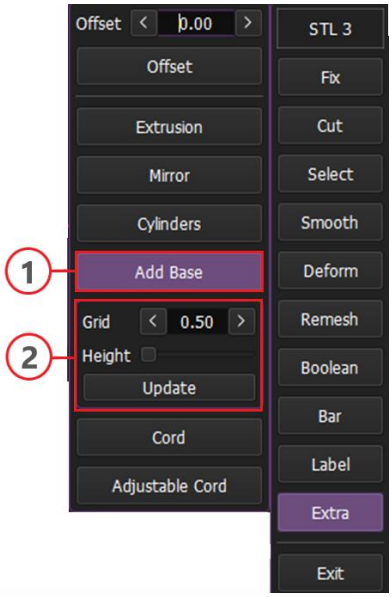
Click **Apply** and **Move to Project** (**3**) and created cylinder/s will be transferred to the project as a separate object.



Hint!	To create a group of cylinders, use the **Select** tool to select the necessary area on the STL object. After, click **Extra** and select **Cylinders**. Set up the necessary cylinder parameters and click **Move to Project** to generate the group of cylinders.

Add Base (1)

This tool is designed to create digital models of patient scans.



Step 1

Click **Extra** in the Edit menu and Click **Add Base** to add a vertical base under the open STL surface.

Step 2

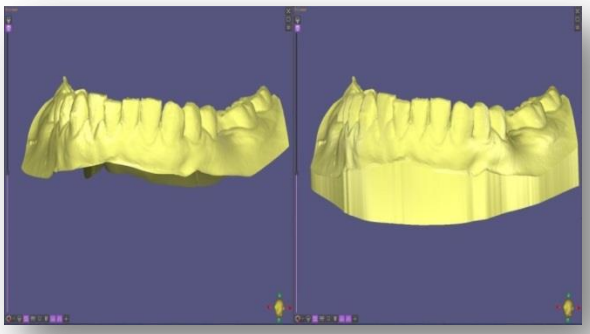
Set the necessary parameters (**2**):

- **Grid** parameter is responsible for marking the triangles on the platform. The smaller this value, the smoother the surface of the base.
- **Height** slider adjusts the height of the base.

Step 3

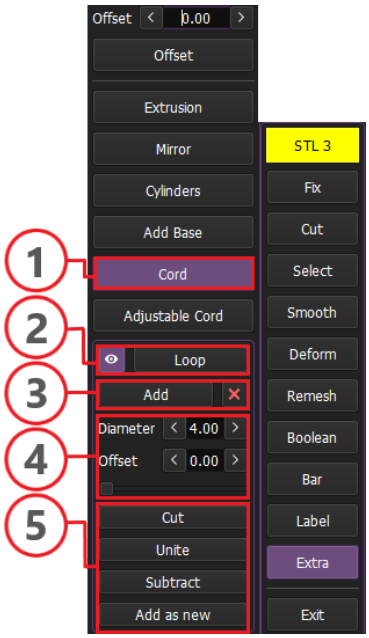
Click the **Update** button.

Note!	The result mimics the traditional based plaster dental model in appearance and can be easily 3D printed.



Cord (1)

The **Cord** tool generates a round cord on the selected surface.



Step 1

Click **Extra** in the Edit menu and click **Cord**.

Step 2

Set (**4**):

- **Diameter** adjusts the cord's diameter (mm)
- **Offset** adjusts the distance between the cord's center and the object's surface (mm). A positive value moves the cord outward, while a negative value moves it inward.

Step 3

Click **Add** to add cord. Point the cursor at the desired area on the mesh and click the left mouse button to place the first point, and then draw a cord by dragging the mouse and clicking to place more points.
You must put at least three dots to make cord visualized in a **3D** rendering window.
To create a closed cord, click **Loop** (**2**).
Click  to delete cord (**3**).

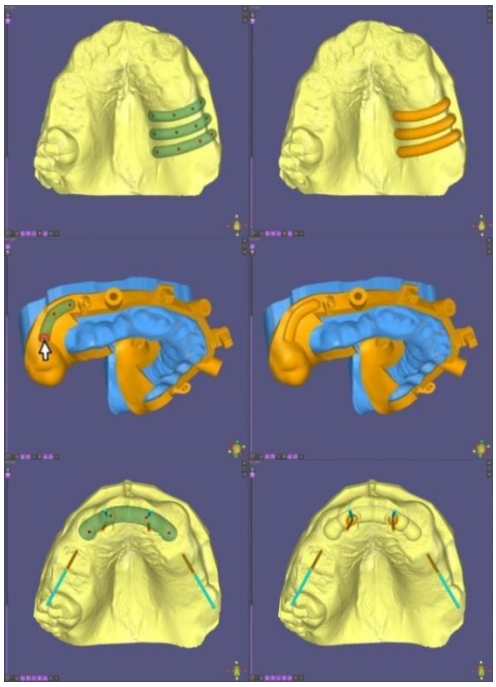
Step 4

Choose one of the Cord tool (**5**):
Cut to remove the part of the cord located inside the STL object

Unite to unite the cord with the STL object.

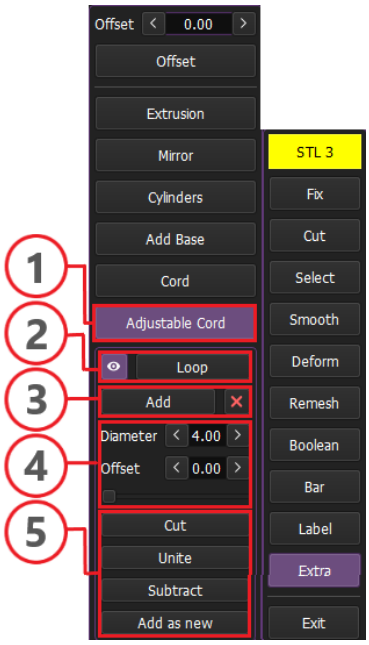
Subtract to subtract the cord from the STL object.

Add as new to add a cord as new STL surface.



Adjustable Cord (1)

The **Adjustable Cord** tool is used to create a custom-shaped round cord that can be positioned freely in 3D space. Unlike the standard **Cord** tool, which strictly follows the STL surface during drawing, the **Adjustable Cord** allows more flexibility.



Step 1

Click **Extra** in the Edit menu and click **Adjustable Cord**.

Step 2

Set (**4**):

- **Diameter** adjusts the cord's radius (mm)
- **Offset** adjusts the distance between the cord's center and the object's surface (mm). A positive value moves the cord outward, while a negative value moves it inward.

Step 3

Click **Add** to add cord. Point the cursor at the desired area on the mesh and click the left mouse button to place the first point, and then draw a cord by dragging the mouse and clicking to place more points.
You must put at least three dots to make cord visualized in a **3D** rendering window.

To create a closed cord, click **Loop** (**2**).
Click  to delete cord (**3**).

Step 4

Choose one of the Adjustable Cord tool (**5**).

Cut to remove the part of the cord located inside the STL object

Unite to unite the cord with the STL object.

Subtract to subtract the cord from the STL object.

Add as new to add a cord as new STL surface.



Hint!

Using **Adjustable Cord** tool can be useful in creating various structures, for example, internal irrigation channels inside the guide.

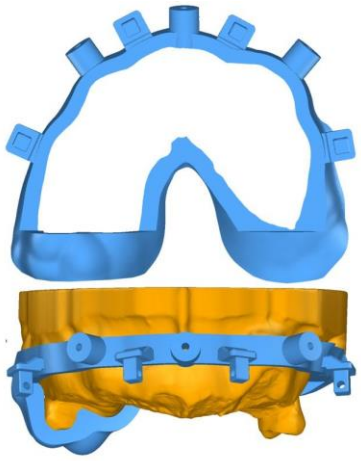
ANNEX H

STACKABLE SURGICAL GUIDE

CHAPTER I STACKABLE SURGICAL GUIDE COMPONENTS

FIXATION BASE

The Fixation Base's initial function is bone reduction. The upper edge of the Fixation Base is created to indicate the level to which the bone needs to be reduced. The second function of the Fixation Base is to support the components: Osteotomy Guide, Carrier Guide, PMMA, Ceramic or (Provisional) Prosthetic.



PIN GUIDE

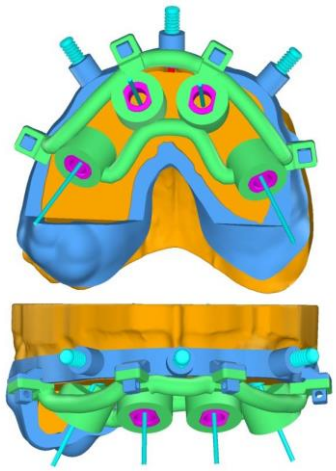
Pin Guide's only purpose is to deliver the Fixation Base and maintain its position while the fixation pins are set.

Dentate Pin Guides seat on the teeth are verified via revision windows and ensure the surgery starts in the correct position. With edentulous patients, the Pin Guide will seat exactly like the denture and verify the bite and ultimate tooth position.



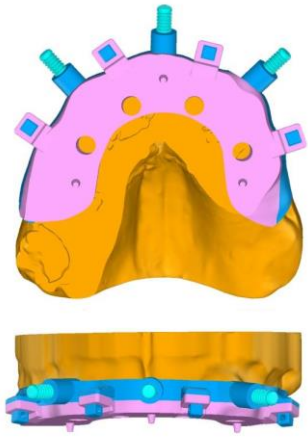
OSTEOTOMY GUIDE

Osteotomy Guide controls the implant depth, trajectory, and indexing (rotation) during osteotomy creation. It seats into the Fixation Base and is fixed using Locks.



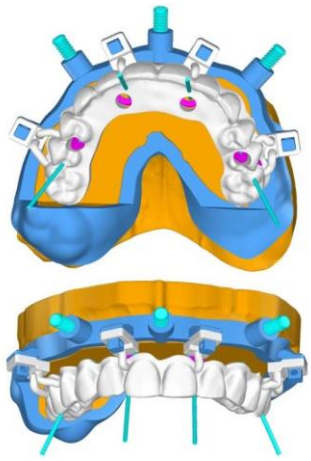
CARRIER GUIDE (OPTION)

One function of the Carrier Guide is to serve as a tissue gap between the top of the bone reduction to the bottom of the Prosthetic. The Carrier Guide remains in the mouth through the prosthetic conversion. The two or three clear plastic pegs on the Carrier Guide deliver the prosthetic in the proper position as planned.



PROVISIONAL PROSTHETIC, PRINTED TRY-IN

The Provisional is a strong and esthetic prototype for the final restoration. It is designed for immediate load and extended use. The Provisional is delivered the day of surgery and remains in use until the final prosthetic delivery.



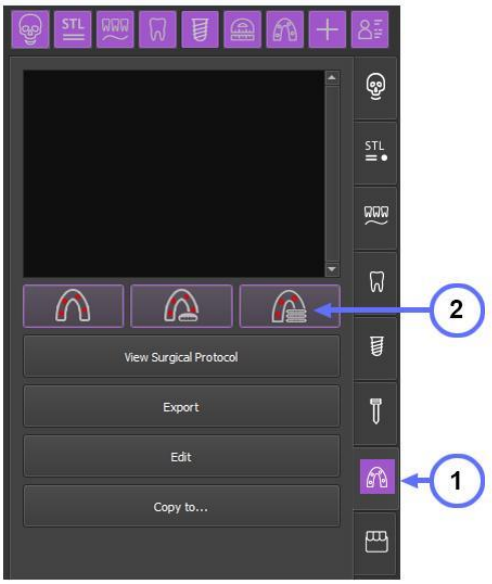
CHAPTER II

This chapter describes how to create a stackable surgical guide using the step-by-step workflow.

Preparation Step

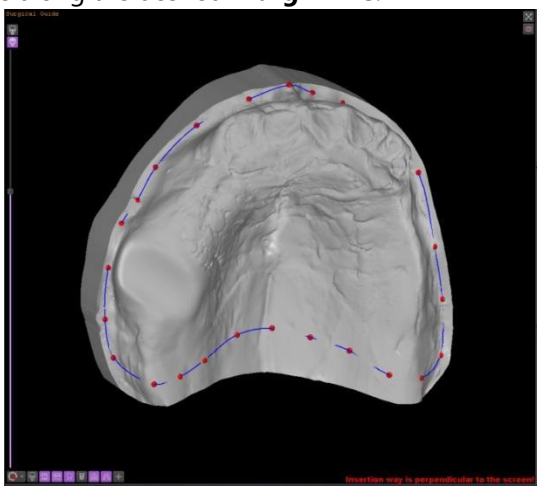
Click **Surgical Guides** (**1**) in the Tab Panel menu to open the tab.

In this window, click Make Stackable Surgical Guide (**2**) to open the surgical guide settings menu.



Step 1
Initiating Surgical Guide Design

Define the borders of the surgical guide by clicking the scanning surface along the desired **margin line**.



Then double-click to close a surgical guide margin line.

Now you can move points on a **margin line** by dragging and moving them with your mouse.

Using the surgical guide settings menu, you can define settings for the surgical guide and its representation in the 3D-rendering view.

Select a surgical guide insertion direction - Mandible/Maxilla (**1**).

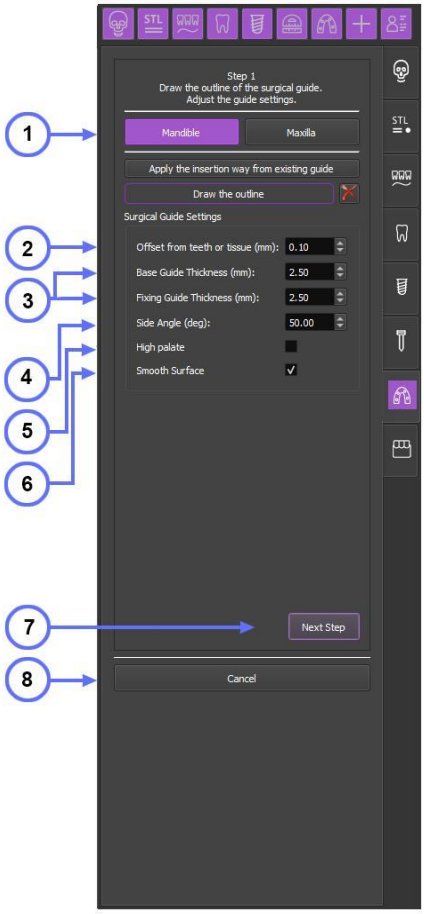
Use the Guide Offset (mm), Thickness (mm), Side angle (deg) settings to define the design properties of the surgical guide (**2, 3, 4**).

Select "High palate" if the palate is unusually high and narrow (**5**).

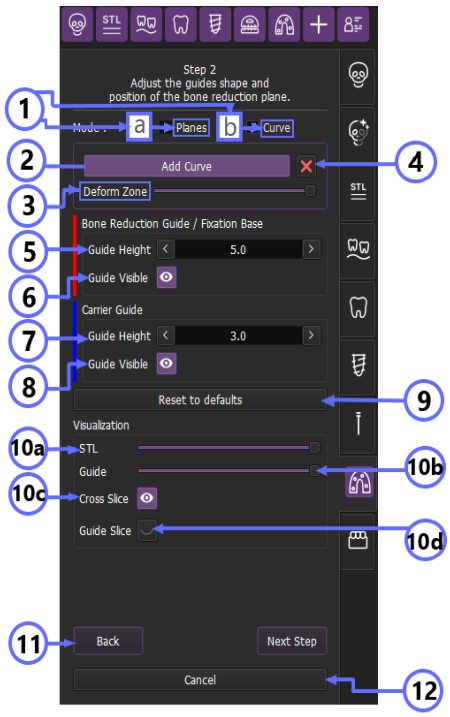
Select "Smooth Surface" for smoothing the top of the guide surface (**6**).

In Step windows, clicking **Next Step** saves the settings for this work step and takes you to the next step (**7**).

Clicking **Cancel** closes the window but discards all settings specified since you have started the step (**8**).



Step 2
Plane Settings



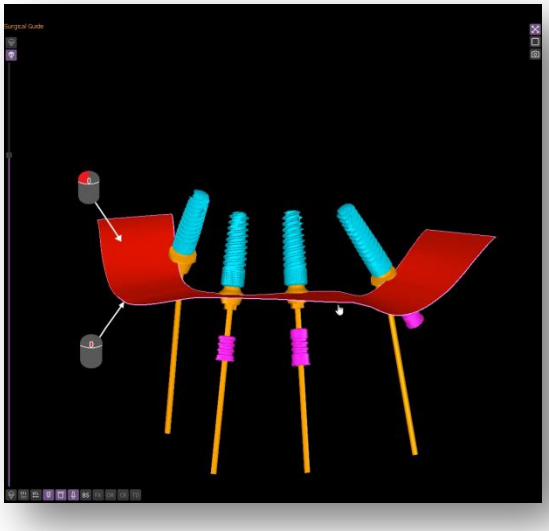
Using the Fixation base, Bone reduction Guide, Pin guide and Carrier guide settings menu for the surgical guide:

Select a Bone reduction guide surface design – Planes or Curve (**see Top Figure -1a,b**);
Choose either **Planes** or **Curve** for the Bone Reduction Guide surface design based on the occlusal platform arrangement. Use "Planes" if all implants are aligned in the same plane (**see Top Figure 1a**).
Use Curve If the occlusal platforms of the implants are not in the same plane (**see Top Figure -1b)**

If Curve mode is chosen, prepare and add a curve by following these steps:

Step 1
In the axial slice window, adjust the slice from occlusal to apical until all implants are visible to define the base trimming level.

Step 2
In the axial window trace the curve along the buccal surface of the proto-guide, similar to creating a panoramic arch. Complete the tracing with a double-click and adjust the curve width if needed. To delete or reset the drawn curve, use the appropriate buttons on the workflow panel (**see Top Figure -2, 3**).



Step 3
In the Volume window, bend the curve by dragging the middle of the curve while holding the left mouse button. Turn off the base guide visualization to enhance visibility. Adjust the extent of the deformation using the **Deform Zone** slider (**see Top Figure -3, Down Figure**).

Step 4
In the Volume window, adjust the curve's edge height by placing the cursor on the edge and using the scroll wheel to align it with the occlusal surface of the implants. Adjust the extent of the deformation using the **Deform Zone** slider (**see Top Figure -3, Down Figure**). Repeat this adjustment for the posterior edge if necessary.

For both Plane and Curve modes, set the following settings:
Use the **Guide Height** settings to define the height of the Fixation Base (**Top Figure - 2**);
Show/hide the Fixation Base plane by clicking the checkbox (**see Top Figure 3**);
Use the **Guide Height** settings to define the height of the Carrier guide (**see Top Figure - 4**);

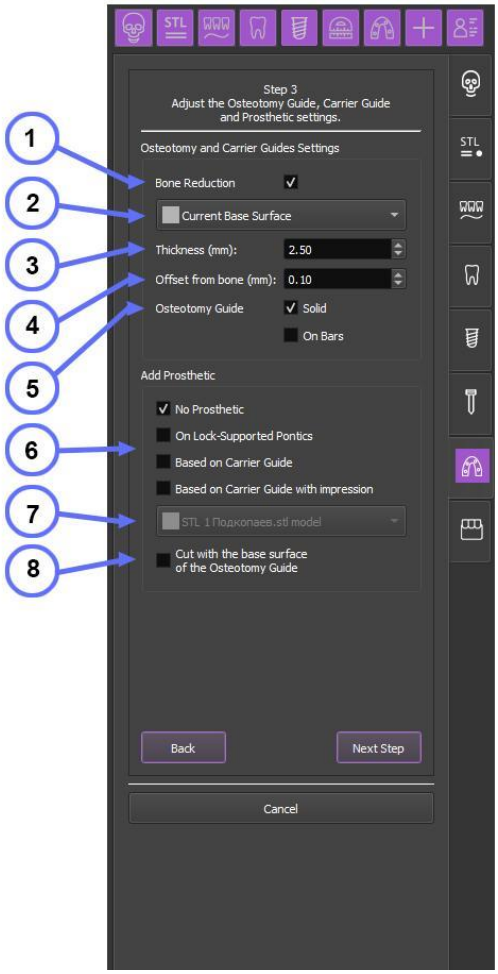
Show/hide the Carrier Guide plane by clicking the checkbox (**see Top Figure - 5**);
You can also reset the default values for the Fixation base height (5 mm) and Carrier guide height (3 mm) (**see Top Figure - 6**);
Use the sliders "**STL**" and "**Guide**" transparency to define the STL and Guide surface opacity (**see Top Figure - 7a, 7b**);
Show/hide the Cross Slice plane by clicking the checkbox (**see Top Figure - 7c**);
Show/hide the Guide Slice plane by clicking the checkbox (**see Top Figure - 7d**);
In the Step window, clicking **Back** returns you to the previous step (**see Top Figure - 8**);
In Step windows, clicking **Next Step** saves the settings for this work step and takes you to the next step; Clicking **Cancel** closes the window but discards all settings specified since you have started the step (**see Top Figure - 9**).

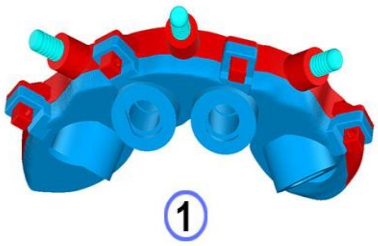
Step 3
Osteotomy guide, Carrier guide and Prosthetic design settings

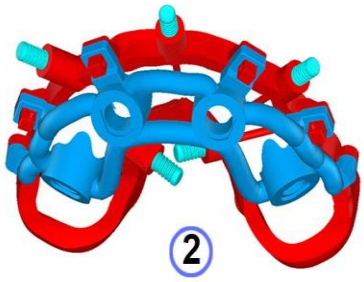
Using the Osteotomy guide / Carrier guide / Prosthetic settings menu for the surgical guide:
If you want to use the bone reduction option select the **Bone reduction** checkbox (**see Top Figure - 1**);
Select the surface you will use for the osteotomy guide from the list of the STL (**see Top Figure - 2**);
Use the **Thickness** (mm) settings to define the thickness of the Osteotomy guide (**see Top Figure - 3**);
Use the **Offset from bone** (mm) settings to define the offset from bone for the Osteotomy guide (**see Top Figure 4**);
Select the desired Osteotomy guide type using the two checkboxes "solid" (**see Top Figure 5 and Down Figure - 1**) and "on bars" (on lock-supported pontics (**see Top Figure - 5 and Down Figure - 2**).

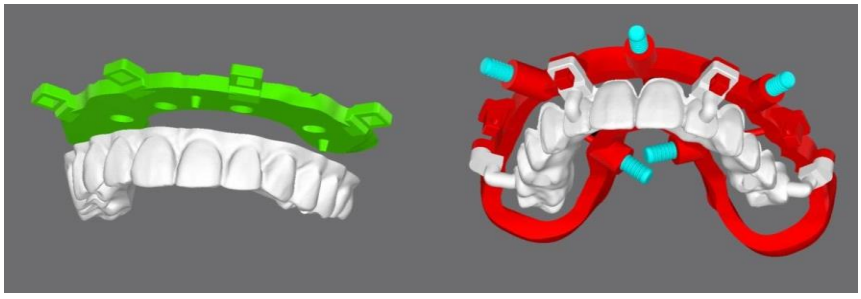
Select the desired Prosthetic type using the corresponding checkboxes (**see Top Figure - 6**);

Select the surface you will use for the prosthetic from the list of the STL (**see Top Figure - 7**);
Select the checkbox and the prosthetic will be cut off automatically and will be based on the position and shape of the surface you selected in the Bone Reduction settings, when finalizing the surgical guide design (**see Top Figure - 8**);
In Step windows, clicking **Next Step** saves the settings for this work step and takes you to the next step;
Clicking **Cancel** closes the window but discards all settings specified since you have started the step.



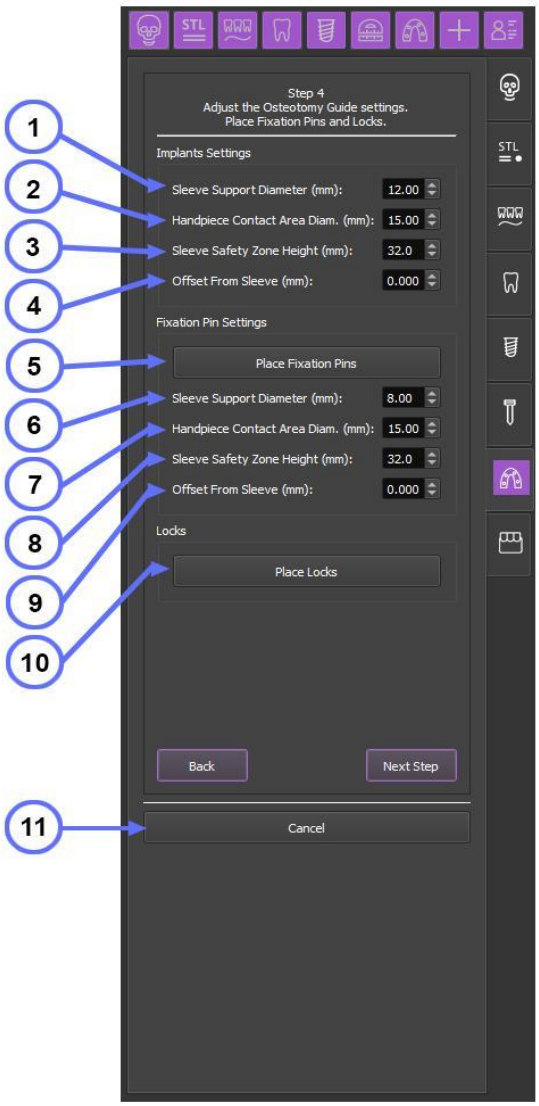






Step 4
Implant-supported and Pin-supported area settings

Using the Implant-supported and Pin-supported area settings menu:



Use the **Sleeve support diameter** (mm) settings to define the diameter of the implant sleeve support (**1**);

Use the **Handpiece contact area diameter** (mm) settings to define the diameter of the contact area (**2**);

Use the **Sleeve safety zone height** (mm) settings to define the sleeve safety zone height of the implant sleeve support (**3**);

Use the **Offset from sleeve** (mm) settings to define the offset between the sleeve and the inner surface of the implant sleeve support (**4**);

In this step, you define the pin position. (**5**) Select and refine the pin position by moving and/or rotating the pin (see 3D Exacto Instruction for Use);

Use the **Sleeve support diameter** (mm) settings to define the diameter of the pin sleeve support (**6**);

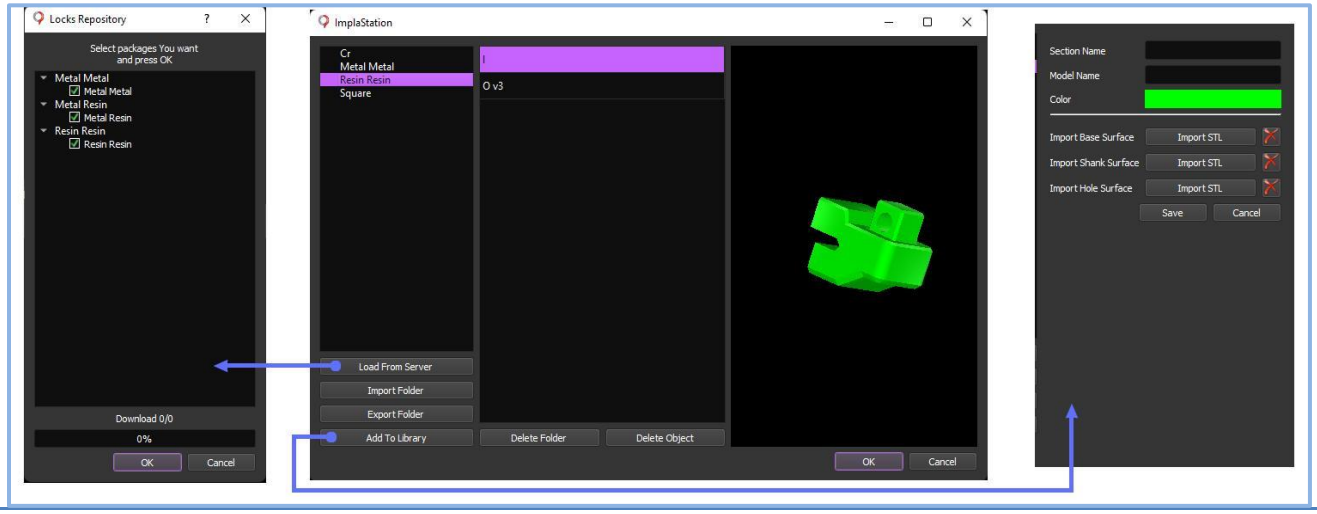
Use the **Handpiece contact area diameter** (mm) settings to define the diameter of the contact area (**7**);

Use the **Sleeve safety zone height** (mm) settings to define the sleeve safety zone height of the pin sleeve support (**8**);

Use the **Offset from sleeve** (mm) settings to define the offset between the sleeve and the inner surface of the pin sleeve support (**9**);

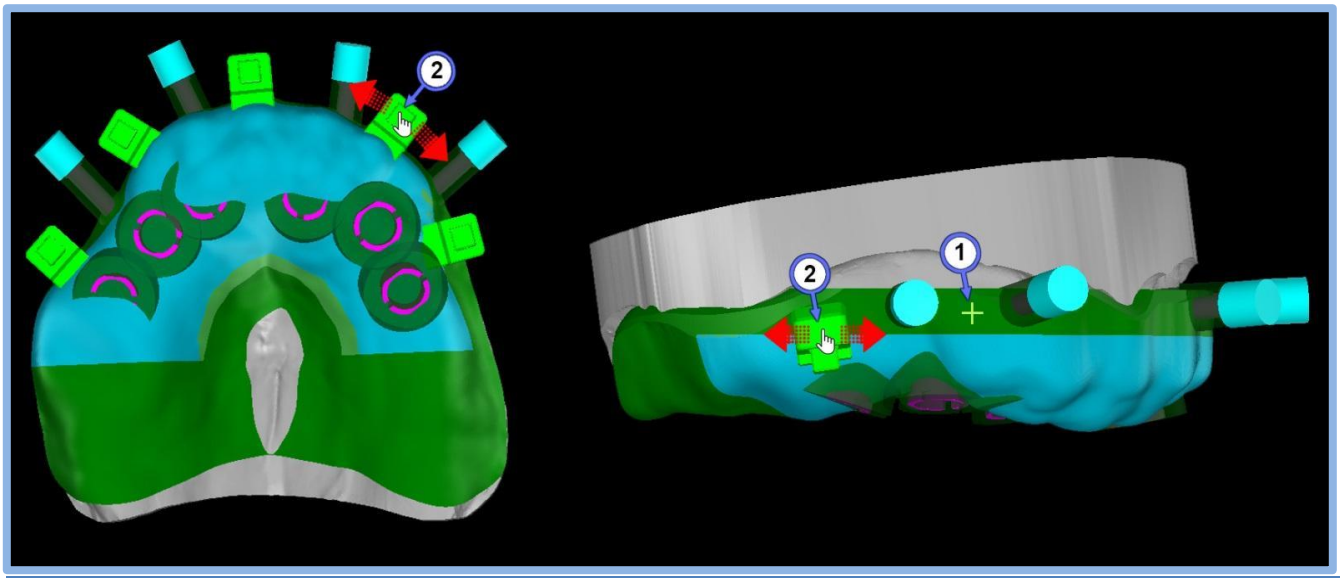
Select (**10**), place (**1**) and refine the lock position by moving the lock (**2**);

Using the **Locks** menu you can load the different types of locks.
Click “Load From Server”, select checkbox and click OK. Use “Add to library” button to add a custom STL file of the lock from your PC.
Enter the section name, model name and you can choose the color of the STL file.
Then import STL files of the base surface, shank surface and hole (the subtraction body) from a selected source using the "Import STL" buttons.



Lock library downloading

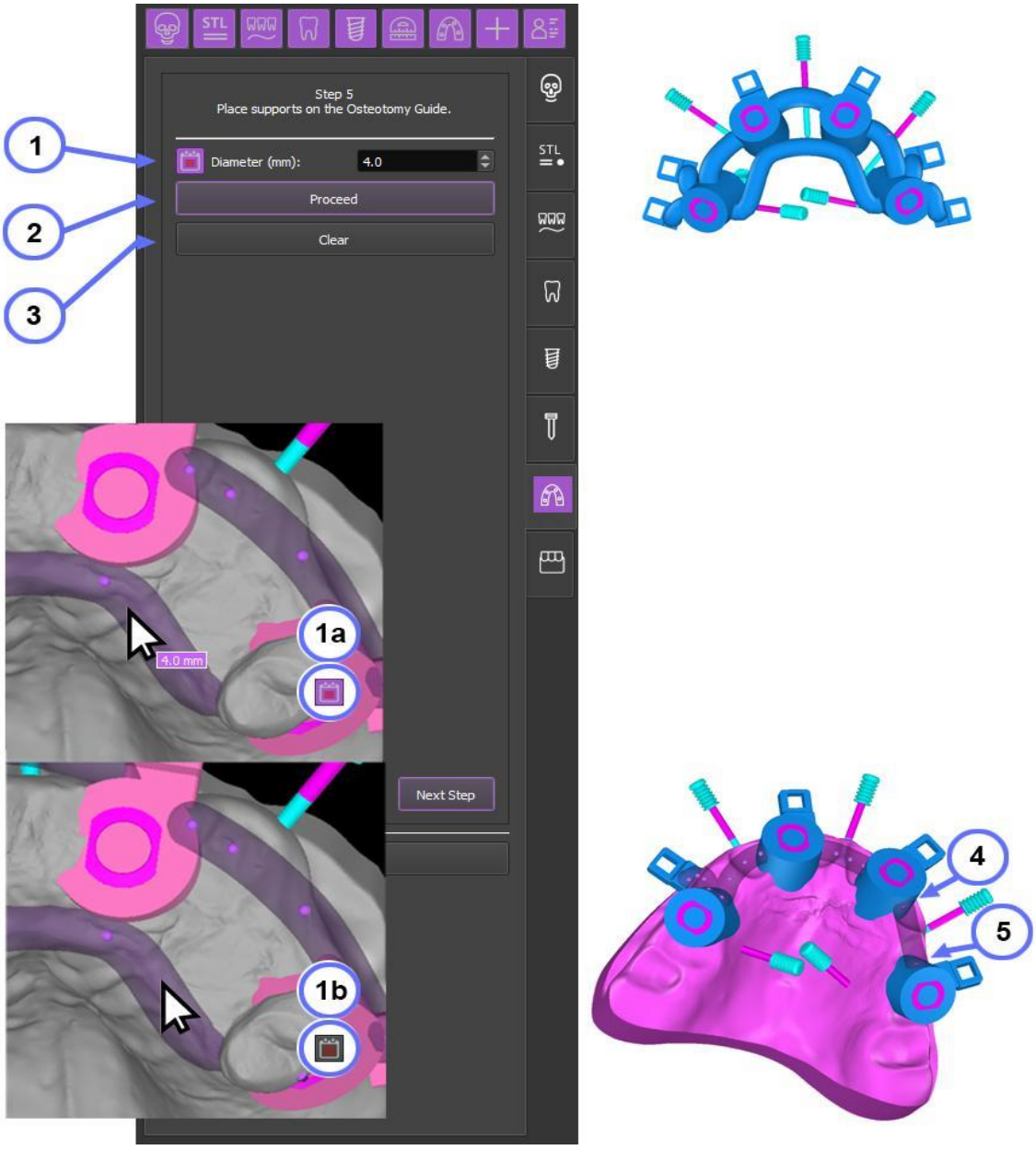
Move the cursor to the desired position on the fixation base with your mouse and left-click to set the lock (**1**);
Once you have set a lock position, you can still change its position by dragging it with your mouse. To delete a lock, click the right mouse button on the lock and then delete (**2**).



Refining the lock position.

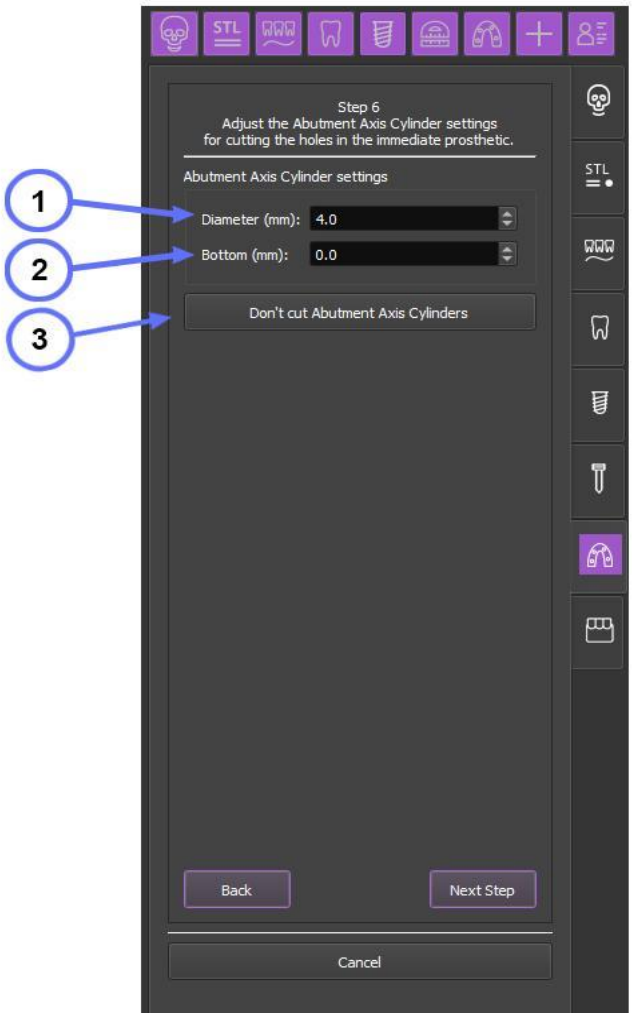
Step 5
Osteotomy Guide on the lock-supported bars design settings

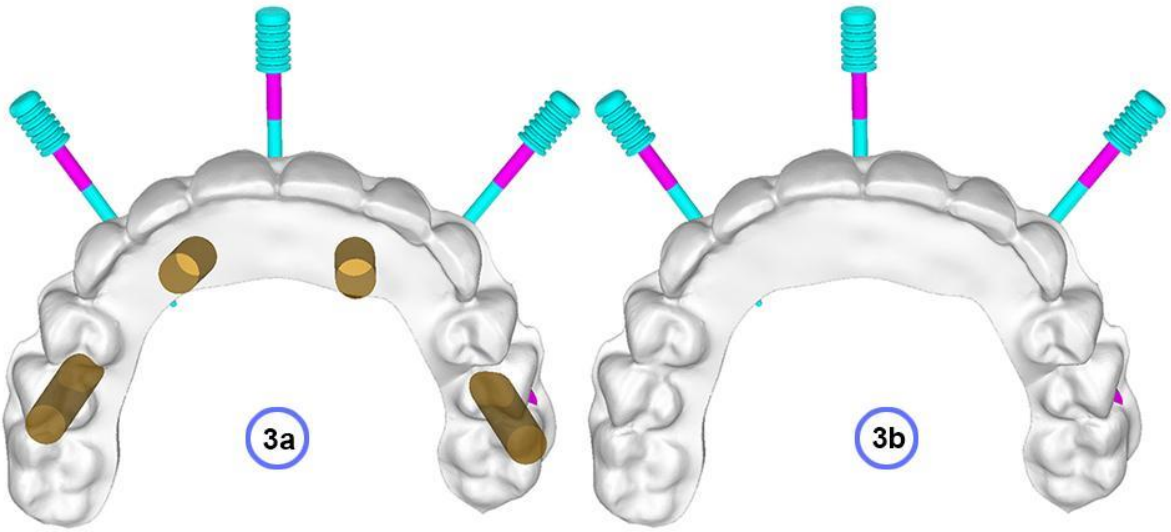
This step is only available if you have selected the option **On Bars** in the Osteotomy Guide Settings step (**Step 3**).
Use the **Diameter (mm)** settings to define the diameter of the lock-supported pontic.
Show/hide the visualization of the numerical value of the pontic diameter by clicking the checkbox (**1a, 1b);**
Once you have set all pontics for the osteotomy guide you can click **Proceed** and proceed with the workflow (**2);**
Click **Clear** to abort the support pontic placement step (**3**)**;**
Click a position at the sleeve support area and when you move your mouse away from this point, a purple solid bar is displayed between the pink point and the mouse cursor position. Click a position at the desired surface to set the bar at this selected area (**4, 5**). Now you can change the shape and diameter of the bar by dragging points and scrolling the mouse will.



Step 6
Temporary cylinder area settings menu

You can select to set up the diameter (**1**) and bottom (**2**) of the holes for the Multi-Unit Prosthetic Temporary Cylinders, around the abutment axis. To skip this step, click the "Don't cut Abutment Axis Cylinders" button (**3, 3a, 3b**).





Step 7
Carrier guide design settings

This step is only available if you have selected the option **Based on Carrier Guide** in the Carrier Guide Settings step (**Step 3**).
The transparent plastic pegs on the Carrier Guide deliver the Provisional Prosthetic in the proper position as planned.
Move the cursor to the desired position on the carrier guide with your mouse and left-click to set the peg (**a**);
Once you have set a peg, you can still change its position by dragging it with your mouse. To delete a peg, click the right mouse button on the peg and then delete (**b**).
Use the **Peg Height (mm)** settings to define the height of the peg (**1**);

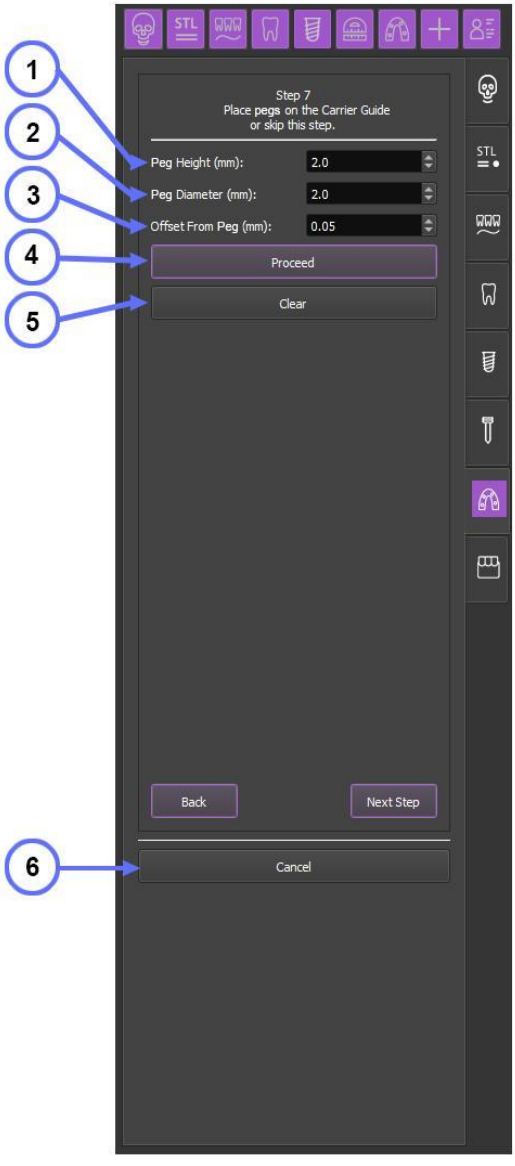
Use the **Peg Diameter (mm)** settings to define the diameter of the peg basis (**2**);

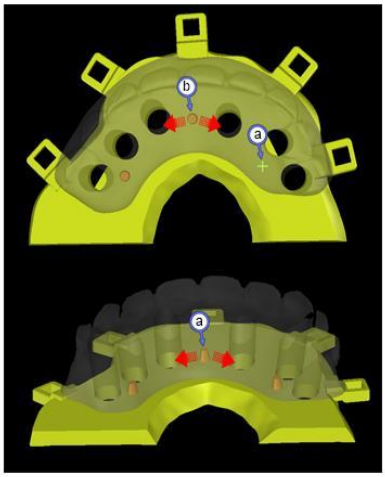
Use the **Offset from Peg (mm)** settings to define the offset between the peg and the inner surface of the Provisional Prosthetic (**3**);

Once you have set all pegs for the carrier guide you can click **Proceed** and proceed in the workflow (**4**);

Click **Clear** to abort the peg placement step workflow (**5**);

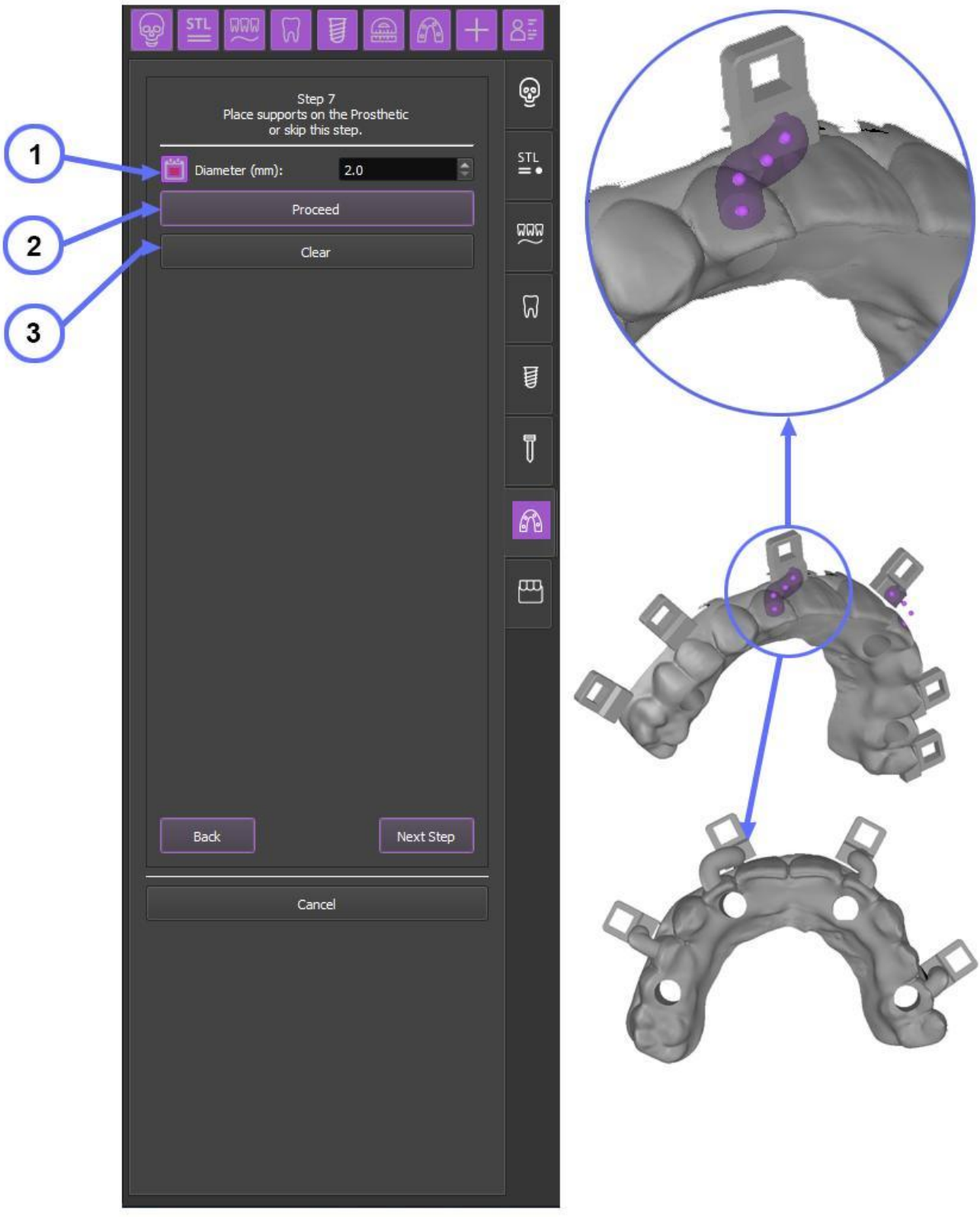
Clicking **Cancel** closes the window but discards all settings specified since you have started the step (**6**).





Step 7
Provisional Prosthetic on the lock-supported pontics settings

This step is only available if you have selected the option **On Lock-Supported Pontics** in the Prosthetic Settings step (**Step 3**). Use the same techniques as described in **Step 5**.



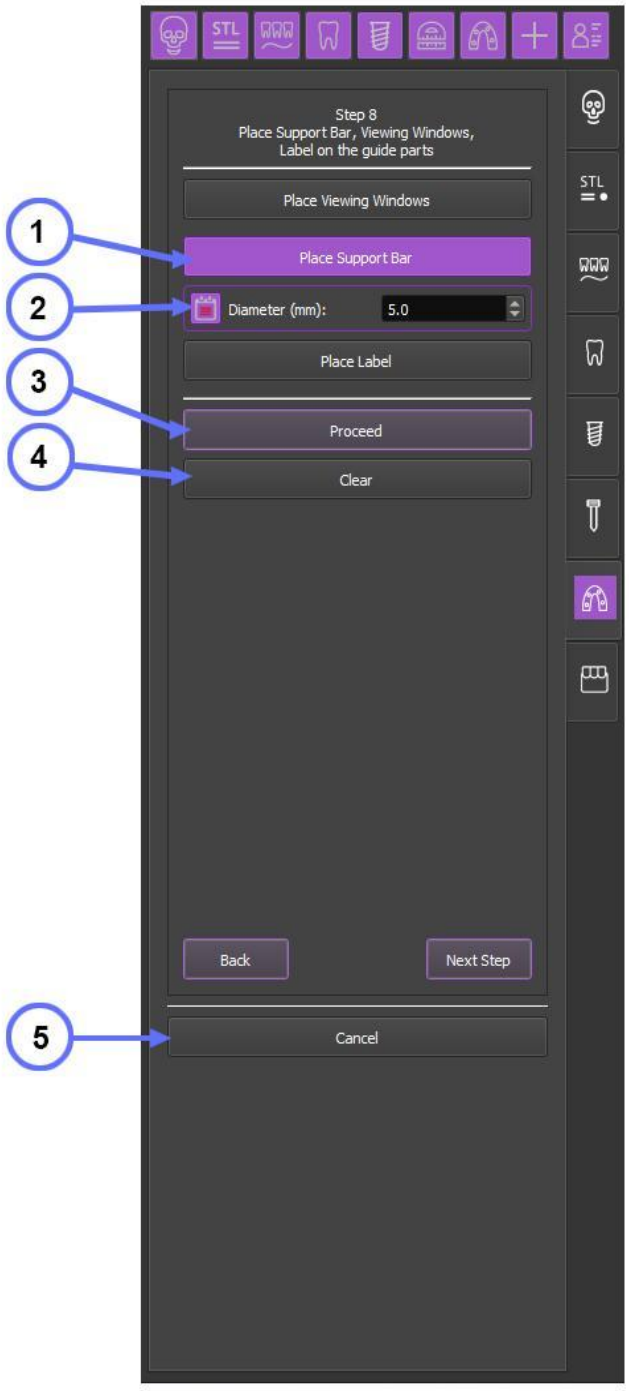
Step 8
Support Bar settings

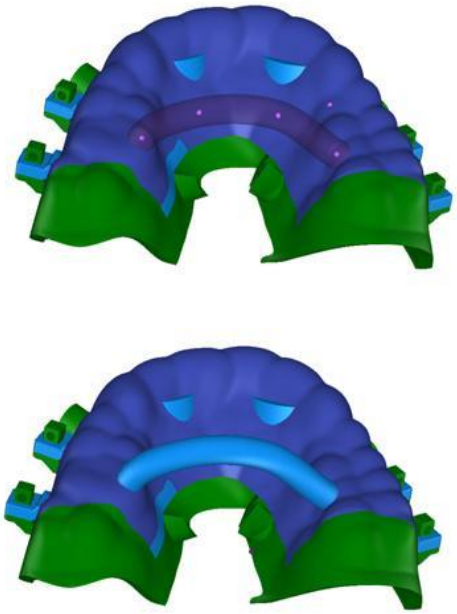
You can add a support bar to the surgical stackable guide's surface. Use the same techniques as described in **Step 5**.

Select **Place Support Bar** in the menu (**1**);

Click a position to the desired position on the stackable guide and when you move your mouse away from this point, a purple solid bar is displayed between the pink point and the mouse cursor position. Click a position at the desired surface to set the bar at this selected area. Now you can change the shape and diameter **(2)** of the bar by dragging points and scrolling the mouse will;

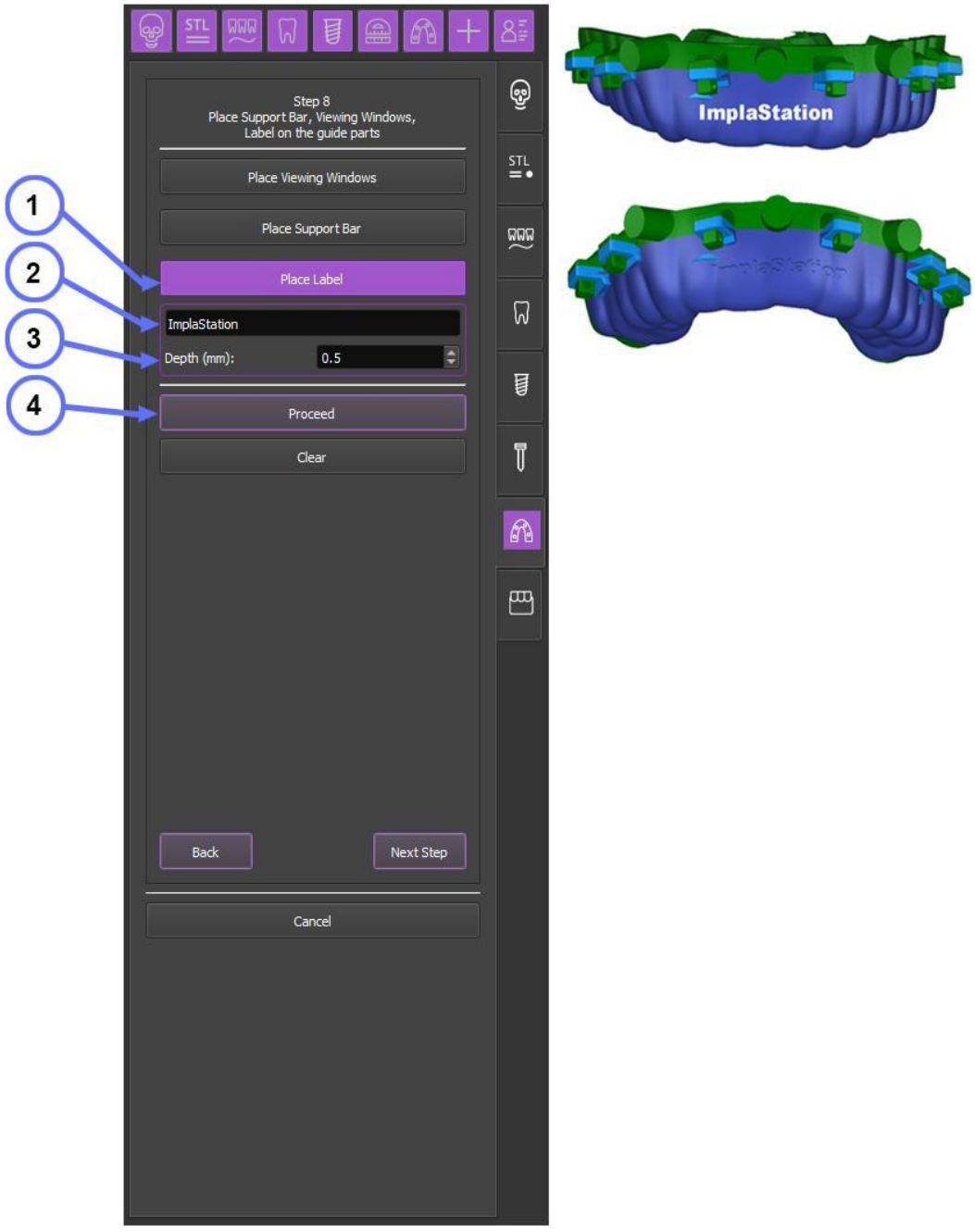
To delete a Support Bar, right-click on it and delete.





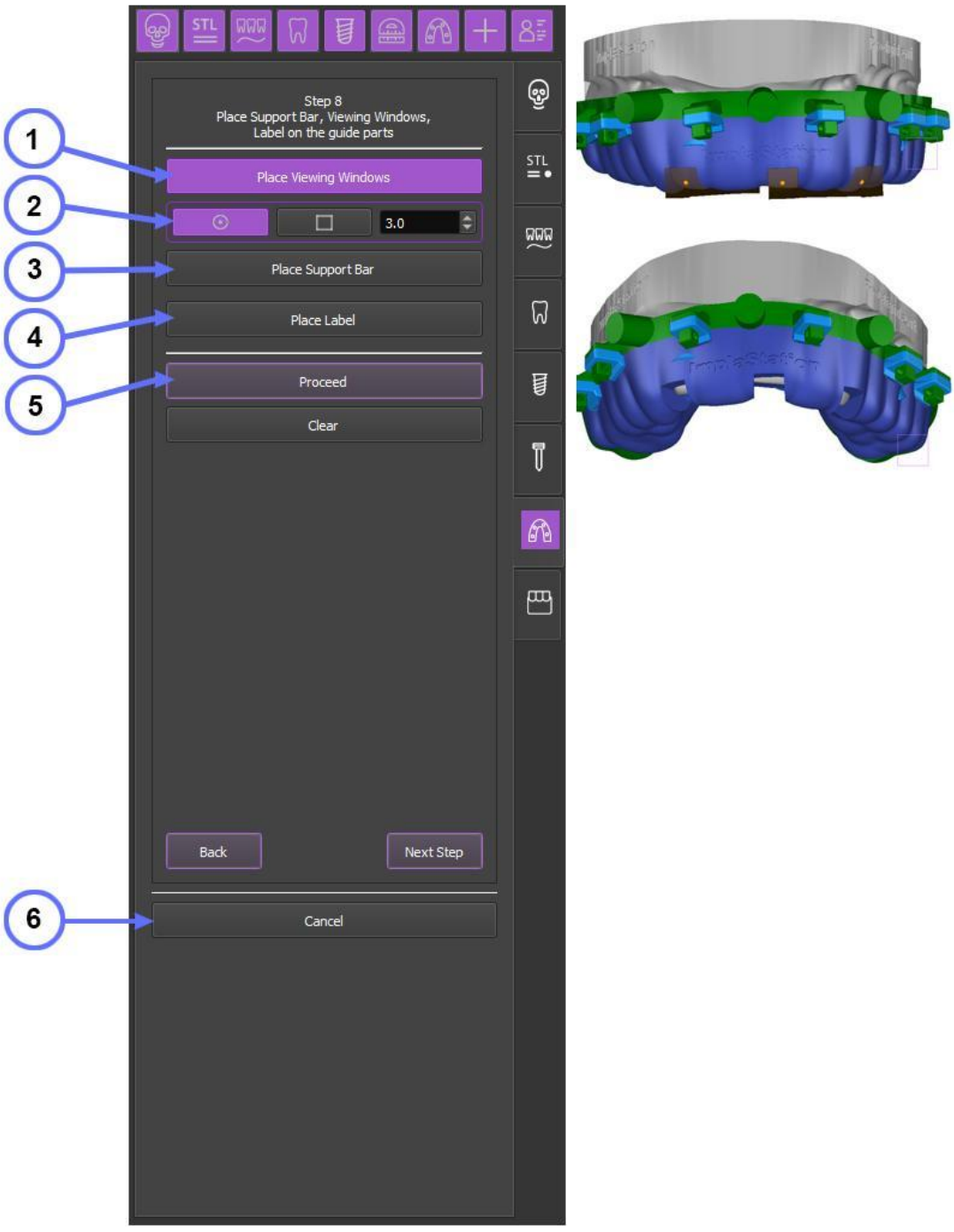
Step 8
Label text settings

You can add a text label to the surgical stackable guide's surface.
Figure 2.8.1.1 Label text settings.
Select **Place Label** in the menu (**1**);
Type the desired text into the text field (**2**);
Using the window below the text field, you can change the depth (**3**);
Move the text to the desired position on the stackable guide surface and click to set it, then proceed (**4**).
To delete a text, right-click on it and delete.



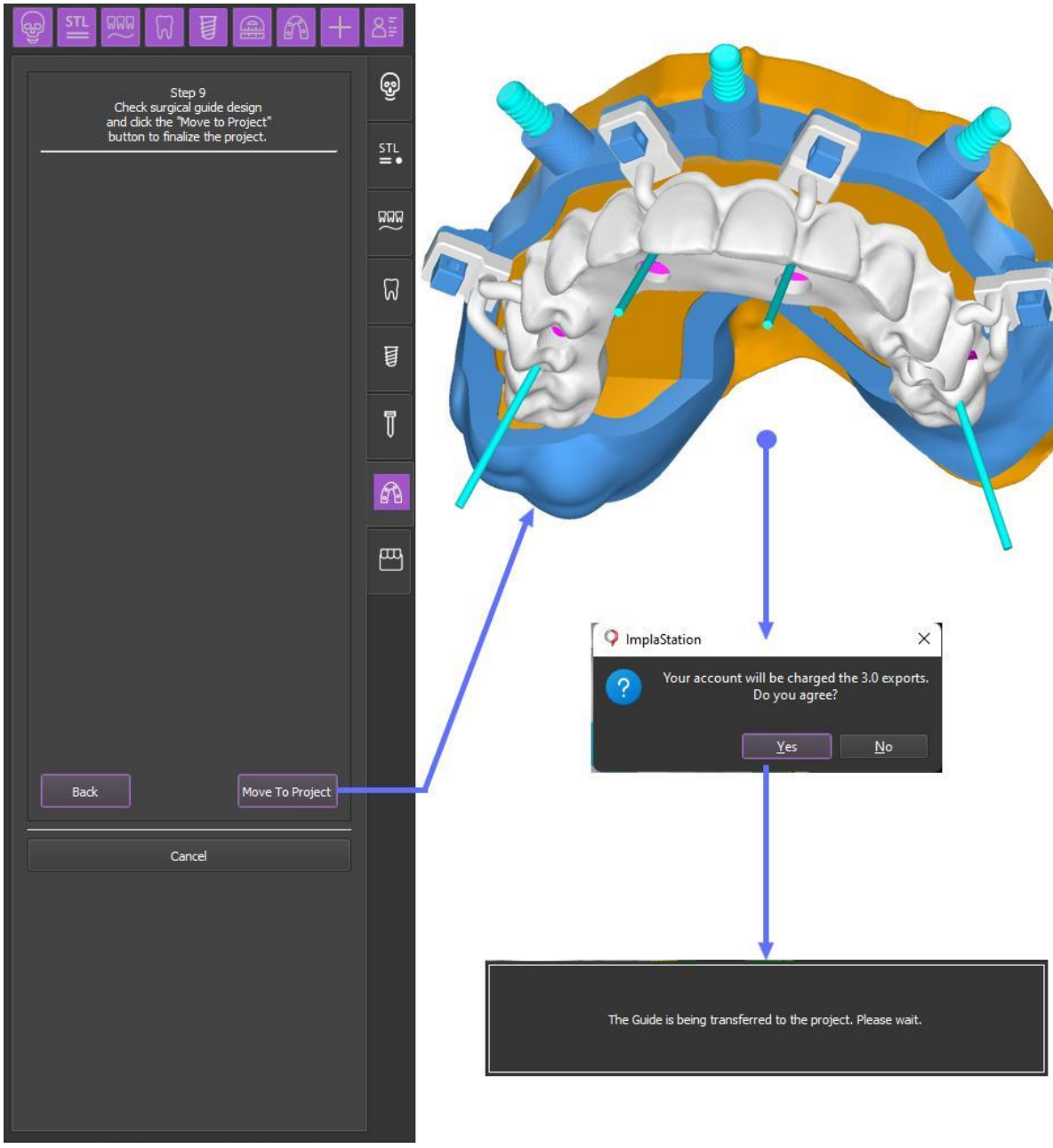
Step 8
Revision window settings

You can also add a revision window(s) to the surgical stackable guide's surface.
Select **Place Viewing Window** in the menu (**1**);
Move the window shape (**2**) to the desired position on the stackable guide surface and click to set it.
To delete a revision window, right-click on it and delete.



Step 9
Finalizing the Project

Planning result files (STL files and PDF files) are created in the planning result files generation and finalizing step. You can check the result of your planning in the views and finalize the project by clicking the Move to the project button or clicking Back if you are in doubt or not satisfied with the result.



Note! Your account will be charged the 3 exports.

DISCLAIMER

The 3D Exacto Stackable Guide software and all content found on it are provided on an “as is” and “as available” basis.

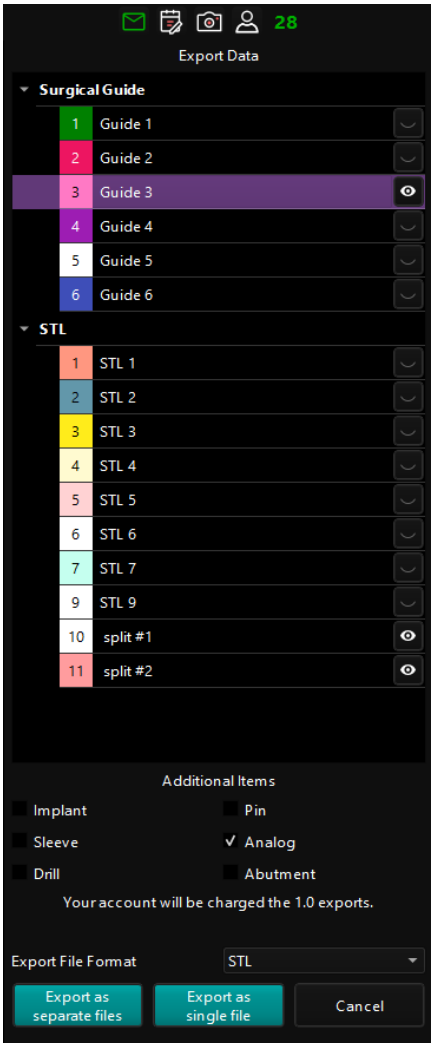
Prodigident Inc. will not be liable for any loss, whether such loss is direct, indirect, special or consequential, suffered by any party as a result of their use of the 3D Exacto Stackable Guide software.

Any downloading, uploading or editing of material on the 3D Exacto Stackable Guide software is done at the user’s own risk and the user will be solely responsible for any damage to any persons or loss of data that results from such activities.

ANNEX I

Export

The **Export** function allows you to save surgical guides and STL surfaces from 3D Exacto to your computer for further processing or manufacturing. The Export menu activates by clicking **Export** button in the menu bar at the top of the screen. The Export Data panel will open on the right side of the screen.



Step 1

Open the Export Panel.

Step 2

Select items to export

The Export Data panel displays two collapsible **lists**:

- Surgical Guide – all created surgical guides
- STL – all STL surfaces available in the project.

Click an item to select it. To select multiple items, use standard selection controls:

- **Ctrl + Click** to select multiple individual items
- **Shift + Click** to select a contiguous range of items

Step 3

In the Additional Items section at the bottom of the panel, activate the checkboxes for any associated elements you want to include in the export.

Choose the export file format. Use the Export File Format dropdown to select the desired format:

- STL
- OBJ

Hint!	Exporting Additional Items consumes 1 export credit per session, regardless of the number of items selected.

Step 4

If multiple items are selected, the software will prompt you to choose how to package the output:

- Export as separate files – each selected item is saved as an individual file
- Export as single file – all selected items are merged and saved as one combined file

Click the appropriate button to complete the export.

Labeling


About application
MD
ImplaStation v5.5.0
www.implastation.com
REF
IMPL-5.5.0
UDI
+D165IMPL1/$$+7550S
#
build017, Galimplant 3D Exacto
Prodigident, Inc.
123 Flinn str., Batavia, IL 60510, USA
SigmaTech Nordic ApS
Soumisvej 4, 1927 Frederiksberg C, Denmark
NUEVA GALIMPLANT, SLU
C/ Benigno Quiroga, 90, 27600, Sarria, Lugo, SPAIN
K210241
ARTG 530243
Windows macOS
Made in USA
Copyright © 2026 ProDigiDent

This Instructions for Use has been prepared according to the regulatory requirements of the FDA (United States), the MDR (European Union), and the TGA (Australia).

Galimplant

3D Exacto

NUEVA GALIMPLANT, SLU
C/ Benigno Quiroga, 90,
27600, Sarria, Lugo,
SPAIN

FDA 510(k) Cleared, No. K210241
ARTG Identifier 530243