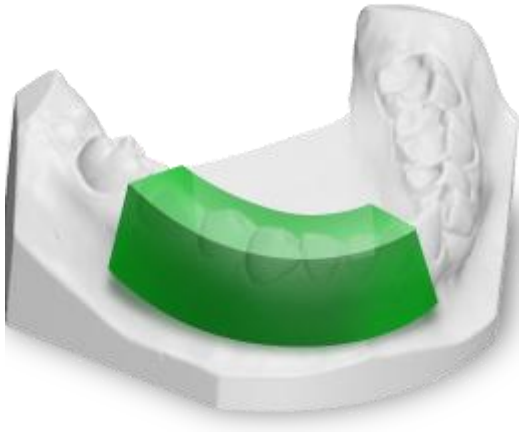

Appliance Designer 2012™

User Manual

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Introduction

Appliance Designer is a flexible program designed as a toolbox to create most of the types of orthodontic appliances. Its structure allows a maximum of flexibility in the design process:

- The basic tools can be applied in any sequence
- Any model can be exported for production (i.e. either a modified dentition model used as master or the appliance itself)
- Any combination of the active 3D models is supported (addition or subtraction)
- STL library items can be applied to any 3D models
- Predefined profiles and parameters are supported
- Full export and import of design template are supported to ensure productivity and consistency of designs
- Virtual articulation allows appliance design optimized for the patient's specific occlusion

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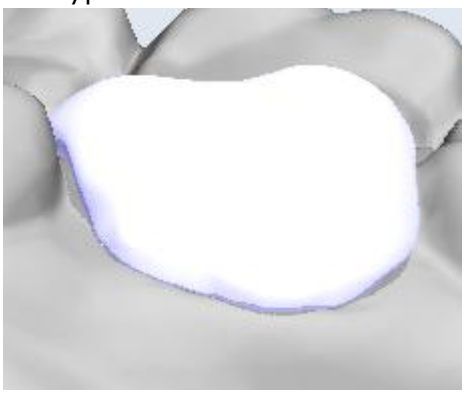
Appliance Designer Glossary

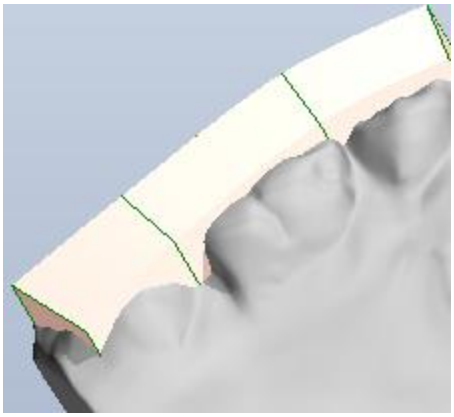
- **Part**: a part is an individual element of an appliance; it consists of components and features and will typically be manufactured individually. For example, one appliance can be made of two separate parts: one for the upper jaw and one for the lower which will mounted together.
- **Component**: a component is a building block for the appliance.

The 5 main component types are:

Shell: wrap surface with a fixed offset distance.



Bar: a 3D model based on a 2D profile, allows to apply predefined parameters and geometries.



Combine models: adding or subtracting 2 models geometries

Remove undercuts: fills undercuts to simulate e.g. waxing on models

Load model: load any STL to use it as visual guide, or combine it with the appliance

Features: indicate the specific design parameters applied to a component.
Additional features, which can be applied to any active 3D model (ie not just to components) are:

- **Additional Features**:

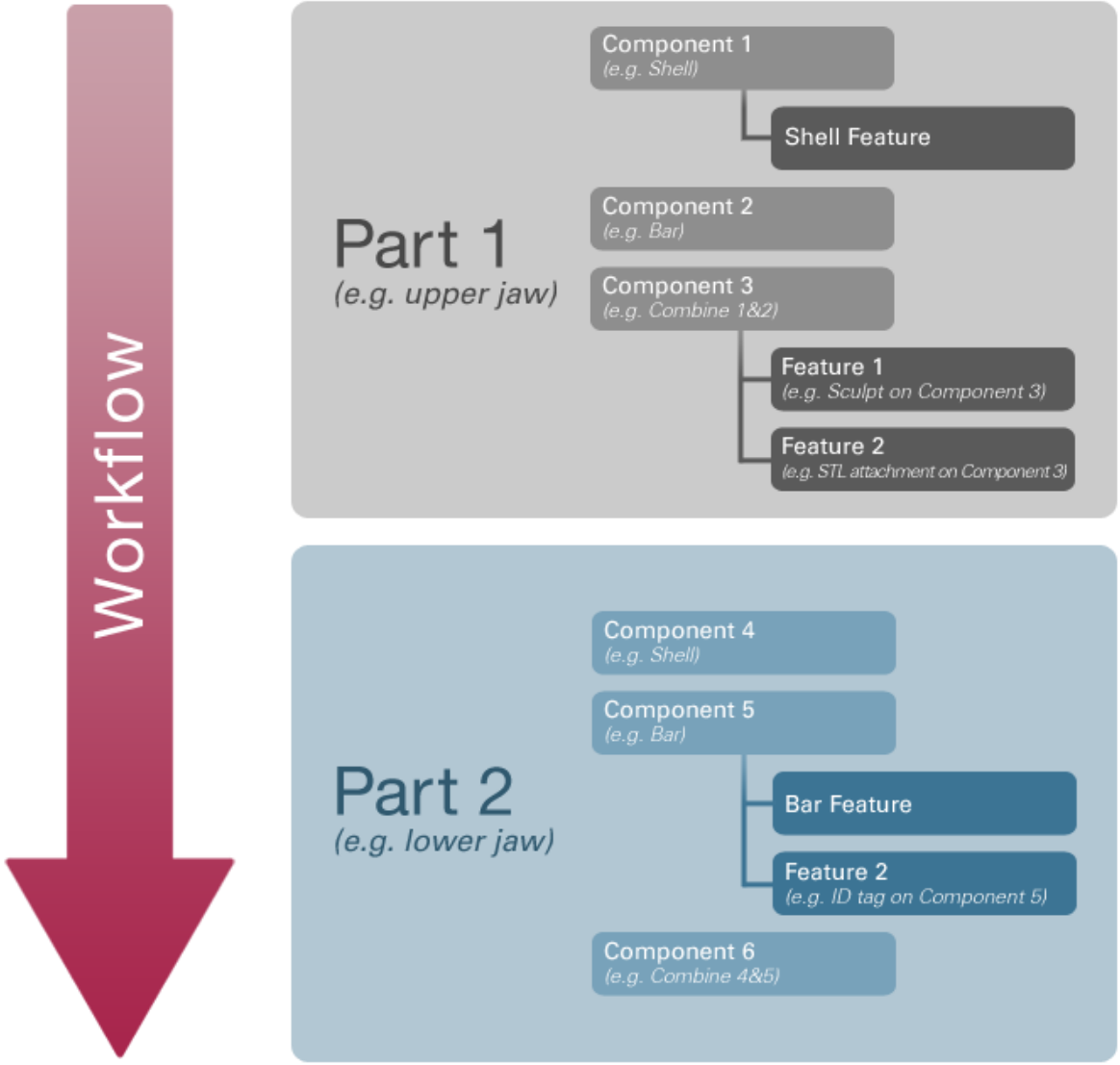
Modify model: a sculpting toolkit to add, remove material, or smooth the model's surface.

Attachments: allows to place an STL object from the library to the active 3D model.

ID tags: places a unique ID tag on the active 3d model to distinguish it.

Workflows: sequence of components and features used to create an appliance or part.

Template: a template allows to reapply a given workflow to another patient model set.



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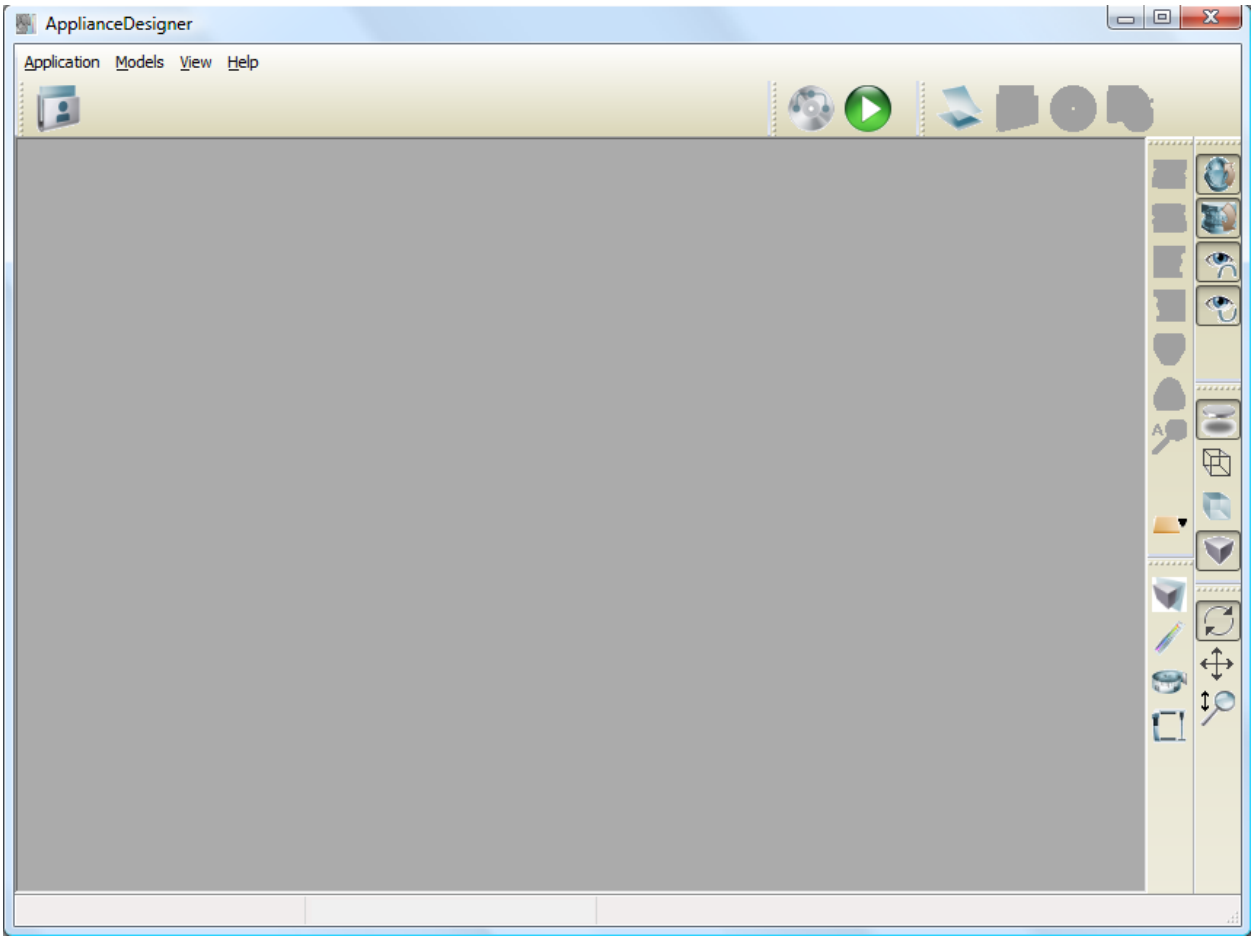
1 Getting Started

This section describes the principles of managing the models in ApplianceDesigner™ .

The chapter is divided into three parts: the first part is about a new patient entry, the second is for working on already created cases, and the third is for running an independent session with no patient database attachments.

1.1 Executing ApplianceDesigner

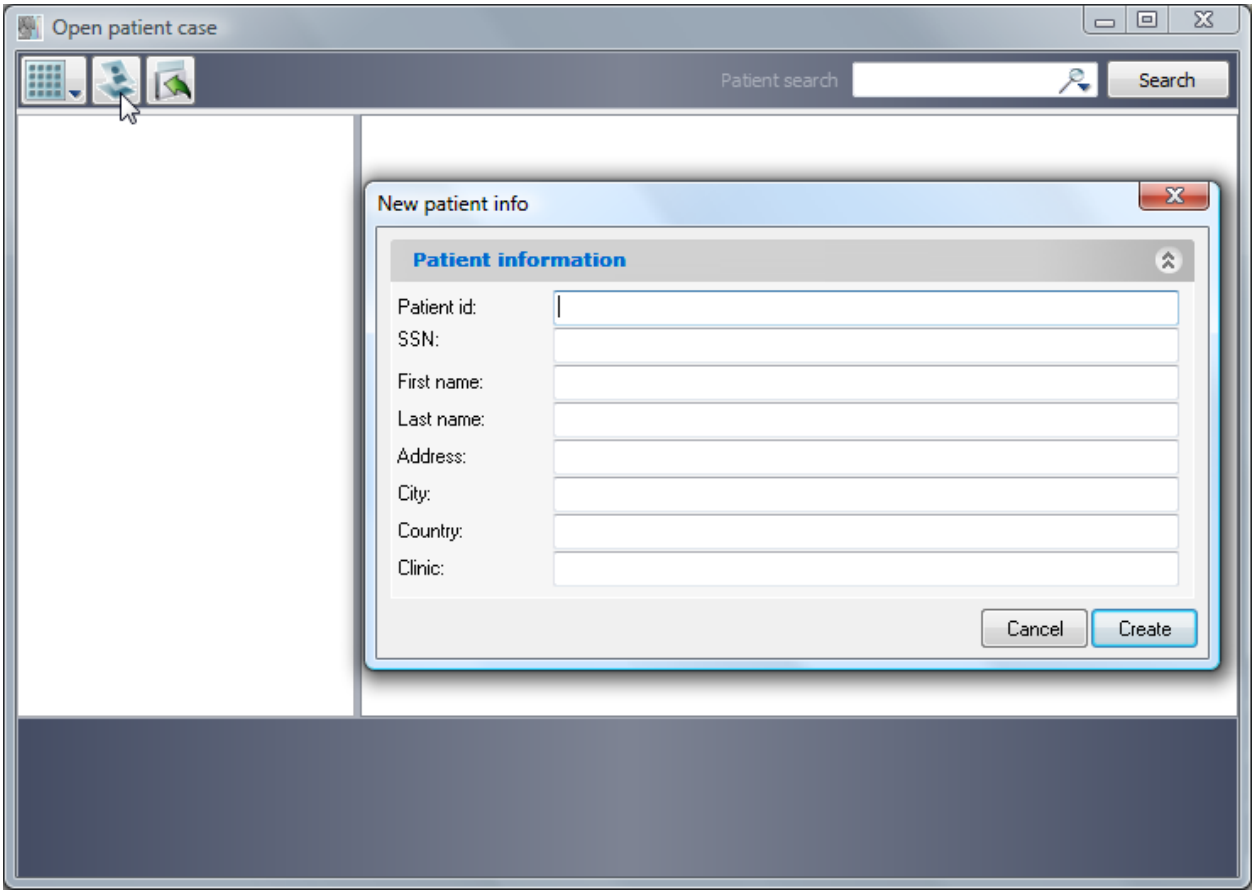
ApplianceDesigner ™ is started by double-clicking the ApplianceDesigner™ desktop icon  or via the Windows ™ Start menu: *Start->Programs->3Shape->ApplianceDesigner*. When the application starts, the following screen appears.



1.2 Creating a New Database Entry

1.2.1 New Patient

Before starting an analysis session, a patient database needs to be created. To create a new patient, click on the ***Patient model set***  button in the Main toolbar and select the ***New patient***  button. A *New patient info* form will appear.



1.2.2 New Model Set

The **New model set** step creates the set for the current patient by allowing you to fill in a dialog box. Click the ***New model set***  button to get the *New patient model set info* window:

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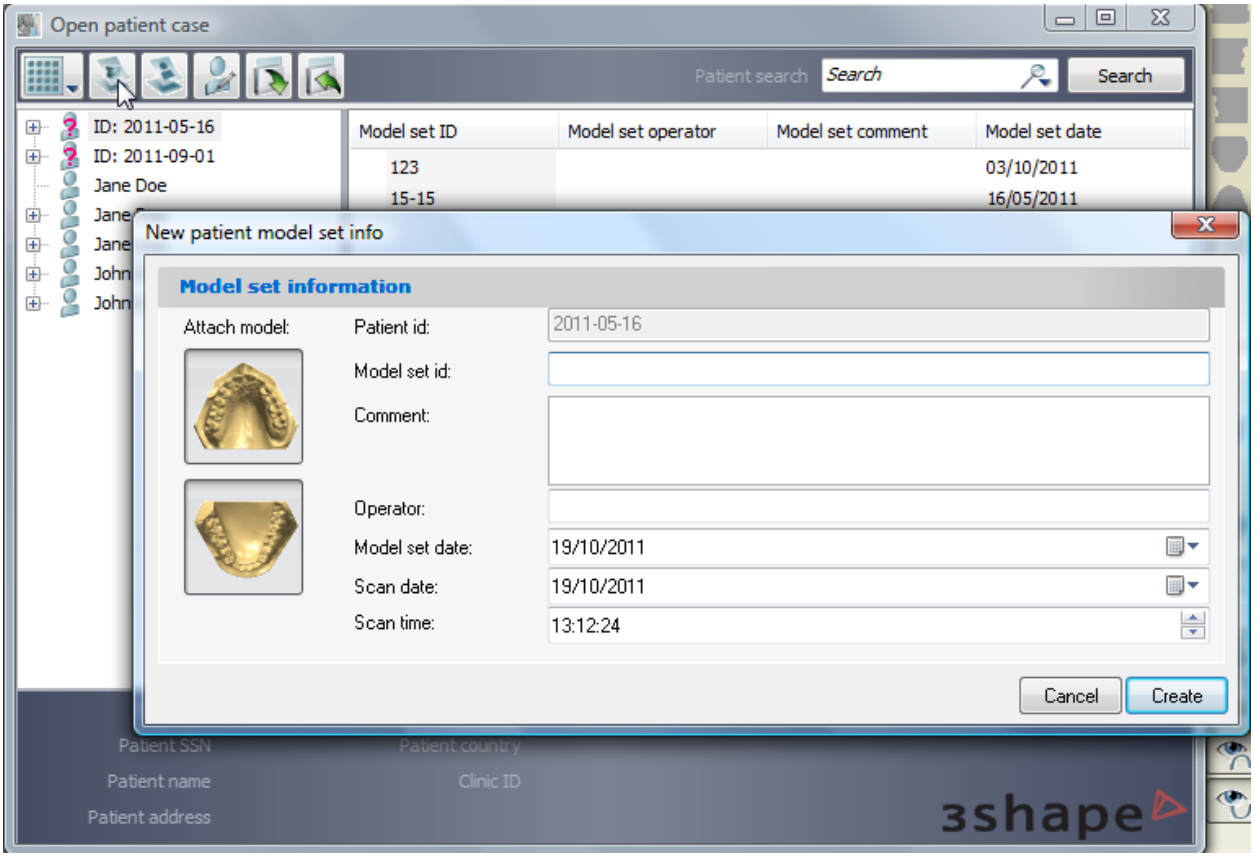


In the section **Attach model** you can select the new model set intended for import. Click ***Create*** to save the entered data. The description of the fields in the form is provided below:

Model set ID – Corresponds to the ID string that uniquely identifies the model set for a particular patient.
Comment – Any comments related to the order or the model can be added here.
Operator – Fill in the name or the code of the operator performing model set analysis.
Model set date – The date the model was created (the default date can be modified).
Scan date and Scan time – The Date and time the scan was taken (the default date and time can be modified).

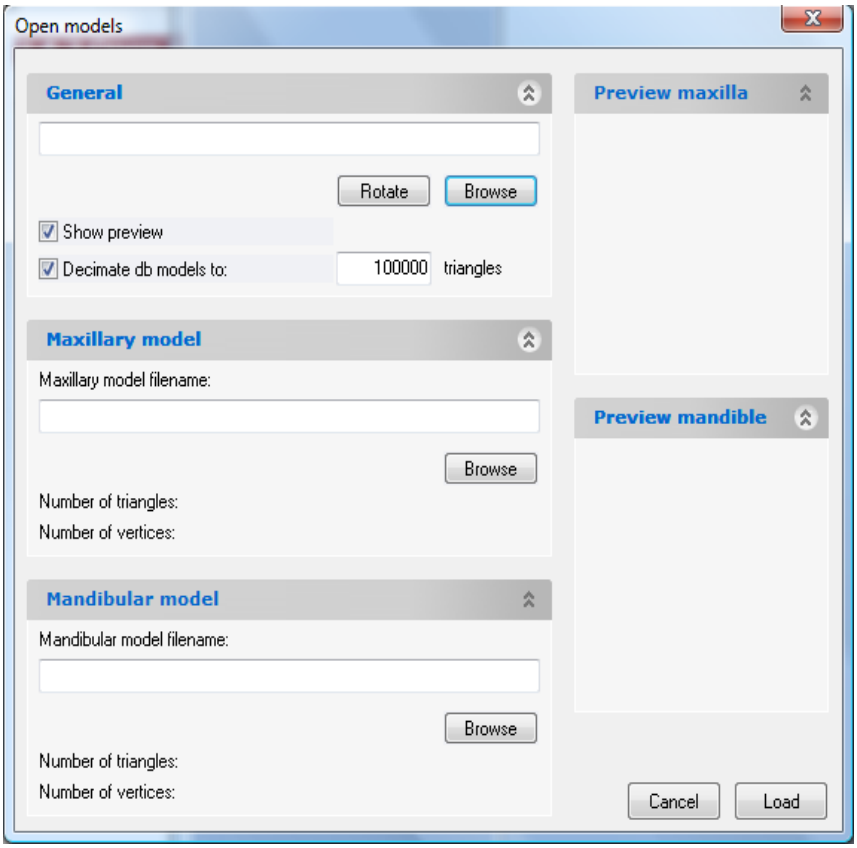
Note: The information specified in **New patient** and **New patient model set** appears at the bottom left corner of the session window for reference.

1.2.3 Open Models for Patient

Please follow the steps described below to open the existing models:

Step 1: **Choose the models for the form**

Select the ***Open models***  button in the Main toolbar. The *Open models* form appears:



General:
Select either of the 2 models to automatically locate the complimentary model and open the model set. They can be loaded in three formats: .**stl**, .**wrl**, .**dcm**.

Maxillary model:
Choose a maxillary model only, to load into the system.

Mandibular model:
Choose a mandibular model only, to load into the system.

Step 2: Load the models to the form

To browse for a model set in the Orthodontics Control Panel, use the Browse buttons under each of the blank fields. The default path to the folder with files of the patient database is C:\Program Files\3Shape\OrthoData.

- You may browse exclusively for the maxillary or mandibular models using the **Maxillary** model or **Mandibular** model fields of the window.
- If both parts of a model set are required, then it is possible to either select them using the option described above or to browse for any of the two models in the General field. The system automatically locates the pair model in the same folder to form a properly occluded double-cast for further inspection.

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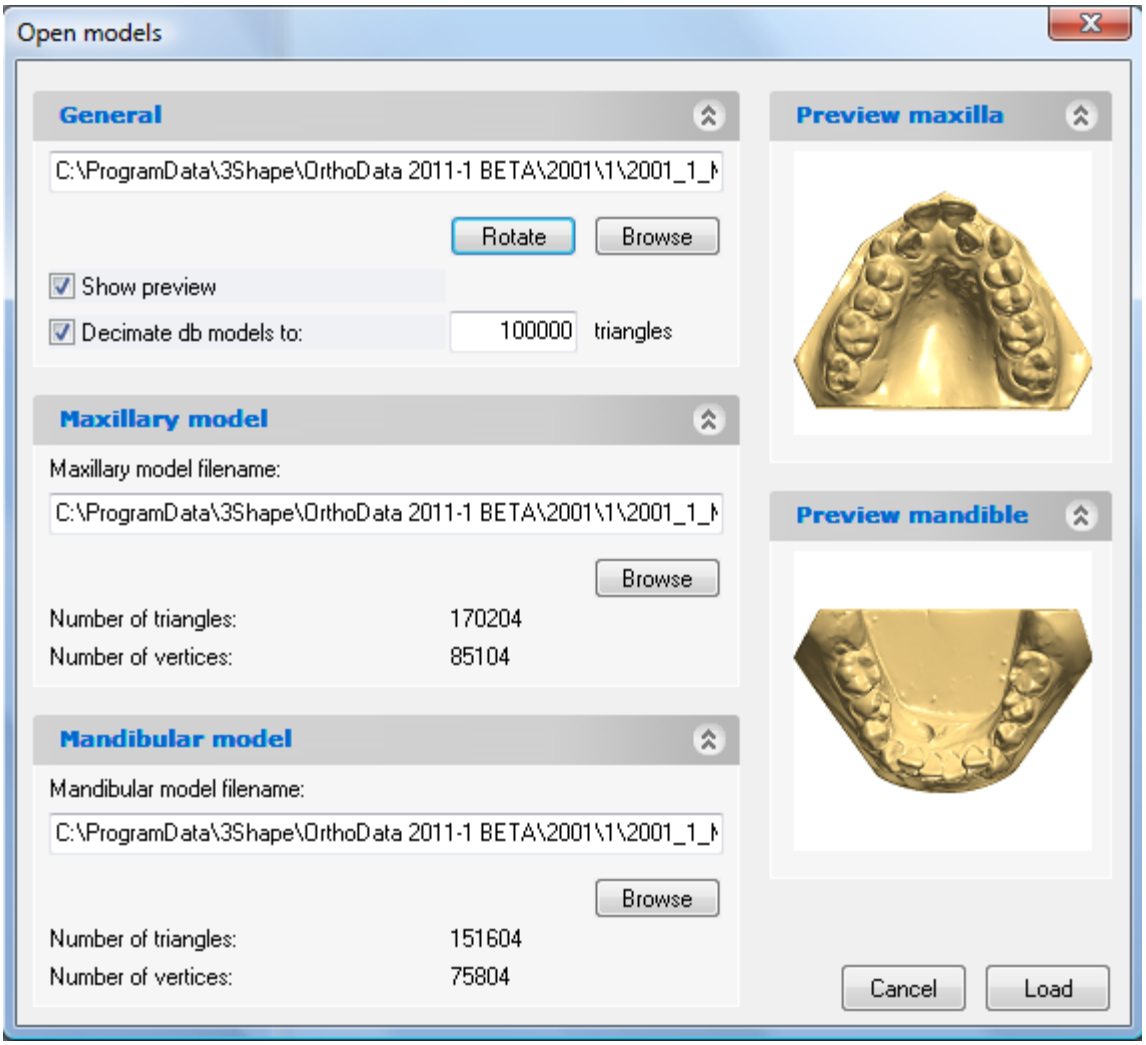


Step 3: Check ***Show preview*** to see the selections made in the preview boxes to the right of the window.

Step 4: Check **Decimate db models to**...**triangles** to decrease the number of triangles on the model to a predefined value.

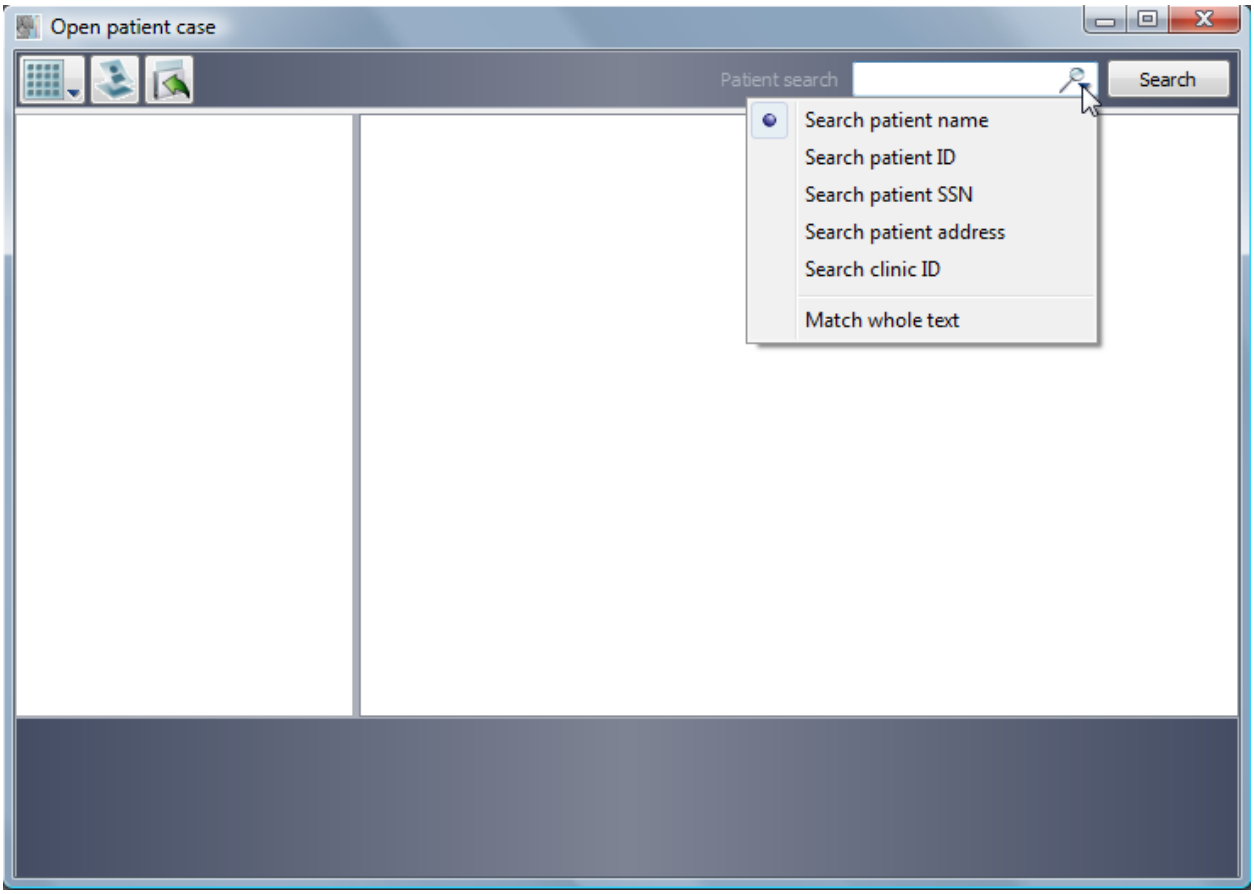
Step 5: Click ***Rotate*** to set the appropriate rotation angle for the relational positioning of the maxillary and the mandibular models. In the new window which pops up immediately after you push the Rotate button, move the Rotation degrees sliding bar (minimal value being -180°, maximal 180°) to bring the rotation angle to the necessary value. View the changes made in the preview boxes provided, to help you to control the maxillary, mandibular and the occluded modes rotation.

Step 6: Click ***Load*** to display the selected model(s) in the session window and start the analysis session.

1.3 Returning to the Database Entry

1.3.1 Patient Model Set

Another option to start analysis session is to open an existing patient model set by clicking the ***Patient model set***  button in the Main toolbar. Then the empty *Open patient case* window pops up suggesting to search for a patient.



To view model sets for inspection, search for them using the search bar in the right corner of the window, which includes various criteria to search with for the necessary patient model set(s) (e.g. patient's ID or address).
To see all available model sets set in the Control Panel, click ***Search*** leaving the search bar blank. A list of patients sorted by their IDs will appear under **Patient Selection**, which would offer model set(s) for further inspection.
The system employs several indications displayed in the main window to help you look for the necessary model sets in the catalogue:

-  Clicking on one of the patient ID (name) in the catalogue will display the ID, operators, comments, model set dates and scan dates associated with all the model sets for the patient ID selected.

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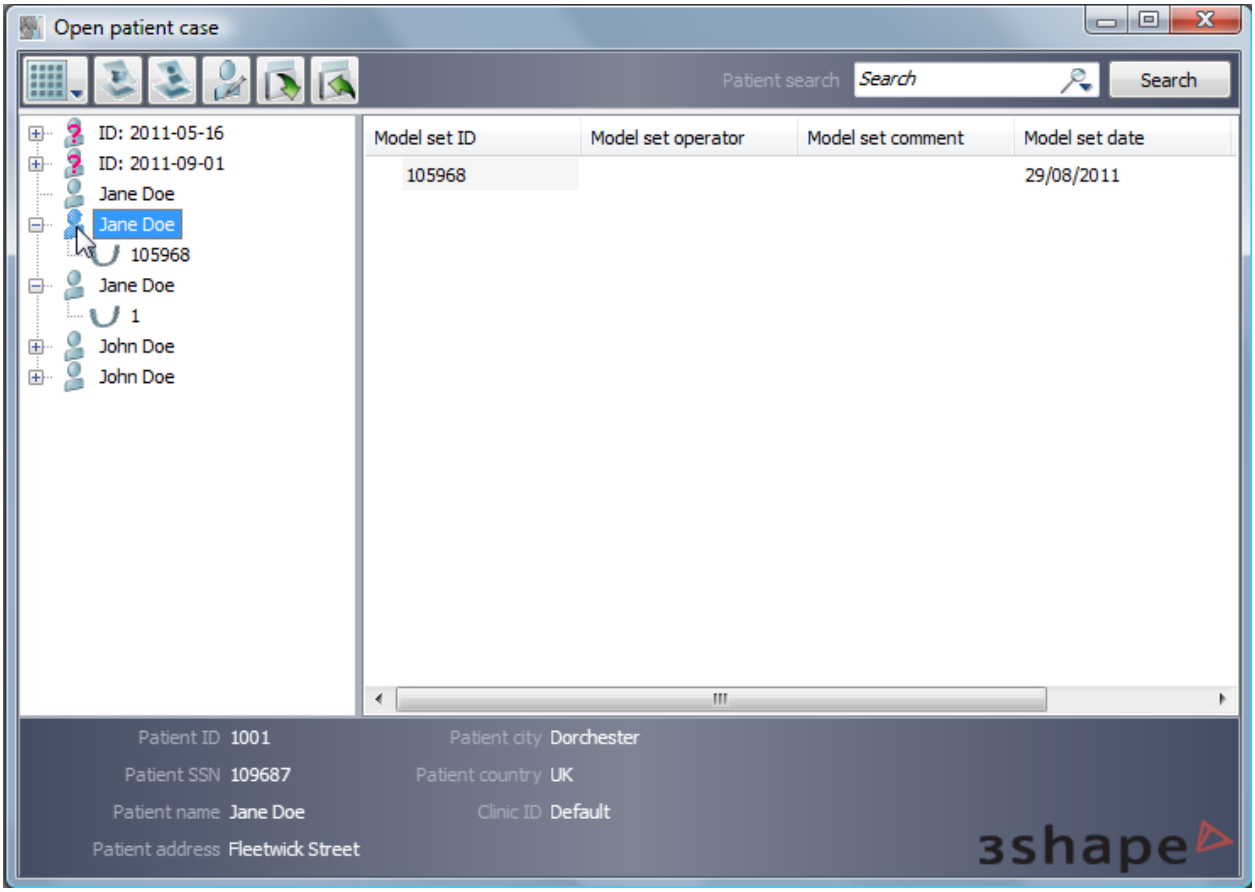
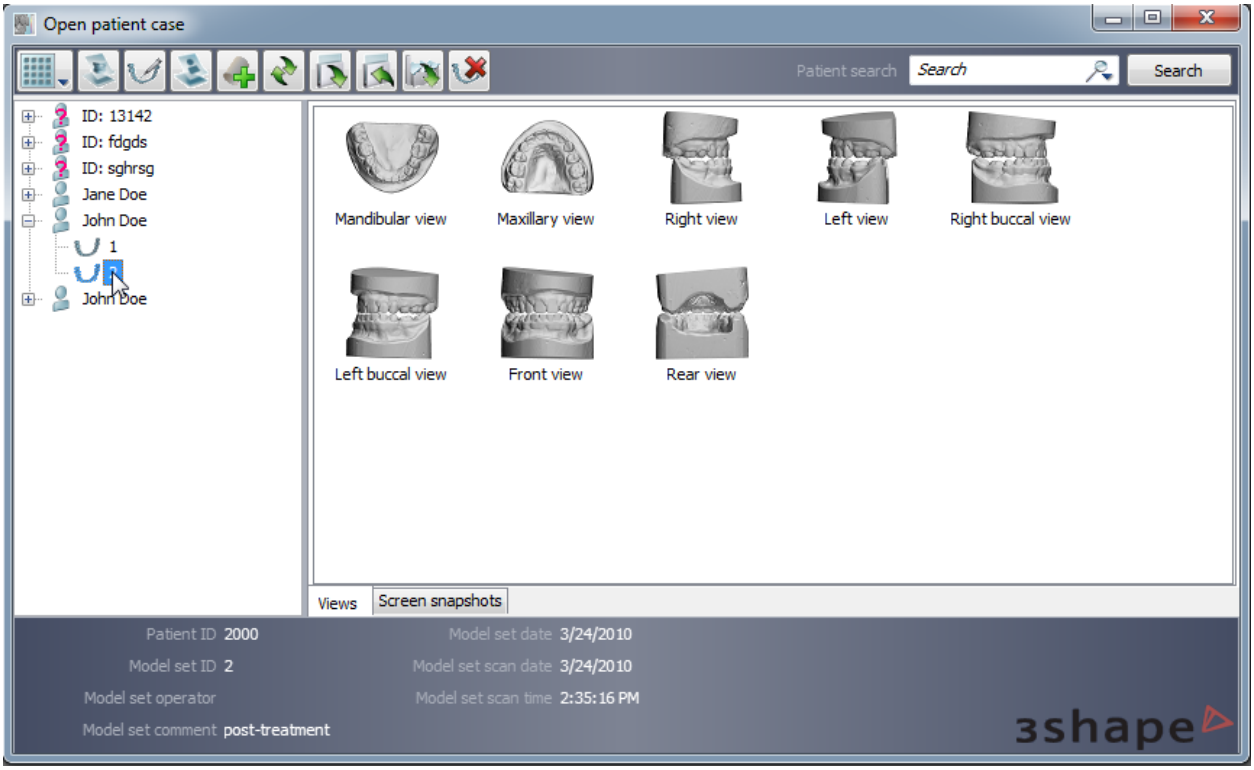
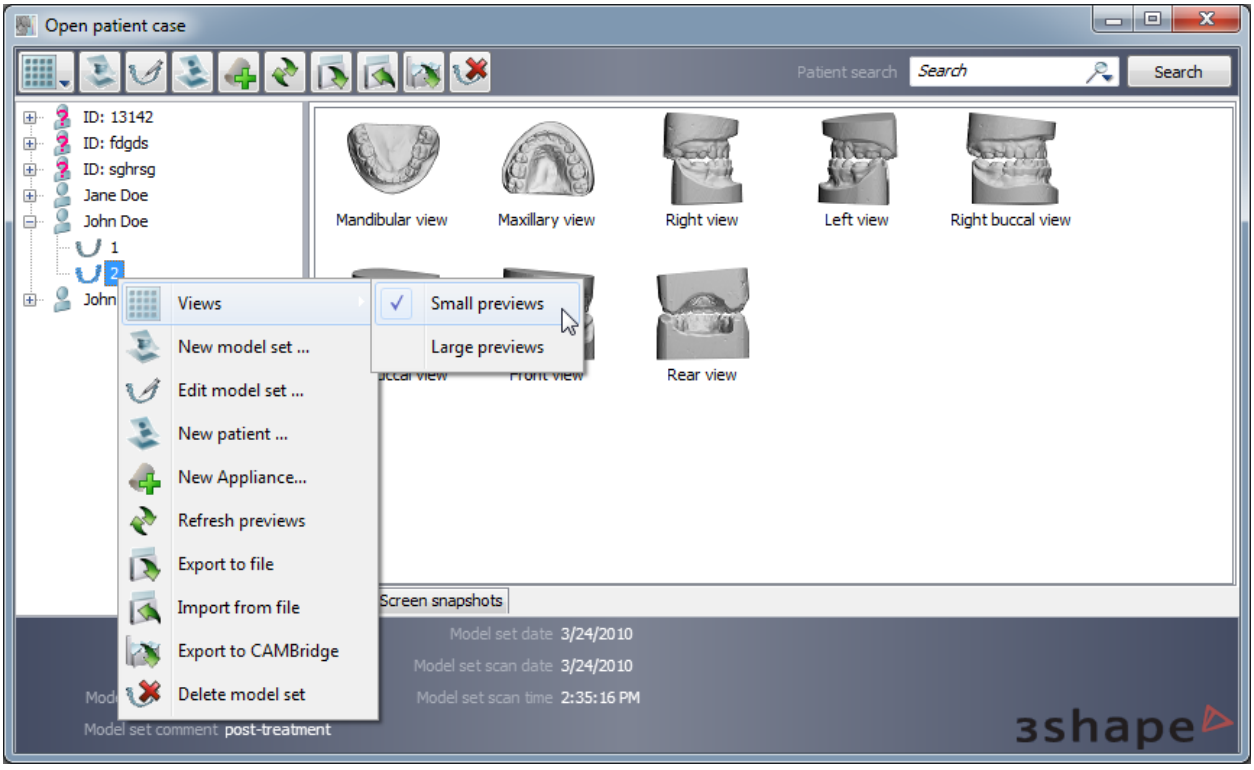


- Unfolding the patient model set tree further and clicking on one of the model IDs (e.g. 2), representing the selected model set, will display seven views corresponding to this particular model.



Note: There are two preview options available at this stage: small previews and large previews. The size of the large preview images can be set in the Ortho Control Panel. Click on the ***Views*** button on the top toolbar to select the most convenient viewing option.



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Throughout advancing to the last node of the selection tree, the system assists you in tracing the order selection, as well as in managing the active selection with available options. There are two sections in the window which perform this task:

- The dark grey field at the bottom of the window summarizes the information related to the active selection.
- The top toolbar window modifies its content according to the active selection:

a) While you are at the **Patient selection**, the only option available at this stage is:


New patient creates a new patient in the patient database (see the chapter New patient).


Import from file button permits to import models from other folders.

b) Selecting a patient model set adds tree buttons:

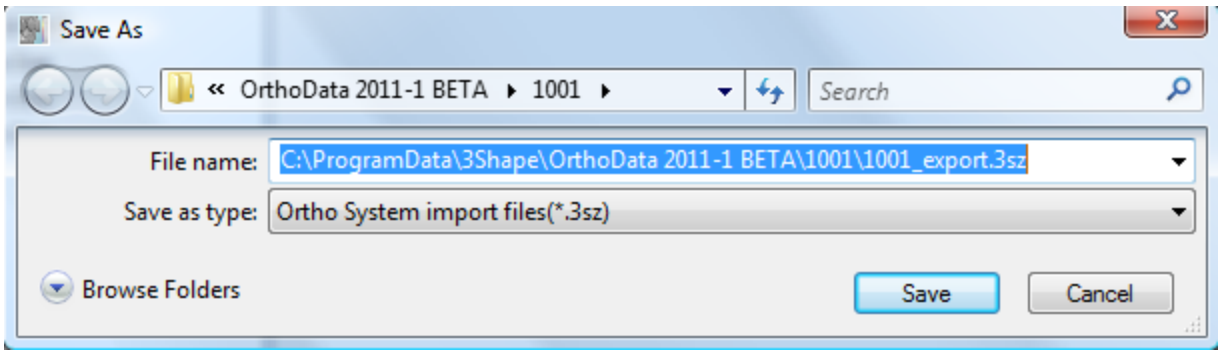

New model set allows you to create a new patient model set for the selected patient (see the chapter New model set).


Edit patient options allows you to open the existing patient information form of the selected patient for editing. Click ***Change*** to save changes.

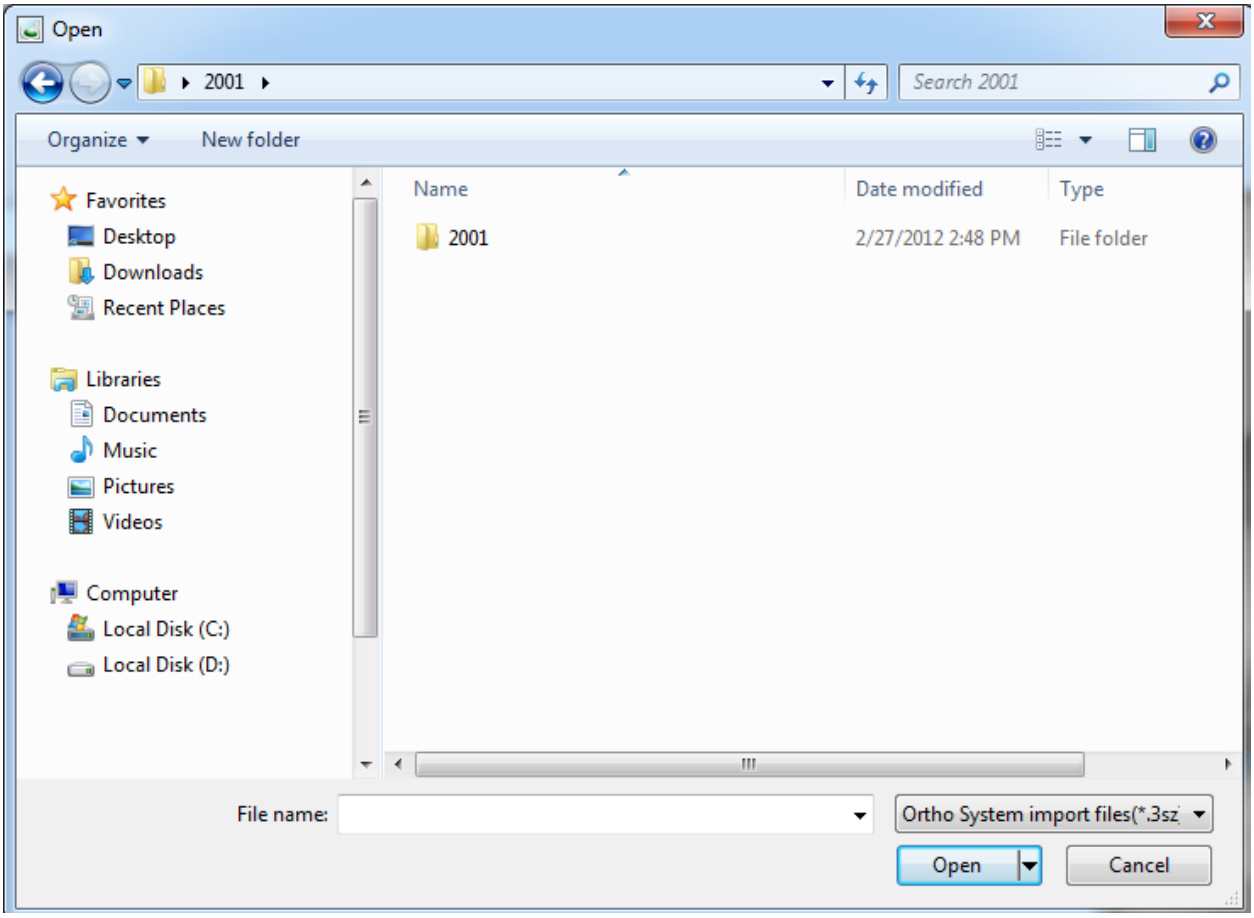

Export to file allows you to save the selected file in the selected folder.


Delete patient allows you to remove the selected patient from the list.

When you select the ***Export to file***  button you will open the window, where you can select the folder where it will be exported to. The format for the exported files is .3sz.



When you select the ***Import from file***  button you will open the window, where you can select the folder where the file will be imported. The format for imported files is .3sz.



c) Selecting a patient case adds two new buttons designed to help with managing an individual model set:

 ***Open model set*** button produces just the same effect as a double left-click on the selected model, i.e. it submits the selected model to the analysis process opening it in the session window.

 ***Edit model set*** button opens the existing patient case information form and suggests editing the fields which were previously filled in. Click ***Change*** to save changes.

 ***New Appliance*** button allows you to create appliance for the selected patient case.

 ***Refresh*** previews option refreshes visualization of the settings changed in the Ortho Control Panel.

 ***Export to CAMBrigde*** lets you open the preview of the selected model.

 ***Delete model set*** button removes the selected model set from the list.

When you click the ***New Appliance***  button the dialogue window appears.

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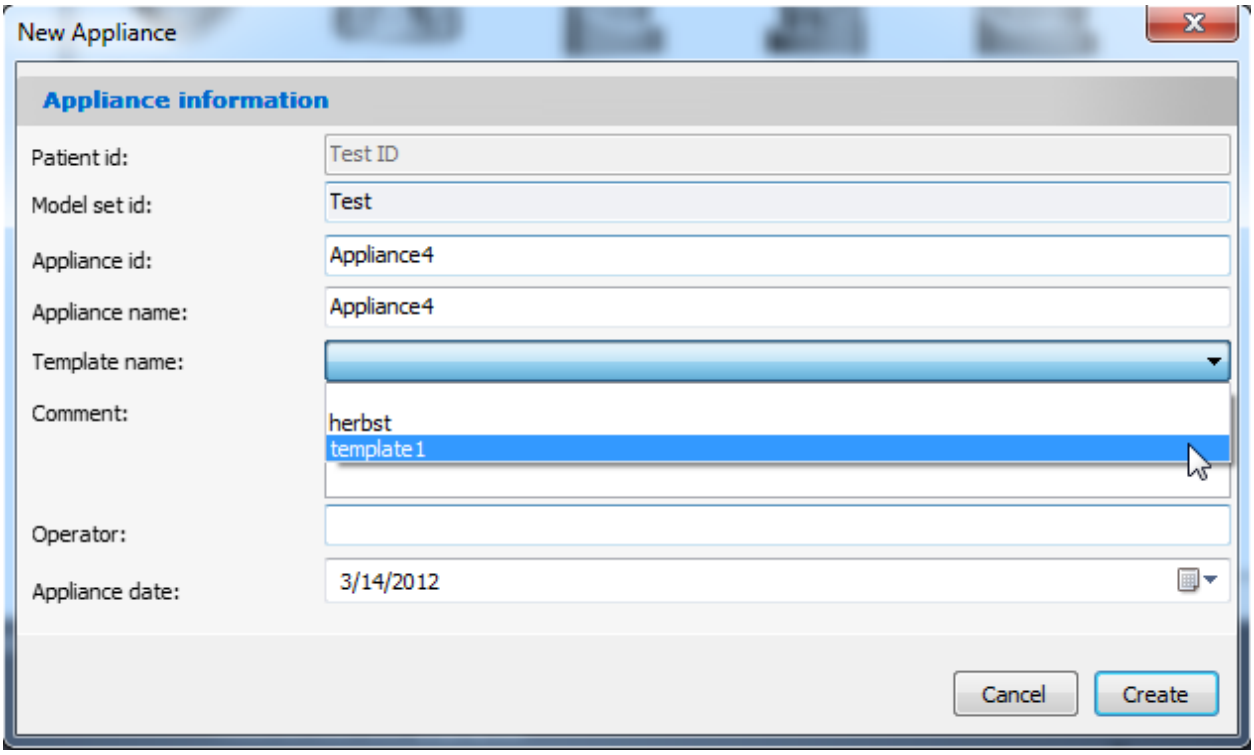
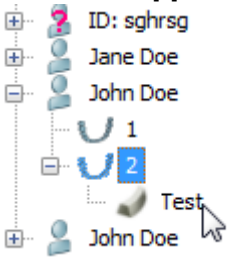
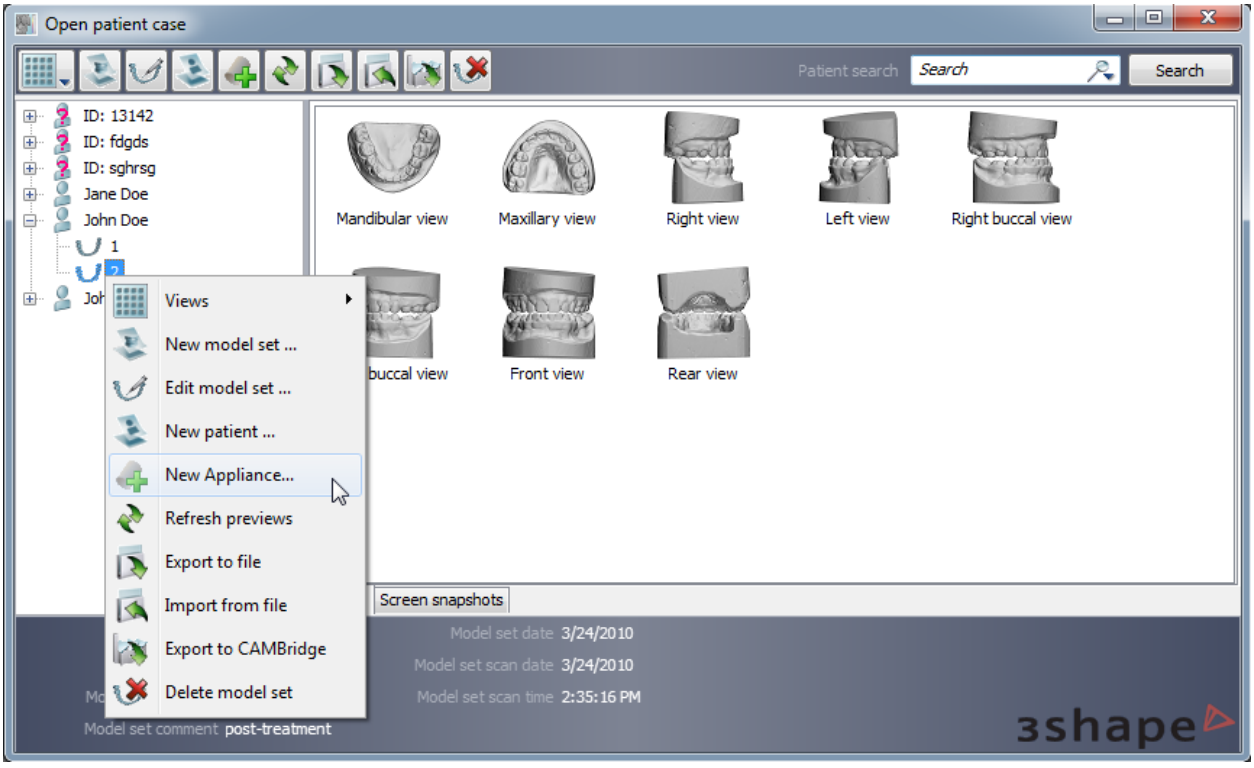


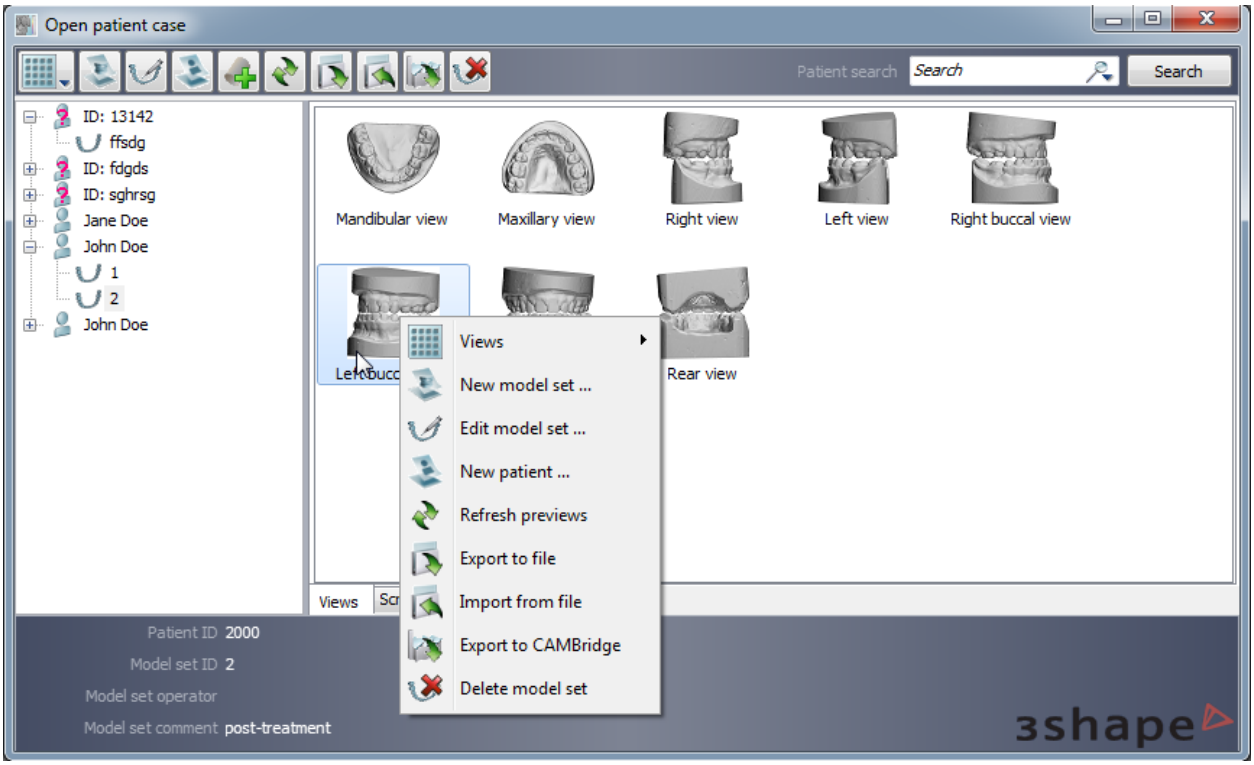
It is possible to select the template from the drop down menu. All the templates imported to the programme will be displayed here.
Enter the **Appliance id** and the **name** and click ***Create*** to proceed.



After creating the new appliance you can observe it in the *Open patient* form list. Right-click the **Appliance** icon select ***Modify appliance info***... button from the popup menu to edit the appliance information or ***Delete Appliance*** button to remove the appliance.

When you select the ***Export to CAMbridge*** button you can observe the preview of the selected model in the *Model preview* window.
All of the options described above may also be accessed by right-clicking on the active selection label (top) or image (bottom):





1.3.2 Open Model Set

In order to complete this step, it is required that there is a patient case created (see the chapter New Patient). Having opened a patient model set in the session window (see the

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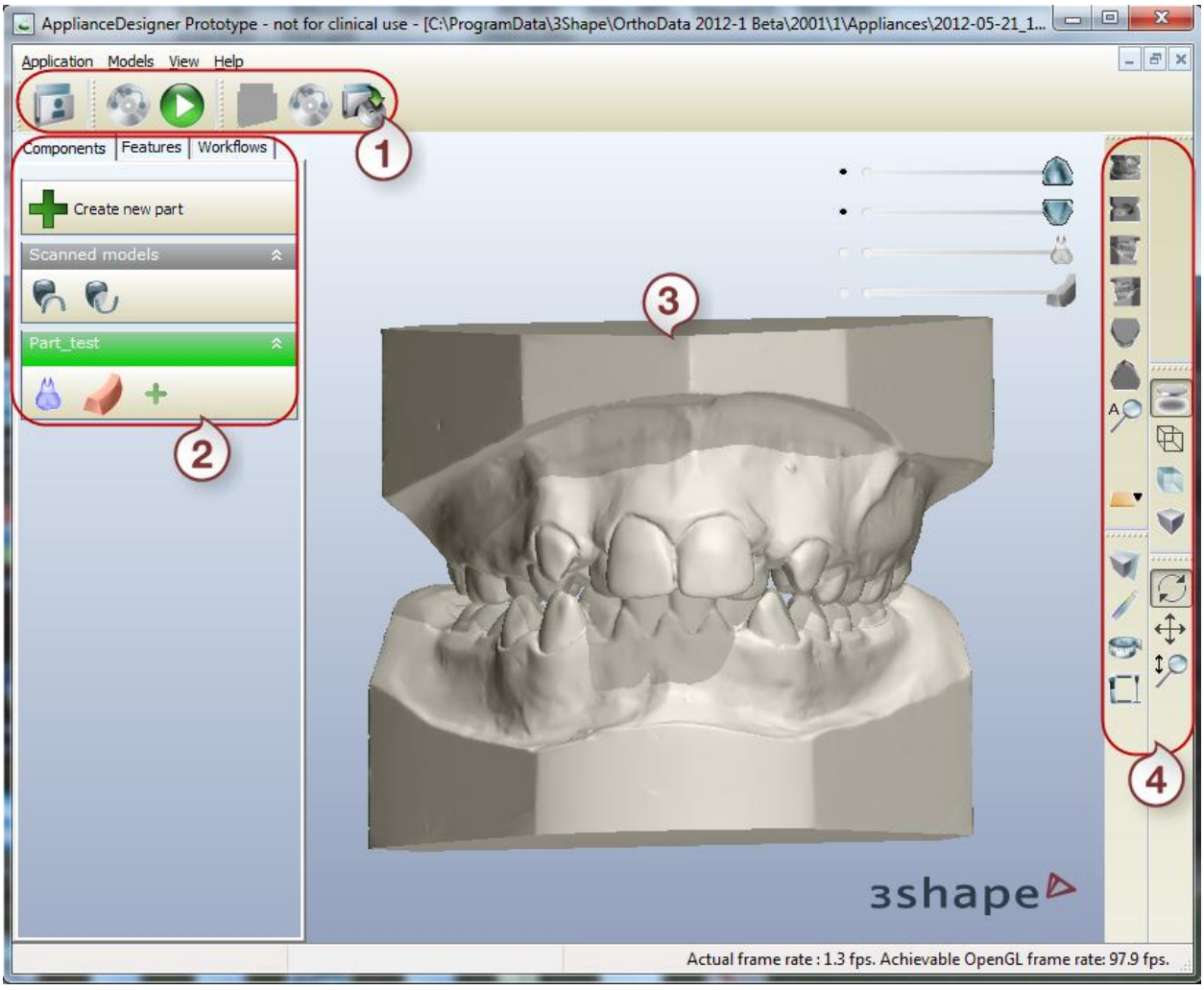
chapter Patient Model Set) you can use the ***Open model set***  option to see the record of just the patient case specific to the model opened. The function is provided in order to open a different model set for the same patient and to compare models while making a complete analysis of a single model set.

2 Appliance Designer User Interface

The user interface comprises of the following four sections as indicated on the image below:

1. Main toolbar
2. Functional panel
3. Session window
4. Visualization toolbar



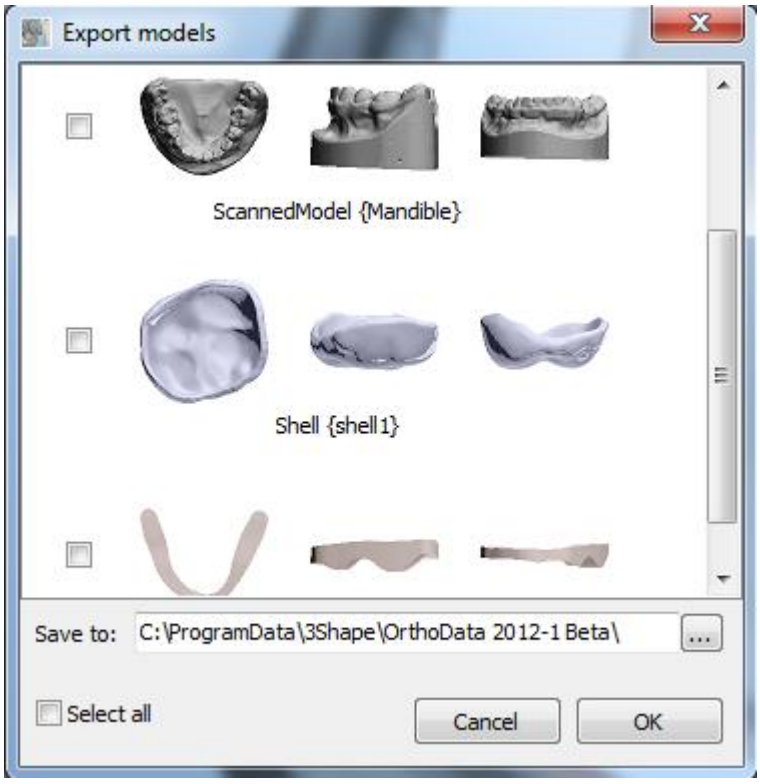
The following sections describe all the user interface components in details.

2.1 Main Toolbar

The Main toolbar contains functionality relevant to managing the analysis and the processes of design:

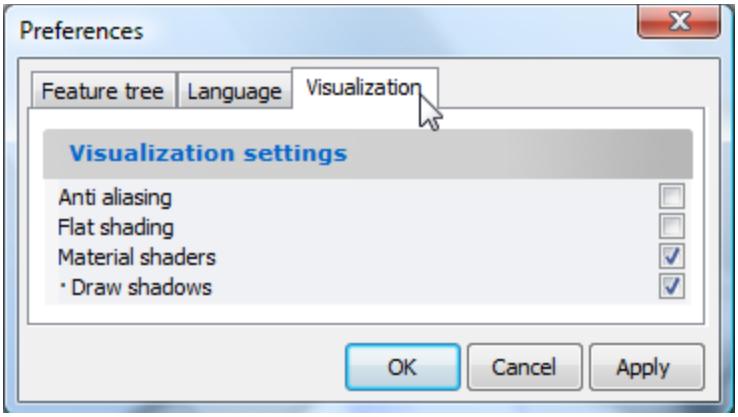
Icon	Action
	***Patient model set*** - Loads previously created patient model sets from the patient database.
	***Templates*** - manages the templates for the program.
	***Rebuild all components*** - allows to rebuild all the loaded components.
	***Open models*** - Opens model(s) required for inspection and imports them into the system.
	***Save session*** - Saves the current analysis session to default file.
	***Export models as...*** - Saves the current analysis session to the selected file.

Once you have selected the ***Export models as...*** button, the **Export models** window appears, displaying all the models and components:

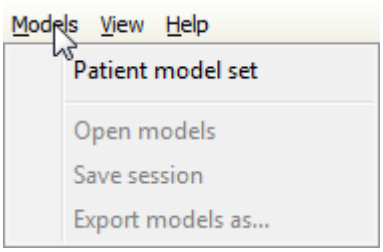


Select the path for the saved models and click ***OK***.

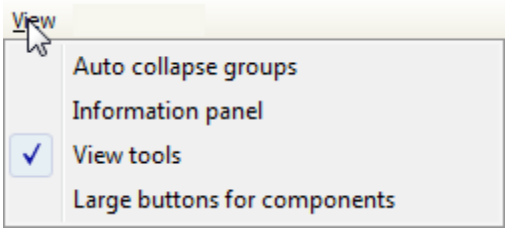
It is possible to customize the general application settings. Go to *Application->Preferences*. The *Preferences* window appears:



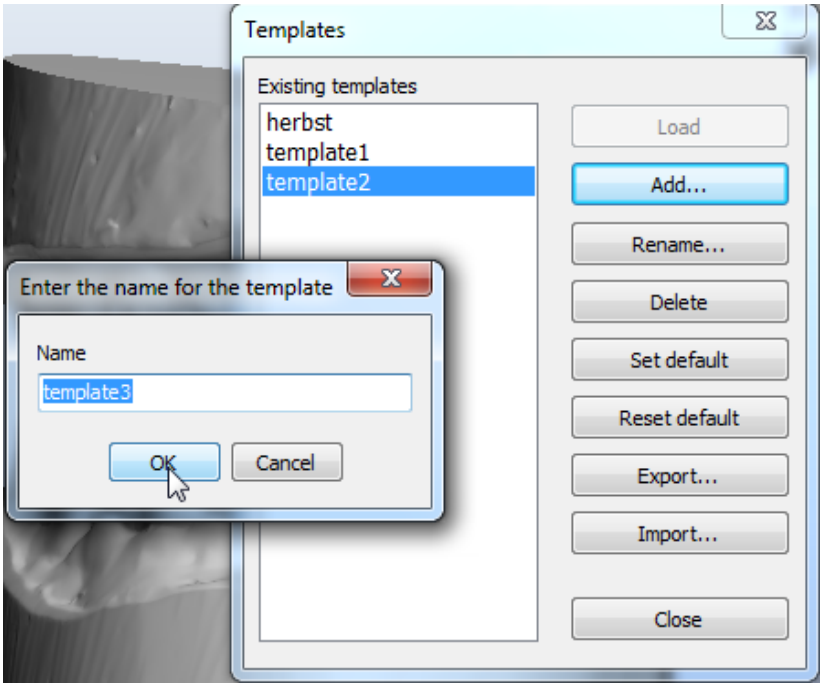
The *Preferences* window posses three tabs.
In the **Feature** tree set the parameters for the **Features** tab in the ApplianceDesigner.
Language tab allows you to select the language of the program from the drop-down menu.
Visualization tab offers a choice of checkboxes to customize the view of the model in the program.
Click ***OK*** to save the settings.



The ***Models*** button opens the list of options that duplicate the functions in the Main toolbar



The ***View*** button allows you to customize the look of the **Components** tab.



It is possible to operate with templates in the program.

Click the  button to open the *Templates* window.
You can ***Add...*** to the list of the **Existing templates**.
Type the **Name** of the new suit and click ***OK***.
Use the operation buttons to the right to customize the section.
It is possible to ***Export/Import...*** by selecting the path of export/import.

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Note: while a template will allow to save and reapply all design parameters, and components and features, you will still need to define the components (e.g. cutting splines or control points) on the new patient's model set.

2.2 Visualization Toolbar

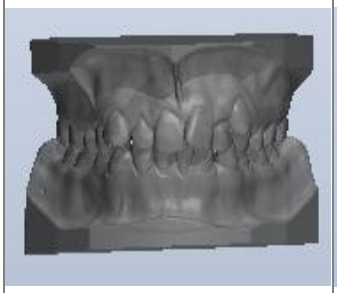
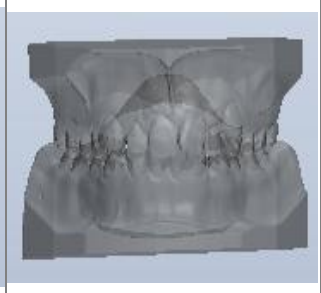
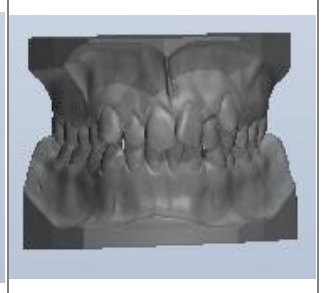
Finding the right point of view to work on a 3D model most effectively can be difficult and time-consuming. ApplianceDesigner comes with a set of tools aimed at helping you find the most advantageous visualization. The visualization tools are grouped in the **Visualization** toolbar and allow to:

- Rotate the 3D view to a preset angle: this allows you to view the models from the front, back, top, bottom etc.
- Set the position of objects on the screen for analytical inspection of the models in relation to each other.
- Set the visibility of objects visualizing a part of the virtual cast, which allows for a more detailed inspection of particular objects on the virtual cast.
- Set the objects transparency for the current operation to see or hide their internal structure.
- Switch to custom rotation and panning mode enabling rotation and free movement of objects on the screen.

The buttons in the **Visualization** toolbar are presented in table below:

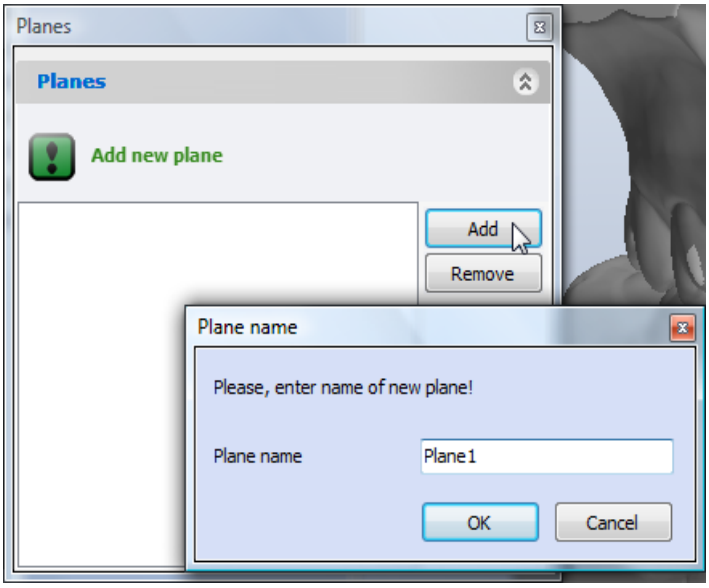
Icon	Action
	**View** – The view buttons allow you to switch between a number of predefined viewpoints: • ***Front view*** • ***Rear view*** • ***Right side view*** • ***Left side view*** • ***Top view*** • ***Bottom view***
	***Zoom all*** – Sets the model to the default view, size and position on the screen leaving its current rotation angle intact.
	***Additional plane view*** - allows to view the models according to predefined planes.
	***Cross-section*** - Allows to create the 2D cross-section of the model at any stage of the workflow.
	***Show undercuts*** - Marks the undercuts with the colors according to the set color scheme
	***3D measurements*** - Allows you to perform measurements on the model.
	***Articulator*** - simulates the movements of the model in the articulator.
	***Axis locked rotation*** – Locks the rotation to the x, y or z axes.
	***Pan view*** – When selected, right-click and drag to pan the view.

	***Zoom view*** – When selected, right-click and drag to zoom in and out.
	***Show material and shadows*** – Marks the cast segments with the corresponding material colors and toggles on and off the function of shadow visualization.
	***Show wireframe*** – Shows the triangulation of the virtual casts.
	***Show as transparent*** – Changes the degree of transparency of the virtual casts.
	***Show perspective*** – Transforms the model into a realistic 3D object, where the distant objects appear smaller than the closer ones.

			
Show material and shadows	*Show wireframe*	*Show as transparent*	*Show perspective*

2.3 Additional Plane Views


Click the ***Additional plane views*** button on the Visualization toolbar to open the option. Select **Planes configuration** from the drop-down menu



Click the ***Add*** button on the *Planes window* to create a new plane. Give a name to the new plane, place three blue arrows on the model to mark the plane area:

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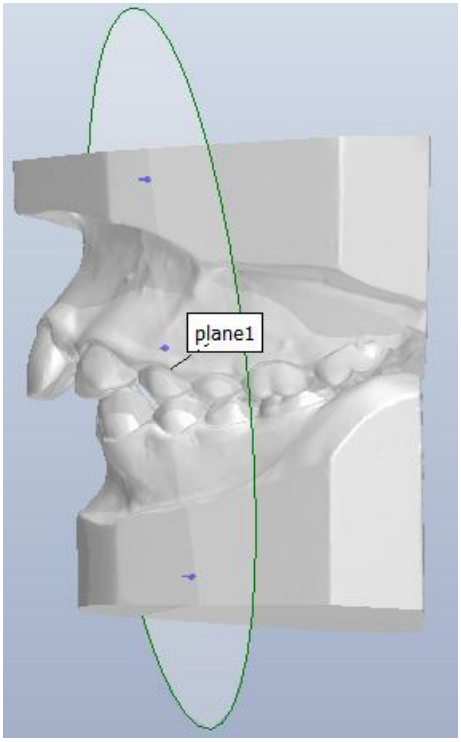
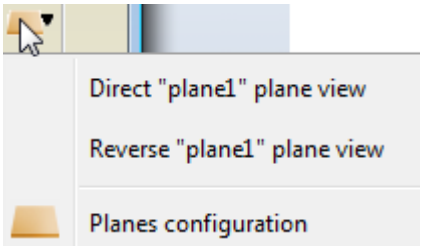


The plane is annotated with its name and turns yellow when selected.
Click ***OK*** to save it.

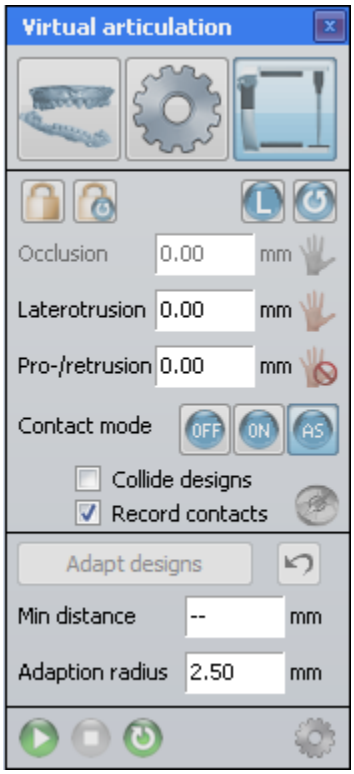


It is possible to select now different views according to the created plane.

Note: You can apply any custom planes created (and saved) in OrthoAnalyzer in Appliance Designer by choosing them from the list.

3 Interactive Tools

Interactive Tools: These tools positioned on the **Visualization** toolbar can be activated at any step of the analysis and planning process.
Activation: In order to avoid conflicts in user input between the open Function and the chosen tool (articulator, 3d measurements or 2D Cross section) and to be able to toggle between them, click on the tools' window to activate the given tool or on the function window. The active feature window is marked in blue as illustrated.



3.1 Cross-section

The ***Cross-section*** button in the **Visualization** toolbar opens the 2D Cross-section preview in the bottom right corner. It can be accessed at any time during the modeling process to inspect the design

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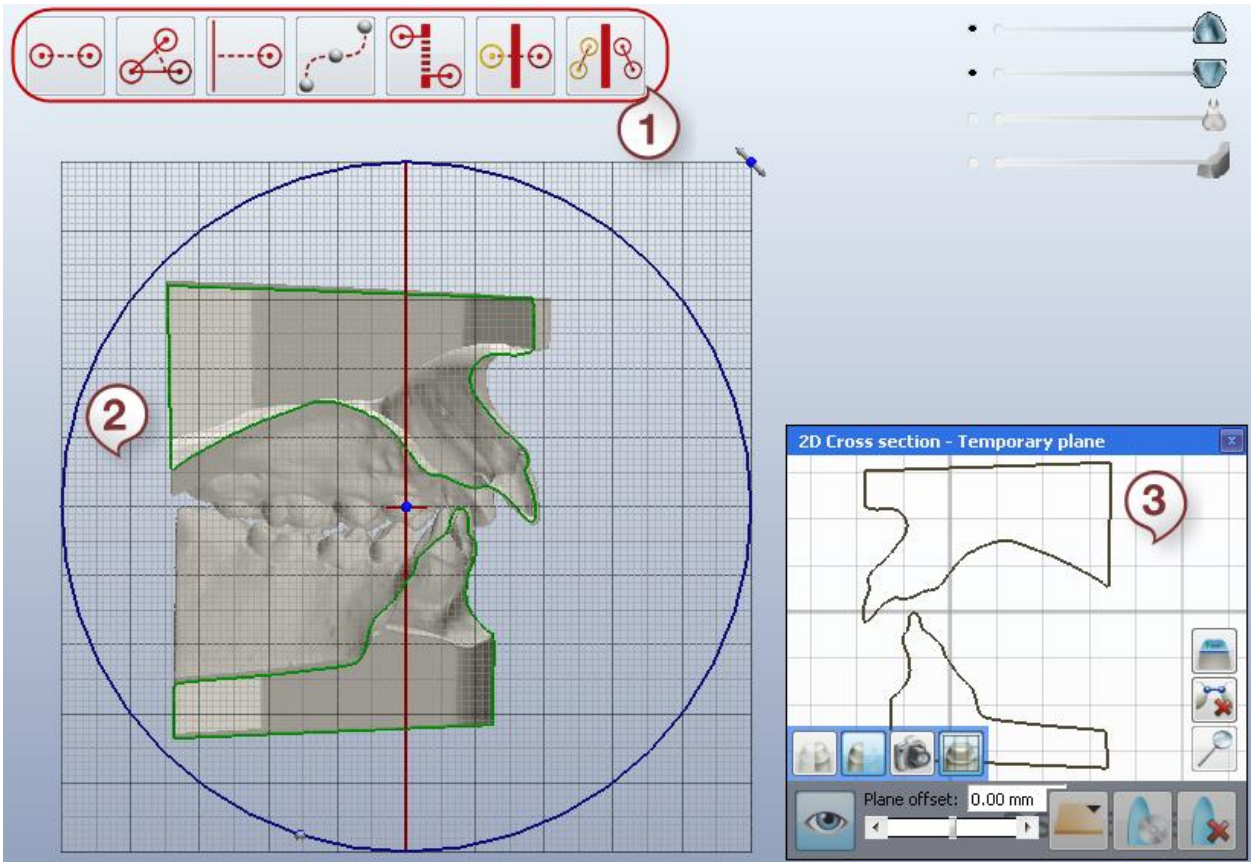
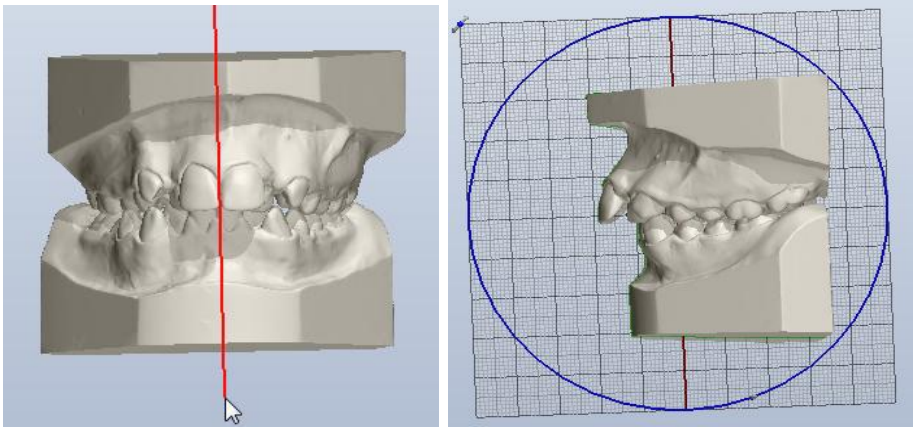


1. *Measuring buttons*
2. *Cross-section plane*
   2D Cross section
3. *Temporary plane*
   window



Step 1: Draw the Cross-section plane

Draw the red line to define the cross-section plane by holding the left key of the mouse. Single click at the line creates the cross-section.
It is also possible to define a cross-section plane by placing three points.

Use the manipulation buttons on the 2D Cross-section - *Temporary plane* window **(2)**:

	***View buttons*** - unfold the number of options that customize the view of the cross-section plane:
• 	***Clipping plane*** - ***Left*** - Hides the right side of the model in relation to the cross-section plane.
• 	***Clipping plane*** - ***Right*** - Hides the left side of the model in relation to the cross-section plane.
• 	***Fixed camera*** - allows to fix the plane in one position and move the model instead.
• 	***Grid*** - turns on/off the grid of the cross-section plane.

	***Standard planes*** - Allows you to select the plane through which the cross-section plane will go.
	***Save plane*** - Allows you to save the current plane*
	***Clear View*** - Removes cross-section.
	***Measure area*** - Allows you to measure distance on the model in the 2D Cross-section window.
	***Clear measure*** - Deletes the measures.
	***Zoom all*** - Allows you to zoom the model.

Note: You can reuse and apply any custom plane created (and saved) in OrthoAnalyzer in Appliance Designer by choosing them from the list of **Standard** planes.

* Once you have selected the ***Save plane***  option you get the popup window. Choose the ***Save as new plane*** option and confirm the action by giving the **Name** to the new plane and clicking ***Save***:

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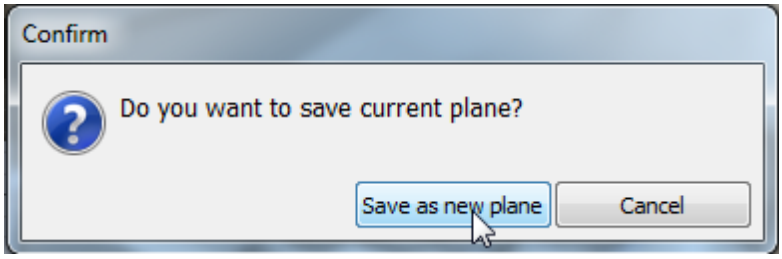
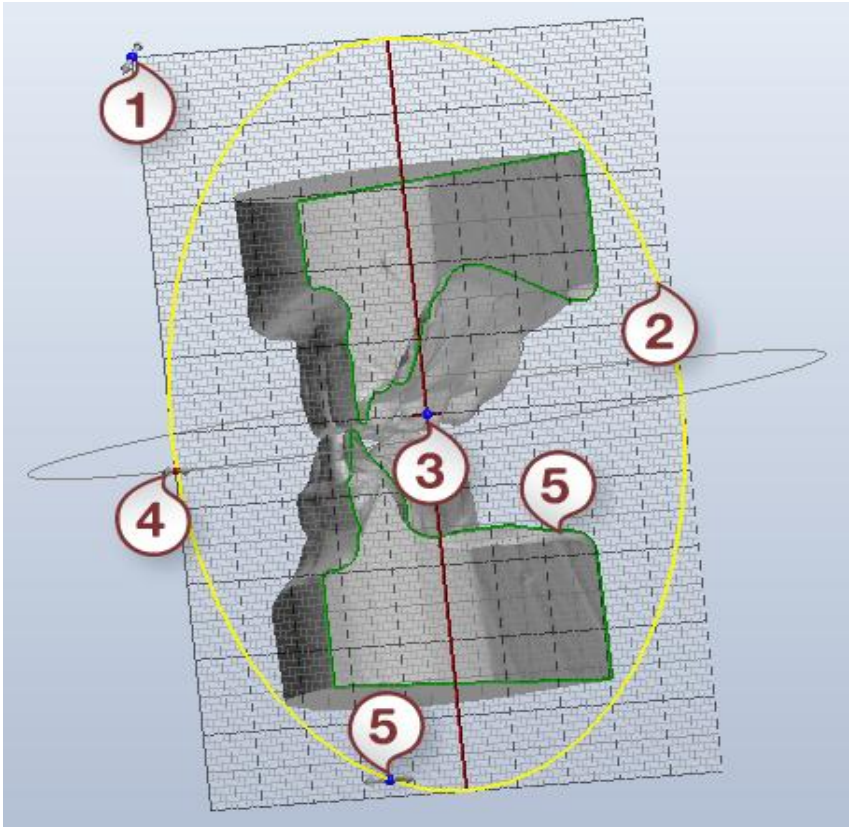


You can select any saved plane in ***Standard planes*** section.



Step 2: Move the cross-section plane:

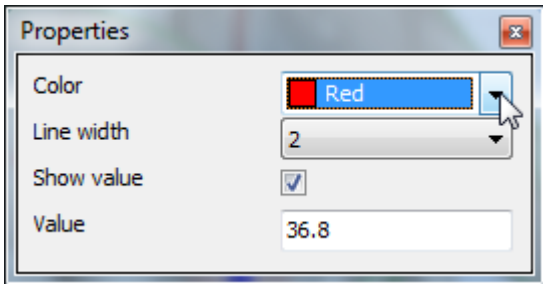
1 - The control point located on the plane corner rotates the plane clockwise and anti-clockwise without moving it or changing its slope. Thereby, you can select the most appropriate position of the plane for making on-grid measurements.
2,4 - Change the slope of the plane along the perpendicular cross-section plane which is perpendicular to it. To visualize this additional plane, activate the blue circle by clicking on it to make it turn yellow. When the cursor is positioned on the yellow circle with its own red control point indicating the movement path is displayed. While pressing the mouse button, drag the control point to change the slope of the cross-section plane.
3 - The central point moves the plane around the model leaving the slope of the plane unchanged.
5 - Plane translation: the control point located on the blue circle defining the plane

moves the plane back and forth without changing its slope. Adjust the plane position by pulling the **Plane offset** slider

on the *Temporary plane* window **(2)**.

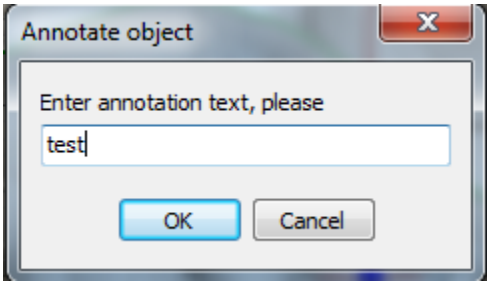
Length measurement requires two points. Measures length from point to point.

It is possible to customize the view of the measurement by right-clicking it and selecting Properties from the popup menu:



Select **Color** or the measurement, set the **Line width** from the drop-down menu and indicate the **Value** by entering the parameters manually in the appropriate editbox.

You can move the measurement along the grid by dragging the control point (turn white when selected). It is possible to annotate the control point and the measurement (if selected turns white). Right-click the desired object, select **Annotate** from the popup menu and enter the annotation text in the popup window:



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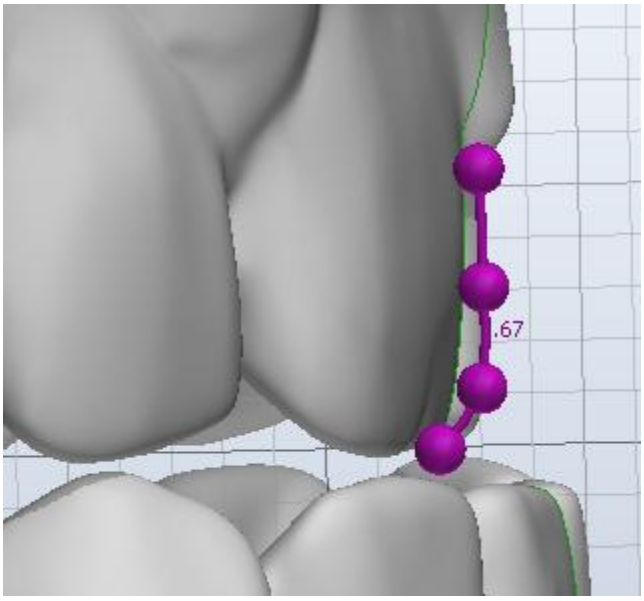
Angle measurement requires three points. With angle measurement, the second point becomes a vertex of the angle.

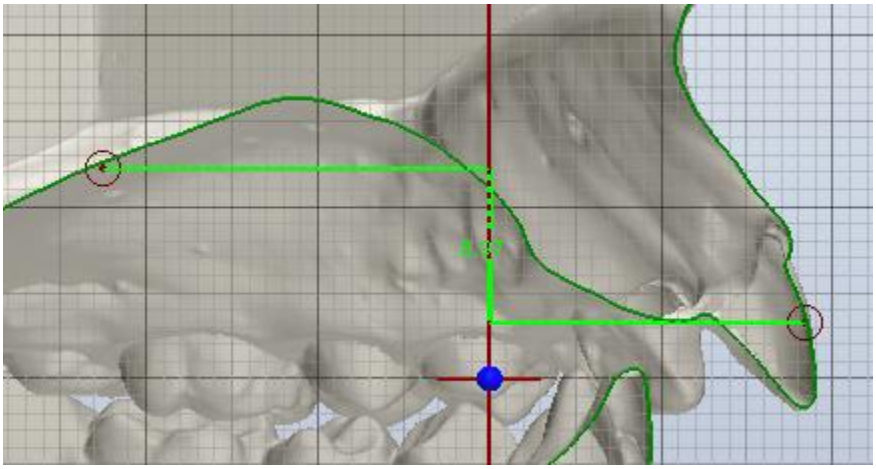
There are some extra options for **Angle** measurement in the Properties option. It is possible to select the **Revert angle** or edit the angle parameters manually.

Distance to measurement requires one point on the desired measurement and one point on the grid.

Spline measurement allows you to measure curved areas with the help of the spline.



Distance along centre line requires two points. Measures the distance in relation to the centre line.



Symmetry points requires one point. Automatically places the second point symmetrically to the selected one along the center line.

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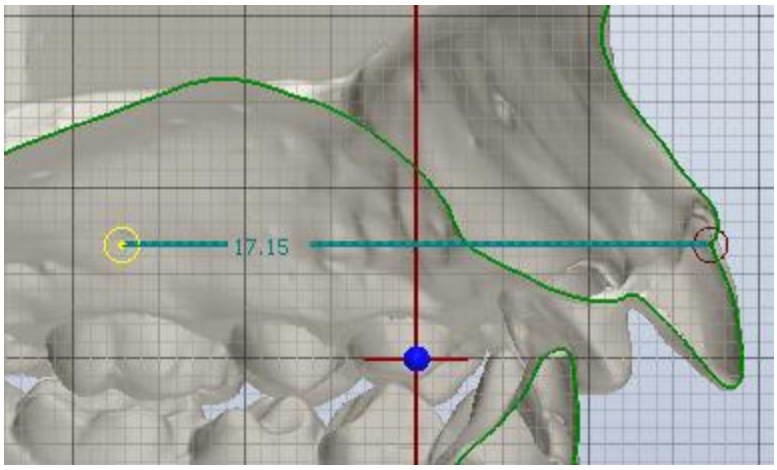
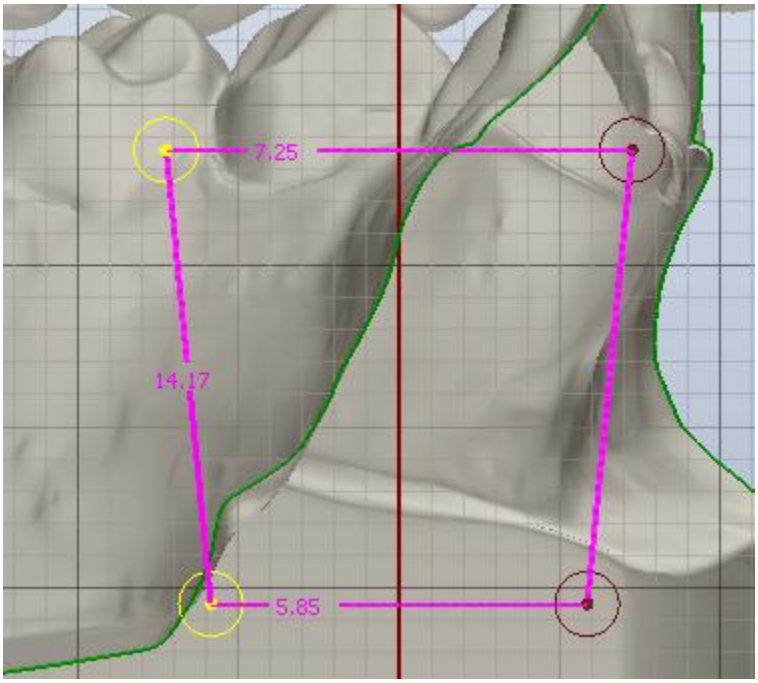
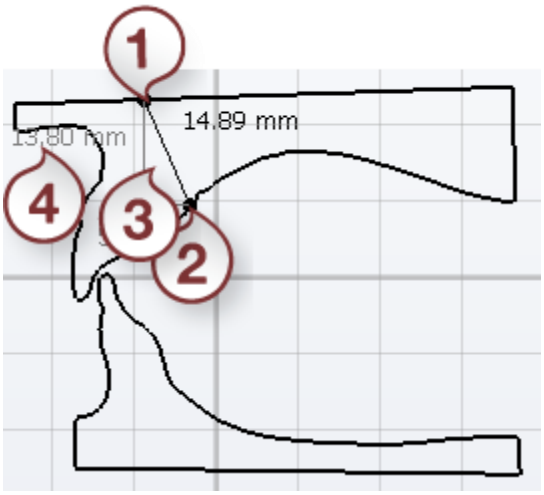


Symmetry guide requires three points.



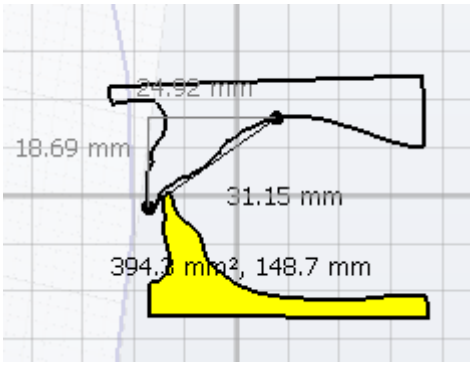
For ***Symmetry guide*** it is possible to **Show additional line** and display **Distance to the center line** by marking the appropriate options in the right-click menu.

The 2D cross section allows to take measurements automatically at the intersection of the chosen plane and 3D model in screen. All measuring points snap to the intersection. It also allows to take measures of area and perimeter defined by this cross-section.
The measurements placed on the model in the 2D cross section window are automatically mirrored on the model in 3D.

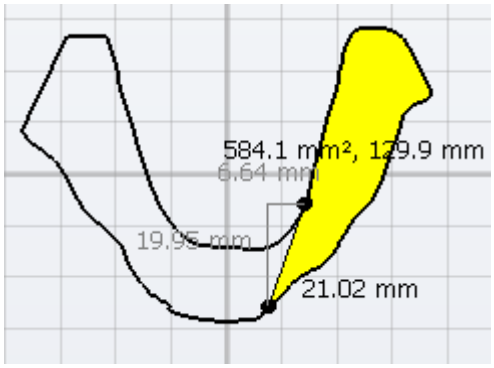


1, 2 – The points placed on the cross-section are automatically attached to the beige contour of the cross-section.
3 – The system automatically builds the right-angled triangle using the selected measurement line as its lateral face.
4 – The measurements of the other faces of this triangle are also displayed on the cross section.

To clear the measurements made, click the  button.



Click the ***Measure Area***  button and place the cursor over the area you want to measure. The area becomes colored yellow and its value is calculated automatically. Note that the second value is the perimeter of the cross section.



It is possible to open this function on any stage of your modelling process.

3.2 Remove Undercuts

Remove undercuts fills undercuts to simulate e.g. waxing on models.

To set the **Remove undercuts** option follow these steps:

▶ **Step 1: set the remove undercuts properties**

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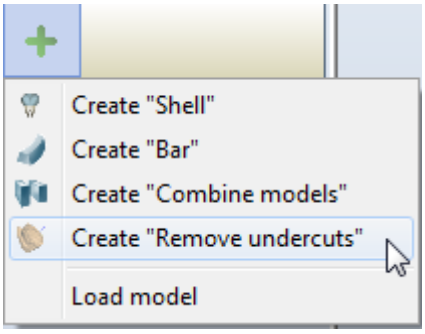
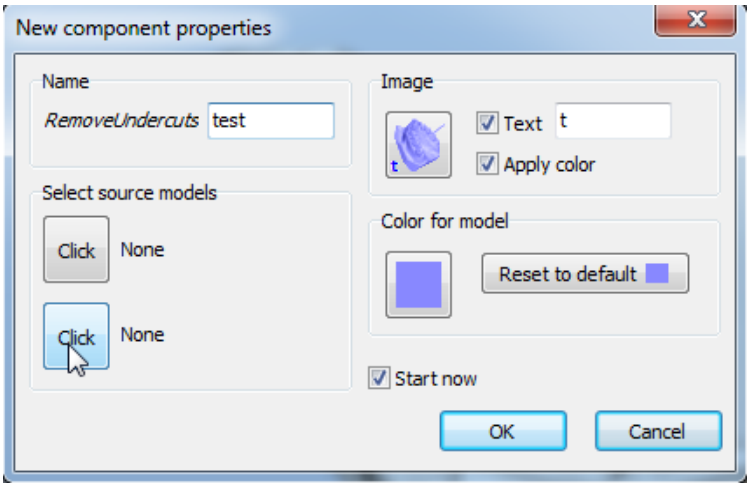


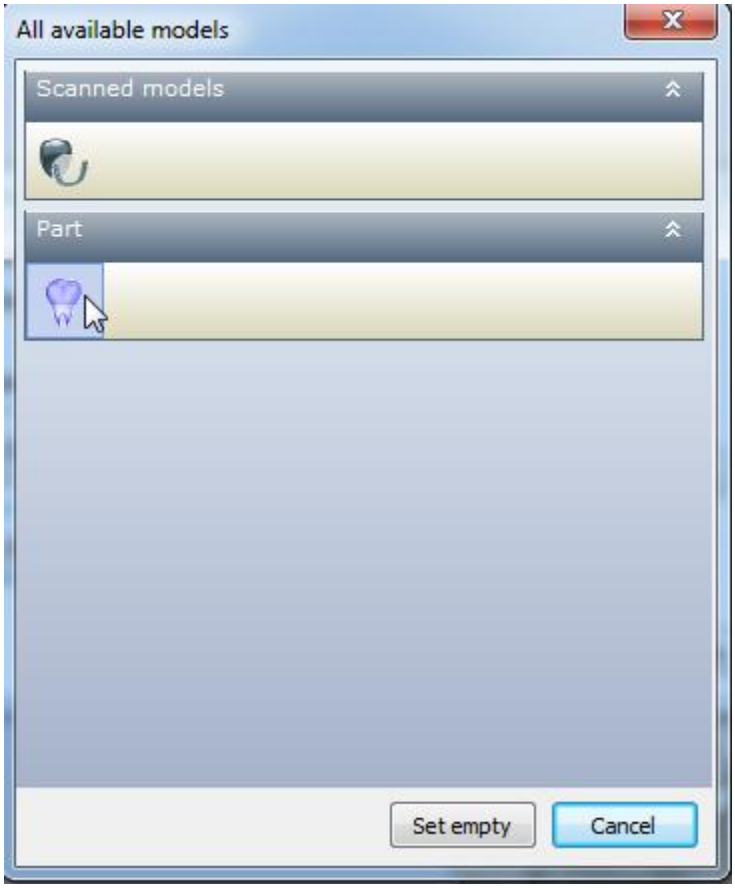
In the newly created part section click the  button in the **Part** section and select the **Create "Remove undercuts"** option from the drop-down list (see the image to the left).



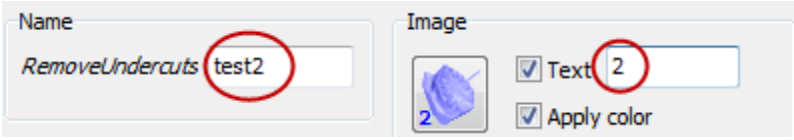
Indicate the Name of the new component.

Important: the name of the component corresponds to the name of the Output model!

It is possible to select source models on which the component is to be built by pressing the ***Click*** button. The *All available models* window pop ups(see the image below):



Note: Appliance Designer™ allows you to build any component on either the tooth model, or on any other component created previously.



Select the model from the list or click **Set empty** to leave the source model unidentified.
Fill in the **Text** that will be displayed on the ***Remove undercuts*** icon to the left. The last sign in the **Name** section will be duplicated in the **Text** section (see the image to the left).
Apply color, if checked, allows you to apply the selected color to the image

Step 2: remove undercuts

- *set the insertion direction*

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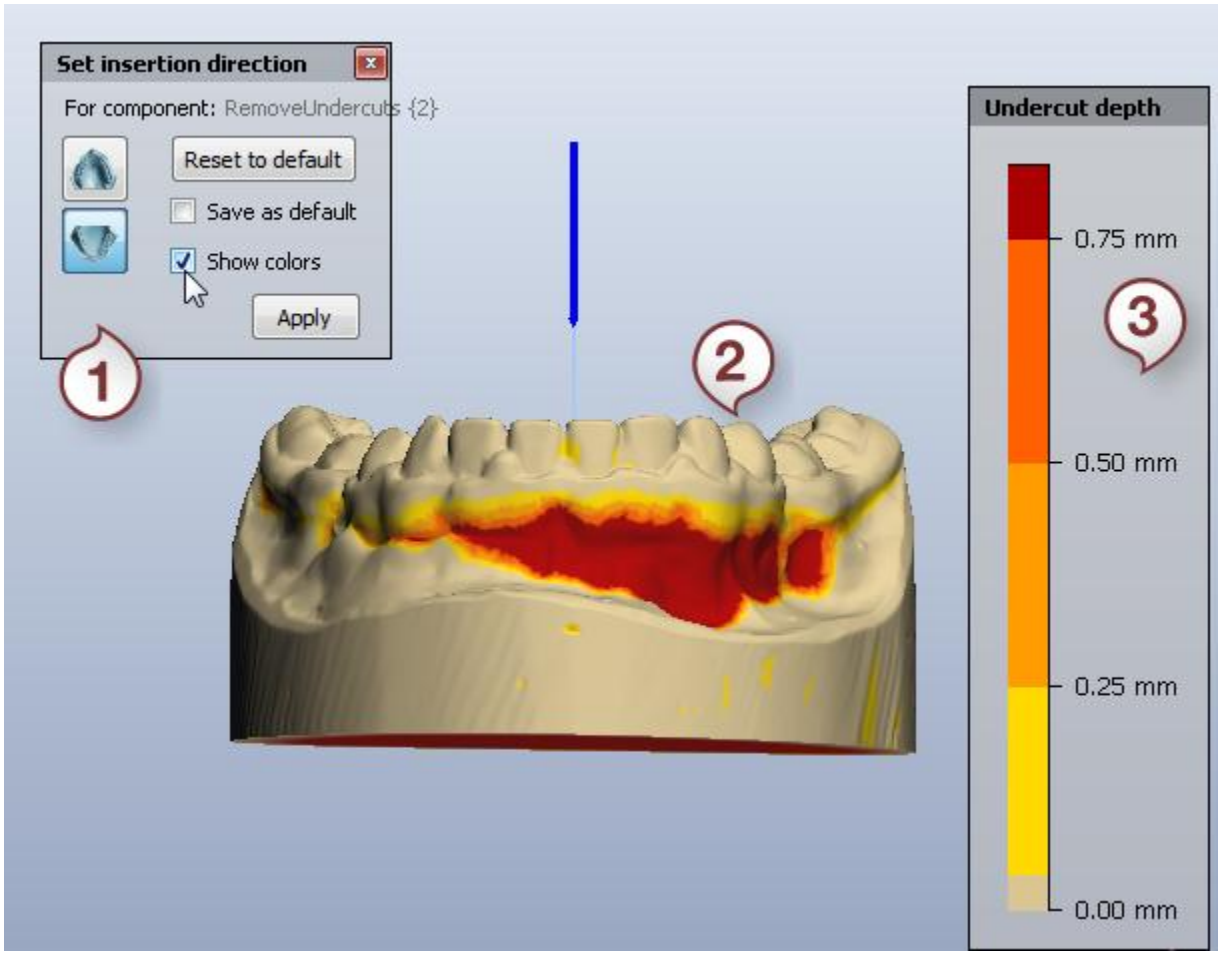
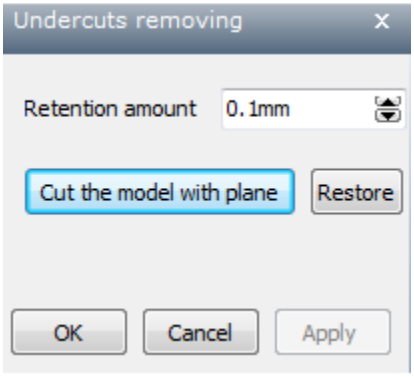
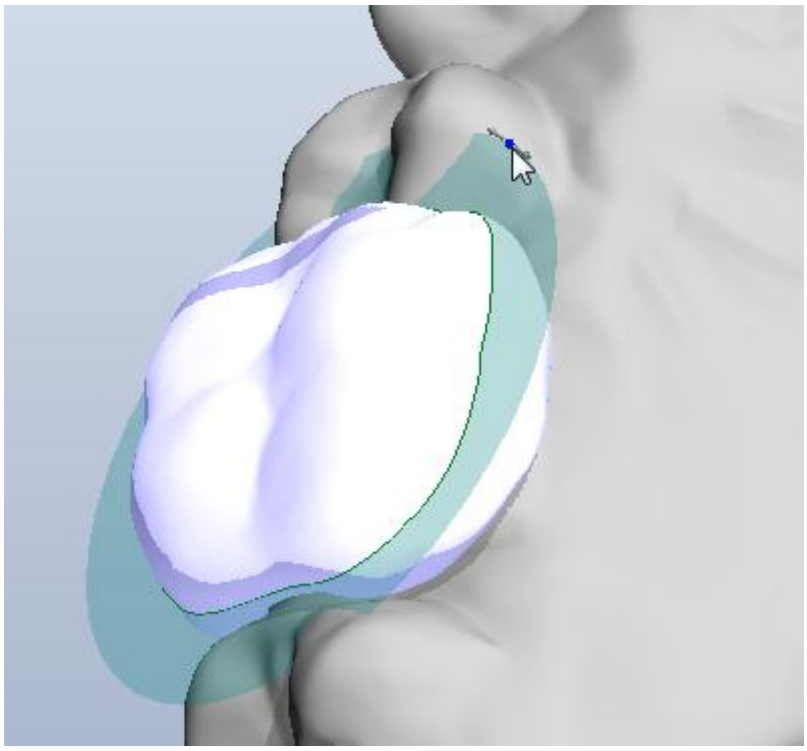
Once you get to the **Features** tab, you can customize the parameters for removing undercuts in the *Set insertion direction* window **(1)**.
Save as default, if checked, allows to set the selected insertion direction as default.
Show colors - indicates the areas on the model according to the **Undercut depth** color scheme **(3)**. You can observe the areas that will be removed on the model **(2)**.
Reset to default - allows you to undo the adjustment of the insertion direction back to the default one.



- *remove undercuts*



It is possible to indicate **Retention amount** in the appropriate edit box on both steps.
The ***Cut the model with plane*** button is activated on this step. Adjust the position of the cutting plane by dragging it and click the button.
It is possible to undo the cutting by clicking the ***Restore*** button.



Click ***OK*** to apply changes.
Click ***Apply*** to save the results.

3.3 3D Measurements and Custom Analysis

The ***Measurements and Custom Analysis*** option allows you to perform and edit the desired 3D measurements on the model.

Landmarks

Landmarks represent simple (**red**) points that define angles, occlusion planes, cross-sections, sections for measurements etc., thus, making up to all the analysis objects enabled by the system.

The arrow-like surface markers help you navigate around the landmarks:

Blue arrows define a passive selection.

Yellow arrows indicate the selected objects. Their positions can be changed by dragging the corresponding marker.

The arrow turns white when you place the cursor on it.

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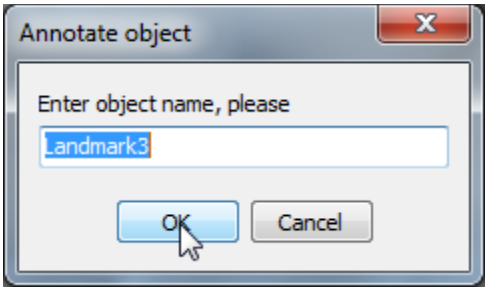
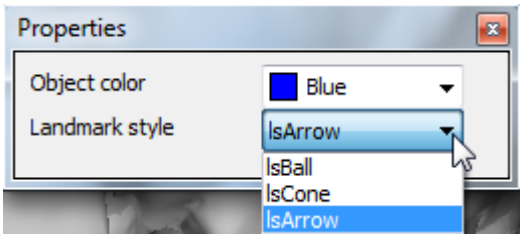
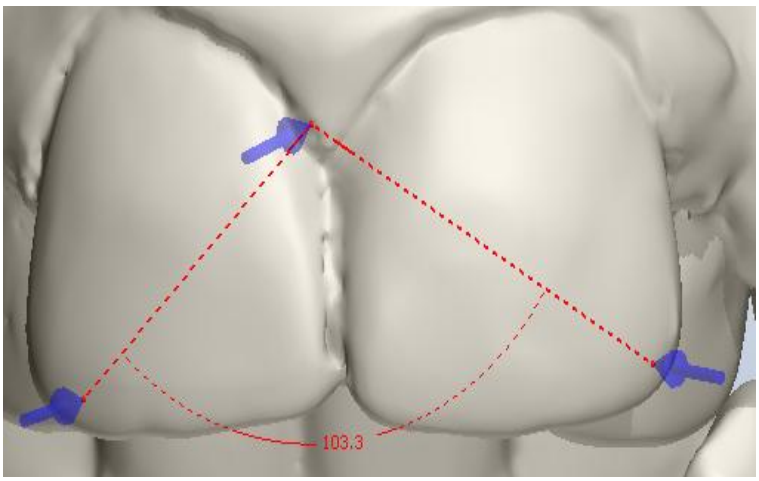


Annotate the landmark by right-clicking it and selection the appropriate option from the popup menu. Enter the name of the annotation and click ***OK***.

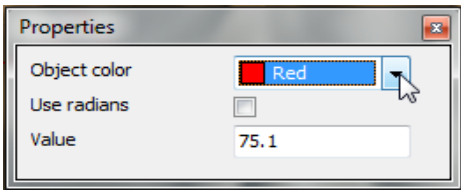


Customize the view of the landmarks by right-clicking a landmark and selecting the color and form of the landmark from the drop-down menu.

 - ***Angles***

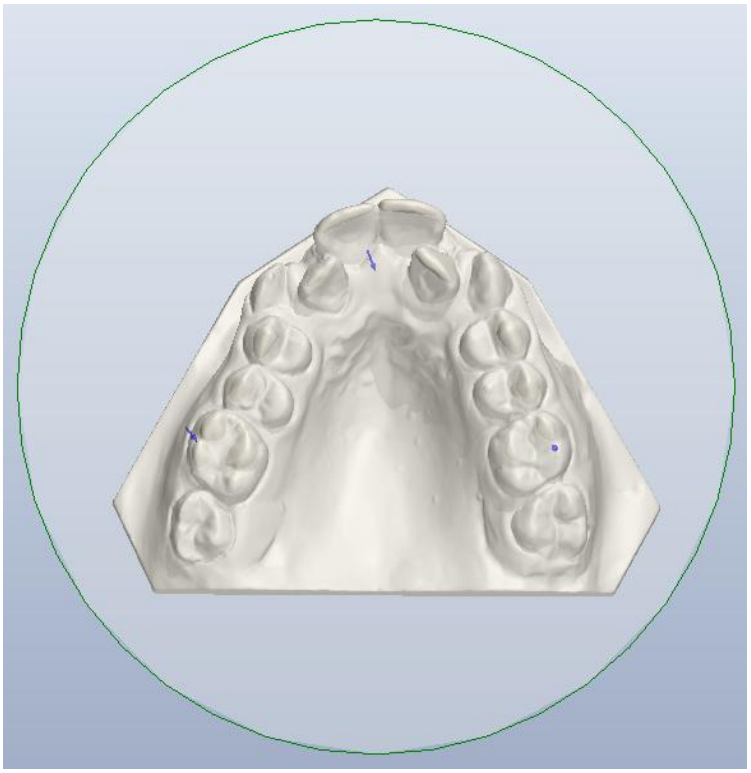


Angles are defined by three landmarks and the measurements which are immediately displayed.
It is possible to customize the view of the measurement by right-clicking it and selecting Properties from the pop-up menu:

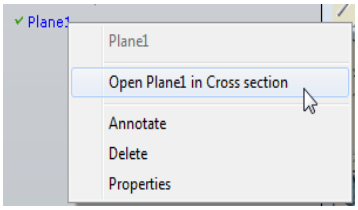


You can open this pop-up menu by right-clicking the appropriate measurement in the Custom objects window.

 - ***Planes***



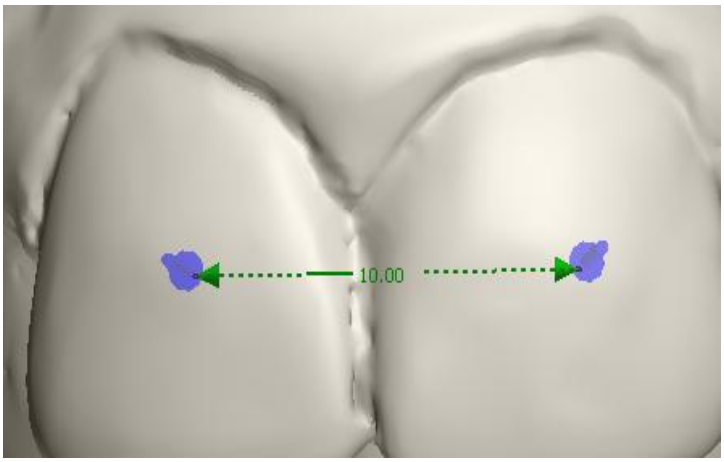
Set a plane by positioning three landmarks on the model.
It is possible to customize the plane by right-clicking its name in the **Custom objects** window:



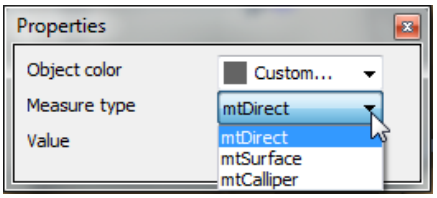
Measures

-  ***- direct***

Place two landmarks on the surface of the model to make the measurement. It is possible to edit it by right-clicking the measurement line (if place a cursor on it turns **white**, if selected turns **yellow**).



Place two landmarks on the surface of the model to make the measurement. It is possible to edit it by right-clicking the measurement line (if place a cursor on it turns **white**, if selected turns **yellow**).
You can select the **Measure type** from the drop-down menu after right-clicking the measurement and select **Properties**:



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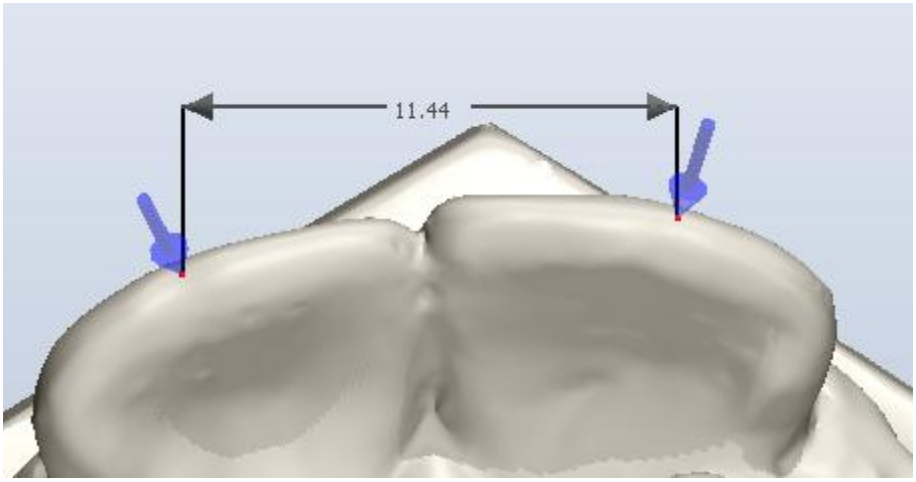
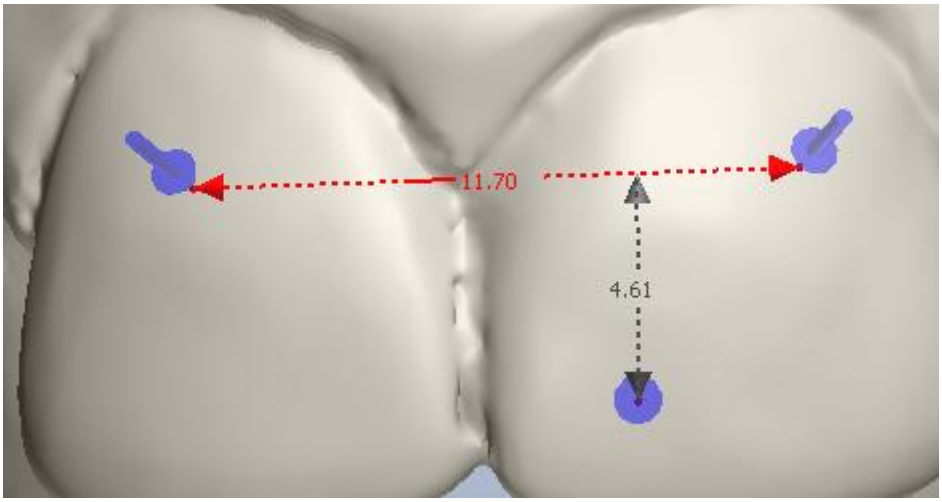
-  ***- calliper***



Indicate the measurement by two landmarks replicating a physical calliper. Customize the measurement by right-clicking the landmark and opening the popup menu.

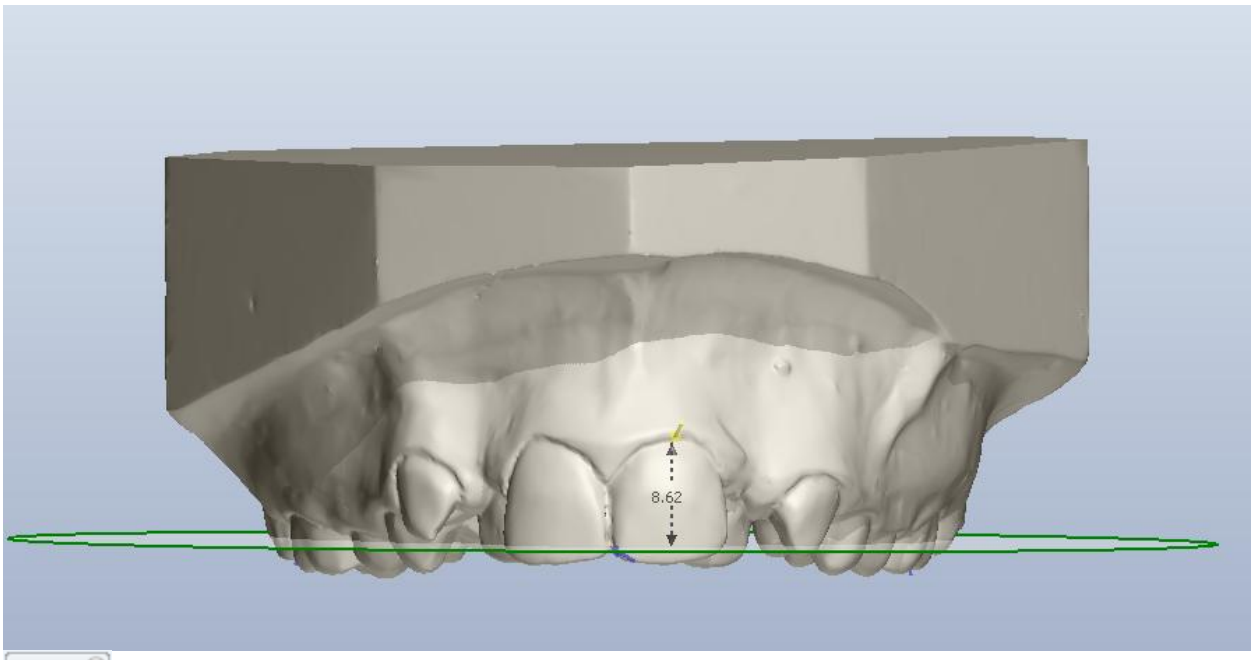
 ***- Distance to line***



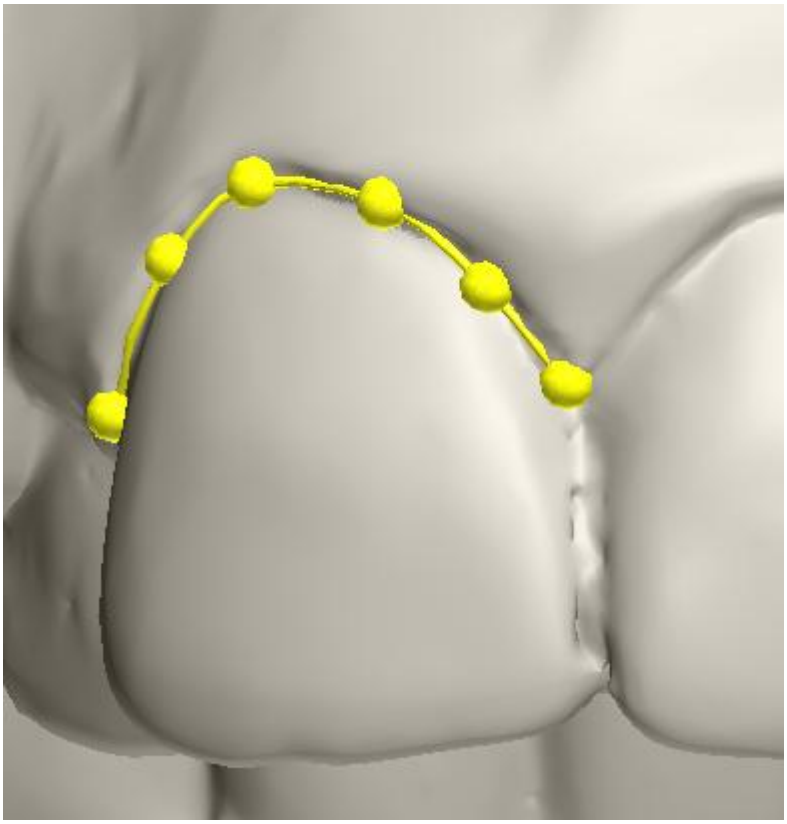
To make this measurement: create the line->place the landmark at the area you want to make the measurement from->click at the selected line

 - ***Distance to plane***

To measure the distance to plane create a plane->place a landmark on the area you want to measure the distance from->click the **green** border of the plane or the annotation (if there is):



 - ***Custom spline***



To make this measurement place a few control points that define the spline. Double-click the last control point to close the spline.

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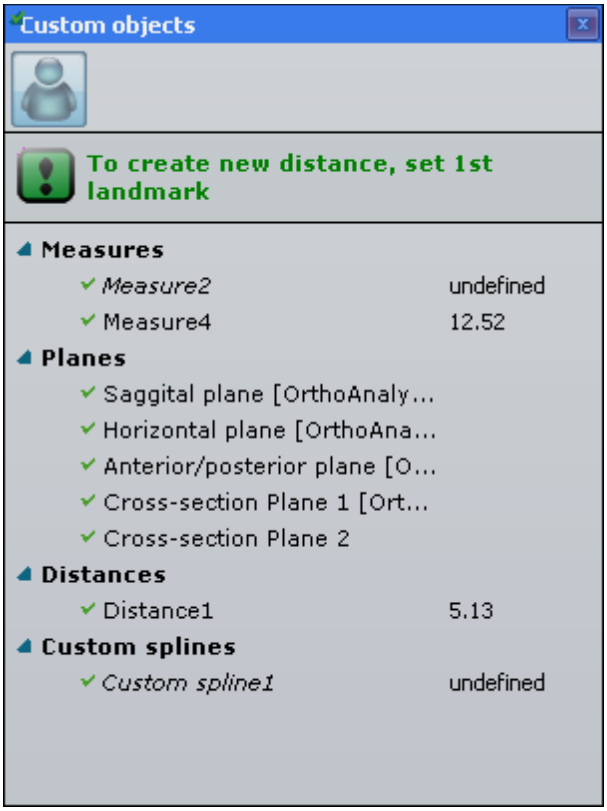


To hide the measurement, click the **green** tick in front of the measurement/plane. The selected object is highlighted **blue**. To fold/unfold the groups click the arrow in front of its name.

3.4 Articulator

The **Articulator** option is activated with the ***Articulator*** button in the Visualization toolbar during the design process.

Step 1: Modify the alignment of the jaws

You can set the jaws in the desired positions by clicking the ***Modify*** button in the Virtual Articulation window and moving the control points with the cursor for a better fit (see the images below).

Occlusal change - The indicated value of displacement in the occlusion direction is relative to the original static occlusal and shown in millimeters. It can be set directly by clicking the value and keying in the number directly or selecting it with the buttons next to the value.

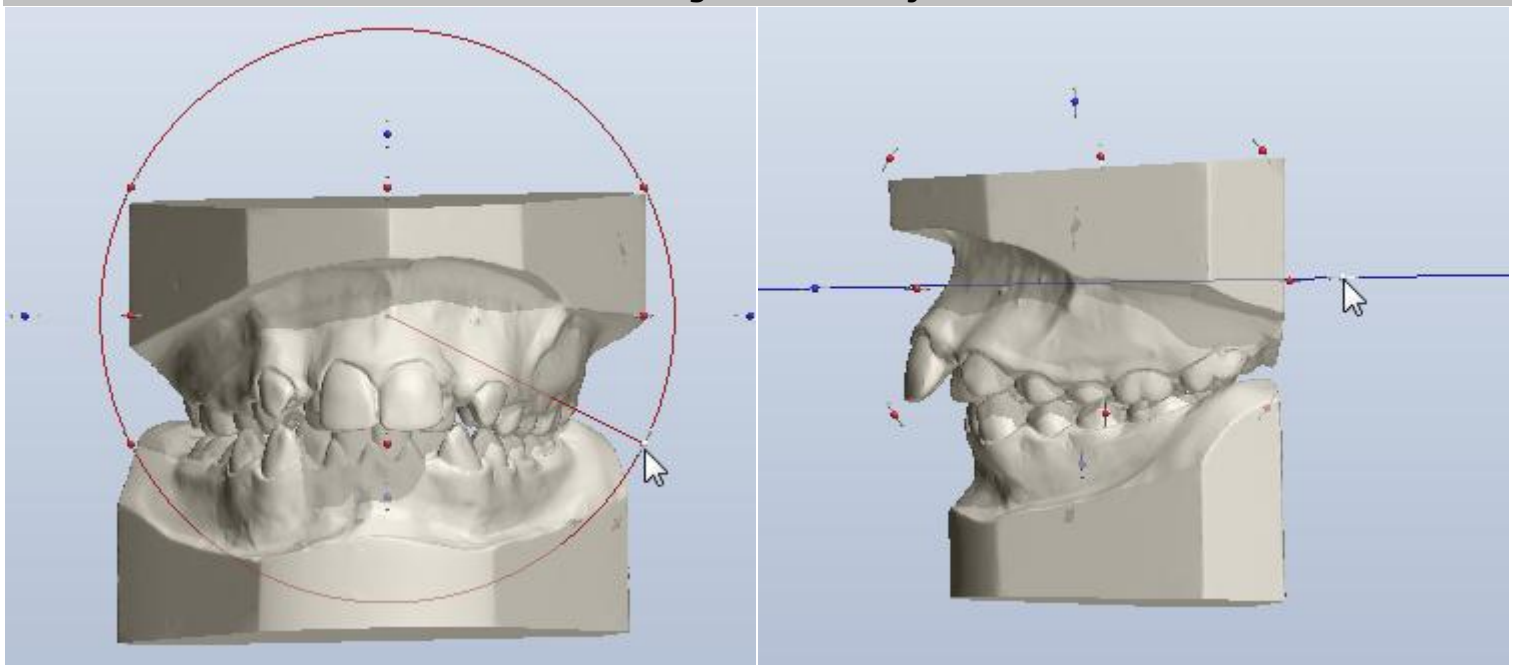
Use the buttons in the *Virtual articulation* window to undo your changes to the static occlusion **(1)** or update the static occlusion with the current alignment of the jaws, when you are

satisfied with the results **(2).**

Click the ***Optimize occlusion*** button to align jaws automatically.

The **red** control point allows to rotate the jaw. The **blue** control point allows you to move the jaw up and down:

Set the alignment of the jaws



Step 2: Setup articulator model and adjust placement

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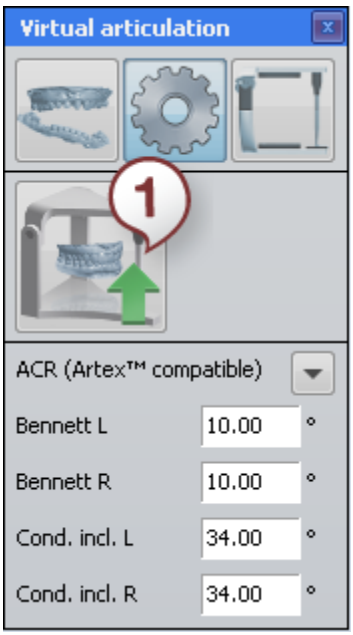
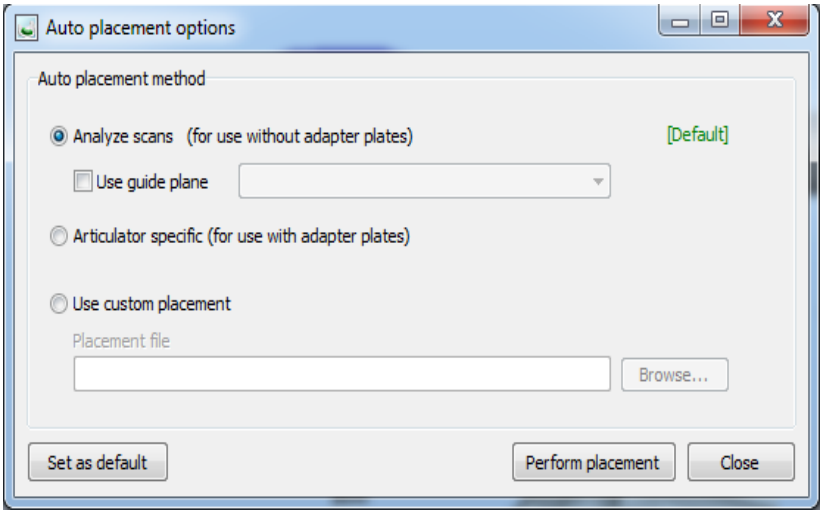


Click the  button on the *Virtual articulator* window to start the setup.
Select the type of the articulator from the drop-down menu clicking the arrow button  .

It is possible to place the model in the articulator according to the **Occlusion plane**, selected from the popup menu. Set up the desired plan first in the Additional plane views section.

L Bennett, **R Bennett**, **L cond. incl.**, **R cond. incl.** are articulator model parameters, they can be set by clicking the appropriate editboxes and entering their parameters manually or selecting them with the buttons next to the values.

The ***Auto placement*** **(1)** button allows you to customize the default placement method and perform auto placement of the model. Click it to open the *Auto placement options* window:



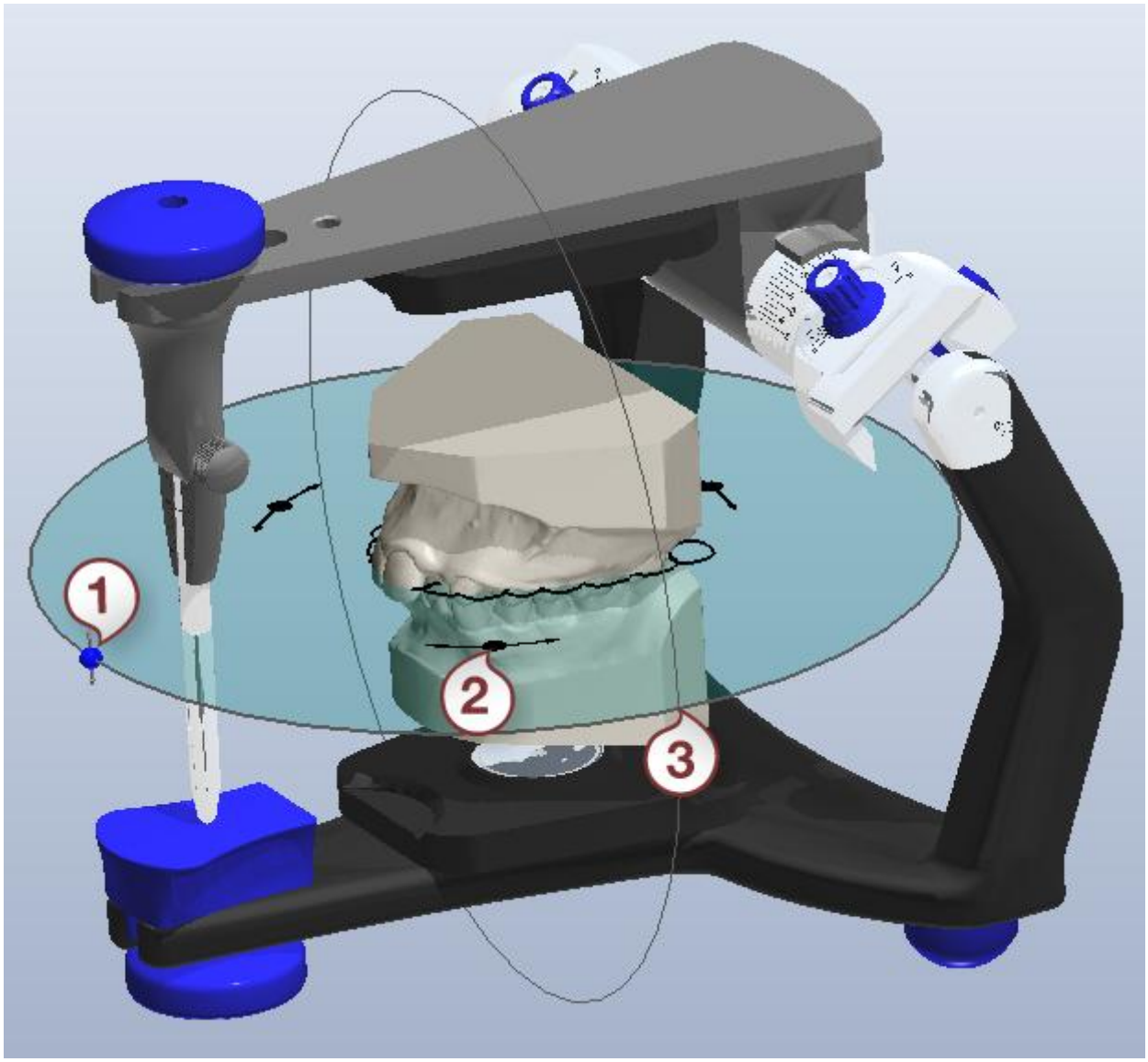
You will need to click the ***Browse*** button and select your placement file to use.
The ***Set as default*** button saves you current auto placement method to use as the default one for future orders.

The ***Perform placement*** button initiates the auto placements process with the chosen options.
Analyze scans is selected for the models without adaptor plates . You can load predefined planes created in e.g. OrthoAnalyzer, such as the occlusion plane as default position in the articulator by checking the **User Guide plane** box and selecting the plane from the drop-down list.
Use custom placement is used when the adaptor plates are included.

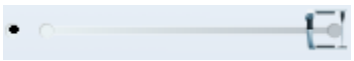
Click the ***Browse*** button

to set the ***Placement file*** path.

The placement can be adjusted manually by moving the blue plane. The plane can be moved up/down and rotated with the cursor by the indicated points to find the correct position. It is also possible to translate the plane by dragging it. You can change the tilt of the plane by dragging its rim:



1. *Moves the articulator up and down*
2. *Rotates the articulator*
3. *Change the angle of the articulator position*

Use the visibility slider  in the upper right corner of the screen to customize the view of the articulator.

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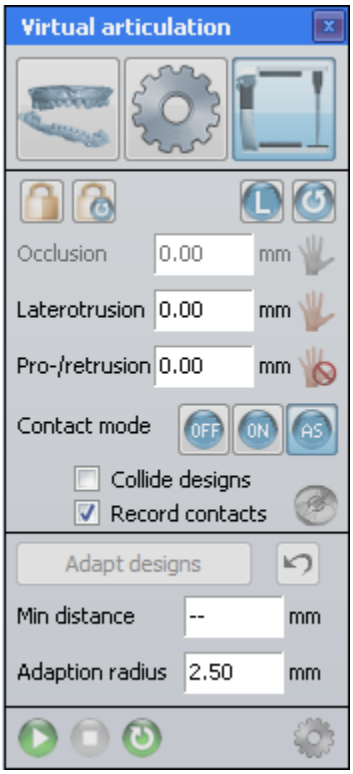
Step 3: Use articulator to evaluate and adapt design

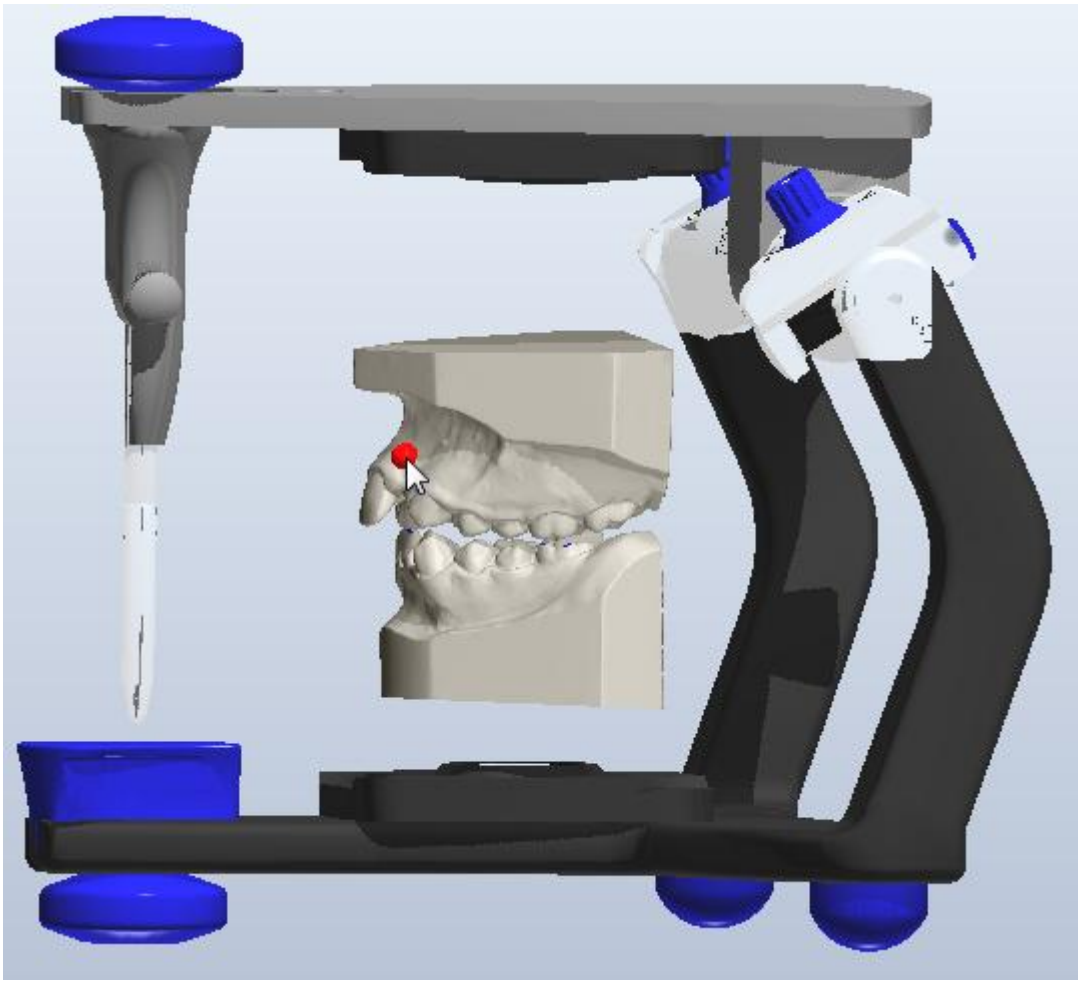


Click the  on the *Virtual articulator* window to open the settings tab.

To set **Occlusion**, **Laterotrusion** and **Pro-/retrusion** you can:

- Edit the parameters in the appropriate edit boxes in the *Virtual articulation* window.
- Move the jaws by holding the left button on the mouse (the **red** control point appears and the parameters are updated automatically).

Click the locked  button to change it to the unlocked  to free the movement of the required plane. Place the cursor over the articulator/model and drag it while keeping the mouse button pressed:



With the ***Upper*** / ***Lower*** jaw button you can select the jaw you want to move with when resetting the articulator to the static occlusion.

Click to lock the modelling position of the articulator. To set the articulator in modelling position click .

There are three options in the **Articulate** function:

- **Contact mode** - ***on/off/auto shut (AS)*** indicates the position of jaws in relation to their penetration.
- **Collide designs** - indicates in color the collisions of the jaws.

The ***Occlusal compass*** button allows to color the areas of collision according the color map **(2)**:

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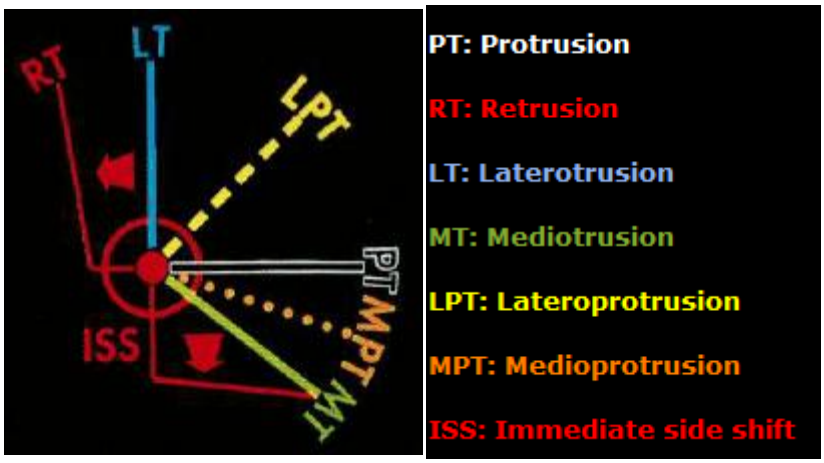
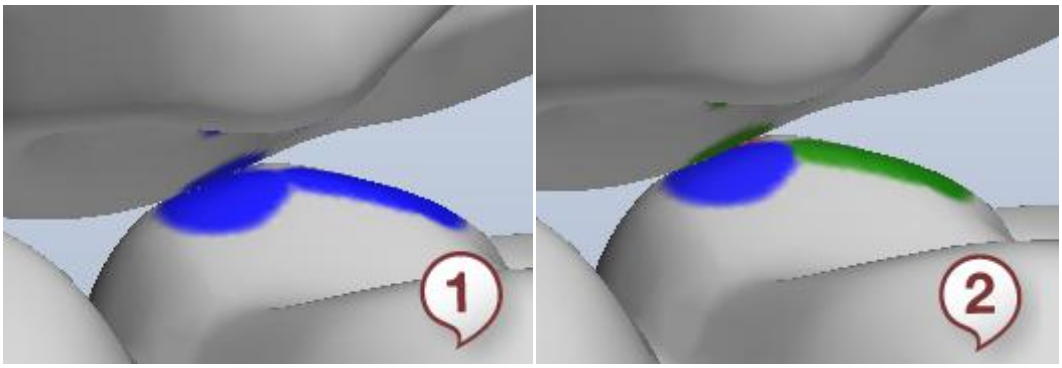


The legend of the compass

Disabled it colors the areas in blue **(1)**:

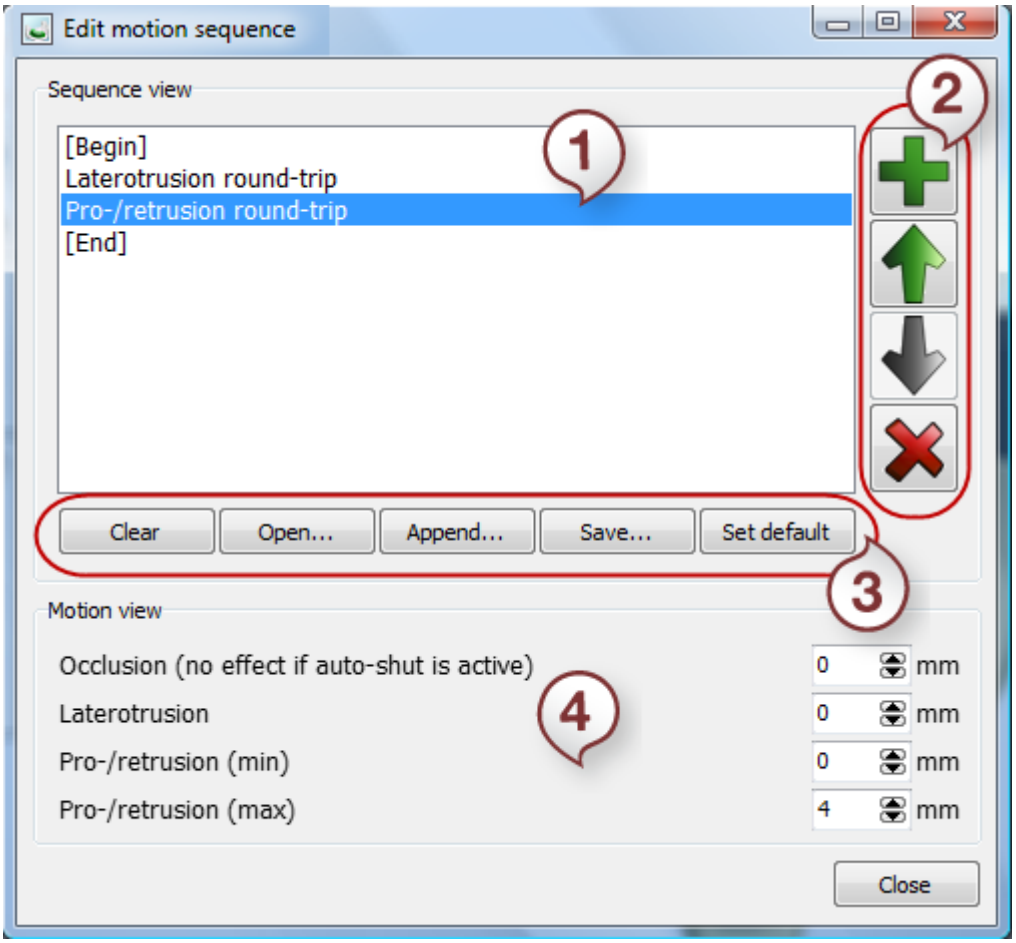


- **Record contacts** - if checked, enables the **Adapt design** section. Indicate the **Min distance** and **Adaption radius** by setting the parameters in the appropriate edit boxes.

The operation can be undone with the ***Undo*** button.

Step 4: Simulate articulation

It is possible to imitate the articulation motions by clicking the button to start the articulation. Click to pause the process or to stop it. To repeat the process click **.** Click the button to customize the motion sequence:



1. *List of sequences*
2. *Edit buttons*
3. *Operation buttons*
4. *Motion view section*

The **List of sequences** can be customized with the **Operation buttons**. ***Add new sequence***, ***Move up/down*** the list or ***Delete*** from the list.

3.4.1 How to Adapt a Parts Design in the Articulator

After creating the **Appliance** it is possible to adapt its design with the help of the **Articulator**. Please follow these steps:

▶ Step 1: Open the component you want to adapt

Click the **Feature** tab and select the component you want to work with:

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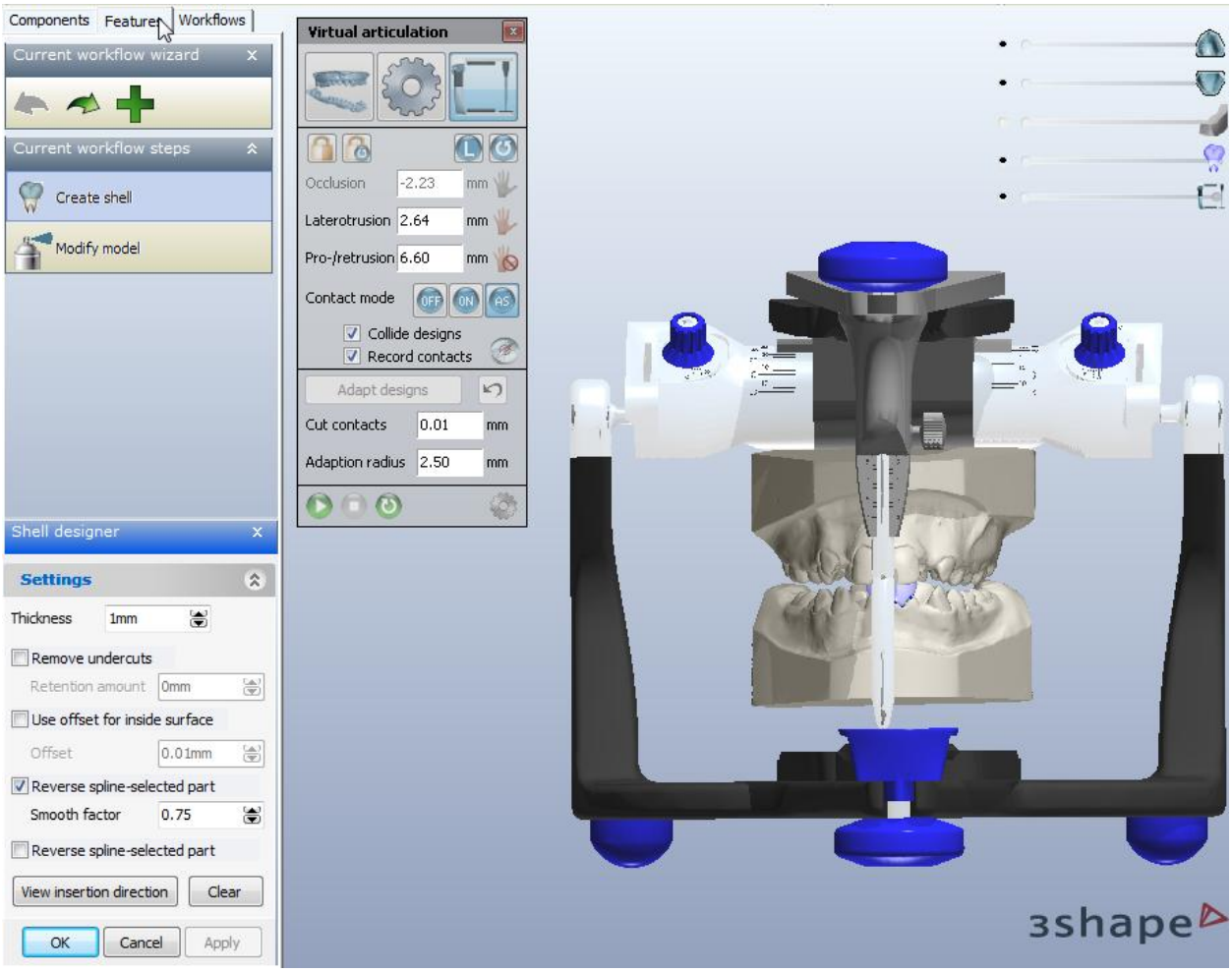
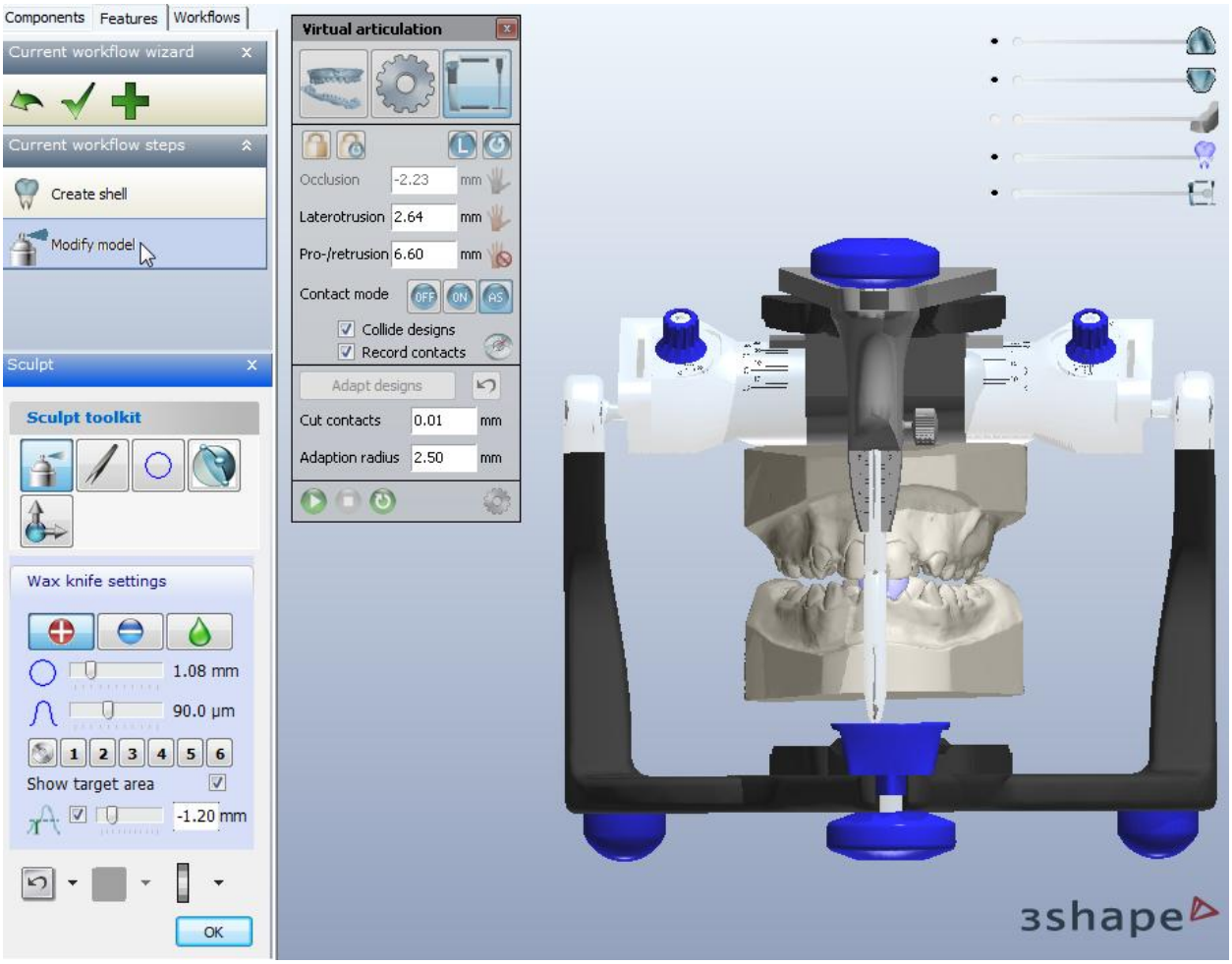


Step 2: Activate the Modify Model option

Select the **Modify model** option to open the **Sculpt:**



Note: The active **Sculpt** option is a **must** for the further process.

Step 3: Articulate

Activate the **Collide designs** and the **0cclusial compass** to view the problem areas. Move the jaws in the articulator by holding the pressed left mouse button and changing the position of the red control point:

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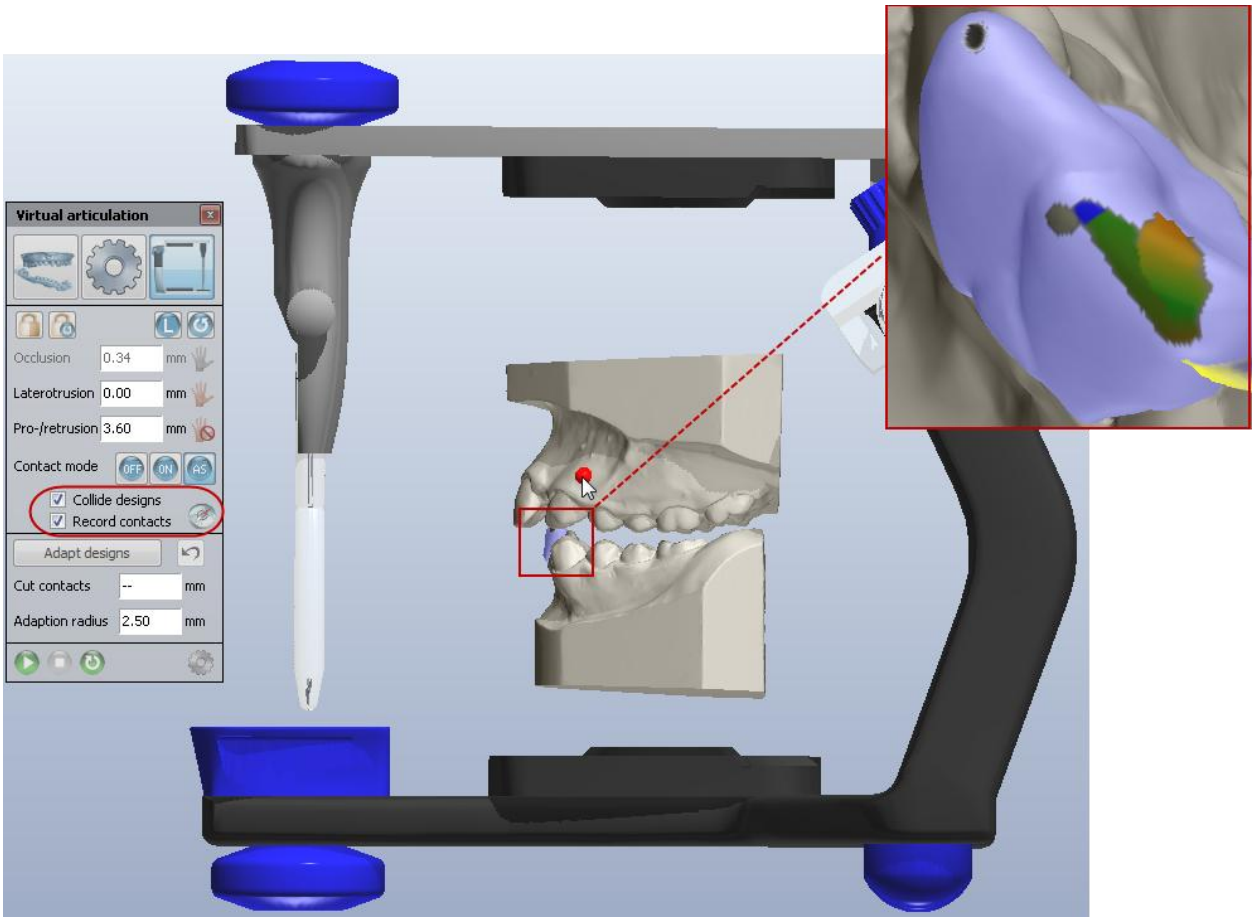
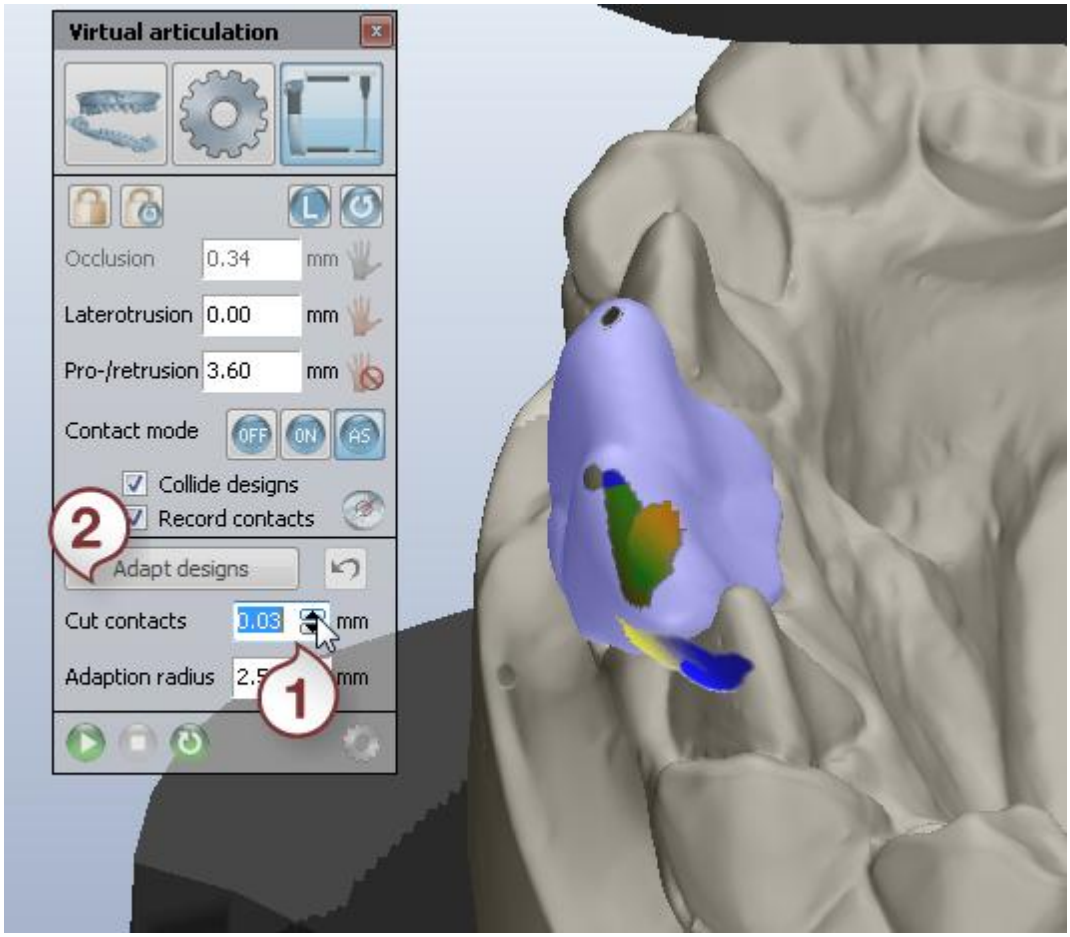


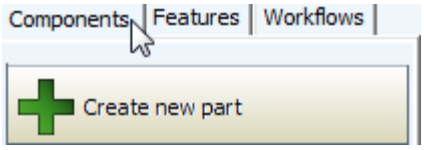
Step 4: Adapt the appliance

Enter the desired parameters for the **Cut contacts** in the appropriate editbox **(1)** and click the activated now ***Adapt design*** button to finish the process **(2)**:



4 Components Tab

Components are the main building blocks used to create an appliance part.



The main **Components** are: Shell, Bar, Combine models, Remove undercuts and Load model. The **Components** tab presents the section where it is possible to operate with the appliances for the model.

4.1 Part

Parts are the main elements of an appliance. Using parts allows for more overview in the design process and to create a file that will be exported individually. For example, for an appliance that consists of a maxillary and mandibular element, two parts will be created individually

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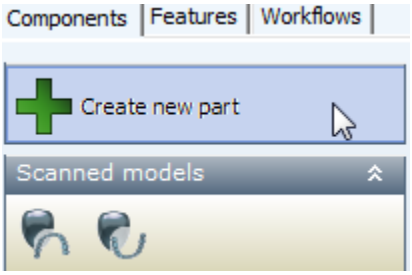
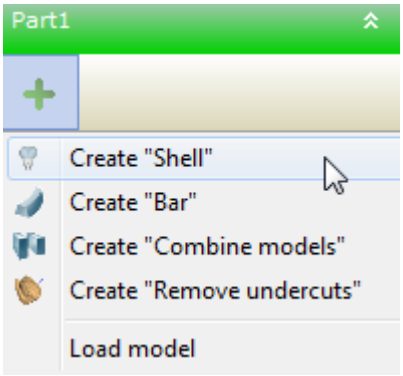


Click the ***Create new part*** button, give a name to the part and click ***OK***.

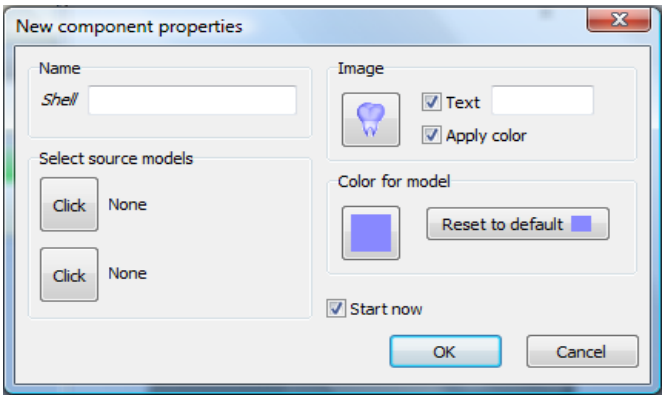
4.1.1 Shell

A **Shell** is a wrap surface with a fixed offset distance.
To create a **Shell** you need to proceed with the following steps:

Step 1: set the new component properties



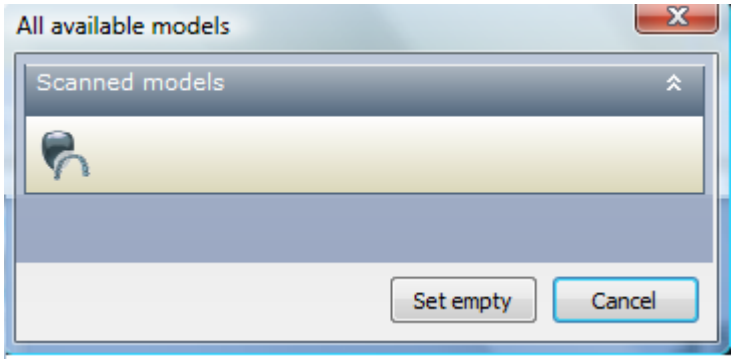
In the newly created part section click the button in the **Part** section and select the ***Create shell*** option from the drop-down list (see the image to the left).



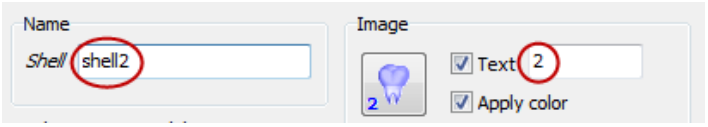
Indicate the **Name** of the shell.

Important: the name of the component corresponds to the name of the Output model!

It is possible to **select source models** on which the component is to be built by pressing the ***Click*** button. The *All available models* window pop ups(see the image below):

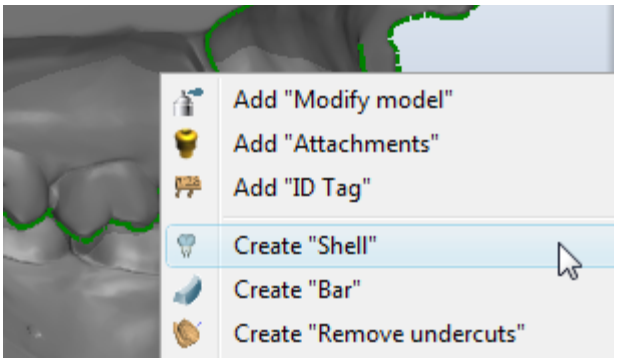


Note: Appliance Designer™ allows you to build any component on either the tooth model, or on any other component created previously.



Select the model from the list or click ***Set empty*** to leave the source model unidentified.
Fill in the **Text** that will be displayed on the ***Shell*** icon to the left. The last sign in the **Name** section will be duplicated in the **Text** section (see the image to the left).
Apply color, if checked, allows you to apply the selected color to the image icon.
Customize the color of the model click the colored square and select the desired color from the palette. To get back to the default color click the ***Reset to default*** button.

Start now if checked allows you to be immediately taken to the **Features** tab for the selected component.
Click ***OK*** to save the settings.



Alternatively, you can right-click at the desired model and open the drop-down menu duplicating the options in the **Part** tab.

Step 2: Defining a shell on a model

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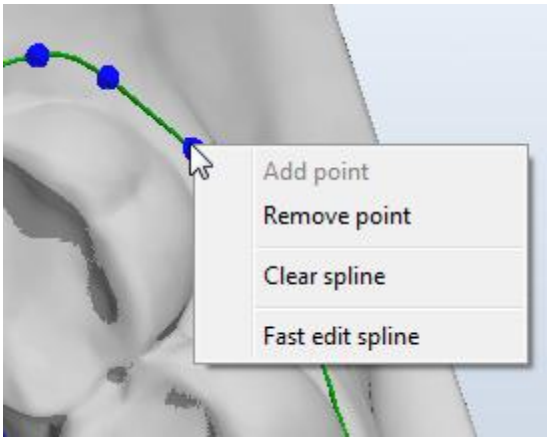
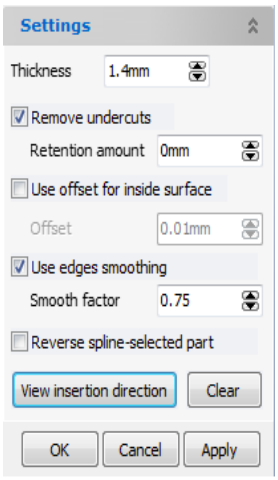
You need to indicate the area of the shell on the model. Place the control points to create a spline that will define the shell.

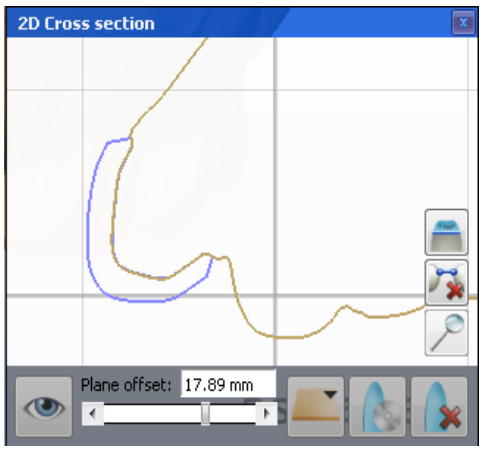


Double-click the first control point on the spline to close it. It is possible to edit the spline by right-clicking at it and selecting the appropriate options from the pop-up menu.

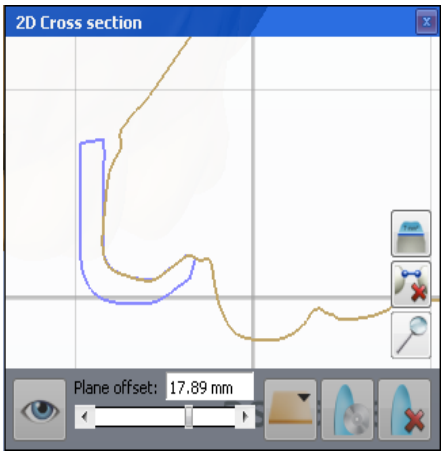
From the Shell features, it is possible to set the parameters of the shell in the *Settings* window:
Thickness edit box displays the thickness parameters of the shell.

Remove undercuts, if checked, removes all the undercuts visible from the set insertion direction to ensure easy insertion of the shell on the underlying model.





without Remove undercuts



with Remove undercuts

Retention amount allows you to define a retention factor that will compensate for the removed undercut, depending on the material to be used for manufacturing. If it is 0 then all the undercuts are removed. The larger the parameter,the more retention is provided for, in spite of the undercut removal.

Use offset for inside surface - indicates the parameters of the offset inside the shell.

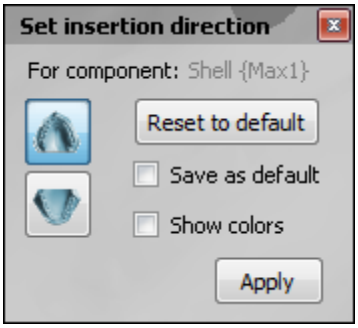
Offset - Allows you to set the offset parameters in the edit box.

Use edges smoothing - If checked, sets smoothing for the edges of the shell.

Smooth factor - Allows to indicate the parameters for smoothing in the edit box as a factor of the shell thickness.

Reverse spline-selected part - Reverses the splines placement that the opposite area that originally selected will be shelled.

Click the ***View insertion direction*** button to observe the insertion direction indicated with the blue arrow. The *Set direction* window appears:



Show colors, if checked, displays the undercuts marked by different colors according to the color scheme in the right bottom. This option is generally used for the **Remove undercuts** step.
Click ***Apply*** to change the insertion direction according to the view on screen.

4.1.2 Bar

A **Bar** is a 3D model based on a 2D profile that allows to apply predefined parameters and geometries.

Note: to be able to use Bar component, you need to define some 2D profiles of bars in the Ortho Control Panel (see the Ortho Control Panel manual for more information).

To create a **Bar** you need follow the procedure:

Step 1: define the new component properties.

This step is identical to the one from the Shell setting.

Step 2: set a bar on the model

- ***Set the insertion direction***

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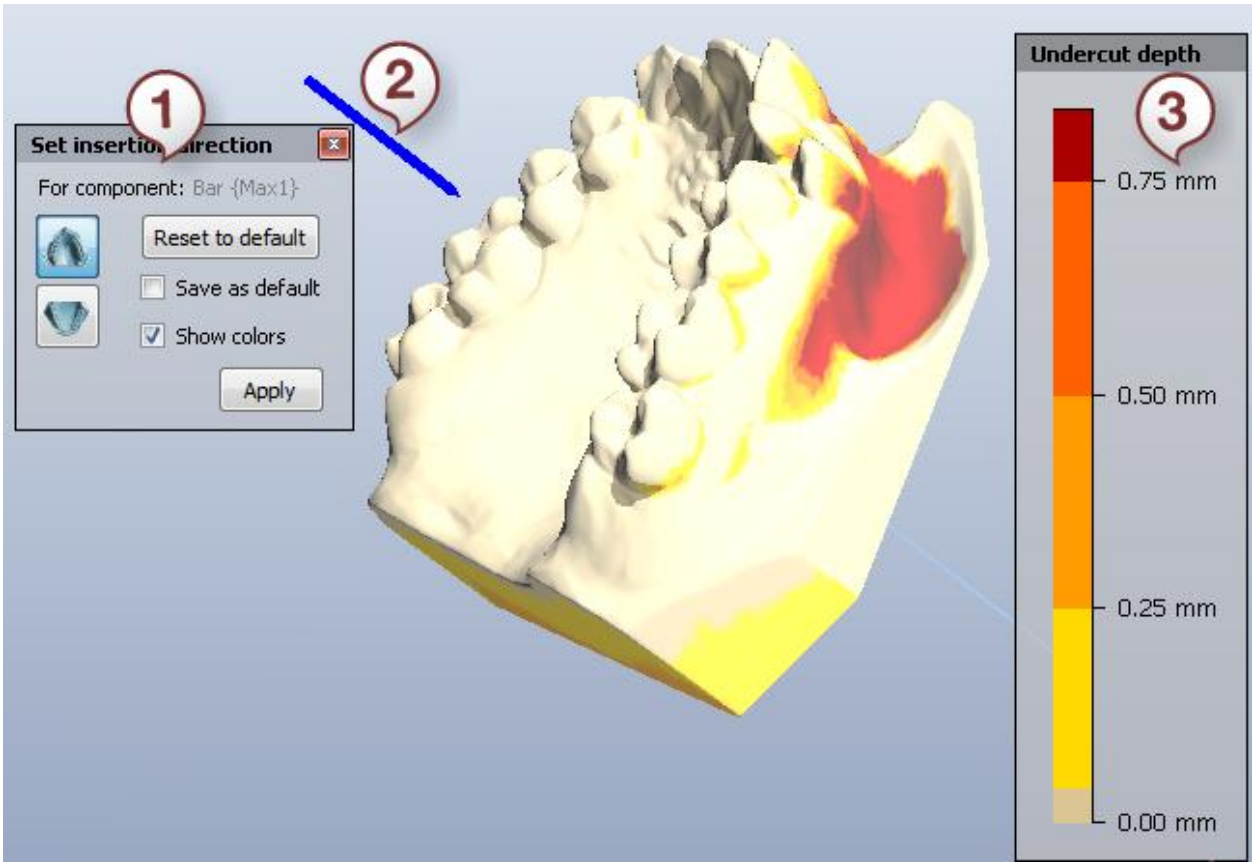


Indicate the insertion direction **(2)** in the *Set insertion direction* window **(1)** and mark the **Show colors** checkbox to observe the undercuts. The *Undercut depth* color scheme will be displayed in the right corner **(3)**.

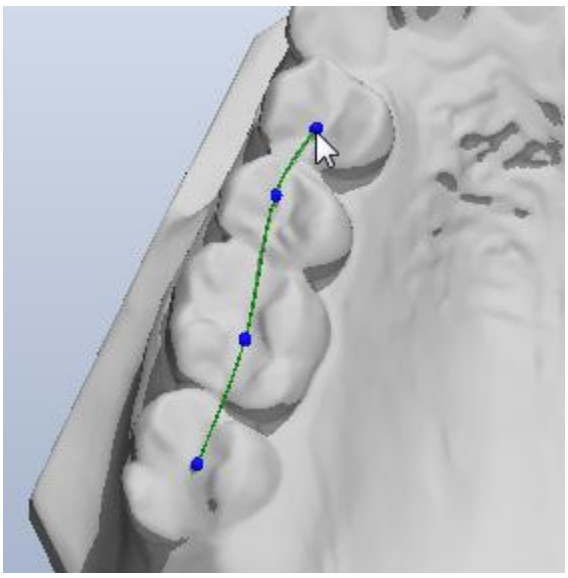
- ***Define spline***

Click the ***Define spline*** button to move to the next step or click .

1. Central spline tab

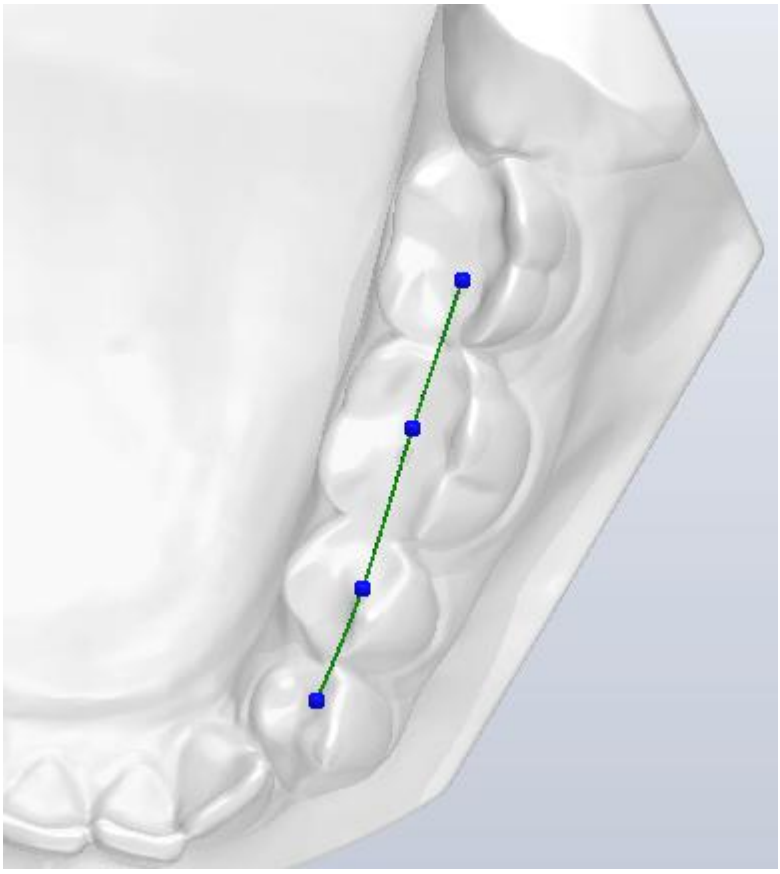
There are two ways of placing the spline:

3D spline on models

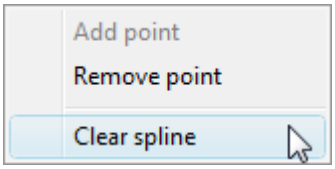


Place the spline in the desired area by simply clicking at the surface of the model.
Click ***Next*** to continue.

2D spline on models



To place the central spline in 2D mode select the plane from the **Plane** drop-down menu (you have to create the desired planes before this. For more information see the Additional Plane Views section).
Adjust the plane in the **Offset** edit box.
Place the spline of the plane and click ***Next***. To redo the spline click ***Clear spline***. Otherwise, right-click the spline to edit it:



2. Fixed-on-model spline

This option allows to create a secondary spline to limit the lower part of the bar. Click ***Next*** to move to the following step.

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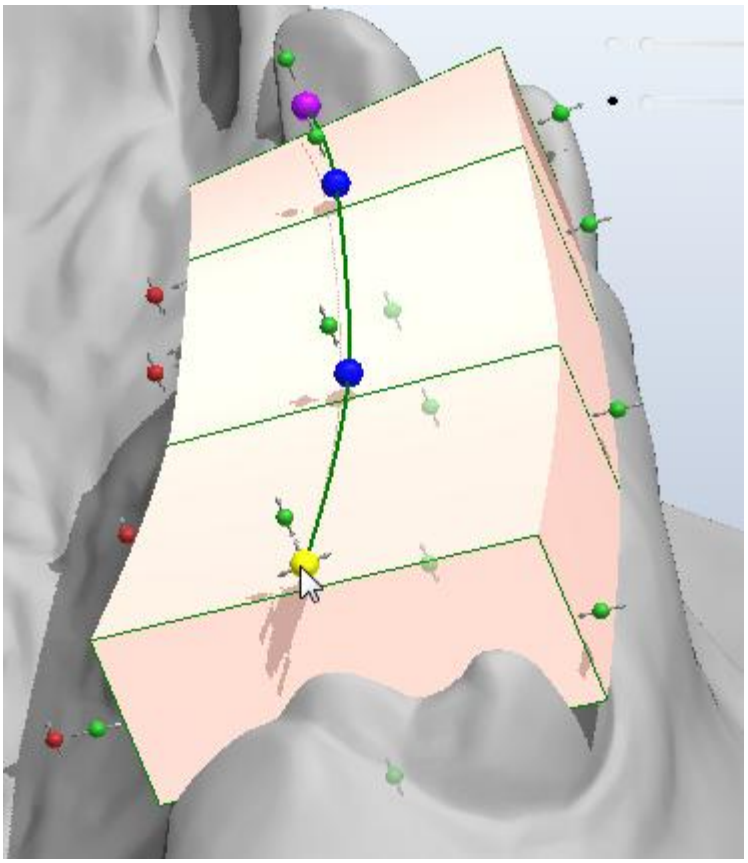
-  ***Create Bar***

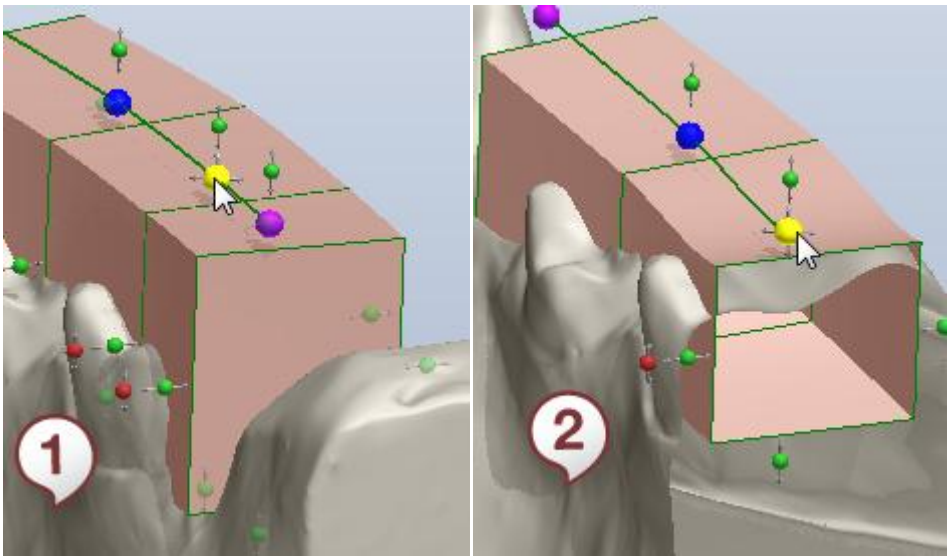
Click the ***Create Bar***  button. The bar on the model if formed. Align it using the control points:

Important: the **Bar** should be created previously in the Control Panel (see the Ortho Control Panel manual for more information).



Hold the right button of the mouse and pull the control point (becomes **yellow** when active) to the desired position. Use **red** control points to rotate the bar segments round their axis. **Green** points allow you to squeeze/stretch the bar. **Blue** control points move the bar up/down in relation to the model surface. **Purple** control points stretch/shorten the general length of the bar.

Double click at the blue control point cuts the bar at the control point for easy inspection **(2)**. To undo the cutting double click the control point once again **(1)**:

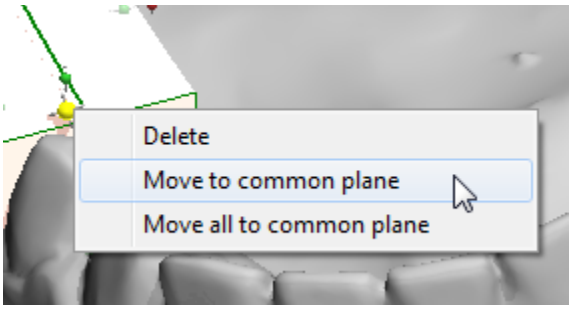


Click ***Adjust width*** to set automatically the bar width to that of the underlying model.

Select the type of the **Bar ending** from the drop-down menu (**Flat** or **Wrap around**).

In case you have selected the **Fixed-on-model** spline type, mark the **Use fixed-on-model** checkbox. **Fit bar surface to spline**, if checked, automatically aligns the bar to the secondary spline.

Press ***Shift*** to select all the control points.
Press ***Ctrl*** to select some points only.



Right-click the selected control point to open the popup menu. With this option it is possible to move all selected control points to a common plane.

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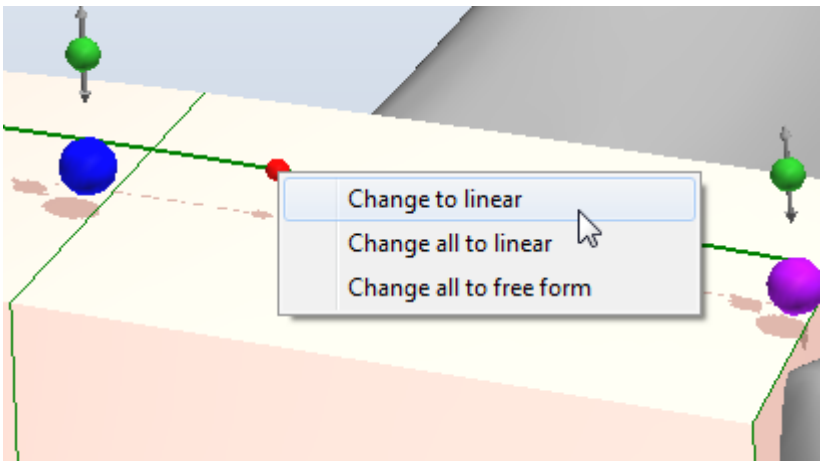


Once you place a mouse over the **green** segment a **red** control point appears. Right-click the point to open the pop-up menu.
The **linear** option creates straight segments only and the control points are positioned between them.
The **free form** option resets all the segment to their original states.

- ***Finish***

Click the ***Finish*** button to finalize the process.

Select one of the four options enlisted in the section.

Merge with model upper/lower part is available only for the bars created with the fixed-on-model spline.

Combine bar with other model. Indicate the **Remove undercuts retention amount** parameters in the edit box for the **Substract reference model** option.

Click ***OK*** to apply the adjustments and exit the current feature step.

Click ***Apply*** to make the changes but remain on the current step.

4.1.3 Combine Models

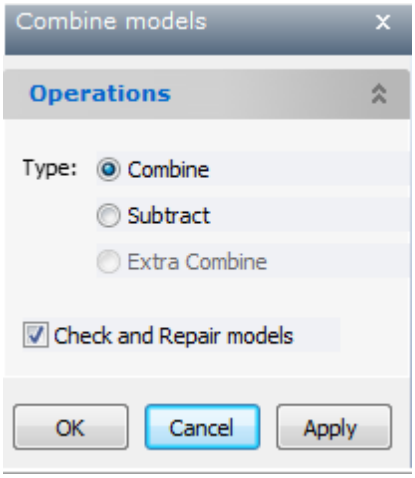
Combine models feature allows you adding or subtracting 2 models geometries.
To use this option follow the next steps:

Step 1: **set the new component properties**

This step has the same procedure as the one for the **Shell** setting (for more information see the Shell chapter).

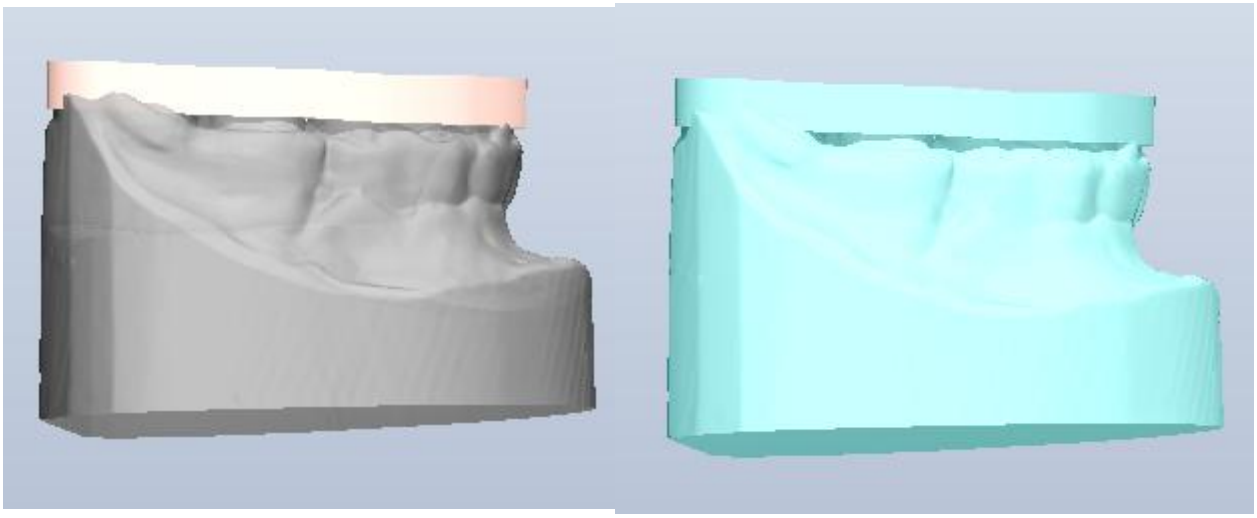
Important: the model you want to subtract from should be selected first from the list of the models.

Step 2: combine the selected models



Use the **Operations** tab in the left bottom corner of the screen to select the type of combination:

- **Combine -** allows you to combine two models' geometry into one new solid model



Bar before Combining *After Combine*

- **Subtract -** will carve one model's geometry according to the other selected model.

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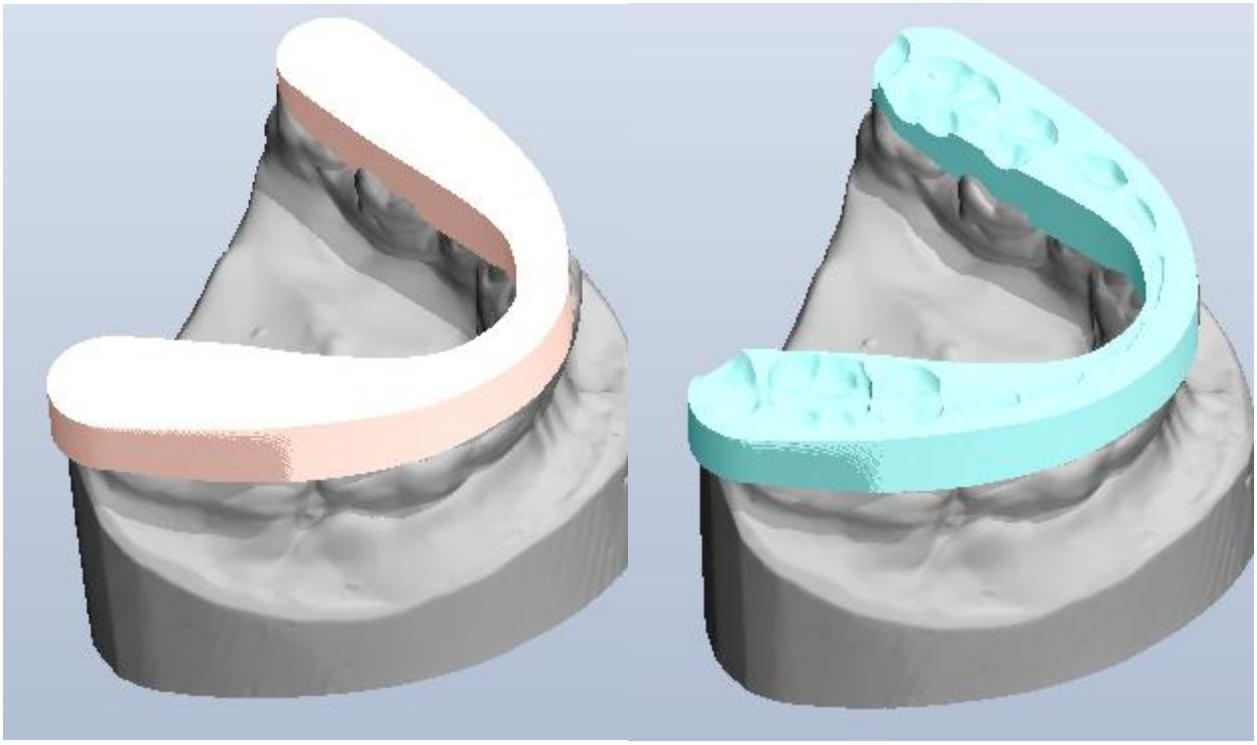
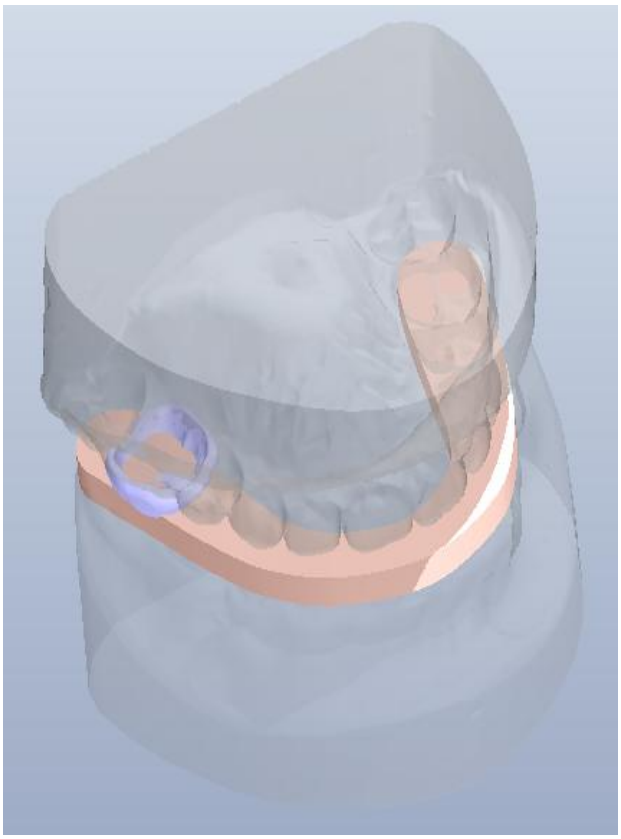
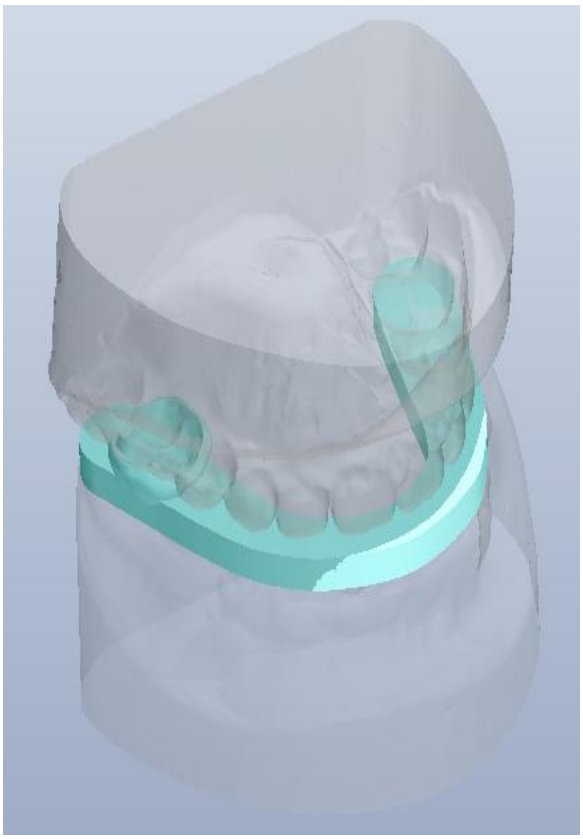


Bar before Subtracting *After Subtracting*

- **Extra Combine -** allows to combine two models while carving the underlying dentition automatically.



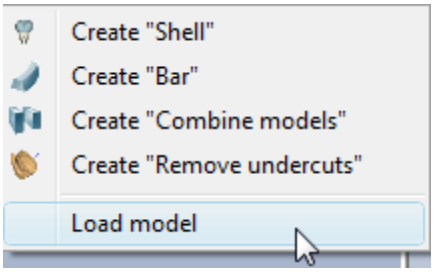
Shell and Bar before Extra Combine



Shell and Bar after Extra Combine

For higher results mark the **Check and Repair models** checkbox.
Click ***OK*** to view the result of the process.
Click ***Apply*** to save the changes.

4.1.5 Load Model



It is possible to load any STL model to the **Components** section. Contrary to the **Attachments** , loaded models don't need to be fixed on another 3D model. They get their own view slider which allows to view them as design guides without affecting the actual design process.

Click  in the Part section and select **Load model** in the popup menu. The *New component properties* window is open (see the settings for it in the Shell section).

In the **Select source model** section indicate the path a model is loaded from.

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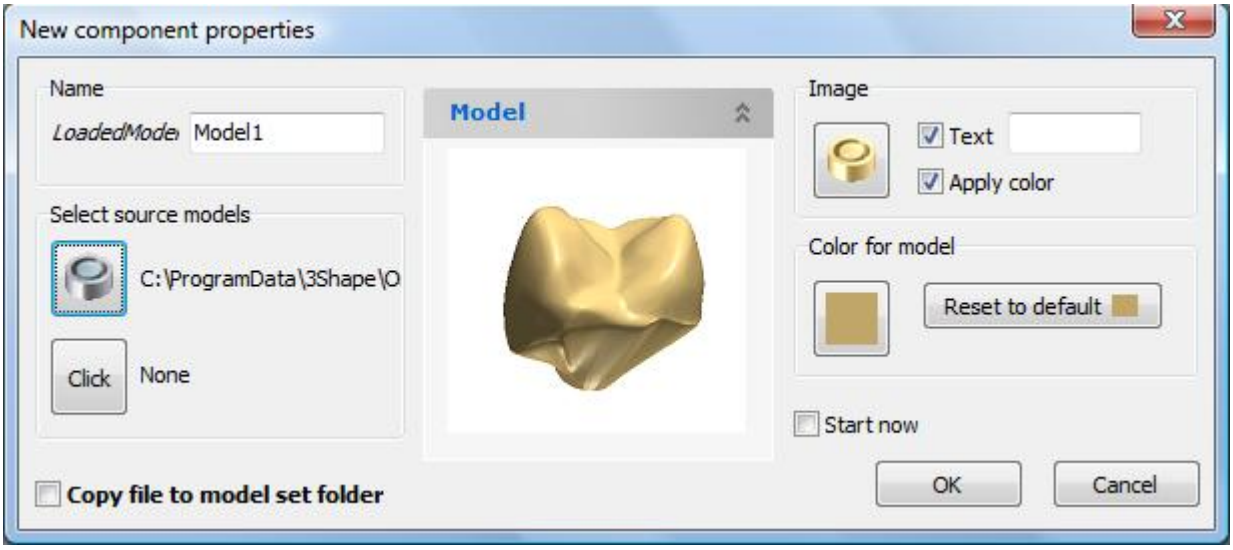
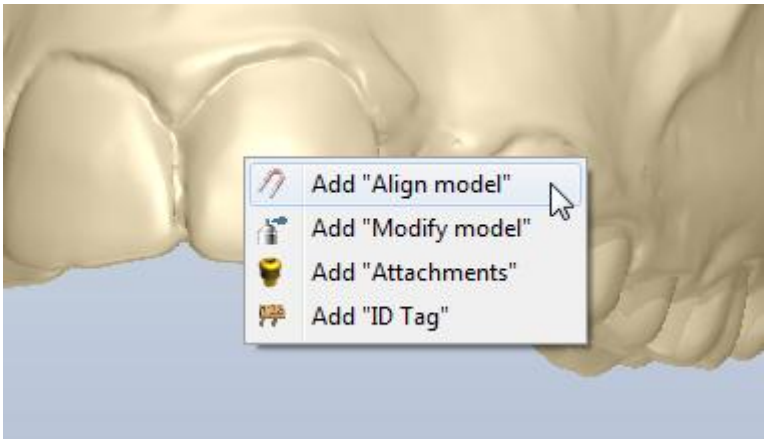


It is possible to observe the model preview. Mark the **Copy file to model set folder** checkbox to save the copy of the model.



After loading model it is possible to align it. Right-click it and select the **Add "Align model"** option in the popup menu. The *Registration* window appears:

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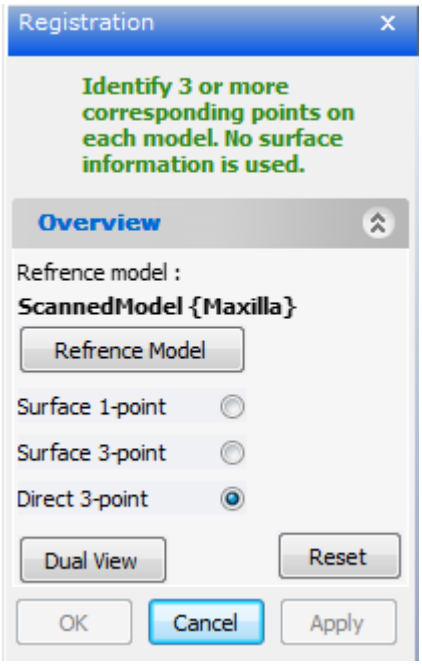
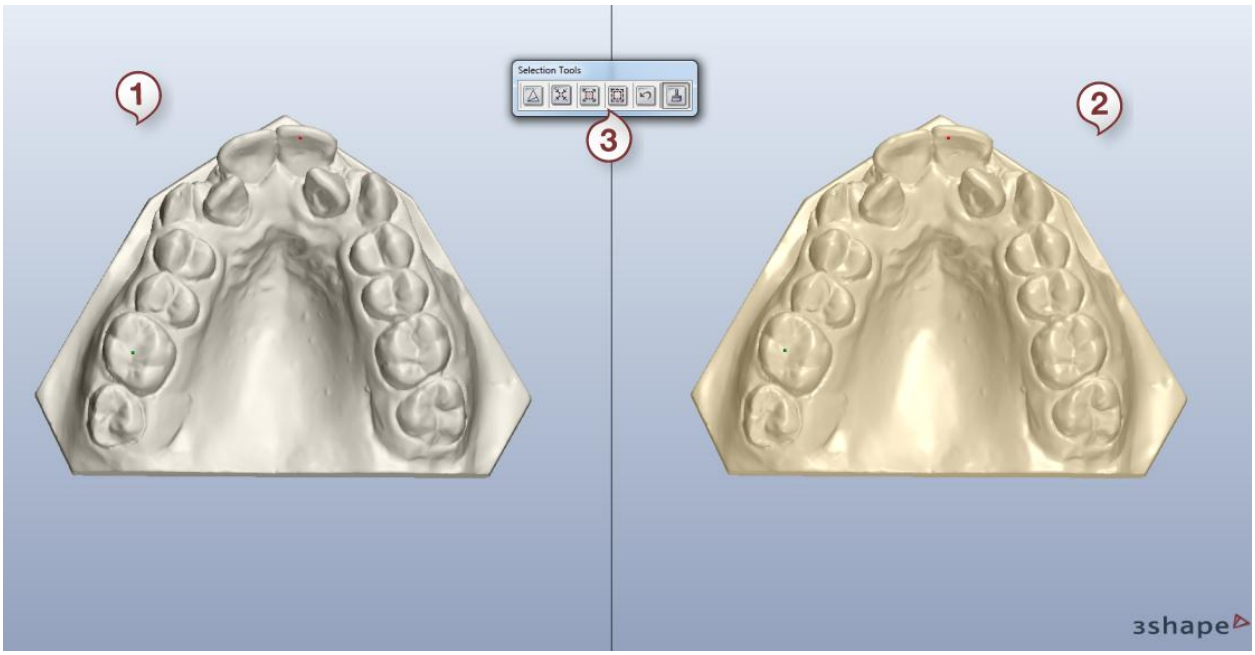
Follow the instructions at the top of the **Registration** window.



The ***Reference model*** button opens the list of the available models.
You can select the type of alignment:

- **Surface 1-point**: Indicate single corresponding points on each model for alignment and optionally select the surface to align with the help of the Selection Tools.
- **Surface 3-point**: Indicate three corresponding points on each model for alignment and optionally select the surface to align using the Selection Tools.
- **Direct 3-point**: Indicate three corresponding points on each model for alignment. This method does not use the surface information.

Note: It is advisable to use the ***Dual view*** visualization in order to access both models at the same time. The selection of points can be done on the reference model first and then on the comparative model, or vice versa. Use the ***Reset*** button to delete the reference points if needed.



1. *Reference model*
2. *Model*
3. *Selection tools*

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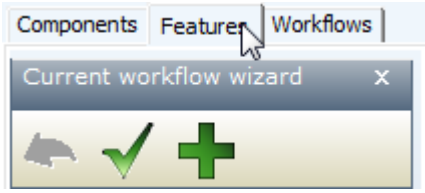
The selection tools can be used in any order and combination. They include the following:

Icon	Function
	***Unselect all*** – Unselects all selections
	***Reduce selection*** – Reduces the selected area by excluding the neighboring areas from the selection.
	***Extend selection*** – Extends the selected area by adding the neighboring areas to the current selection.
	***Invert selection*** – Unselects the selected and selects the unselected areas of the surface.
	***Undo last selection*** – Makes one step back in the selection operation
	***Paint selection*** – Use the brush tool to "paint" the model reference surface while keeping the left mouse button pressed.

Click ***Apply*** to perform the changes but remain on the current step.
Click ***OK*** to apply the alignment and exit the current option.

5 Features Tab



The **Features** are the parameters of components as described in this section. But there are also additional features that can be applied to an existing component. It is automatically open if you have marked the **Start now** checkbox while creating the new component.

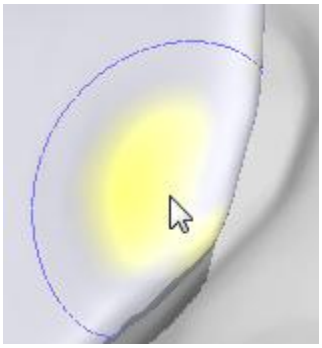
It is possible to use the **Current workflow wizard** operation buttons to customize your steps:

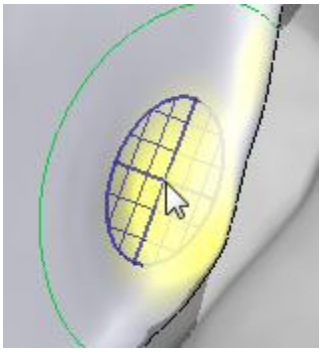
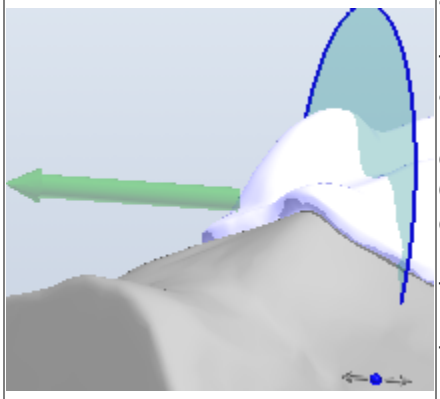
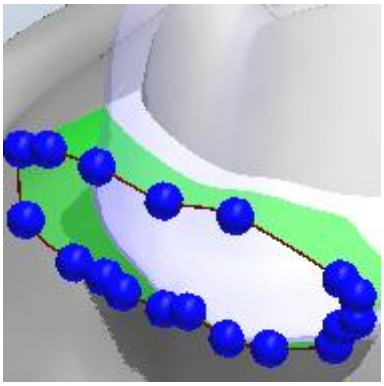
	The ***Previous*** button allows you to undo the changes.
	The ***Next*** button allows you to move to the next step.
	The ***Done*** button finalizes the customization process.
	The ***Add Feature*** button opens the drop down menu with additional features:

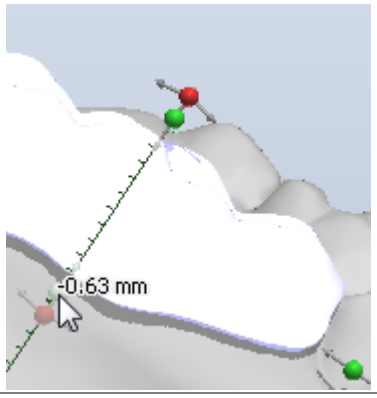
-  **Add "modify model"**

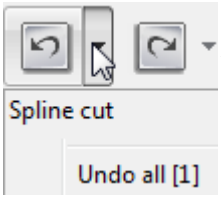
The **modify model** is a sculpting toolbox that allows you to make local changes to the selected component or model's geometry.

This option adds the **Modify model** step to the selected component or model.
This is applicable, when you have created a certain appliance and need to trim it. The *Sculpt window* is open:

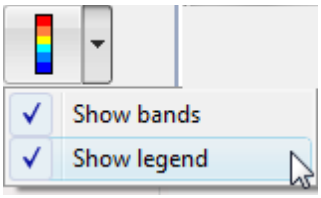
	The ***Wax knife***  option allows you to add or remove surface on the selected area. Click the ***Add***  button in the *Wax knife settings* section to add some material to the scan. The target area circle is marked red. Select the ***Remove***  button to delete extra material or click the ***Smooth***  button. The target area circle is marked blue and green respectively. Adjust the **Radius** and **Level of the marking circle** by pulling the corresponding slide bars. It is possible to use the already customized settings from the list of six variants. Activate the desired mode by

	clicking at its number in the list. To change the setting for the mode set the parameters in the *Wax knife settings section* and click the ***Save*** button. Mark the ***Show target area*** checkbox to display the marking circle.
	The ***Morphing*** option allows to force in and out the curve on the surface. You can set the radius of the marking circle by pulling the ***Radius*** slidebar. **Note**: the green outline indicates that the model's edge will be affected by the morphing process.
	The ***Plane cut*** option allows to cut the selected parts of the appliance with the plane. Draw the red line by holding the left button of the mouse. The cut plane is created. The green arrow indicates the cutting direction. You can move the cutting plane by pulling the blue control point or set the parameters in the **Offset** edit box. You can select the already created plane by choosing it from the **Default planes** drop-down list. It is possible to adjust the size of the cutting plane by marking the **Local plane cut** checkbox. Set the parameters for **Smooth radius** in the appropriate edit box. If marked the **Show cut-off part** shows the section that was cut. To select another side of the plane cut click **Swap cut direction**. Click ***Clear*** to delete the plane cut.
	The ***Spline cut*** option allows to cut-off the parts of the appliance using the splines. Mark the desired area with the splines. It will be circled with green line and the rest of the model will be transparent. It is possible to ***Save*** parameters for this option or ***Load*** the already existing on another model. Select the **Spline** type. Set the **Fillet radius** and **Plane offset** parameters in the corresponding edit boxes. Use the ***Reverse Direction*** button to change swap the selected area. Mark ***Additional plane cut*** checkbox to create the additional plane to

	Use the ***Global transformations***  option to stretch/squeeze the appliance to the desired length and height. Use the green and to move the appliance. Hold the left button of the mouse and pull the control point in the desired direction. You see the ruler to view the numerical information on the movements you perform. It is possible to rotate the appliance by pulling the red control point. Click ***OK*** to apply the changes.



You can ***Undo/Redo*** the operation by clicking the appropriate buttons. It is possible to view the history of actions performed by clicking the black arrow by each button.



It is possible to observe the changes you perform in colors. Activate the ***Difference map*** by clicking the appropriate button. Open the drop-down menu by clicking the black arrow to view the legend and customize its view.

-  **Add "Attachments"**

Attachments are 3D models that can be added to or subtracted from the surface of any model, or simply used as visual guides for the output models or appliances.

Note: attachment should be added to an **attachment group** through the Ortho Control Panel before they are available in the Appliance Designer.

Click on the desired area and place the attachment. Customize it using the red, green and blue control points:

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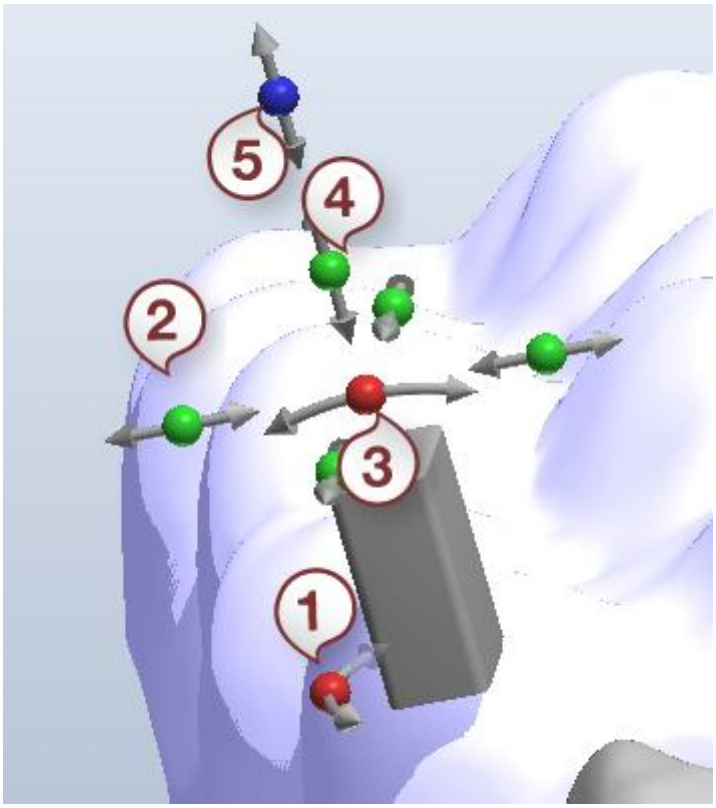


1. *Rotates attachment round its axis*
2. *Changes the width of attachment*
3. *Moves the attachment back and forward*
4. *Changes the height of attachment*
5. *Moves the attachment in relation to the surface of the appliance*

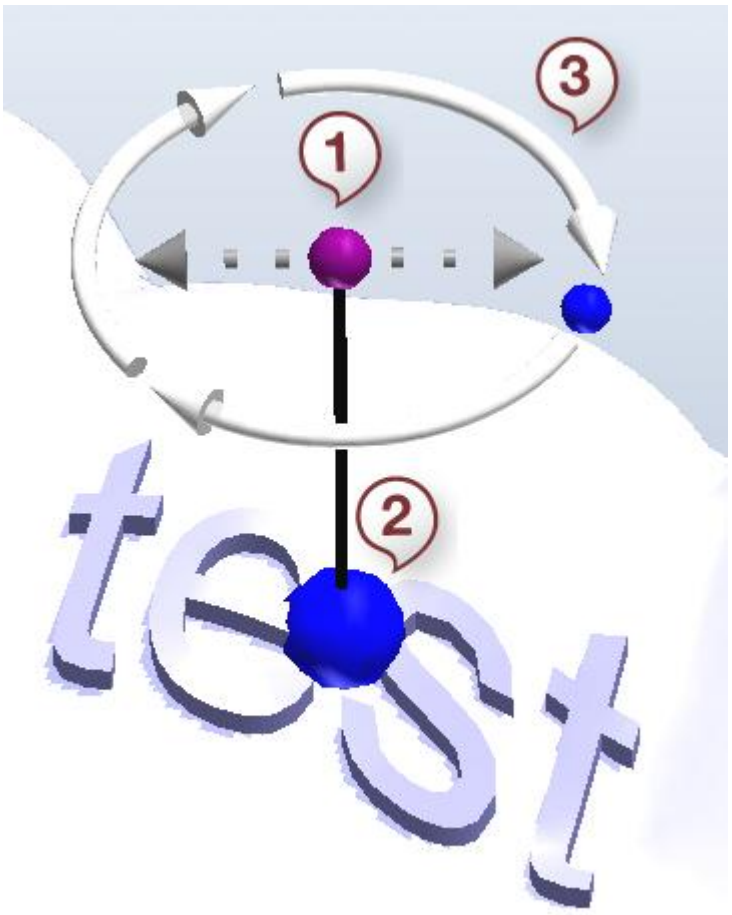
Note: Attachments properties can be edited in the Ortho Control Panel: it is for example possible to lock the scaling parameters (i.e. 2 and 4 in the image above), so the attachment's geometry cannot be changed during the design process. This is particularly useful when attachments are standard physical parts that will be mounted in the designed appliance.

It is possible to select the **Group**, type of **Attachment** and **Default orientation** by choosing them from the corresponding drop-down menu in the *Attachment settings* window.

- **Add "ID tag"**

ID Tags allow you to add easy identification to any output model for easy sorting in the manufacturing equipment.

Click at the desired area on the model. It is possible adjust the ID tag in the desired position with the help of the blue and purple control points on the joystick:



1. *Changes the angle of ID tag position according the surface*
2. *Moves the ID You can setad along the surface of appliance*
3. *Rotates the ID tag round its axis*

You can customize the look of the Id tag in the *ID Tag generation* window. Type the **ID Text** and select the drop-down menu (**Detachable** or **Integrated**).
Indicate the **Text Depth** and **Font Height** in the appropriate edit boxes (for **Integrated** ID Tag type).
Place one control point in the desired area to indicate the detachable ID tag and click ***OK*** to view the result:

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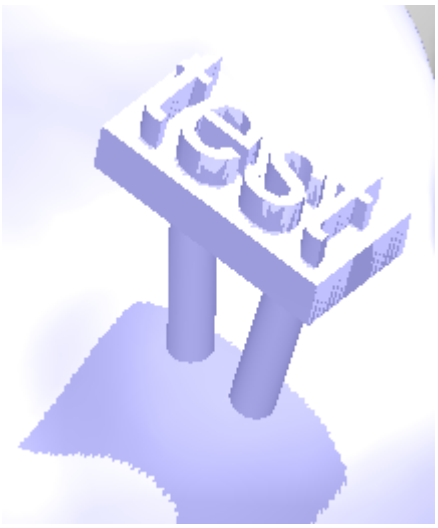
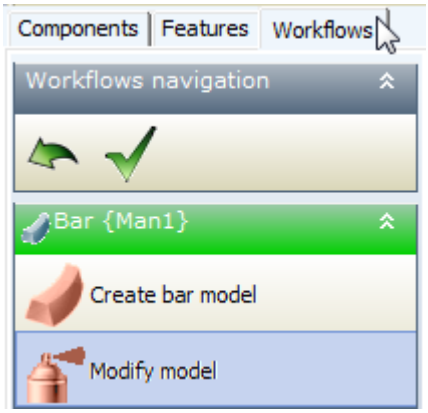


Set **Connectors count** and **Min. distance to Model** in the corresponding edit boxes (for **Detachable** ID Tag type).

6 Workflow Wizard



The **Workflow wizard** section presents the list of actions performed to create the selected part.

It is possible to navigate the list by using the  or  buttons.
Click the arrow buttons in the upper right corner of each section to fold/unfold it.
You can select any step from the list and modify it using its specific settings displayed in the left bottom corner.

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