imes-icore®
Dental & Medical Solutions

Manual

iCAM V5 smart

CAM-Software

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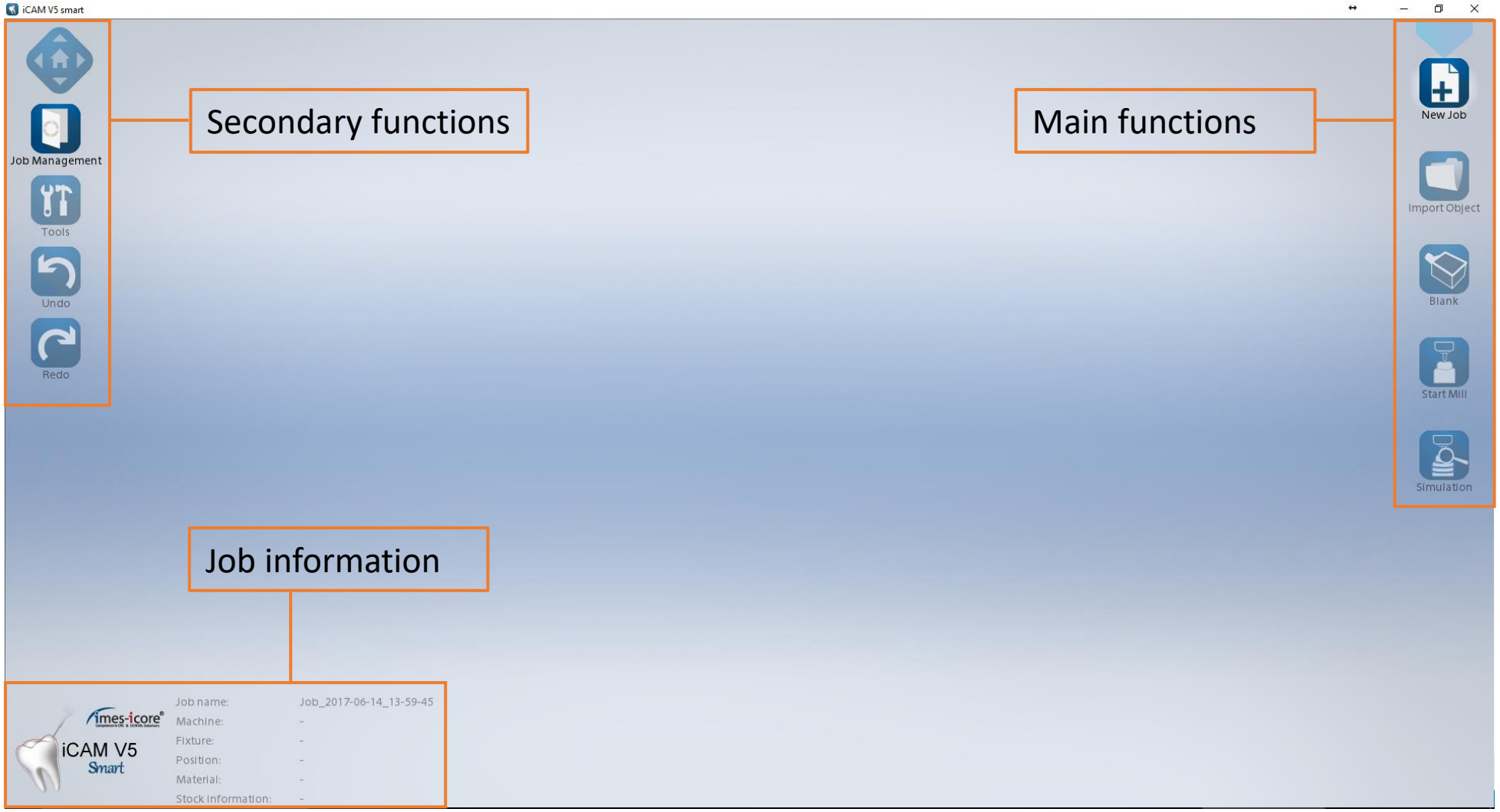
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Machine: -
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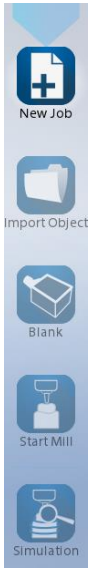
2 Main functions – preparation



A. New job/project

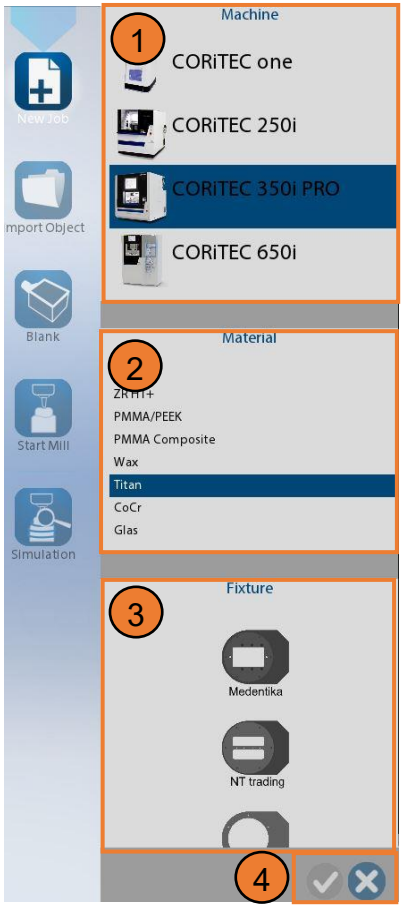
Everything milling job starts with a new project. Once a job is successfully, the project is finished. For the next job(s) a new project is needed. Every project requires the definition machine, material and holder. The stl to mill and the blank will be chosen later on.

1. Choose machine
   This selection depends on the installation. If there is only 1 machine, it will be preselected.

2. Choose material
   This selection depends on the machine. Not all machines use every material.

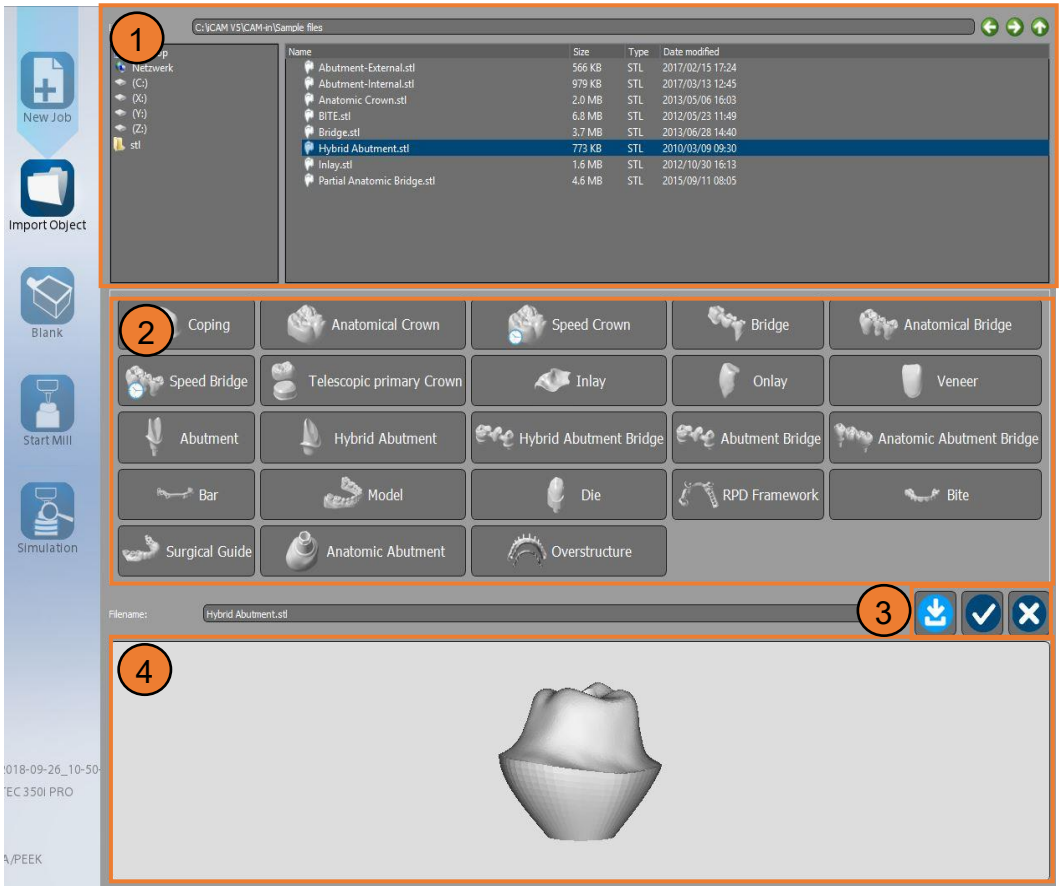
3. Choose holder
   The holder decides which material type is used (disk, premilled or block). In case of premilled or glass ceramics the position has to be chosen in the next lower section of the side bar (it will appear after the holder was chosen).

4. Confirm



B. Open stl

After confirming the new project this side bar will automatically open and prompt to choose



1. stl
   The top part works like a windows explorer. Usually the preferred folder will be pre-selected

2. object type
   The object type decides the milling steps (see chapter 10).

3. and confirm it
   - left imports and leaves this window open
   - middle imports and closes this window
   - right aborts

4. stl-preview
   This last part shows a preview of the stl. Click on it to open a window which shows a rotatable 3D preview of the stl. Further information in the window will show
   - optimized height and optimization angle (dependant on the max angle and checkable box)
   - Height Z scaled (depending on the scaling factor e.g. 1,25 for ZR)

If a blank is already selected, the heights of the stl will be shown and coloured depending on their value
- Red: stl is too big for the blank
- Yellow: stl is smaller than the blank but nearly the same
- Green: stl fits into to the blank

C. Blank

This window opens automatically after the stl selection. It will show a list of blanks that are available for the project depending on your selection in section **A**.

1. shows minimal required blank height
   the blank height should be bigger else parts of the job may be outside the blank and not be milled

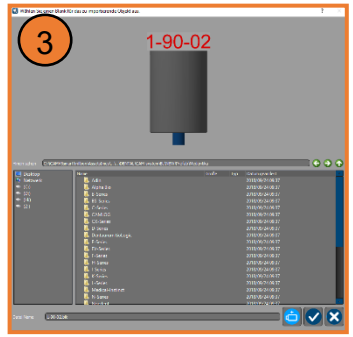
2. prompts you to choose blank if disks are used
   (click on a symbol next to the height to use)

   
   creates new blank and a pop up will ask for
   - blank name
   - LOT number
   - scaling factor if necessary (in case of e.g. ZR)

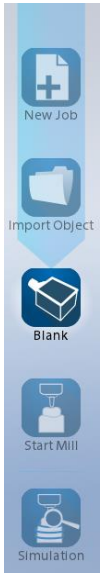
   
   opens existing blank (view page 9)
   (Some materials have more than one disk type. Those will be saved in different folders. That is why not all blanks may be shown in the submenu.)

- prompts to choose a block if a glass ceramics holder is used
  click on the  next to the block to be used (last number of its size equals its height)

3. prompts you to choose the matching blank for your stl when using premilled abutment
   In case of Medentika and nt-trading the software is able to choose the blank automatically if the necessary meta-data is provided.



(only one ‘prompt’ will appear depending on the necessary blank)



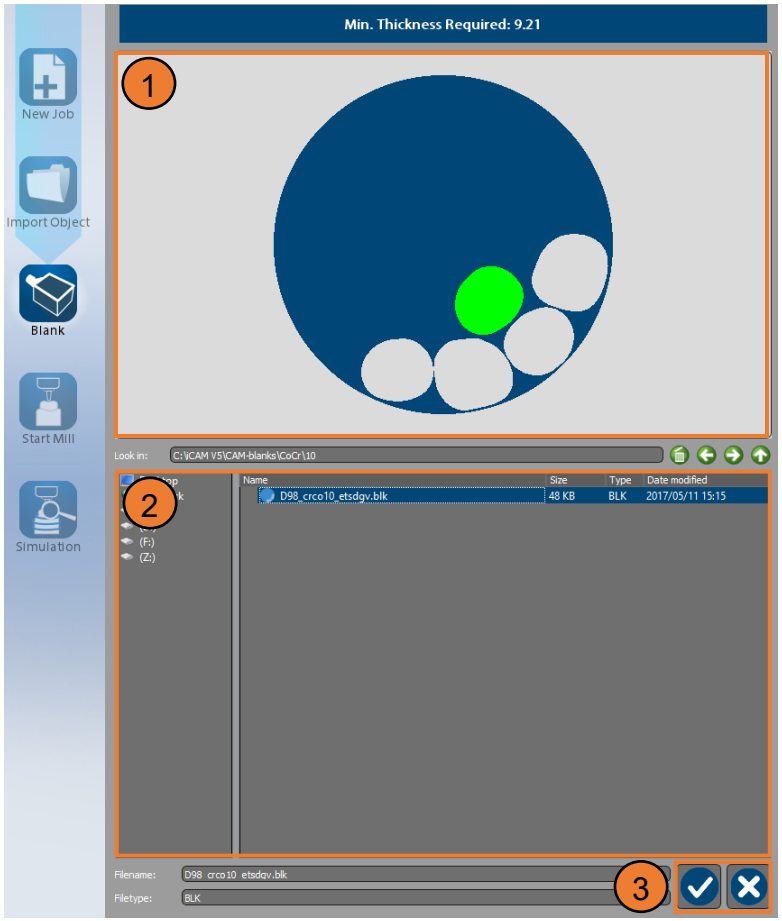
C. Blank – open existing blank

This submenu opens if a number symbol was clicked. 

1. blank preview
   This preview shows the selected blank with all existing holes in it. Position(s) of current stl(s) will be highlighted in green. If it is impossible to position all stls with at least 1mm gap to every other and the blank limitation, a message will appear which says that not everything fits inside this blank.

2. blank selection
   The blank selection works in this case like the windows explorer. The folder with the chosen height and blank type will be preselected. Click on one blank from the list to get a preview (see above). Since different blank types (e.g. multilayers) are saved in separate folders, not all possible blanks of the chosen height may be shown.

3. confirm selection
   Double clicking the selected blank also works.



3 Editing the Job

Once all of the above selections have been done the chosen job will be shown in the blank. It might not be in the preferred position or may need some finishing touches. This section will cover the basics whereas chapter 5 will explain secondary functions in depth. First thing to know is the handling with the mouse:

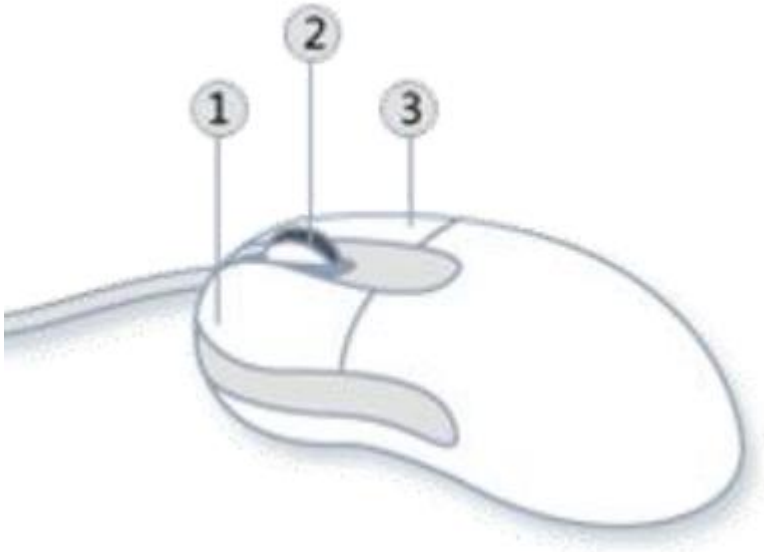
1. Left Button = select

2. Mouse Wheel

a. Click and hold = Move View

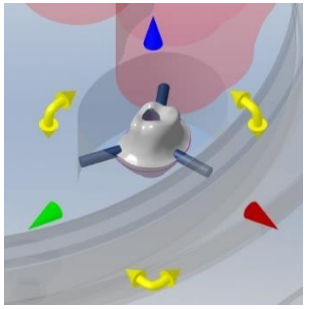
b. Roll = Zoom

3. Right Button = Rotate View



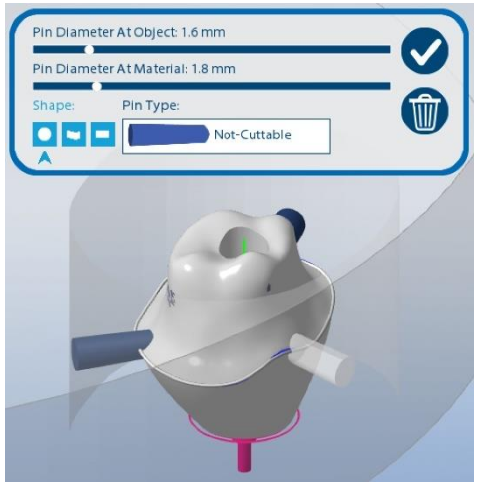
Note: Clicking means a left mouse button click going onwards.

Moving/Rotating



There are two ways of changing the position of the stl: dragging it around or using the arrow buttons.
Dragging it means click on the stl and hold the mouse button. Then simply move the mouse around and the stl will follow. Thus the stl can be positioned in the XY surface. If the blank limit is exceeded, the stl will turn yellow or red.
To use the arrow buttons click on the stl. 3 (if the view involves only 2 room axes) or 6 (if the view involves all 3 room axes) arrow buttons will appear. Click on them and hold the mouse button, then move the mouse. The single arrows (red, green and blue) will make the stl move in one of three room axes, whereas the yellow double arrows will rotate the stl. If the rotation exceeds the possibilities of the machine the stl will turn red (if the mouse button is released at this point, the stl will rotate back to a for the machine possible angle).

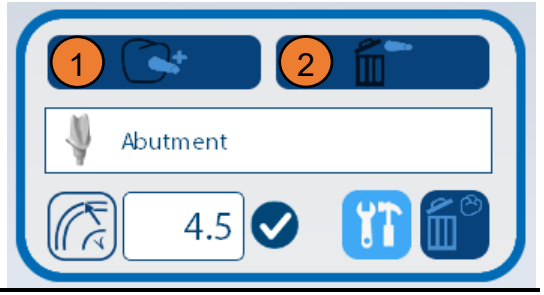
Sprues (Supports/Support Pins)



Clicking on a sprue will highlight it, highlight the equator line in white on the stl, and open the change sprue menu at the top center of the screen. In the change sprue menu the size, shape and cutability can be edited.
The size is usually a predefined value of decent size due to experience. If the sprues are too small they may not support the milling process.
The shape can be edited to round, flowing or square. Flowing means that it will follow the equator line in horizontal shape. The pin type defines wether a sprue will be cut at the end of the milling process:

- Not-Cuttable sprues will not be touched
- Semi-cuttable sprues will be cut 50% in a straight cut
- Cuttable sprues will be removed completely by cutting along the stl surface

Leaving the menu requires a click on the confirmation icon.



Clicking on the stl activitates the window on the right (it will appear in the center top on the screen).

1. Will activate the add sprue menu (it has the same options as the change sprue menu; see middle picture). Moving the mouse on the surface of the stl will show a preview of the sprue that will be positioned. Clicking will add the sprue.
   Leaving the menu requires a click on the confirmation icon.
2. Will delete all sprues of the selected stl.

Changing the object type

It is possible though not recommended (since many identification processes run during the import) to change the object type of the stl. Click on 3 to open a dropdown menu and select another object type in it. More information on object types and how they affect the milling process is written in the document "Necessary-tools-for-strategy-options-in-iCAM-V5-smart-Release4.18.pdf" which is available in the online instructions.

Changing the offset

In the text box 4 you can enter a value and click confirmation to change the size of the offset area of a stl. The predefined size is due to experience and necessity for certain operations thus it is not recommended to change this value.

Deleting a stl

Click on a stl to select it. If more than one stl shall be selected, press and hold the CTRL button and click on other stls to select them. Now either hit the delete button or click on the button next to 6.

5 opens a submenu with different options.

0°-orientation

This option is available for single units (e.g. abutment, crown). Clicking will align the insert direction of the cavity or drilling axis to 0° (and rotate the stl 90° in XY area).

Information

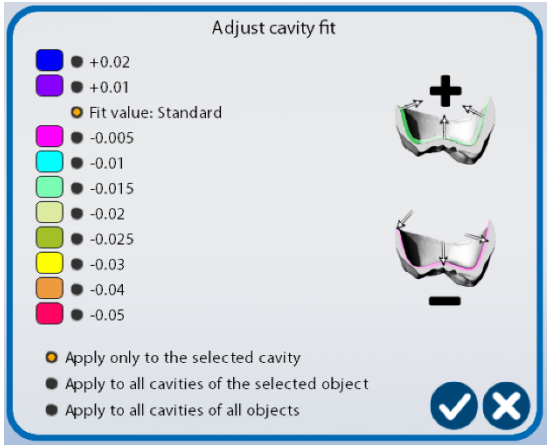
This option shows information shows the stl's name, blank volume and the number of its elements if there are more than one.

Thickness

This option opens a settings window and calculates the wall thickness of the stl. It will then show the thickness colour coded (adjustable) on the inside (cavity or screw channel) of the stl.

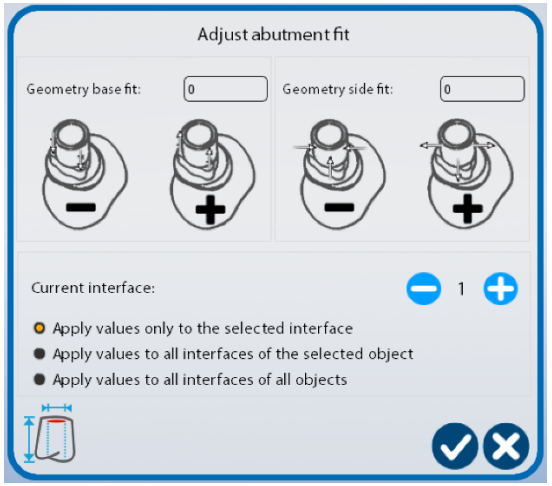
Adjust cavity fit

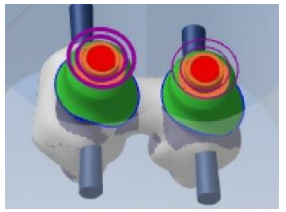
This opens the sub menu shown on the right. It is used to change the fitting of cavities (mostly for telescopic crowns). The ‘standard’ parameter is the fitting dependent on the CAD settings (usually this fits to the base). If it does not fit, another fitting value can be selected to change the cavity size (all values in mm). This will be shown by changing the cavity’s colour corresponding to the chosen value (can be hideable). Bigger values (+ direction) make the fitting tighter, smaller ones (- direction) make the fitting looser.
The change can be used on one cavity (if it was preselected with a click inside the cavity when the stl was selected to get into this submenu), on all cavities of the selected stl, or on all cavities in the project.



Adjust abutment fit

This opens the sub menu shown on the right. It is used to change the fit and position of abutment interfaces. The standard parameter of 0 is the fitting dependent on the CAD library and design. The geometry base fit adjusts the position in axis direction (small arrows indicate the + and – directions). The geometry side fit allows adjustment of the fitting parameter whereas bigger values (+ direction) make the fitting tighter, smaller ones (- direction) make the fitting looser.
The current interface selection is shown, if the stl has more than one interface (and the option ‘apply only to selected interface’ is chosen). It can be used to switch between the different interfaces and highlight the current one with a fat violet base curve (as shown in the lower right picture).
The change can be used on one interface (if it was preselected with a click on the interface when the stl was selected to get into this submenu), on all interfaces of the selected stl, or on all interfaces in the project.





Delete stabilizer

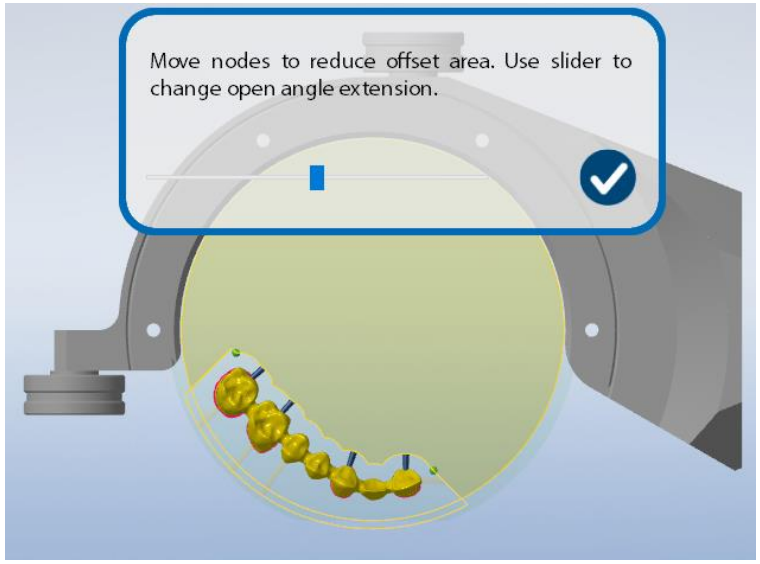
This option is only available if a stabilizer was set. Click on it to delete the stabilizer.

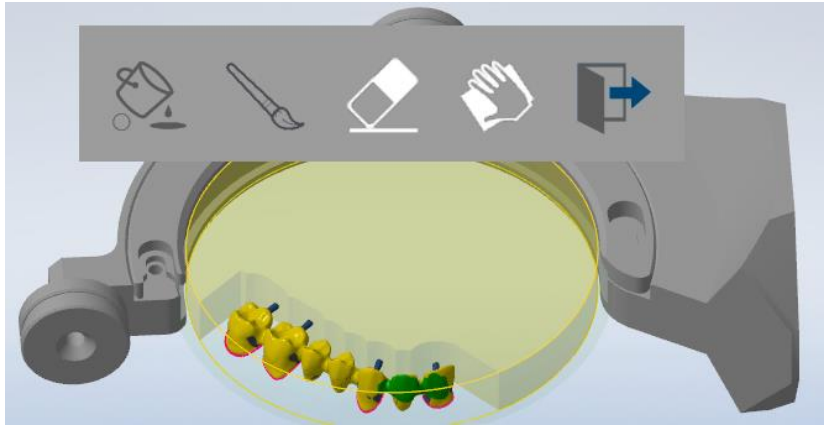
Open C-Clamp border

This operation only appears if a C-Clamp adapter is used. It will open the blank border so that a milling operation from A90 or B90 (depending on the type of C-Clamp) is possible.
After activating the function the border will open, two green dots and a window to customize will appear.

- The green dots are used to set the area of opening
- The slider is used to set the angle of opening

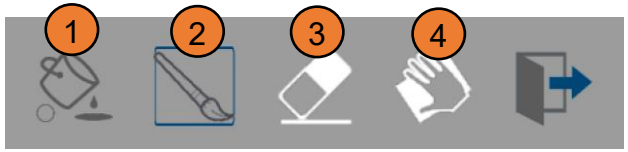
When the setting is confirmed the software will automatically switch to the brush function and a window will appear asking if areas to be milled should be selected. Only brushed areas will be milled from 90°.
To undo this whole operation reclick on its button.



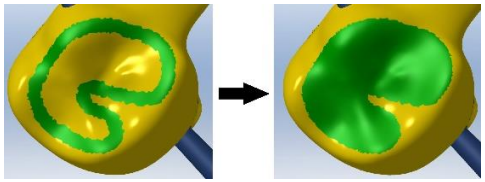


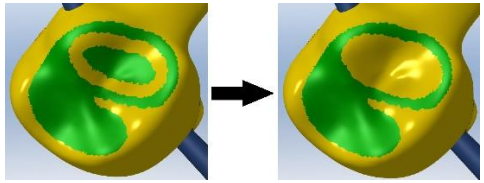
WARNING: At the moment this operation cannot be used at the same time as the brush function (see chapter 7.4).

How to use the brush function:



1. This option can be toggled. It works in combination with 2 and 3. If activated it will fill completed curves with colour or empty them.


in combination with 2


in combination with 3

2. This option is the standard when the select tool is activated. It will turn an area around the mouse pointer coloured. The size of this area is always the same. Zooming just changes the size of the stl compared to the area. Click the mouse and hold it to use the pointer as a brush to colour areas of the stl that require a special milling operation.
3. This option is the opposite of 2. It deletes coloured areas.
4. This option clears all coloured areas from the project.

Select all sprues

This option selects all sprues of the stl and opens the ‘change sprues’ menu.

4 Main functions – Calculation & Simulation

D. Start mill

The first time this menu is opened in a project it will not show options 2 and 3.

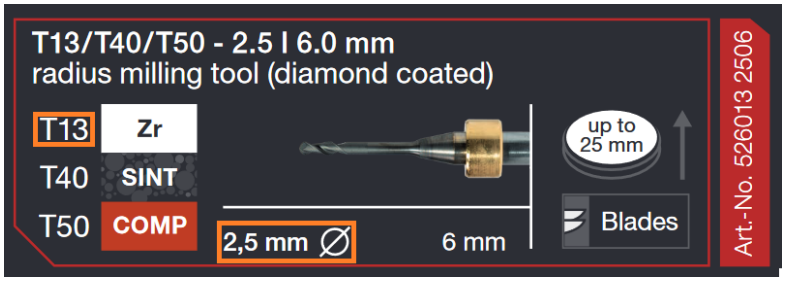
1. Starts the calculation which will generate the NC files for the machine. At 5% a window will open and prompt to choose strategy options (see chapter 10 for more information). After choosing the preferred options the selection must be confirmed. Afterwards a window will open and show the necessary tools for the milling process (see second picture on the right; a list of all imes icore tools can be found here).

   The tools table will show a list of the tools that are necessary for the milling operation. For example the first item in the picture on the right says

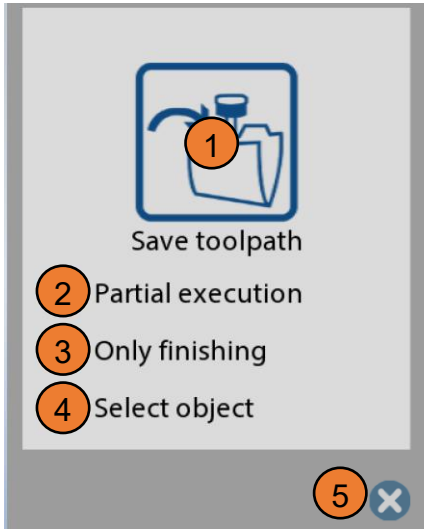
   13 – 2,5mm radius (l=20mm)

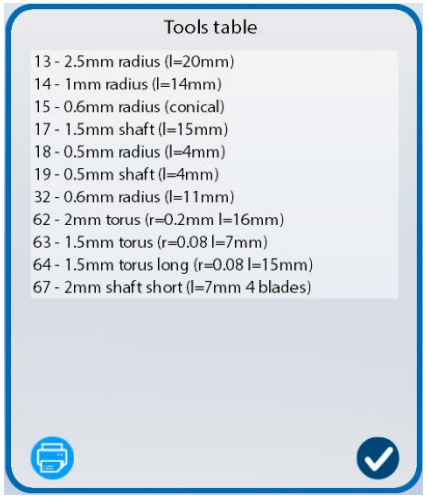
   - 13 – refers to the tool number T13 which is used by the machine to identify the tool (some tools have more than one number depending on the material they are used for).
   - 2,5mm radius – refers to the diameter and form of the tool

   This information can also be found in the catalogue:

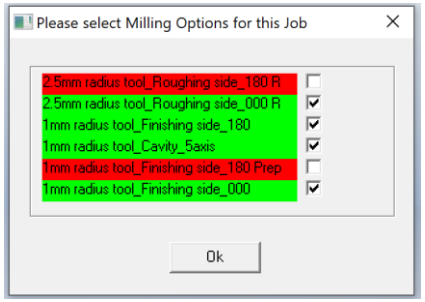


   At the end of the calculation another window will pop up and inform that the calculation is done.





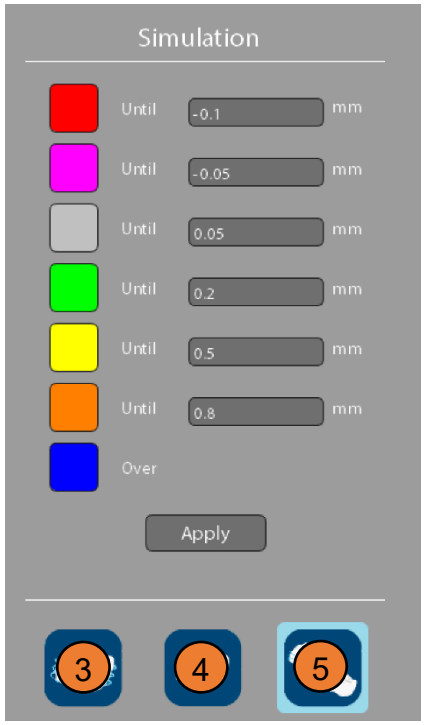
2. Allows a partial execution. This option is necessary if the machine stops milling before the project is done (e.g. due to a power shortage) or if further options shall be used after parts of the project are already milled.
   Since not all steps will be milled (which could lead to severe damage for machine, blank and tools if used incorrectly) a warning will appear. After its confirmation a click on 1 will start the calculation process.
   This time a further window will appear after the strategy options selection (see right). It shows all steps that will be used to mill the project. The selection works by clicking on the steps themselves. Green highlighted steps will be milled while red highlighted ones will be excluded. It is recommended to get some training about this feature before using it.
3. Enables the 'only finishing' option. Imes icore strategies use it to redo the 1,5mm and 1mm finishing steps for cavities. This feature is commonly used together with the 'adjust cavity fit' option (see page 13).
   Afterwards click on 1 to start the calculation.
4. Allows the selection of stl in the project. The selected stl will then be calculated separately in their own project. It can be used to first nest many stl and then calculate them step by step in case of scheduled production.
5. Closes this menu.



E. Simulation

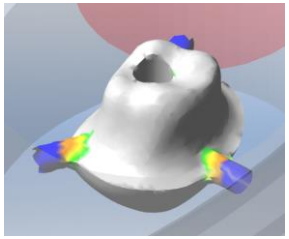
After the calculation is done it is possible to get a preview of what will be milled by using the simulation. There are two different ways to simulate (see top right picture):

1. Will open the kinematic simulation. It will be explained on the next pages.
2. Will show the simulation in the current view of the blank and a new menu will open on the right side (shown in the lower right picture). The colours indicate how much material is left compared to the original stl. It is adjustable by changing the values.

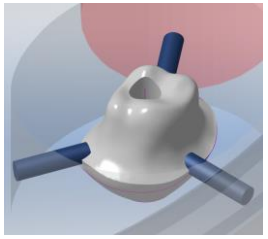


The view can be changed in three ways:

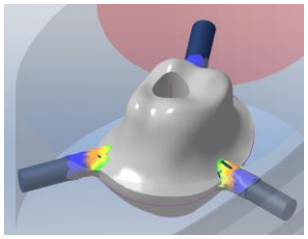
3. Will only show the residual material in the colour scale:



4. Will only show the original stl:

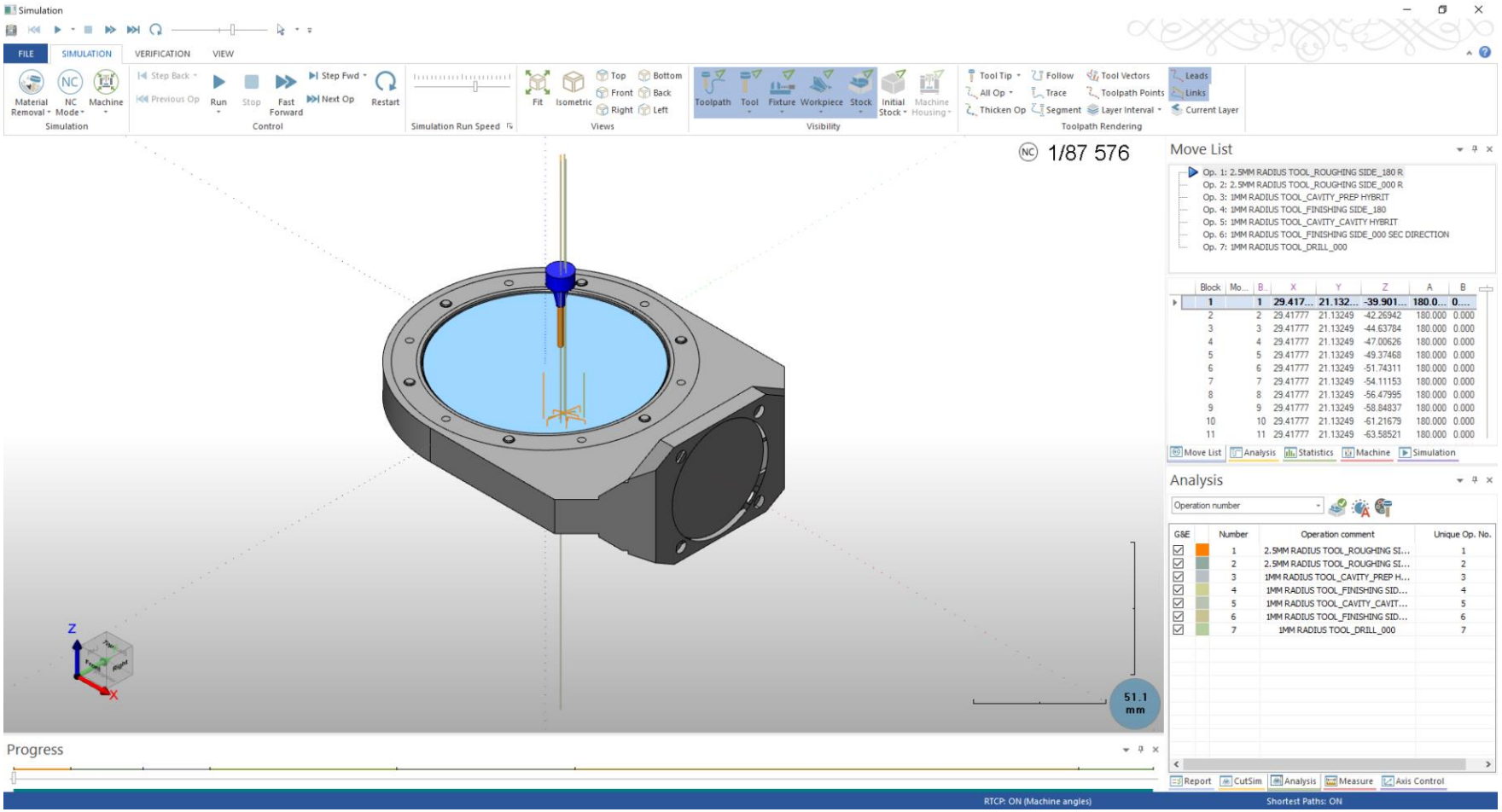


5. Will overlay both previous options with each other:



E. Simulation – cinematic simulation

This is the standard main view of the cinematic simulation. The basic use is to let the simulation run to the end and evaluate if the finished part is satisfactory.
Sometimes the simulation will show collision warnings. These will not translate to errors during the milling process and can be ignored. Only if the simulation shows grave errors (like missing parts) in the finished job, it is necessary to take measures.



E. Simulation – cinematic simulation

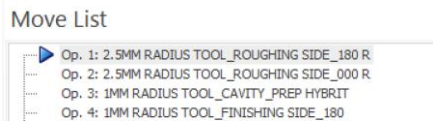
To make the simulation easier to read it is recommended to change the visibility settings so that only the stock is shown. This way the result is what the machine produces and grave errors are not hidden (e.g. by an overlayed stl).
Wether the tool visibility is necessary depends on the way the simulation is run.

Furthermore the view on the right side should show the ‘move list’ and the ‘analysis’. This further reduces unnecessary information.

The mouse functions change in this window:

- Hold left to rotate the view
- Rotate the wheel for zooming
- Hold right to move the view

There are different ways to run the simulation:



- Hit play button to let it run as a video of the milling process. The speed is adjustable with a bar to the right of the play button. Only by using this way is the tool visibility useful.
- Hit ‘next op’ while the simulation is not in video mode to skip to the next step in the milling process (indicated in the move list). The simulation will calculate the end result of the step and show it.
- Click on a step in the ‘move list’ to jump to the beginning of that step. The simulation will calculate the movements and show the result. This is the fastest way to simulate. (Note: clicking on the last step will show what happens up to that step; to show the end result of the milling a click on ‘next op’ is necessary!)

5 Secondary functions

View Buttons

These buttons allow changing the view.

- Click on the house to change the view to the 0° position from above (home view; shortcut F5). Click it a second time to rotate the view to the 0° position from below.
- The arrow buttons rotate the view by 45° in the clicked direction

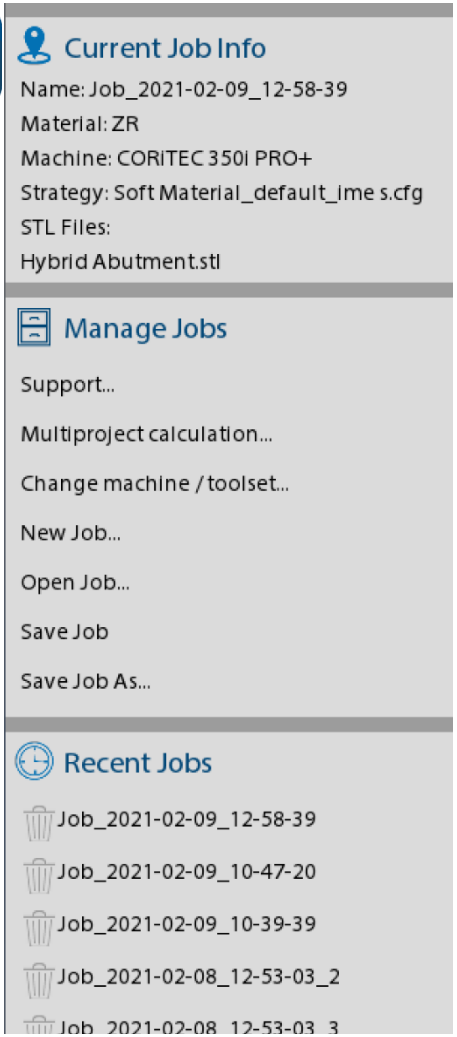
Undo

The undo button can reverse the last action taken.
This might be crucial in case of a necessary recalculation since it is impossible to recalculate, if any job was moved after the calculation (the software will give a warning in this case).
Note: it can only undo actions that have been taken since the last time the project was loaded! Closing the software or loading another project will make undoing all previous actions invalid!

Redo

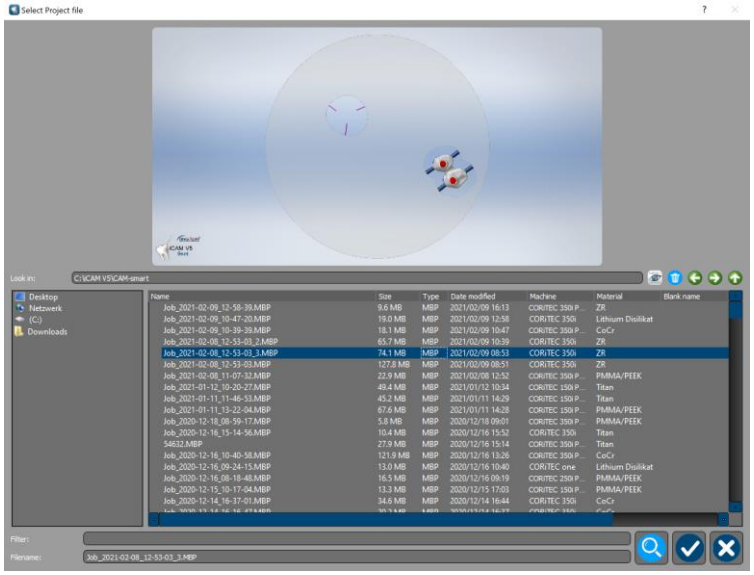
This button can only be used after the ‘undo’ button was used. It reverses the undone effect.
Note: it can only redo undones since the last time the project was loaded! Closing the software or loading another project will make redoing invalid!

6 Secondary functions – job management



The job management is split into three sections:

- The ‘current job info’ contains information about the active project such as its name and used material.
- ‘Recent jobs’ lists the last 5+ projects that were worked on (depending on screen size). Moving the mouse over them shows a preview of the project.
- ‘Manage jobs’ has access to submenus
  - ‘New job…’ does the same as the main menu button and opens a new project.
  - ‘Open job…’ opens a submenu. Depending on the CAM version (standard, comfort, or advanced) there are different possibilities to search and sort the projects in an explorer window. Clicking on a project will show a preview



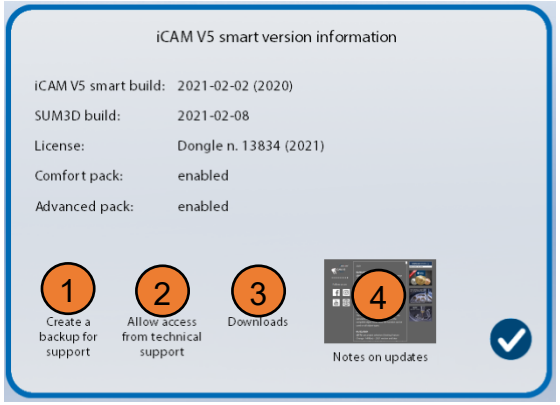
  - ‘Save job’ will save the current status of the project. Though this is usually unnecessary since the software has an auto-save feature.

- ‘Save job as…’ opens a text box. A copy of the current project can be saved by entering a new name and confirming
- The feature ‘change machine/toolset’ is currently not completely included by iCAM V5 smart. Currently it only allows to change between two machines with the same description (e.g. between 350iPRO and 350iPRO+). This option is invisible if it is unusable.
- ‘Multiproject calculation’ allows the NC file calculation of multiple projects simultaneously.

6.1 Support

[This is still about the ‘job management menu’. The section just gets its own chapter since it is important in case of customer problems]

- ‘Support…’ opens the support submenu (it is also available by right clicking on the logo in the upper left corner and then selecting ‘support…’). If the imes icore support is contacted, they will refer to this menu most of the time.
  - Important information are ‘iCAM V5 smart build’ and the ‘licence’. These have to be included in support requests.
  1. Creates a backup of the currently active project. This includes this latest calculation. With this backup the support team can view the project and analyze it in case errors are reported concerning it.
  2. Opens the TeamViewer for remote access. This way the support can show how to handle the software on screen.
  3. Opens the imes icore download page for the support. The section is password protected and not accessible for customers.
  4. Opens the same homepage as the splash which runs at program start. It includes an abstract about updates to the base software. Updates from imes icore are (in the upper right corner) and can be downloaded and installed by everyone. Furthermore it contains information about features and services from imes icore on the right side.

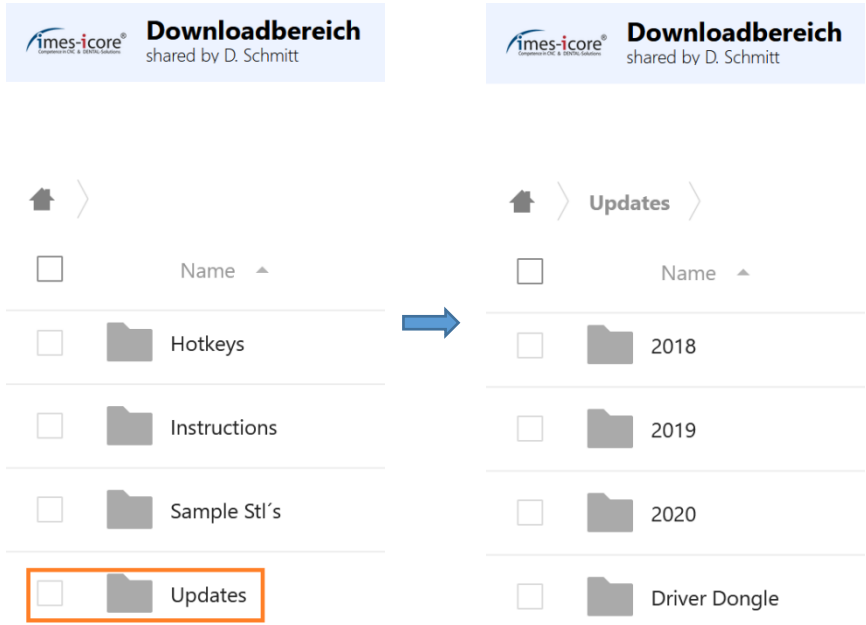


6.2 Installing Updates

Updates can be installed by everyone. It is just necessary to know which version the dongle supports. The version can be found behind the dongle number in the support window (see picture on the right).
As explained in chapter 6.1 point 4 (page 23) the homepage can be opened from the support window or the software start splash.

On the homepage select the ‘download center’ which leads to the imes icore ‘Downloadbereich’ (=download area).

In the ‘Downloadbereich’ select ‘updates’ which will lead to a selection of the version. Here it is recommended to select the latest version that the dongle supports.
Download both files and install them as administrator (option found by right clicking on the files).
Note: it is advised to switch off antivirus software during the installation since it could interfere. Also put iCAM V5 on the whitelist of the antivirus software, else the software might not work properly.



7 Secondary functions – tools

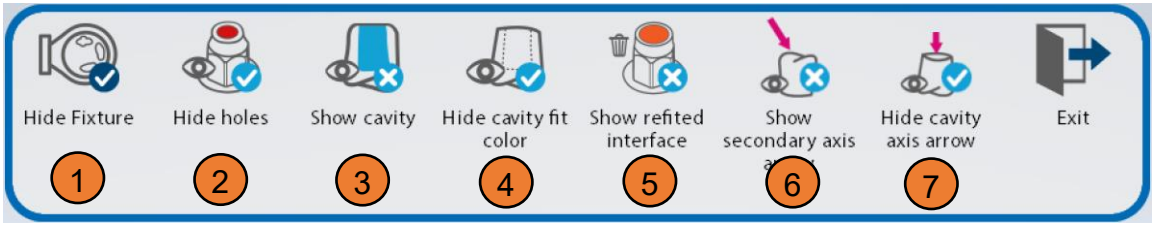
Tools can be used to alter the project in different ways ranging from simple view options to in depth changes of curves for calculations. This chapter tries to explain functions as detailed as possible but it is still recommended to receive personal training before using some of the advanced tools.

7.1 Toolpaths

Toolpaths opens the output folder of NC files for the machine of the current project.

7.2 Show/hide

This tool opens a submenu for viewing options. As the name suggests it toggles certain view options that are hideable. The buttons are responsible for showing/hiding



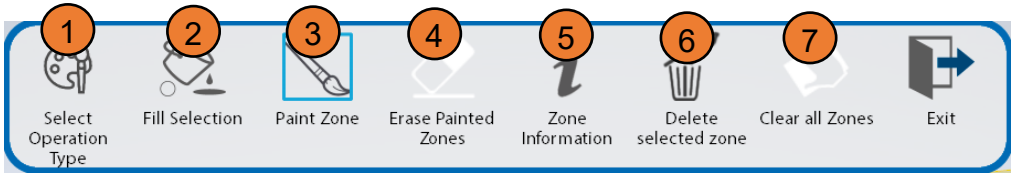
1. the fixture (shortcut CTRL+E).
2. the red caps on holes (shortcut CTRL+R).
3. the gray caps on cavities (shortcut CTRL+G).
4. ‘adjust cavity fit’ colour (see page 13).
5. interfaces which were done using ReFit option (see chapter 7.13).
6. the pink arrow from okklusal which indicates a secondary insert direction (see chapter 7.11.7).
7. the pink arrow from basal at cavities which shows their insert direction.

7.3 Nesting

Clicking this button manually triggers the automatic nesting process. The same happens automatically once a blank is chosen. It tries to fit all imported objects into the blank under the premise that there is at least 1mm space between the offset areas and the blank border.

7.4 Milling Zone Designation

The select tool is also commonly called brush tool because of its symbol and function. It is used to colour areas for special milling operations (e.g. fine fissures on bridges).



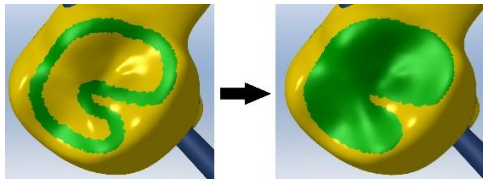
1. This button is used to decide the type of milling

   The chosen areas get an added finish operation with

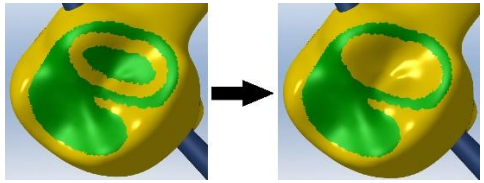
   a. Undercuts manual: 1mm ball
   b. Fissures fine: 0.6mm or 0.3mm conical
   c. Details with T32: 0.6mm ball

   Each area will get a separate colour.

2. This option can be toggled. It works in combination with 3 and 4. If activated it will fill completed curves with colour or empty them.



in combination with 2



in combination with 3

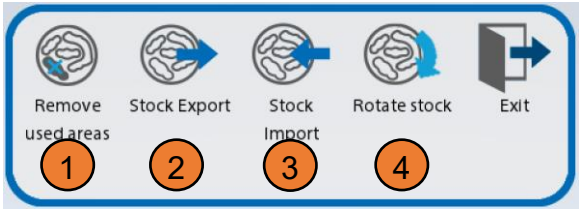
3. This option is the standard when the select tool is activated. It will turn an area around the mouse pointer coloured. The size of this area is always the same. Zooming just changes the size of the stl compared to the area. Click the mouse and hold it to use the pointer as a brush to colour areas of the stl that require a special milling operation.

   Hereby the view decides the insert direction of the milling operation. As long as the type of milling button (1) is not used again, the insert direction can be adjusted by changing the view and clicking on the area again. If the insert direction is angled more than the machine can do, it will instead be changed to the possible maximum in the decided direction.

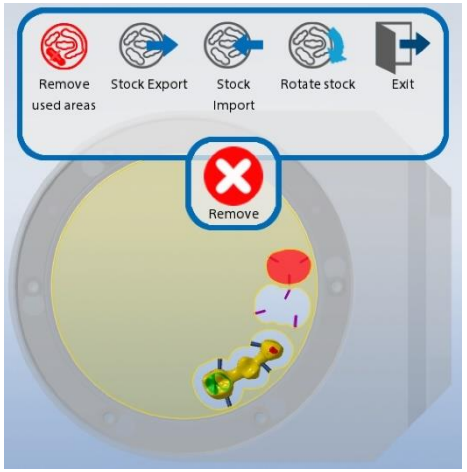
4. This option is the opposite of 3. It deletes coloured areas.
5. If this option is activated, clicking on a coloured area will show which operation (1) will be used on it.
6. This option allows to delete one coloured area.
7. This option clears all coloured areas from the project.

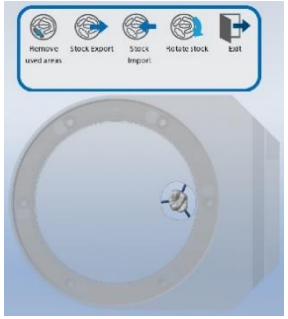
7.5 Reus. blocks

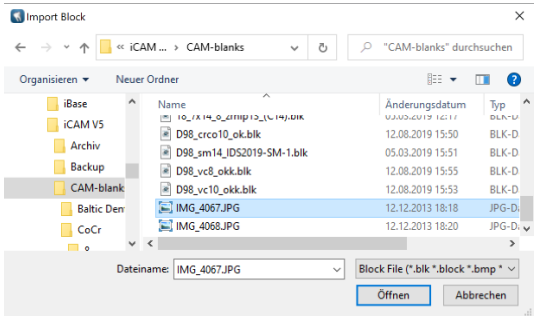
‘Reus. blocks’ is a blank menu. Usually it is only necessary to use, if you want to import a blank that was previously used in another CAM software or delete used areas since all other options are done automatically or in the blank menu (see page 8).

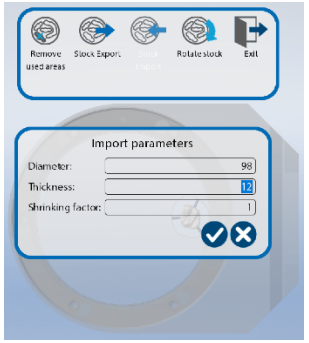


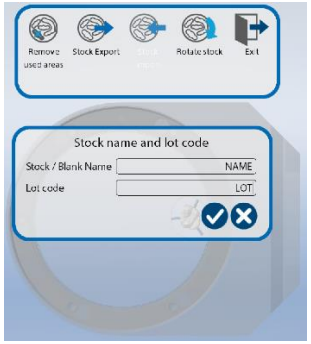
1. This option is used to delete used areas of the currently used blank. When it is activated, click on a used area in the blank. This one will be highlighted red and an option to ‘remove’ it will appear (see picture on the right)
2. This option opens an explorer window with the folder fitting the current blank in material and height. The current blank can then be saved either under its original name or as a copy with a different name.
3. This option lets you either load an iCAM V5 blank or a picture of a blank out of an explorer window. Loading a blank is the same function (but with a different view) as in the blank menu (page 8). Pictures of blanks can be either in jpg or bmp format. They should have high contrast between the background and the blank so that the software can detect the holes appropriately. The pictures below show the process for a 12mm CoCr blank (DO NOT CHANGE THE DIAMETER IN THE PROMPT!):

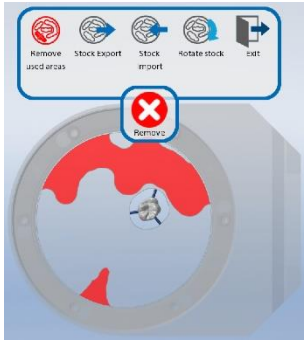










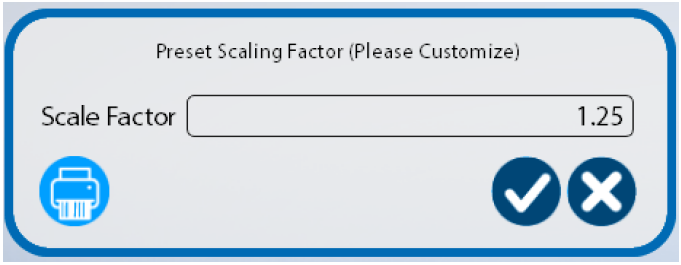


The last picture was edited and the ‘remove used areas’ option was used to highlight what the CAM identified out of the blank picture. The detected areas are close to the picture in this case. Only the small area at the bottom needs to be removed. Leaving a little safety gab to the used area will make this blank usable.

4. This option allows the rotation of the blank around the Z axis. This option is only necessary for 4 axis machines that need to mill with an insert direction that is not 0°. In those cases the CAM will not allow rotation of the stl so the blank has to be rotated to fit the stl. The rotation angle can be chosen freely.

7.6 Shrinkage factor

This tool allows adjusting the shrinkage factor after a blank was created. Click the option and enter the new shrinkage factor in the prompt. This option is only viable for materials that have a shrinkage factor that is not 1 (e.g. ZR).



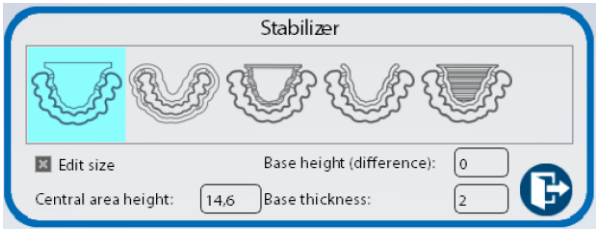
7.7 Stabilizer

This tool adds a stabilizer to ZR works that have at least two units. The different stabilizers have different adjustment options. An important requirement for stabilizers to work is that enough sprues are used to connect the stl to them (from experience: one on each unit). There is no consensus about which type of stabilizer is the best.

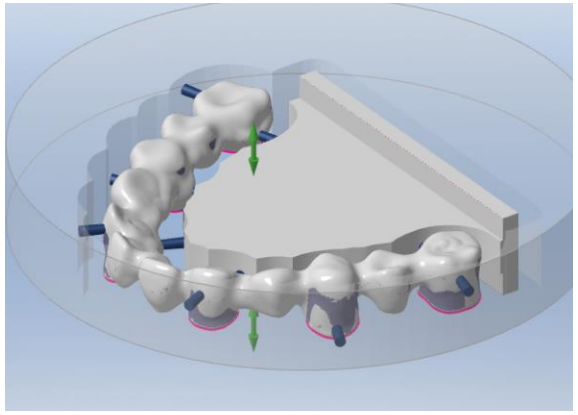
The first option can adjust

- Central area height (CAH) which refers to the thickness of the stabilizer part that connects to the stl.
- Base height (BH) reduces the size of the footing. It needs a negative value (e.g. -2) to work and is a direct reduction from blank height.
- Base thickness (BT) adjusts the portion at the end of the stabilizer which is always at blank height.

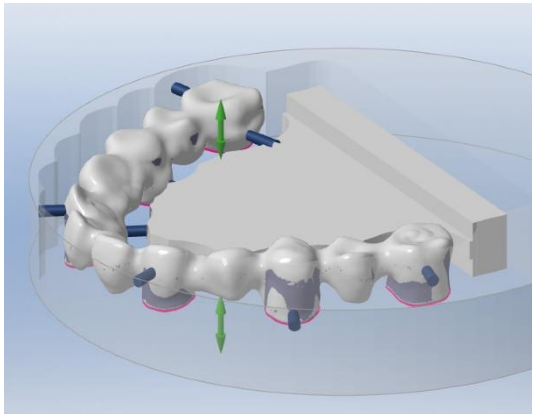
The green arrows allow positioning of the middle area. Click on them and hold then move the mouse to change the position.



Here are two examples:



CAH=6, BH=0, BT=3



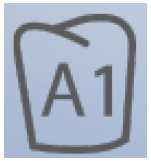
CAH=3, BH=-5, BT=6

The other four options can adjust

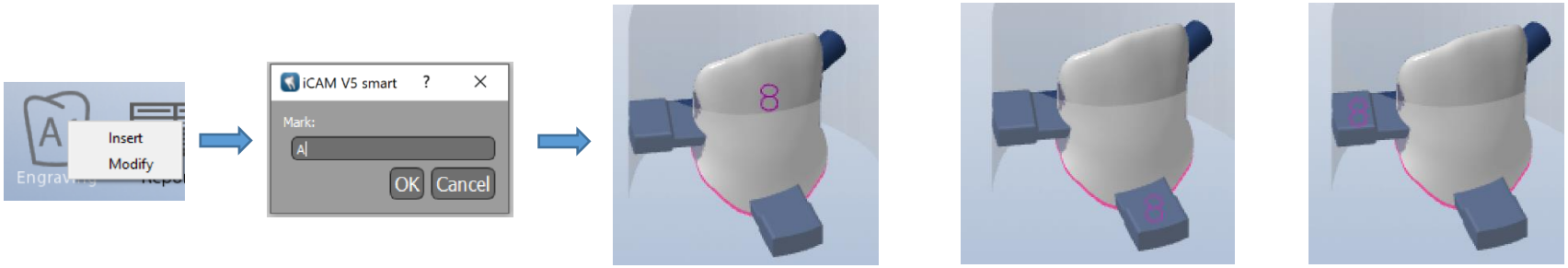
- Thickness of the stabilizing part
- Height of the stabilizer. It needs a negative value to work (e.g. -2).

The green arrows allow positioning of the stabilizer (only viable for height < 0)

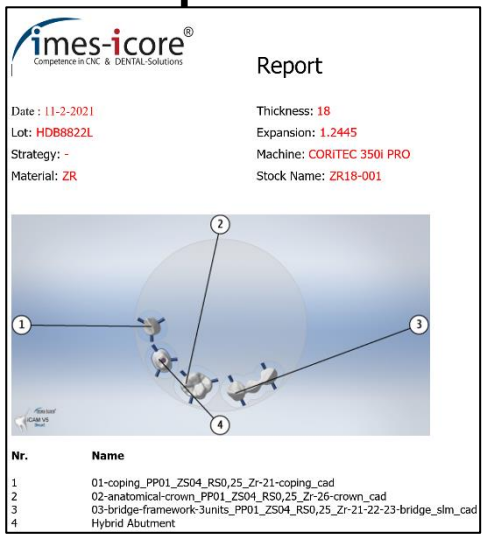
7.8 Engraving



This option allows engraving numbers or letters on stl or their rectangular sprues. Only one engraving is allowed per stl. They can be done occlusal or basal (the insert direction is 0°).
A prompt will appear when the tool is activated. Insert creates a new engraving or replace an existing engraving on a stl (even if it is not in the same position). Modify prompts to choose an existing engraving which will then be replaced by putting another one on the stl.
The three different ways are shown in the pictures below



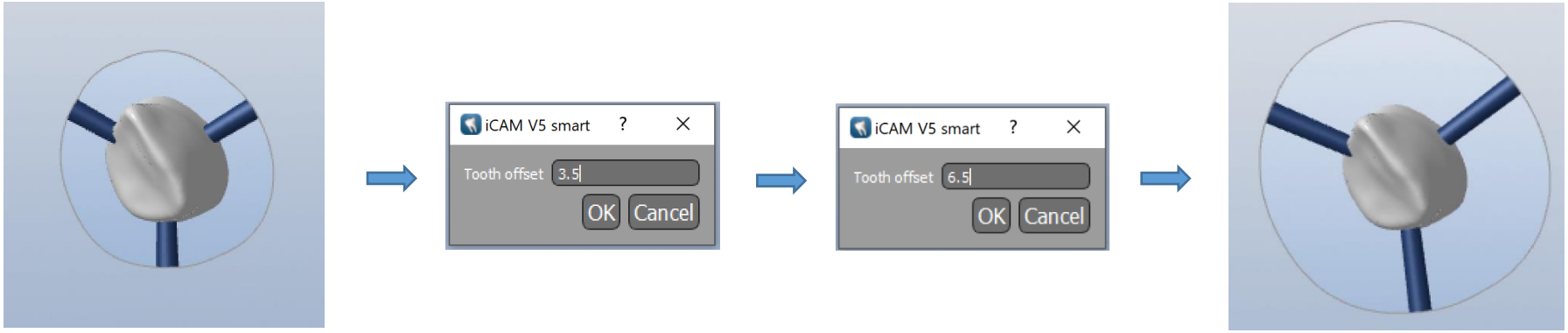
7.9 Report



This tool creates a report for documentation. The current format is a rtf document (other formats like pdf are possible but may require additional programs like Word) which includes information about the active project, its blank and lists the stls together with a picture of the current view (it is recommended to switch to ‘home view’ before creating the report). An example is shown on the left.
The document can be viewed in Microsoft Word and other document editing software. So it can be altered or saved as another file type (even pdf).

7.10 Change offset

This tool allows to change the offset (space between stl and blank). The predefined size is due to experience and necessity for certain operations thus it is not recommended to change this value.
Activate a stl, then click this tool. A prompt to enter the new offset value will appear (the current offset value will be in the text box. After confirming the new value the software will refresh the view and show the new offset together with longer sprues.



7.11 Curves and surfaces

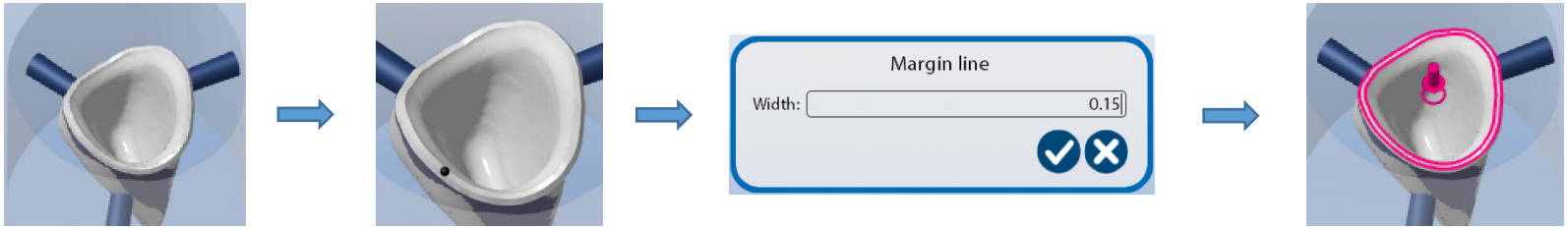
This tool opens its own submenu. All options have a direct influence on the milling process itself. Be advised to proceed with care. This chapter will also explain some details about strategies and their options. Deeper knowledge about this topic will be explained in chapter 10. A feature that is in use will be highlighted orange.

7.11.1 Margin line

Margin lines are necessary to mill high quality cavities. They are mainly used in crowns, bridges and hybrid abutments. A margin line is a bright pink double curve and shows the CAM where exactly cavities lie, which insert direction they have and the thickness of the margin. Usually the automatic detection during import finds all margin lines accurately. Most of the times a margin line is erroneous, it is because the CAD process did not export a decent quality.

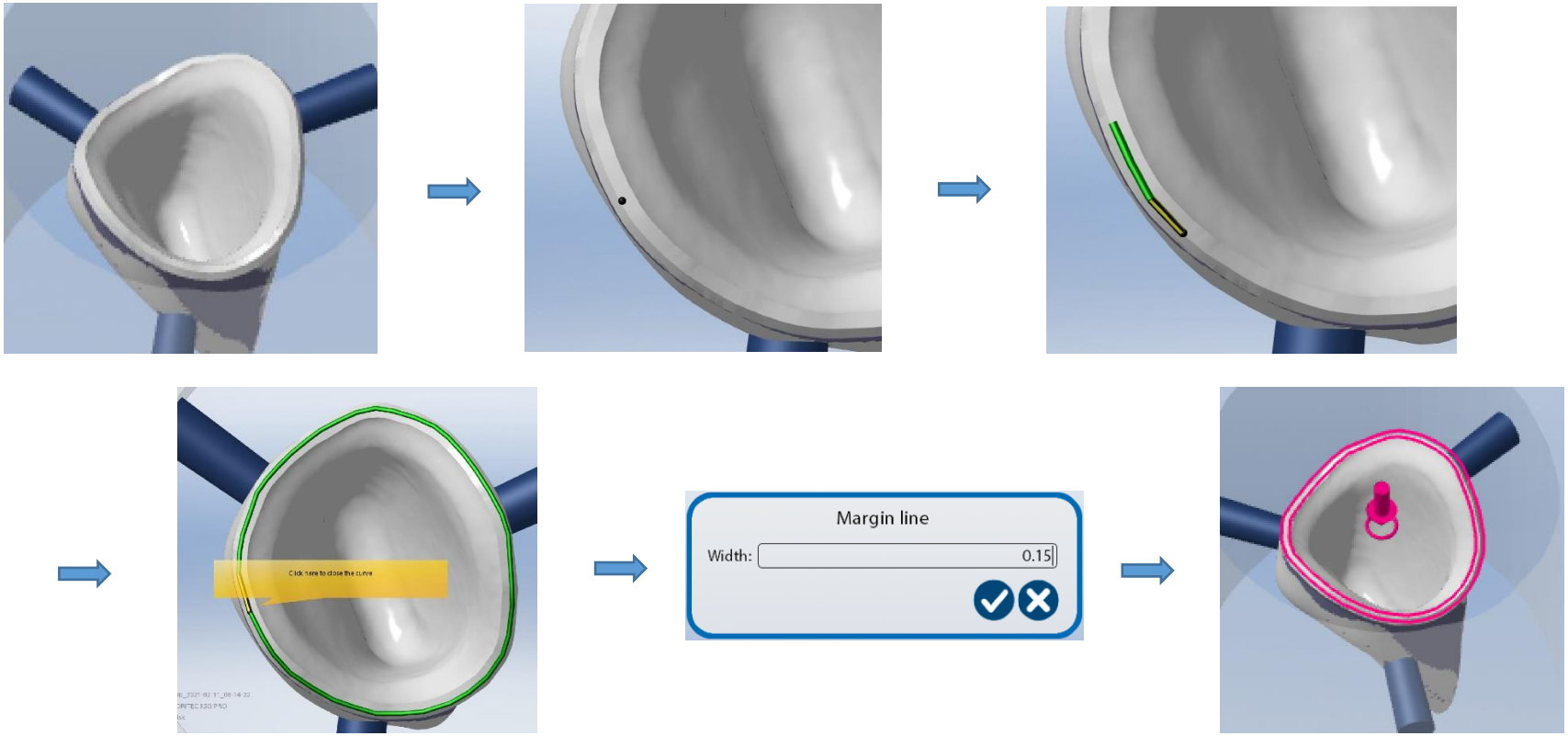
7.11.1.1 Margin line detection

This feature is used by activating it and then clicking on the margin of a cavity (the mouse pointer will show a dark blue spot; see second picture below). After entering the margin thickness (if known, else leave it at the preset) the detection will then work automatically.



7.11.1.2 Manually draw cavity margin lines

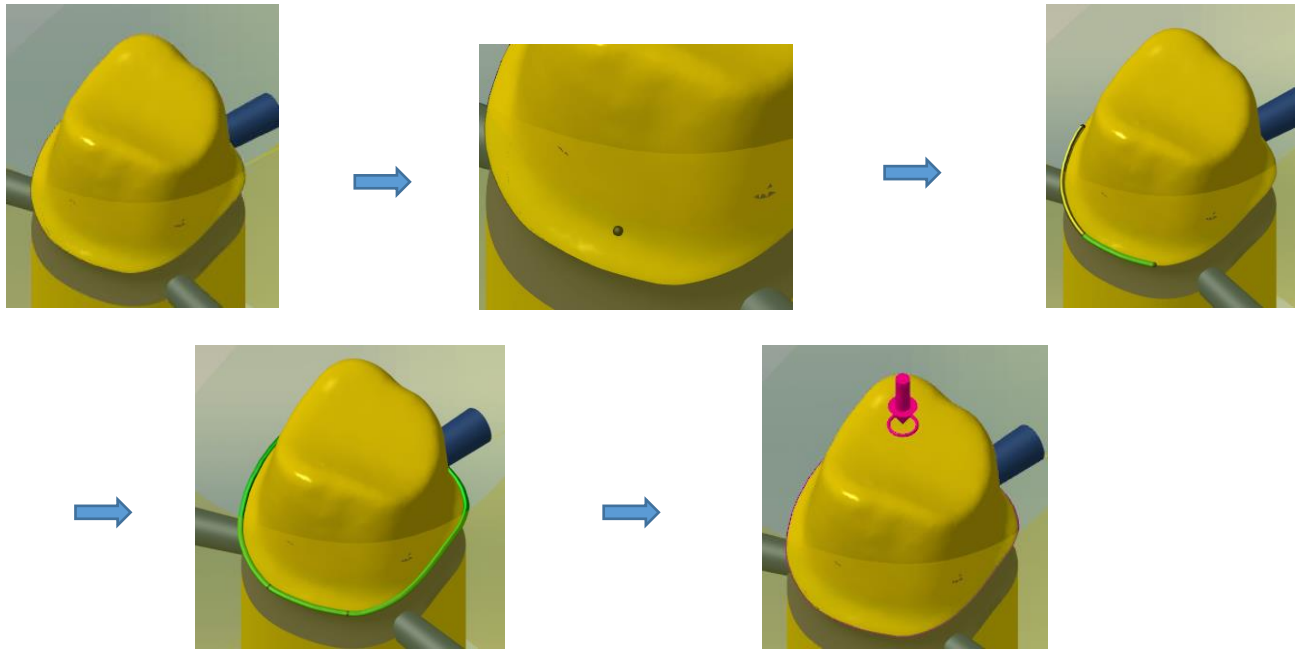
As mentioned in chapter 7.11.1.1 the CAM sometimes cannot detect the margin line automatically. In those cases it is necessary to draw the margin line manually. It is best to rotate the view to point directly inside the cavity which has to be detected and then increase the zooming factor (as shown in the pictures below). Activate the tool (the mouse pointer will show a dark blue spot) and draw the margin line in steps (the software will help by trying to match the line to the margin line). There will be a pop-up once the curve can be closed and a prompt will ask for the margin thickness (insert it if known, else leave it at the preset). The margin line will then be generated by the software.



Note: The margin line will follow the exact points that were drawn. If the steps are too big or the points were at the wrong position, the margin line can “fall” in the cavity. This leads to inaccurate milling and thus fitting problems.

7.11.1.3 Manually draw stump margin lines

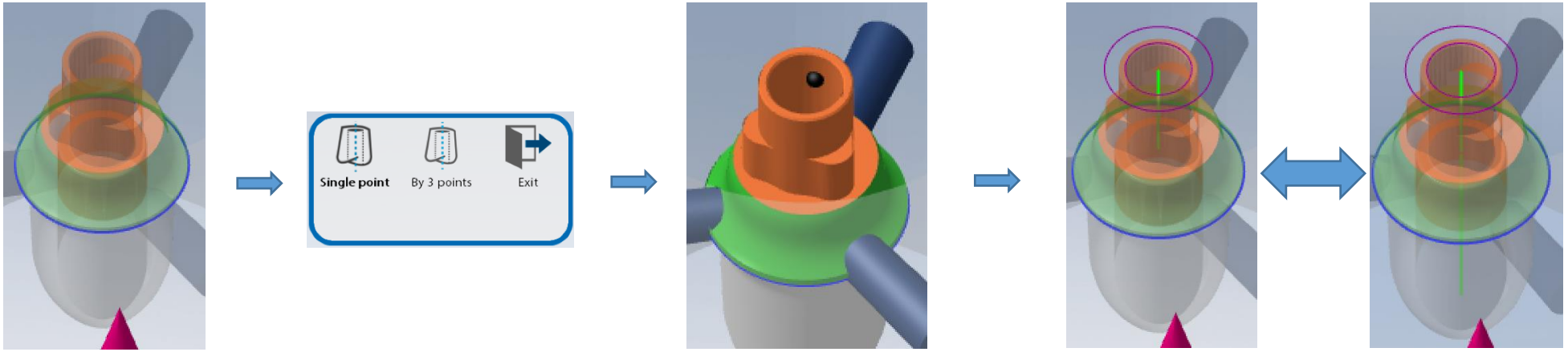
With this tool the margin line of stumps can be generated manually. It works similar to the tool of the last chapter: It is best to rotate the view to point directly inside the cavity which has to be detected and then increase the zooming factor (as shown in the pictures below). Activate the tool (the mouse pointer will show a dark blue spot) and draw the margin line in steps (the software will help by trying to match the line to the margin line). When the curve is closed the confirmation button will switch to blue. The margin line will then be generated with its insert direction by the software.



Hint At the moment this type of margin line is not used in imes-icore strategies.

7.11.2 Cylinder detection

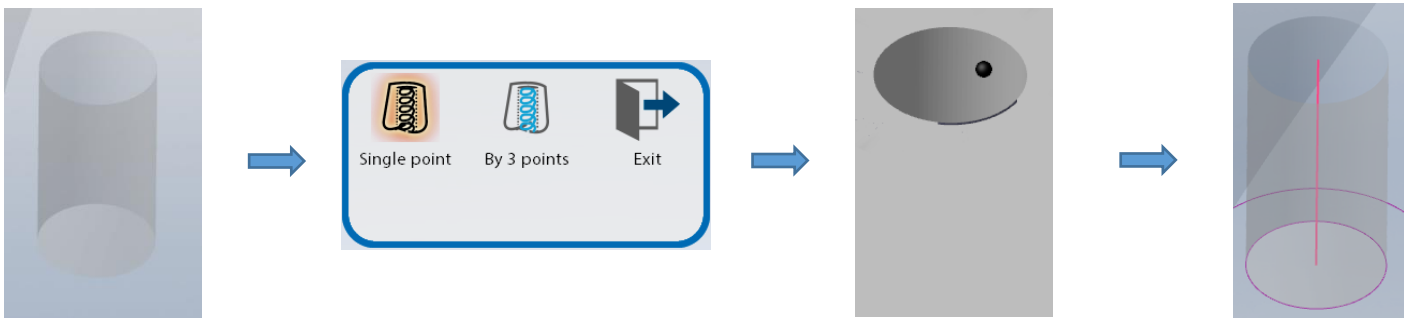
Cylinders are necessary to mill holes and tell the CAM where a hole has to be milled and which diameter and direction the hole has. A cylinder is always perfect in the software with a defined height and circular cross-section (that is why this feature does not work with the titanium base part of a hybrid abutment). If a hole has more than one diameter the software will put more than one cylinder inside it. Cylinders are indicated by a bright green axis. Usually the CAM will automatically detect all necessary cylinders during the import process.
Activating the tool prompts to choose between single or three point detection (usually single point is sufficient). The mouse pointer will show a dark blue spot. Now click inside of the hole (or on three different points) that has to be detected and the software will calculate the axis for this hole.
Note: this procedure might be necessary more than once to completely detect the hole since most holes have more than one diameter. To check this it is recommended to toggle the stl view to transparent (number 9 key) and rotate the view to the side.



Note the difference between these pictures: the left one only has the axis in half of the abutment. This can be changed into the right picture by adding another axis to the second half.

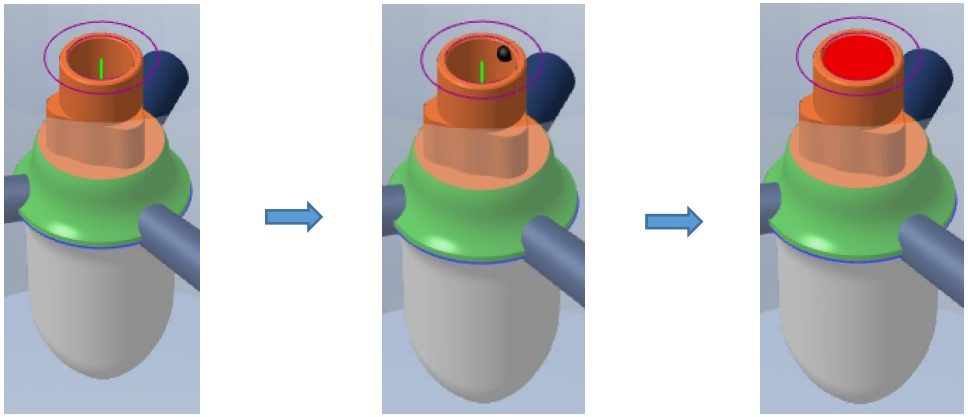
7.11.3 Gewindezylindererkennung

Similar to cylinder detection, thread cylinders can also be detected. The only difference between them and standard cylinders is that they are specially designed for blind holes. The function is identical to the cylinder detection. However, the representation shows a red instead of a green cylinder axis.



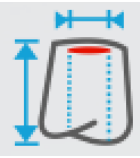
7.11.4 Add cap to cylinder

Red caps are essential to milling holes. They close off the hole for certain milling tasks which guarantees that the process is successful. (Usually a milling process starts with roughing of a big burr. This burr usually cannot reach the small hole and will just leave it unprocessed. Afterwards a small burr is used for a finishing process of the outside including cavities. This burr is set to expect only minimal rest material. If this burr now tries to finish the hole which still has full material it would simply break due to too much stress. Red caps prevent this last part.) This means that every hole (or drill axis) must have a red cap on both ends.
Usually red caps should be visible. If no red caps are visible toggle the view for red caps (either by pressing CTRL+R or use the show/hide tool from chapter 7.2) to see if the red caps are invisible or missing. Add them if they are missing.
Activate this tool and decide wether to automatically detect the cap or draw the border of it manually.
To detect it automatically the mouse pointer will show a dark blue spot. Click near the border of the hole and the software will add the red caps on both ends. In some cases more than one click is necessary. Check if all red caps are present.

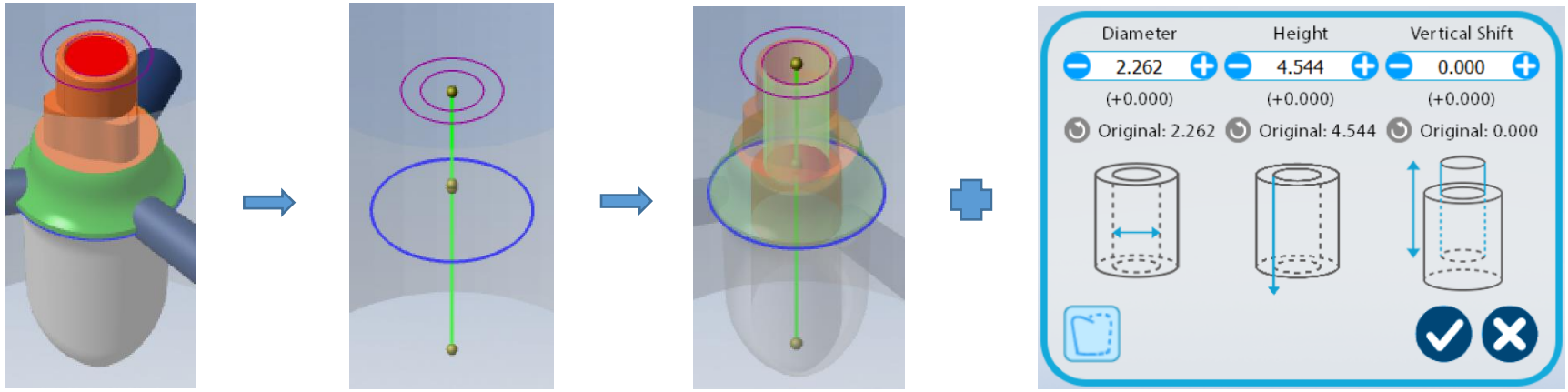


In case of drawing it manually the whole border of the cap has to be drawn.

7.11.5 Change cylinder size

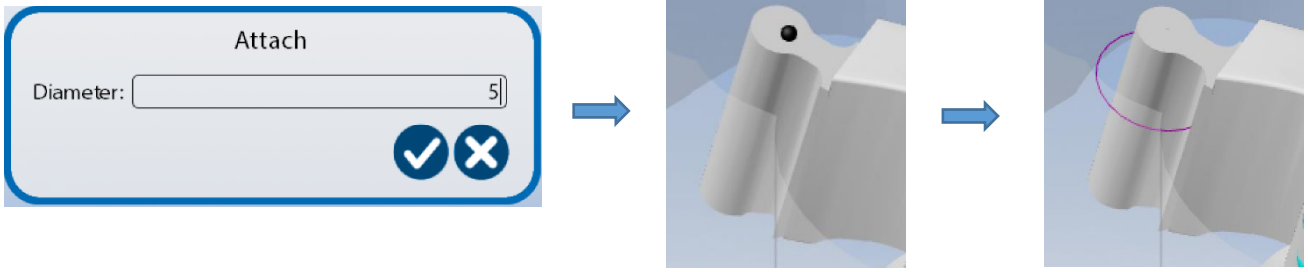


Usually the software detects cylinders in the correct size automatically so that no changes are needed. If a cylinder was not set correctly it can be adjusted with this option.
Activating the tool will switch the view to only show curves. Select the axis of the cylinder which will be adjusted and the software will change the view again. The stl will appear transparent, all curves are shown and the selected cylinder will be shown as a transparent green cylinder. A window will appear in the top center which allows adjustment of diameter, height and vertical shift the cylinder. All changes can instantly be viewed with the transparent green cylinder.



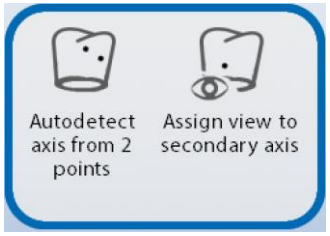
7.11.6 Creation of curves for attachments

Attachments (e.g. preci vertix) desire a polished surface. After a normal milling process they will be left with an equator line where the milling paths from occlusal and basal meet. To enhance the surface the strategies of imes icore include a step to mill attachments from one side only. This feature only works in combination with the tool ‘define a secondary machining axis’ from the next chapter (7.11.7).
Activate this feature and a prompt will appear asking for the diameter of the circular attachment curve. Usually 5 is big enough though it is a value that might require some experimentation. Afterwards the mouse pointer will show a dark blue spot. Click where the center of the circular attachment curve shall be and the curve will be added. Its direction depends on the secondary machining axis (usually shown as a big pink arrow from occlusal which are hideable). The new curve should include all parts of the attachment.



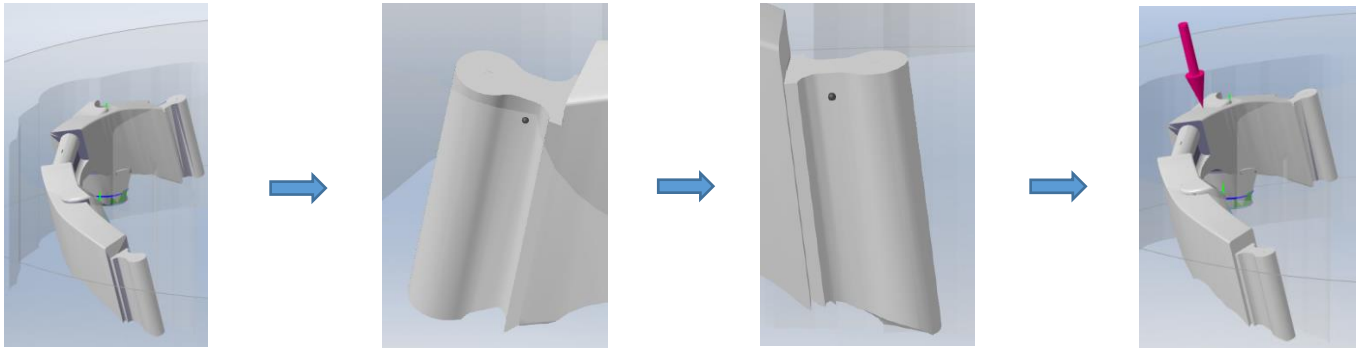
7.11.7 Define a secondary machining axis

A secondary machining axis can be defined to mill a certain operation from a different base angle (which is usually 0) (e.g. attachments; compare chapter 7.11.6).
Defining the secondary axis can be done in two different ways which will be shown when the tool is activated: ‘autodetect from 2 points’ or ‘assign view to secondary axis’.

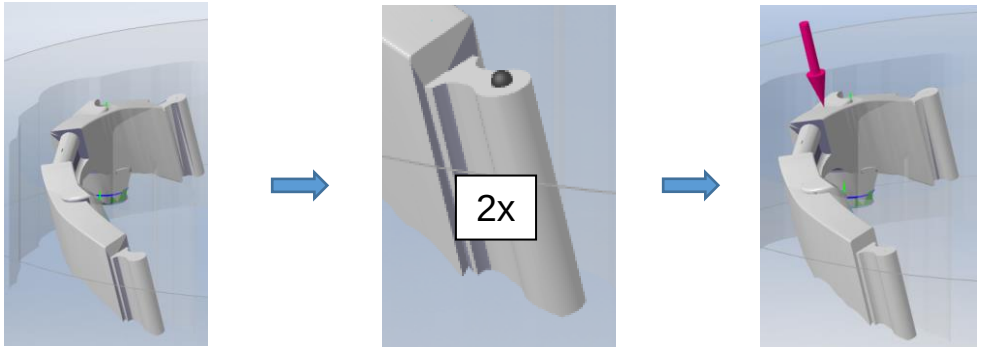


The first option works in two ways (the mouse pointer will show a dark blue spot when activated):

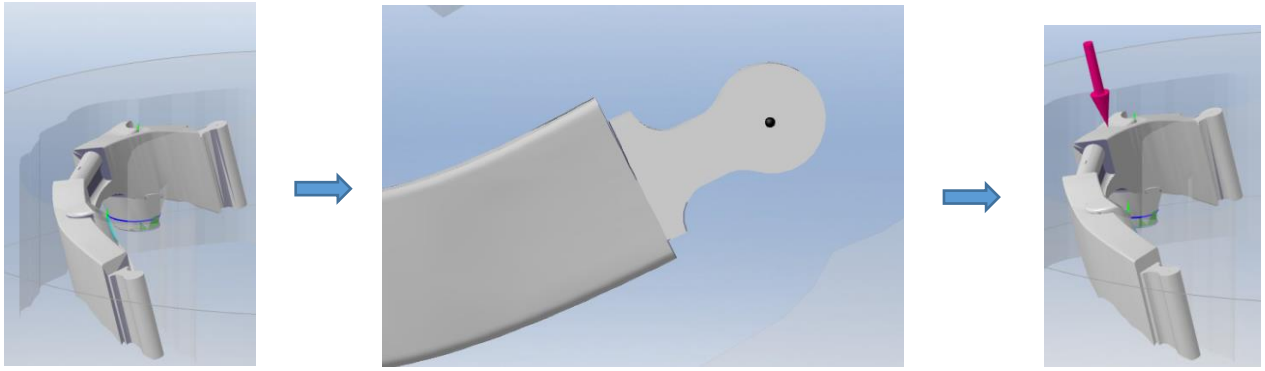
- Click on two opposing parallel even surfaces or



- Click on one point on of an even surface perpendicular to the secondary axis twice (the author recommends using this method since it is much easier to handle)

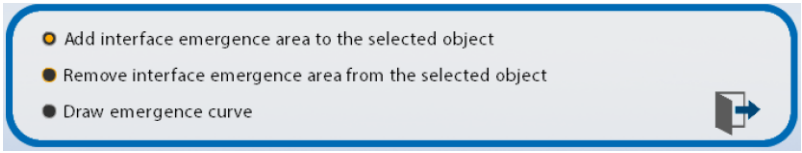


The second option will add a dark blue spot to the mouse pointer. Now the view has to be adjusted to the direction of the secondary machining axis. It does not matter what is viewed and which zooming factor is used. The angle just has to be within the machine's limits. Then the stl has to be clicked that this secondary machining axis should be added to.

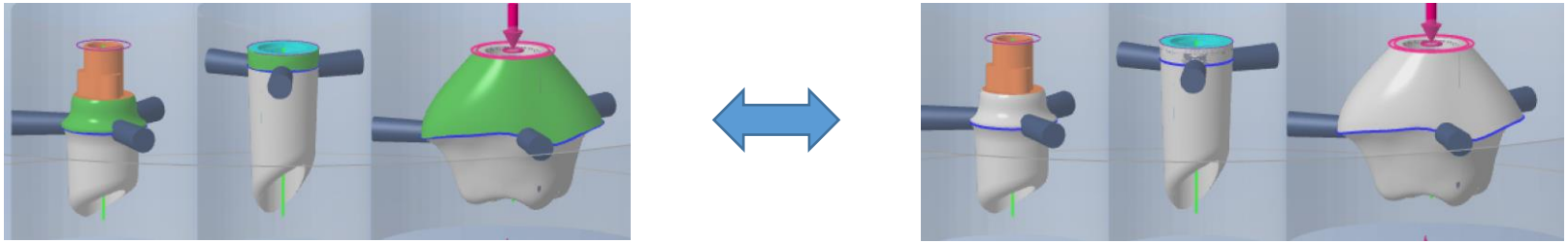


7.11.8 Edit abutment emergence area

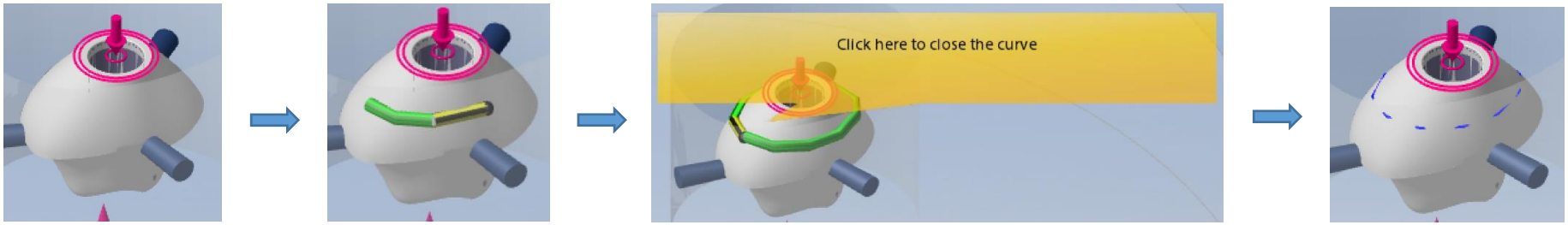
The emergence area of abutments is shown in green colour. Imes icore strategies use a special strategy step to work on this area which raises the surface quality. Its accompanying curve is the blue emergence curve (usually around the widest part of the abutment). An emergence area is always between the emergence curve and the interface.
When the tool is activated, a window will open which specific edit is required.



- The first option allows to add an emergence area between an existing interface and emergence curve by clicking on the stl.
- The second option allows removing emergence areas by clicking on the stl.



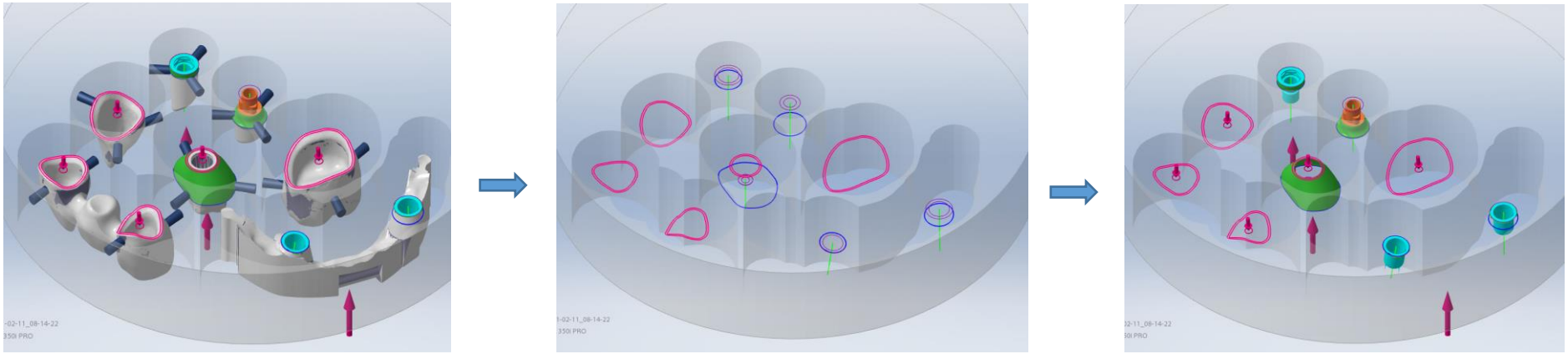
- The last option allows drawing an emergence curve manually. After activation the mouse pointer adds a dark blue spot. Clicking on any point of a stl will start drawing the curve. Draw the curve in small steps. There will be a pop-up once the curve can be closed. The emergence line will be generated by the software once the curve is closed.



7.11.9 Delete curves and cylinder caps

As previous chapters described curves and red caps are necessary for the CAM to calculate the appropriate NC files. Thus they should not be deleted without cause.
Some curves and red caps may be misplaced by the software and need to be cleared or corrected. In those cases activate this tool. The activation has three different levels:

- First click activates the tool so that curves and red caps can be deleted by clicking on them.
- The second click changes the view to show only curves which then can be deleted easier.
- The third click adds insert directions, emergence areas and interfaces to the view. The first can be deleted while the latter two are for easier identification and cannot be deleted with this tool.
- The fourth click deactivates the tool.

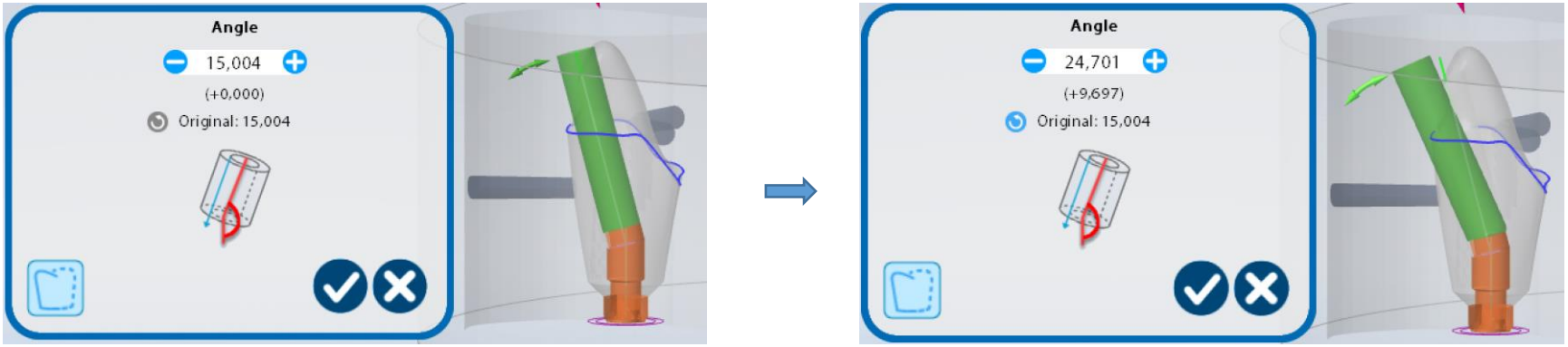


7.11.10 Copy and paste

Copy and paste is a tool that is only available in the 'advanced module'. Activate it, click on an stl and enter the number of copies to be made. The copies will be in the exact same state as the original (only XY coordinates and Z rotation will change during nesting).

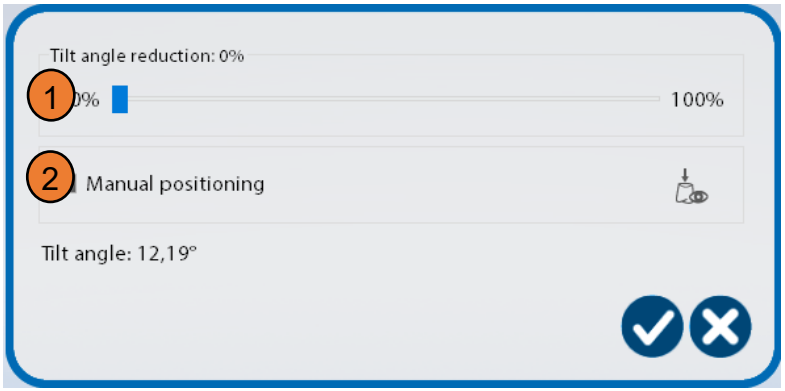
7.11.11 Change hole axis

This tool allows changing the occlusal position of an angled screw channel (though it is recommended to do the whole design in the CAD software).
Activate the tool and click on a stl with an angled screw channel. The stl turns transparent and the screw channel will be highlighted (including its interface). A window will appear showing the current angle and the original angle (which can always be restored). The angle of the screw channel can then be adjusted by either entering a value, using the + and – buttons, or clicking and holding the curved green double arrow and moving the mouse. The software will then generate the new screw channel and close the old one.

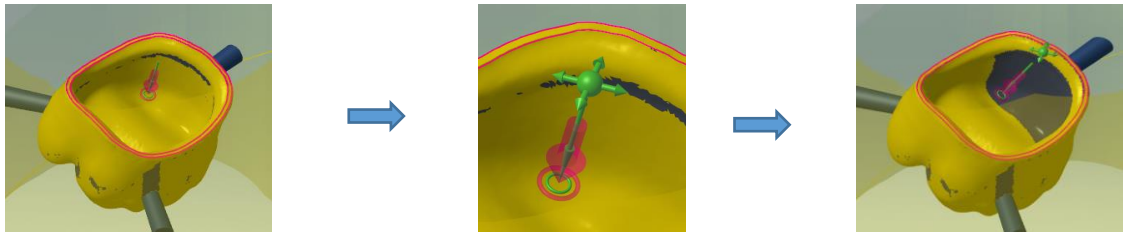


7.11.12 Change insert direction of cavities

This tool is used to change the insert direction of cavities. After activating it, the insert direction (pink arrow) of a cavity has to be selected. A menu with several options will open.



1. This option is used to reduce the insert direction angle of a cavity (100% = insert is 0). The angle of the stl is not touched with this option. This means that potential undercuts will be made which may not be milled.
2. After activating this option, a green arrow cross will appear on the insert direction. By dragging the end of the pink arrow the insert direction can be changed. The angle of the stl is not touched with this option. This means that potential undercuts will be made which may not be milled.

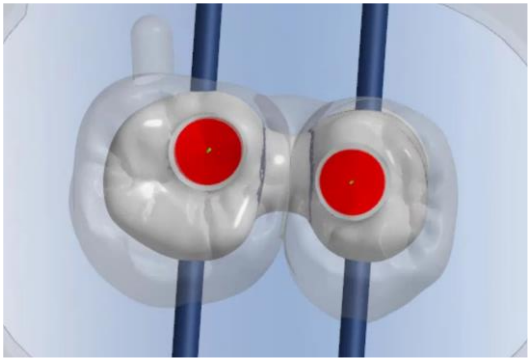


7.11.13 Dimensional verification

Dimensional verification is an advanced tool. Since it is not necessary for processes of iCAM V5 it will not be further explained.

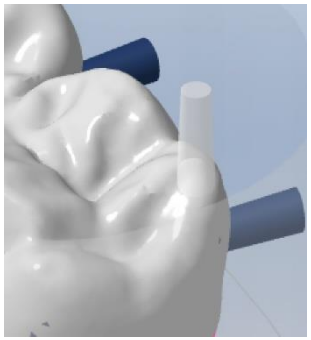
7.11.14 Merge objects

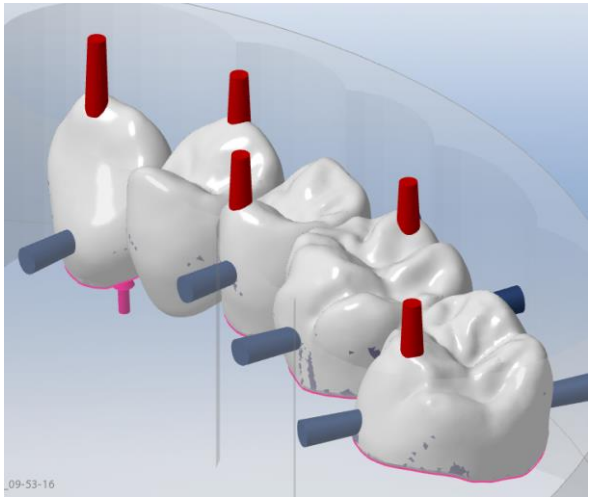
This tool is not yet in use in imes icore strategies. It can be used to combine two stls which represent the same stl in different states. E.g. the base structure of a hybrid abutment could be made for metal. Its overstructure made of PMMA can be added with a second stl. The merge tool then overlays them like shown in the picture. It is then possible to first mill the base layer out of metal. The finished metal work gets a sufficient coating of PMMA which can be milled in a second operation.
More information about this feature will be released when it is available to customers.



7.12 Add vertical support

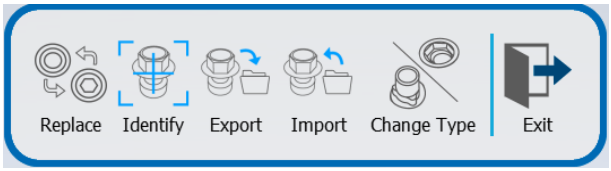
This tool is used to add vertical sprues for the sintering process (e.g. of ZR). The recommendation is to add at least one sprue per unit and make sure that the sprues are distributed so that the stl can rest on them safely.
Activating this tool will open a window in the top center which asks for the dimensions of the vertical supports. After deciding on the dimensions hover the mouse pointer over the stl to see a preview of the sprue. Clicking on the stl will add the sprue.



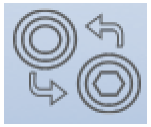


7.13 Connection

The ‘Tools’ category ‘Connection’ contains special editing options for interfaces. If the CAD output cannot be read correctly, the options ‘Identify’ and ‘Change Type’ may help.

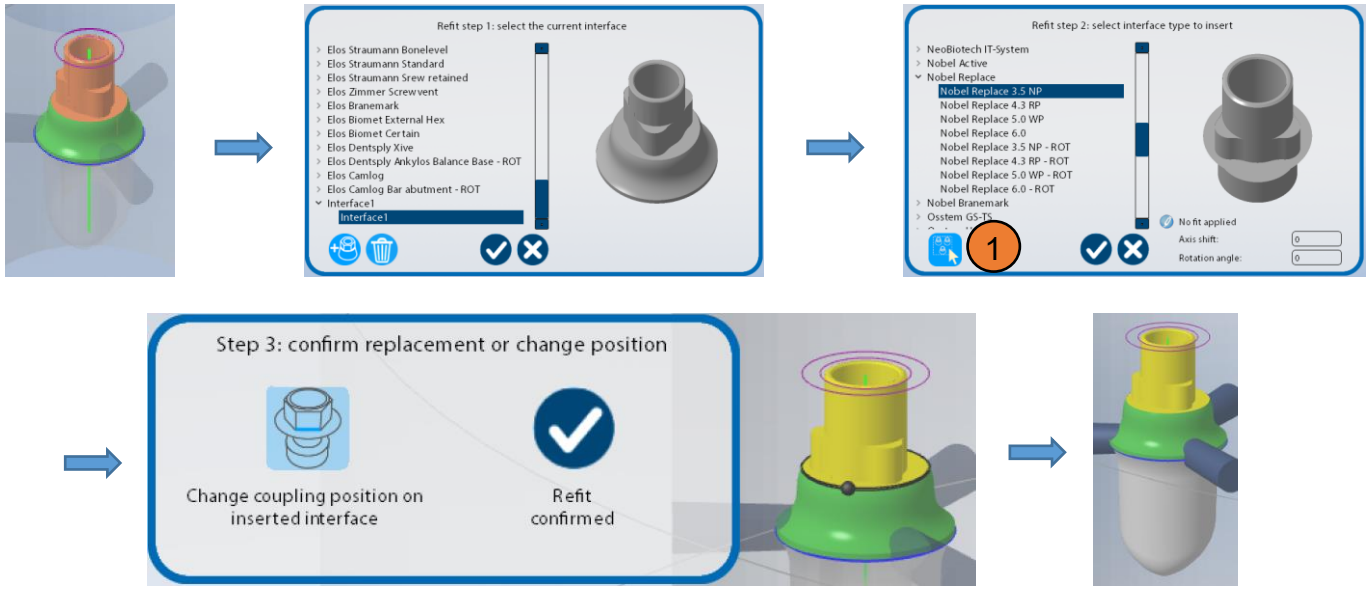


7.13.1 Refit



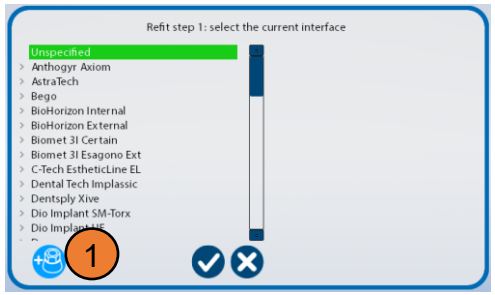
The refit feature is used to replace interfaces of stls with interfaces of the CAM library. In iCAM V5 there are two ways for this feature to work:

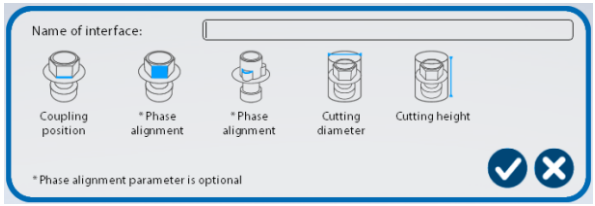
- The ELOS system will replace interfaces automatically during import. It is a special library developed by imes icore for no-lock systems. The system was tested extensively and is guaranteed to produce high quality interfaces for all supported connections.
- The generic CAM library has to be exchanged manually. It works with all detected interfaces but is not guaranteed to produce fitting results.
  When the tool is activated the view will toggle to transparent with opaque interfaces. A window will appear in the top center asking to choose the interface to replace. After confirming the selected interface a window will appear with a list of possible interfaces. Choose the selected interface in the list (usually a self-defined one; see chapter 7.13.2). Then choose the interface type to insert in the next window (if the exact same replacement has to be done for more than one interface it is possible to use the feature 1 to choose the other interfaces before replacing). In the next window it is possible to adjust the coupling position (axial position of the replacement) by a curve. If necessary choose the coupling line and confirm it. The software will then replace the interface.
  Note: most of the time the interfaces will not fit perfectly to each other which leaves gaps in the stl. It is recommended to simulate the milling process before using the milling machine.



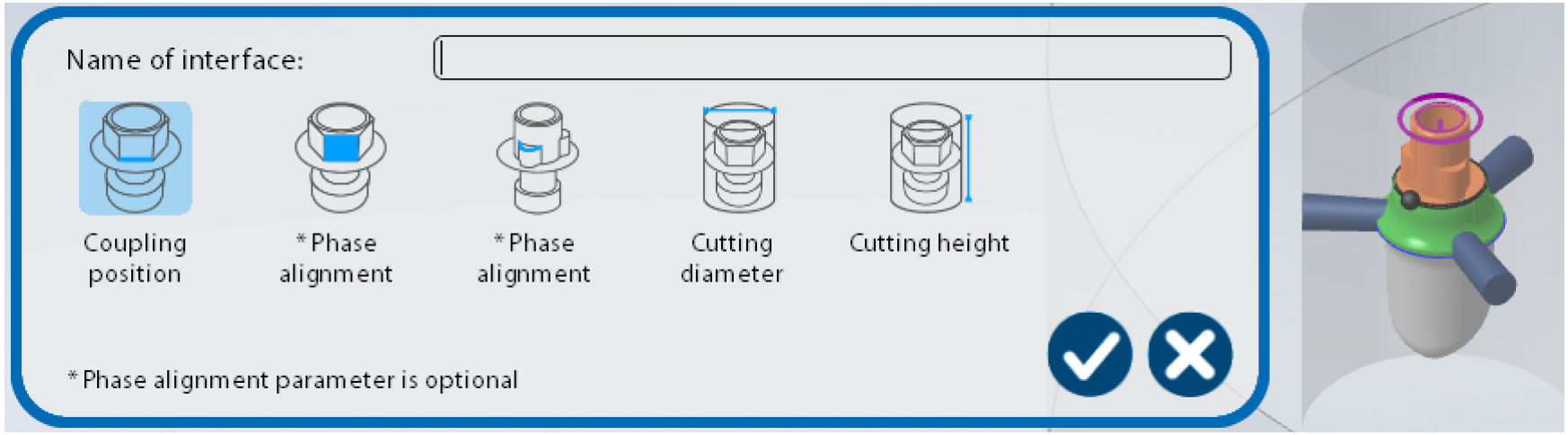
7.13.2 Defining a new interface to replace it

The first time a CAD library interface is used it needs to be taught to the CAM. To do this click on the add interface button in left lower left corner of the window (1). A new window will open asking for the specifics of the interface.

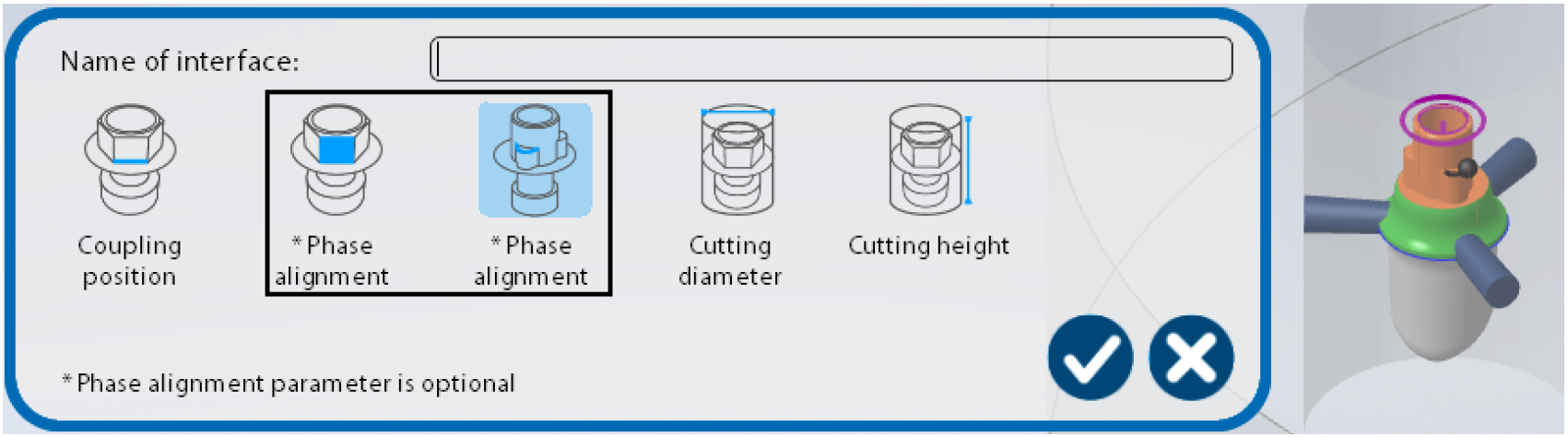




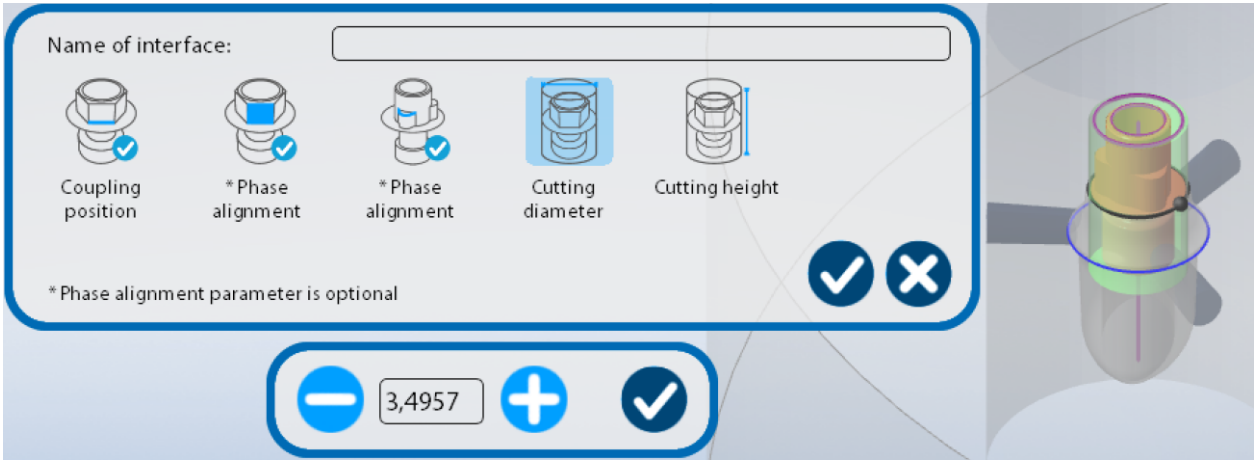
The coupling position is the line connecting the interface and the stl. Different curves will be highlighted during when going over the stl so it is important to choose the correct one (else the refit will not be positioned correctly).

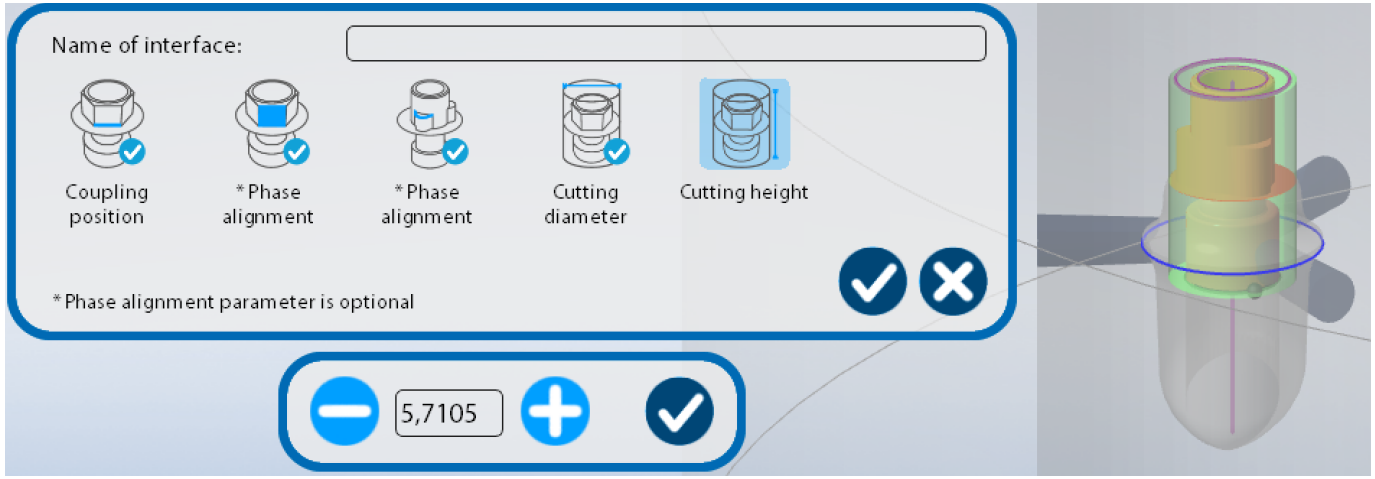


The phase alignment decides the rotary position of lock interfaces (it is irrelevant for no-locks). It can be either a side of a polygon or a curve (as shown in the window).



The cutting diameter and height define where the interface is cut out of the stl. Usually it is sufficient to cut as close to the interface as possible which could equal the coupling line. The cutting diameter can be entered as value or chosen by a curve.
The cutting height has to include the screw position and can also be entered as value or curve.



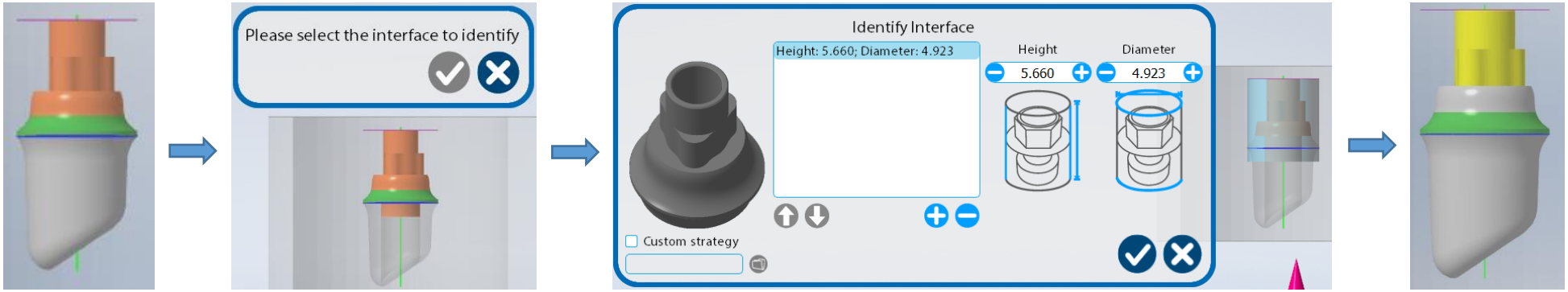


The new interface can be saved after a name was chosen.

7.13.3 Identify

With some CAD outputs there may be problems to correctly identify interfaces during import. In such cases the ‚Identify' function can be used to correct the interface area. This is necessary to make the strategies work perfectly.

Activate the function and all surfaces except the interfaces become transparent. Select the interface to be edited. Afterwards a window appears to set the interface. All changes can be tracked live on the stl. (Optional custom strategy steps can be made with the iCAM V5 expert.) The newly defined interface area is then displayed in yellow.

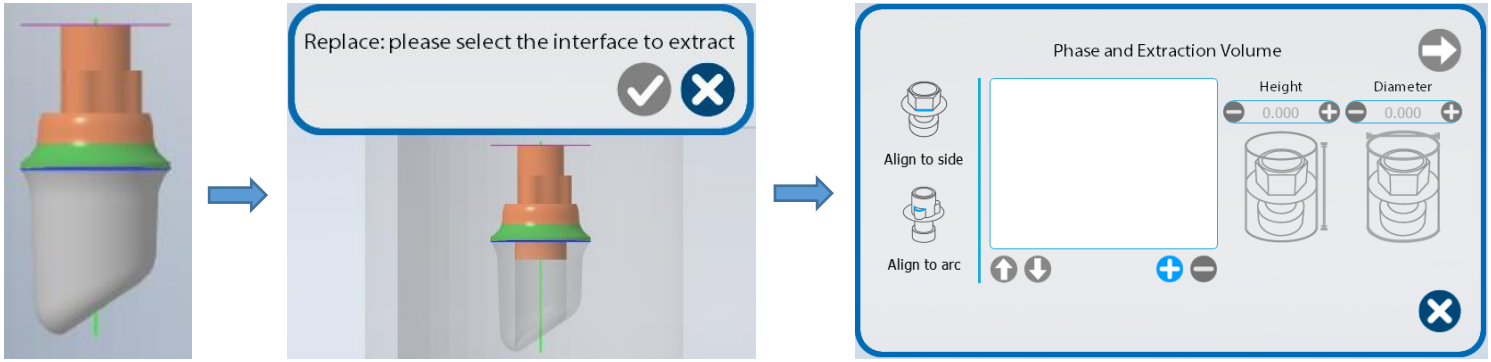


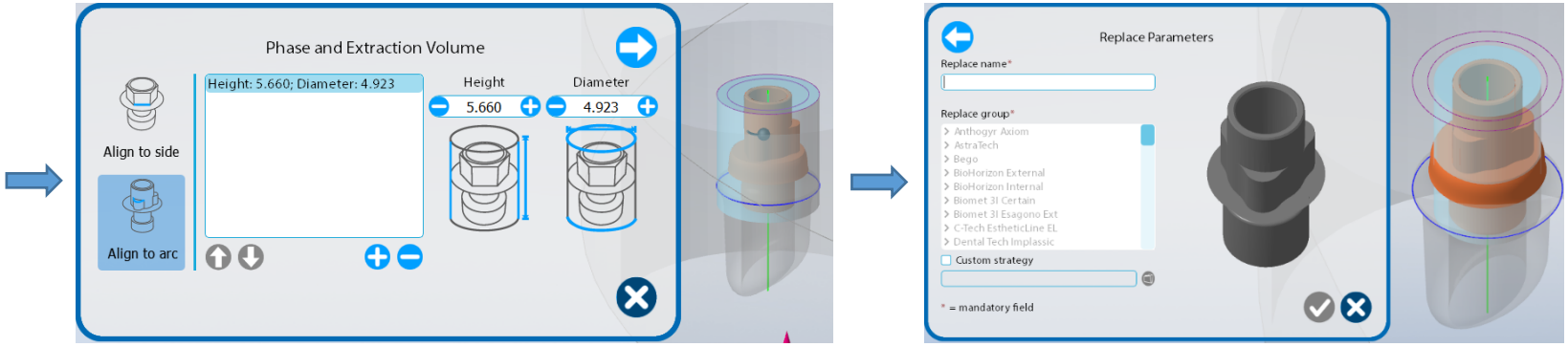
Afterwards it is advisable to use the 'Edit emergence profile' function from chapter 7.11.8 to correct the green area of the emergence profile.

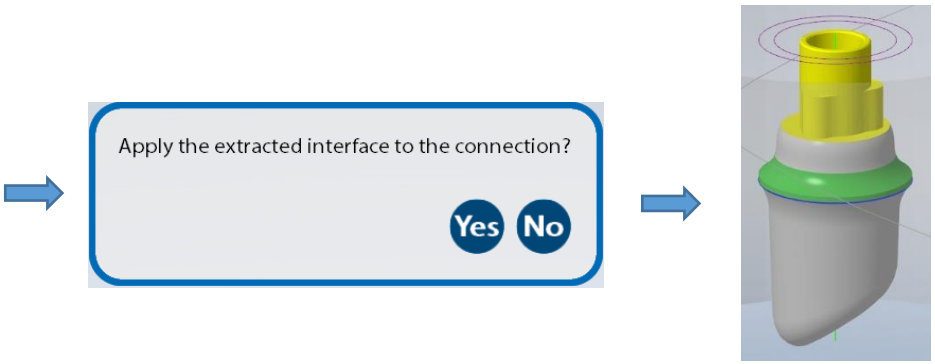
7.13.4 Exportieren

iCAM V5 offers the possibility to export interfaces from stl so that they can then be used with the ReFit function. This means that well-crafted interfaces can be placed on other stl without having to be in the CAD library.

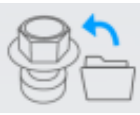
Activat the function and all surfaces except the interfaces become transparent. After selecting the interface to be edited, a menu appears to identify the interface. A click on the highlighted plus activates the cropping function and enables the interface to be identified by height and diameter. It is also possible to assign the position for the rotational lock. Afterwards the interface must be named (if the name of an existing group is placed in the beginning, the interface can be assigned to this group). (Optional custom strategy steps can be made with the iCAM V5 expert.) Finally, the exported interface can directly replace the existing one if the query is confirmed.





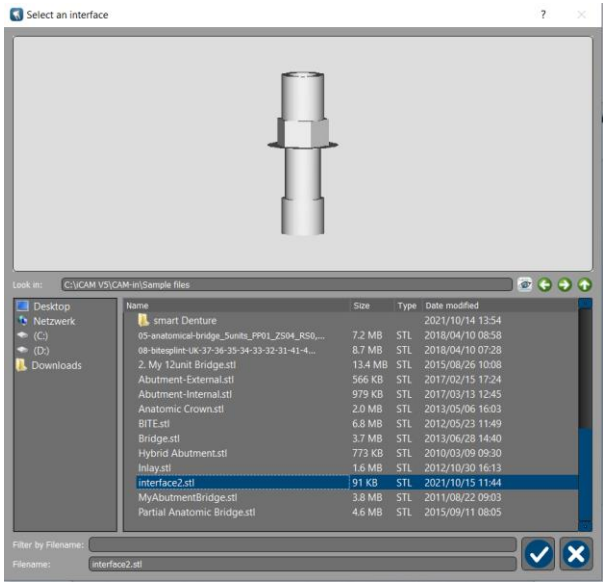

Apply the extracted interface to the connection?
Yes
No

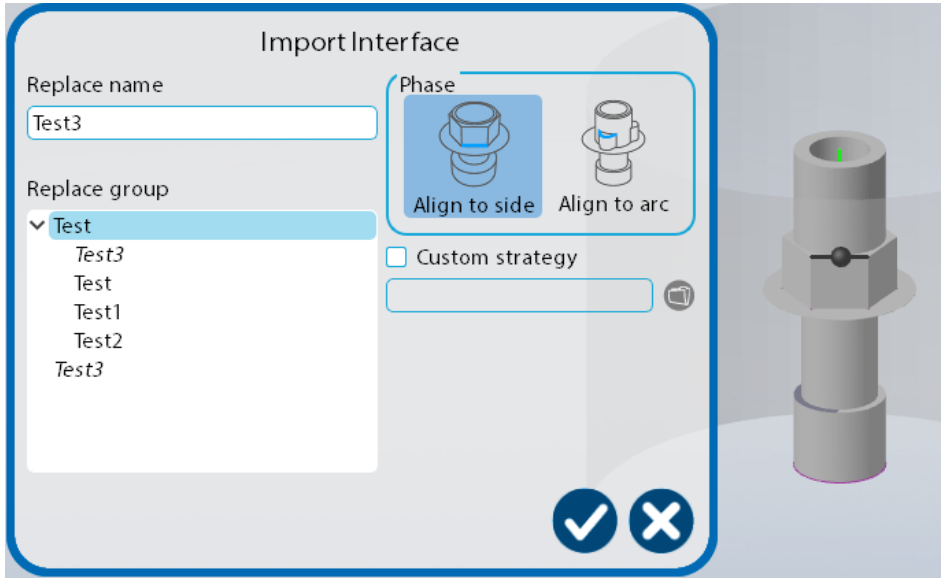
7.13.5 Importieren



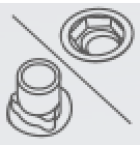
The import function can be used to make a stand-alone interface accessible for iCAM V5. This interface can then be used with the ReFit option. The prerequisite is that the interface is available as a readable file.

Activate the function and an explorer window opens. The interface to be imported must be selected and confirmed. After the software has loaded the interface, it must be named and assigned to a group. It is also possible to assign the position for the rotational lock. (Optional custom strategy steps can be made with the iCAM V5 expert.)





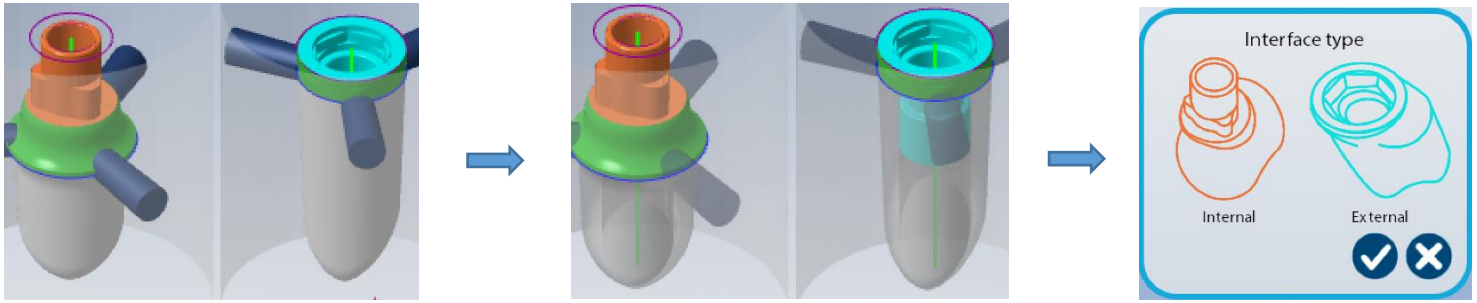
7.13.6 Change interface type



Interfaces exist in two types: external and internal; one is blue the other is orange. It depends on who you ask wether the implant or the abutment is viewed for the naming. Usually the type is automatically detected during import and does not require change.
This function changed since the update of September 2021. The original description will remain in the manual since it has critical information concerning the colour description.

Activate the tool and select an interface. It will change its colour and the software will ask about confirming the change.

The interface type is important for the milling strategy since the two types require different movements to be perfected.
Activate the tool and everything except interfaces will turn transparent. Select an interface and choose the type in the window that appears. The interface will change its colour accordingly. The window also suggests with its coloured pictures what colour the interface requires (only the active one is coloured): Interfaces going inside the abutment have to be blue while interfaces outside the abutment have to be orange as shown in pictures.



8 Shortcuts

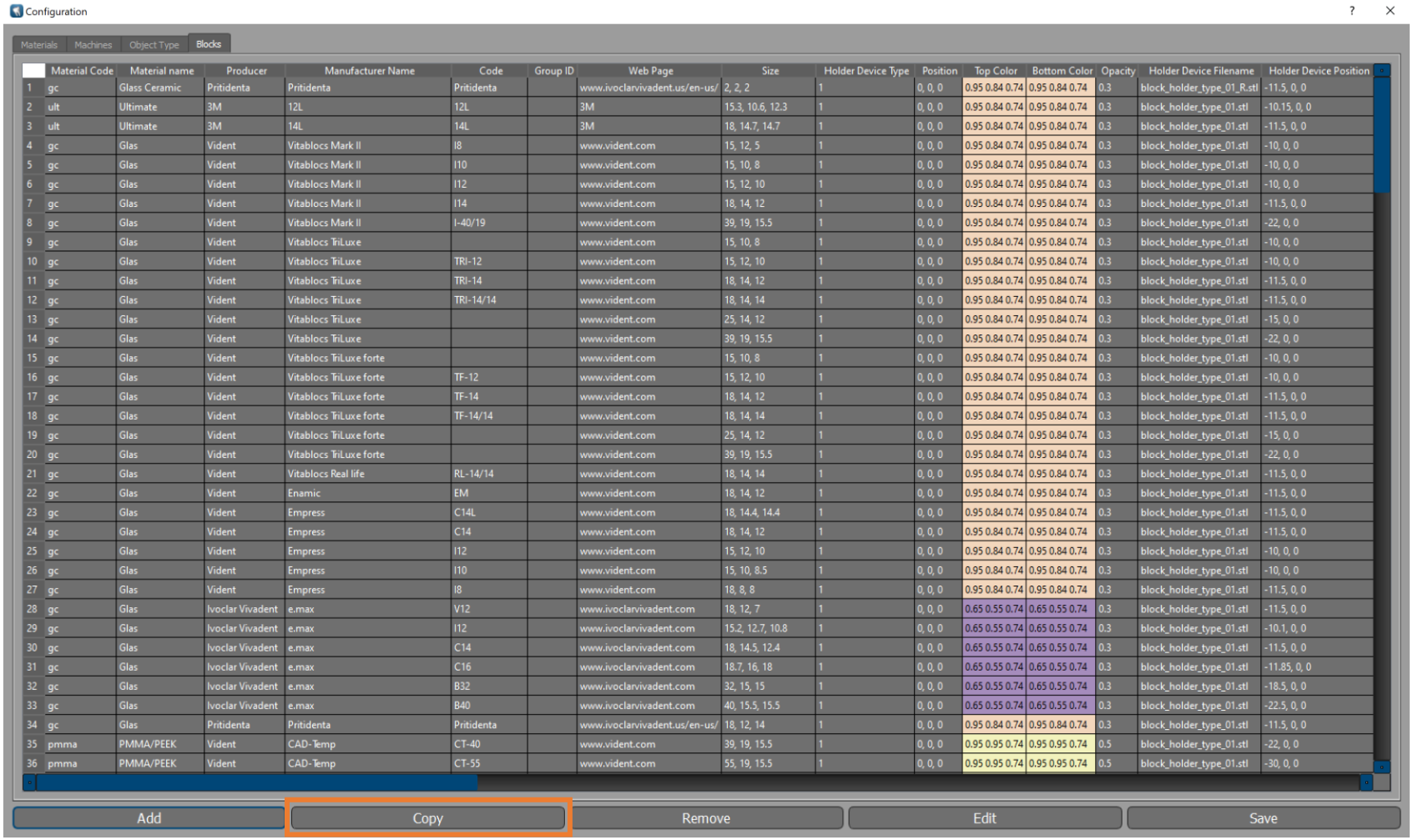
Here is a list of shortcuts usable in iCAM V5 smart

- CTRL+D opens the blank menu
- CTRL+E hides/shows equipment
- CTRL+G hides/shows cavities
- CTRL+R hides/shows red caps
- CTRL+U hides/shows undercuts
- 9 turns stl transparent
- 0 turns stl opaque
- F5 top view
- F6 front view
- F7 side view
- F8 isometric view
- F9 rotates A in view by 180°

9 Adding blanks

Press CRTL+D to open the blank menu.

A blank can be added by using the ‘add’ button and then entering each value. It is much easier to select a blank that is similar to the one to be added and clicking ‘copy’. This way all values are predefined and working; they just need adjustments.



	Material Code	Material name	Producer	Manufacturer Name	Code	Group ID	Web Page	Size	Holder Device Type	Position	Top Color	Bottom Color	Opacity	Holder Device Filename	Holder Device Position
1	gc	Glass Ceramic	Pritidenta	Pritidenta	Pritidenta		www.ivoclarvivadent.us/en-us/	2, 2, 2	1	0, 0, 0	0.95 0.84 0.74	0.95 0.84 0.74	0.3	block_holder_type_01_R.stl	-11.5, 0, 0
2	ult	Ultimate	3M	12L	12L		3M	15.3, 10.6, 12.3	1	0, 0, 0	0.95 0.84 0.74	0.95 0.84 0.74	0.3	block_holder_type_01.stl	-10.15, 0, 0
3	ult	Ultimate	3M	14L	14L		3M	18, 14.7, 14.7	1	0, 0, 0	0.95 0.84 0.74	0.95 0.84 0.74	0.3	block_holder_type_01.stl	-11.5, 0, 0
4	gc	Glas	Vident	Vitablocs Mark II	I8		www.vident.com	15, 12, 5	1	0, 0, 0	0.95 0.84 0.74	0.95 0.84 0.74	0.3	block_holder_type_01.stl	-10, 0, 0
5	gc	Glas	Vident	Vitablocs Mark II	I10		www.vident.com	15, 10, 8	1	0, 0, 0	0.95 0.84 0.74	0.95 0.84 0.74	0.3	block_holder_type_01.stl	-10, 0, 0
6	gc	Glas	Vident	Vitablocs Mark II	I12		www.vident.com	15, 12, 10	1	0, 0, 0	0.95 0.84 0.74	0.95 0.84 0.74	0.3	block_holder_type_01.stl	-10, 0, 0
7	gc	Glas	Vident	Vitablocs Mark II	I14		www.vident.com	18, 14, 12	1	0, 0, 0	0.95 0.84 0.74	0.95 0.84 0.74	0.3	block_holder_type_01.stl	-11.5, 0, 0
8	gc	Glas	Vident	Vitablocs Mark II	I-40/19		www.vident.com	39, 19, 15.5	1	0, 0, 0	0.95 0.84 0.74	0.95 0.84 0.74	0.3	block_holder_type_01.stl	-22, 0, 0
9	gc	Glas	Vident	Vitablocs TriLuxe			www.vident.com	15, 10, 8	1	0, 0, 0	0.95 0.84 0.74	0.95 0.84 0.74	0.3	block_holder_type_01.stl	-10, 0, 0
10	gc	Glas	Vident	Vitablocs TriLuxe	TRI-12		www.vident.com	15, 12, 10	1	0, 0, 0	0.95 0.84 0.74	0.95 0.84 0.74	0.3	block_holder_type_01.stl	-10, 0, 0
11	gc	Glas	Vident	Vitablocs TriLuxe	TRI-14		www.vident.com	18, 14, 12	1	0, 0, 0	0.95 0.84 0.74	0.95 0.84 0.74	0.3	block_holder_type_01.stl	-11.5, 0, 0
12	gc	Glas	Vident	Vitablocs TriLuxe	TRI-14/14		www.vident.com	18, 14, 14	1	0, 0, 0	0.95 0.84 0.74	0.95 0.84 0.74	0.3	block_holder_type_01.stl	-11.5, 0, 0
13	gc	Glas	Vident	Vitablocs TriLuxe			www.vident.com	25, 14, 12	1	0, 0, 0	0.95 0.84 0.74	0.95 0.84 0.74	0.3	block_holder_type_01.stl	-15, 0, 0
14	gc	Glas	Vident	Vitablocs TriLuxe			www.vident.com	39, 19, 15.5	1	0, 0, 0	0.95 0.84 0.74	0.95 0.84 0.74	0.3	block_holder_type_01.stl	-22, 0, 0
15	gc	Glas	Vident	Vitablocs TriLuxe forte			www.vident.com	15, 10, 8	1	0, 0, 0	0.95 0.84 0.74	0.95 0.84 0.74	0.3	block_holder_type_01.stl	-10, 0, 0
16	gc	Glas	Vident	Vitablocs TriLuxe forte	TF-12		www.vident.com	15, 12, 10	1	0, 0, 0	0.95 0.84 0.74	0.95 0.84 0.74	0.3	block_holder_type_01.stl	-10, 0, 0
17	gc	Glas	Vident	Vitablocs TriLuxe forte	TF-14		www.vident.com	18, 14, 12	1	0, 0, 0	0.95 0.84 0.74	0.95 0.84 0.74	0.3	block_holder_type_01.stl	-11.5, 0, 0
18	gc	Glas	Vident	Vitablocs TriLuxe forte	TF-14/14		www.vident.com	18, 14, 14	1	0, 0, 0	0.95 0.84 0.74	0.95 0.84 0.74	0.3	block_holder_type_01.stl	-11.5, 0, 0
19	gc	Glas	Vident	Vitablocs TriLuxe forte			www.vident.com	25, 14, 12	1	0, 0, 0	0.95 0.84 0.74	0.95 0.84 0.74	0.3	block_holder_type_01.stl	-15, 0, 0
20	gc	Glas	Vident	Vitablocs TriLuxe forte			www.vident.com	39, 19, 15.5	1	0, 0, 0	0.95 0.84 0.74	0.95 0.84 0.74	0.3	block_holder_type_01.stl	-22, 0, 0
21	gc	Glas	Vident	Vitablocs Real life	RL-14/14		www.vident.com	18, 14, 14	1	0, 0, 0	0.95 0.84 0.74	0.95 0.84 0.74	0.3	block_holder_type_01.stl	-11.5, 0, 0
22	gc	Glas	Vident	Enamic	EM		www.vident.com	18, 14, 12	1	0, 0, 0	0.95 0.84 0.74	0.95 0.84 0.74	0.3	block_holder_type_01.stl	-11.5, 0, 0
23	gc	Glas	Vident	Empress	C14L		www.vident.com	18, 14.4, 14.4	1	0, 0, 0	0.95 0.84 0.74	0.95 0.84 0.74	0.3	block_holder_type_01.stl	-11.5, 0, 0
24	gc	Glas	Vident	Empress	C14		www.vident.com	18, 14, 12	1	0, 0, 0	0.95 0.84 0.74	0.95 0.84 0.74	0.3	block_holder_type_01.stl	-11.5, 0, 0
25	gc	Glas	Vident	Empress	I12		www.vident.com	15, 12, 10	1	0, 0, 0	0.95 0.84 0.74	0.95 0.84 0.74	0.3	block_holder_type_01.stl	-10, 0, 0
26	gc	Glas	Vident	Empress	I10		www.vident.com	15, 10, 8.5	1	0, 0, 0	0.95 0.84 0.74	0.95 0.84 0.74	0.3	block_holder_type_01.stl	-10, 0, 0
27	gc	Glas	Vident	Empress	I8		www.vident.com	18, 8, 8	1	0, 0, 0	0.95 0.84 0.74	0.95 0.84 0.74	0.3	block_holder_type_01.stl	-11.5, 0, 0
28	gc	Glas	Ivoclar Vivadent	e.max	V12		www.ivoclarvivadent.com	18, 12, 7	1	0, 0, 0	0.65 0.55 0.74	0.65 0.55 0.74	0.3	block_holder_type_01.stl	-11.5, 0, 0
29	gc	Glas	Ivoclar Vivadent	e.max	I12		www.ivoclarvivadent.com	15.2, 12.7, 10.8	1	0, 0, 0	0.65 0.55 0.74	0.65 0.55 0.74	0.3	block_holder_type_01.stl	-10.1, 0, 0
30	gc	Glas	Ivoclar Vivadent	e.max	C14		www.ivoclarvivadent.com	18, 14.5, 12.4	1	0, 0, 0	0.65 0.55 0.74	0.65 0.55 0.74	0.3	block_holder_type_01.stl	-11.5, 0, 0
31	gc	Glas	Ivoclar Vivadent	e.max	C16		www.ivoclarvivadent.com	18.7, 16, 18	1	0, 0, 0	0.65 0.55 0.74	0.65 0.55 0.74	0.3	block_holder_type_01.stl	-11.85, 0, 0
32	gc	Glas	Ivoclar Vivadent	e.max	B32		www.ivoclarvivadent.com	32, 15, 15	1	0, 0, 0	0.65 0.55 0.74	0.65 0.55 0.74	0.3	block_holder_type_01.stl	-18.5, 0, 0
33	gc	Glas	Ivoclar Vivadent	e.max	B40		www.ivoclarvivadent.com	40, 15.5, 15.5	1	0, 0, 0	0.65 0.55 0.74	0.65 0.55 0.74	0.3	block_holder_type_01.stl	-22.5, 0, 0
34	gc	Glas	Pritidenta	Pritidenta	Pritidenta		www.ivoclarvivadent.us/en-us/	18, 12, 14	1	0, 0, 0	0.95 0.84 0.74	0.95 0.84 0.74	0.3	block_holder_type_01.stl	-11.5, 0, 0
35	pmma	PMMA/PEEK	Vident	CAD-Temp	CT-40		www.vident.com	39, 19, 15.5	1	0, 0, 0	0.95 0.95 0.74	0.95 0.95 0.74	0.5	block_holder_type_01.stl	-22, 0, 0
36	pmma	PMMA/PEEK	Vident	CAD-Temp	CT-55		www.vident.com	55, 19, 15.5	1	0, 0, 0	0.95 0.95 0.74	0.95 0.95 0.74	0.5	block_holder_type_01.stl	-30, 0, 0

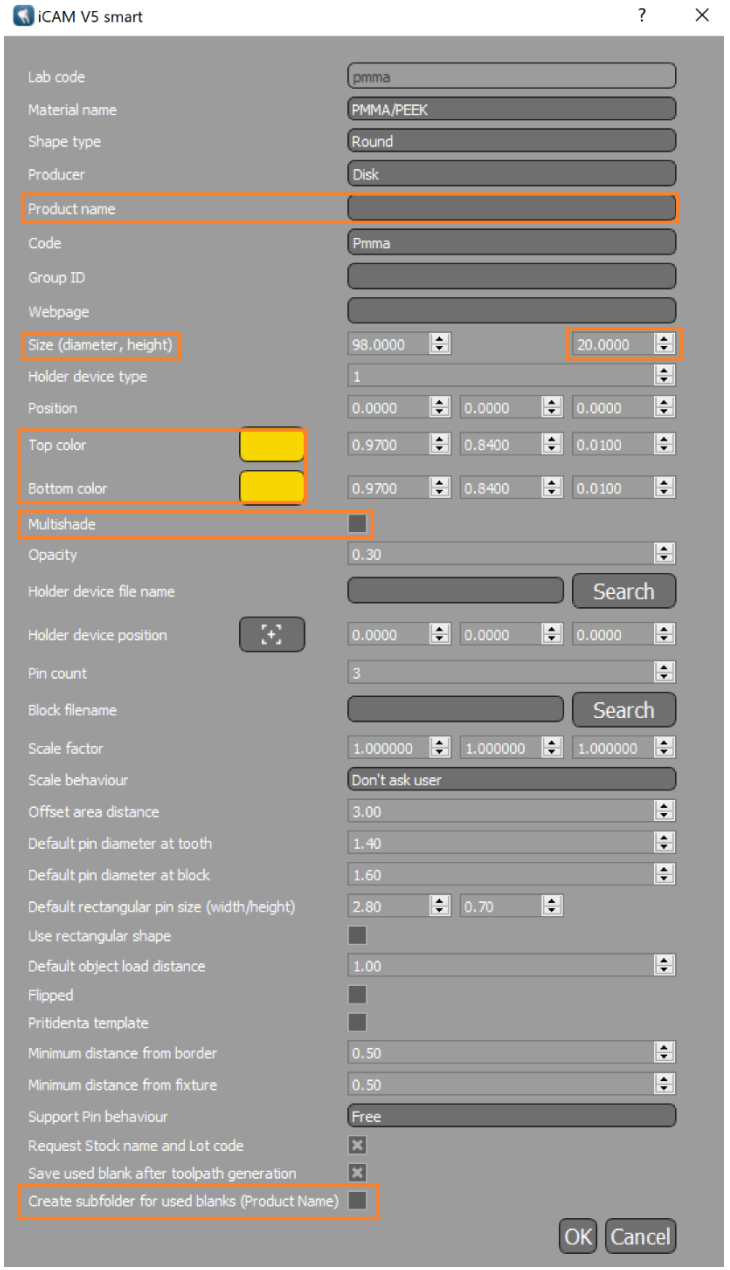
9.1 Adding a disk

After copying a disk the window on the right will open.
It is usually sufficient to simply change the height (do not change diameter!) of the new blank and save it.

If the blank is made from a special producer and is intended to be saved separately from other blanks of the same material group, the ‘product name’ has to be filled and the option ‘create subfolder for used blanks (product name)’ at the bottom has to be selected

The colour of the blank can be changed by changing ‘top color’ and/or ‘bottom colour’. If the blank is a multilayer blank, the option ‘multishade’ can be selected. Afterwards the colour scheme of the blank can be selected by clicking on the new option that appears:

In the new window the colours can be varied as the blank and user request it.



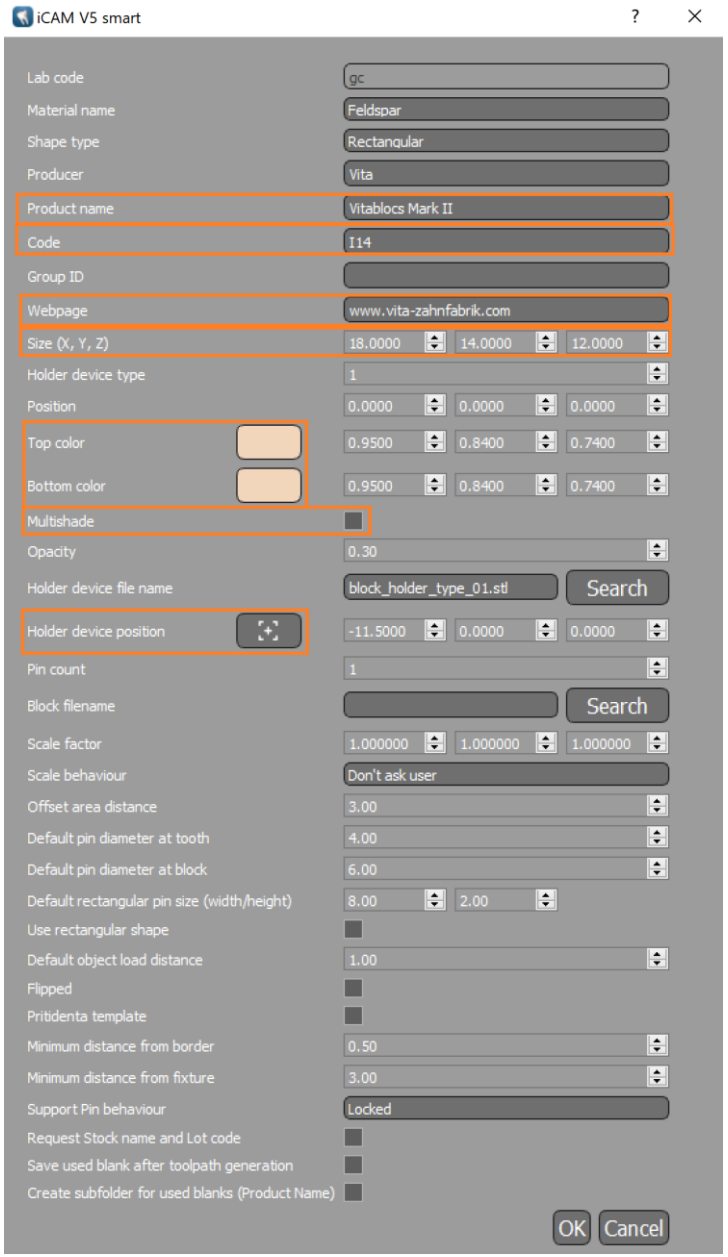
9.2 Adding a block

After copying a disk the window on the right will open.
If the size of the blank has to be changed, it is important to click on ‘holder device position’ afterwards (else the aluminium holder of the blank will be in the wrong position and thus the alignment of the blank in the CAM will be wrong; this would lead to erroneous millings).

The product name, the webpage and the code (usually block size) can be changed according to the blank specifics.

The colour of the blank can be changed by changing ‘top color’ and/or ‘bottom colour’. If the blank is a multilayer blank, the option ‘multishade’ can be selected. Afterwards the colour scheme of the blank can be selected by clicking on the new option that appears:

In the new window the colours can be varied as the blank and user request it.



10 Special Modules

Some function groups of the iCAM V5 require a specific procedure in order to function perfectly.

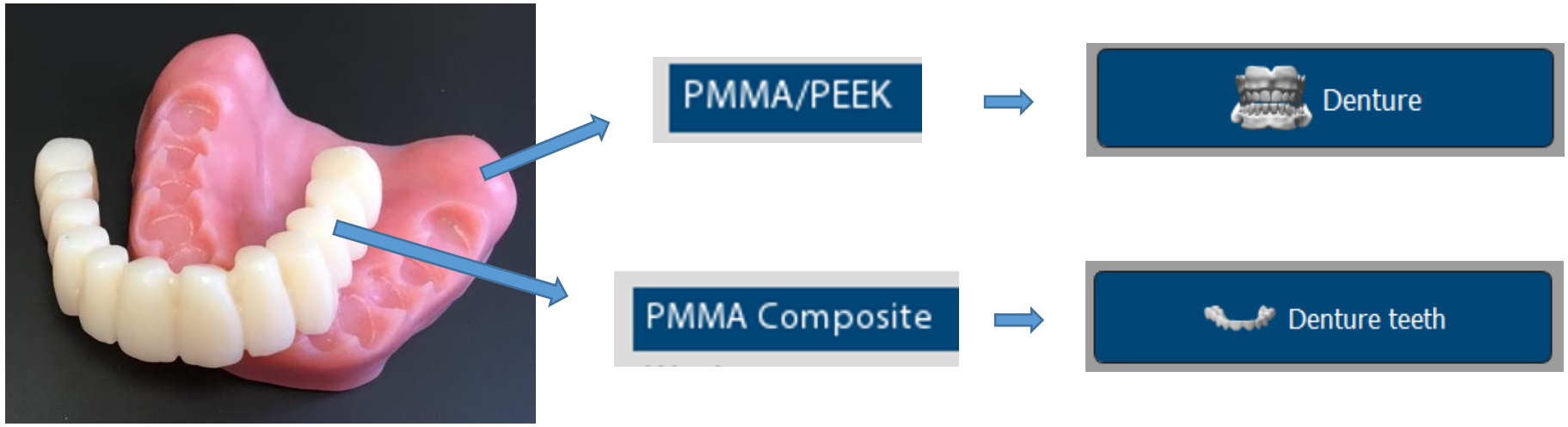
10.1 Denture Module

The denture module offered by imes-icore is based on milling a denture base made of plastic and gluing teeth into it. This can be done in two ways:

1. Mill the base and teeth yourself
2. Mill the prosthesis yourself, use prefabricated teeth (Smart Denture)

10.1.1 Mill custom teeth

For this process, the denture base is milled from PMMA/PEEK using the object type ‘Denture’. In addition customized teeth are milled from PMMA composite as object type ‘Denture teeth’. The results are then merged.



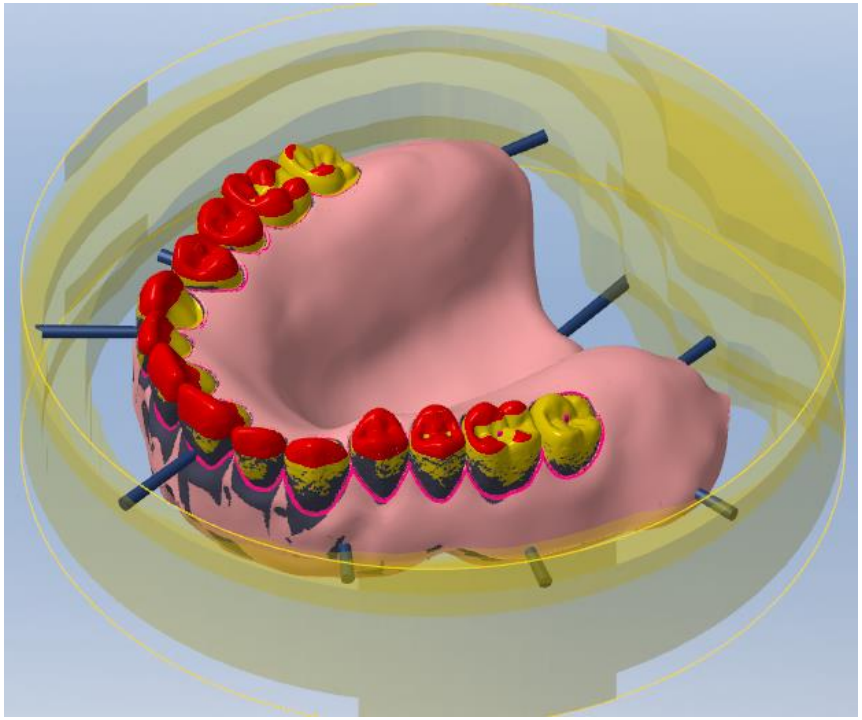
10.1.2 Smart Denture

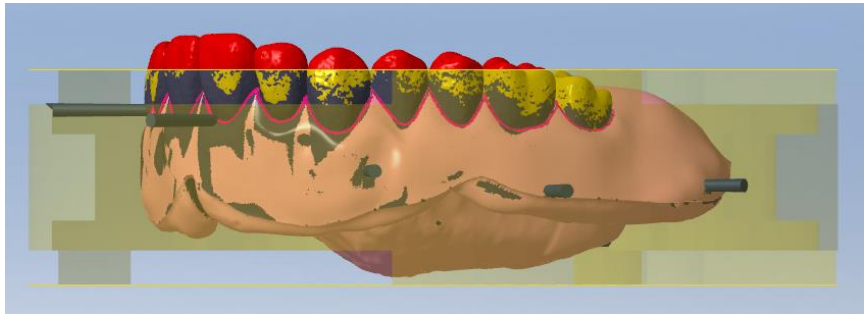
The Smart Denture Module is a process that was developed by imes icore to simplify the use of prefabricated teeth. All Components updates after October 15, 2021 activate it and enable the use of the Smart Denture material, which allows the automated workflow. The following CAD output data is required for this:

ExoCAD
- prostheticbase_cad.stl
- prostheticmonoblock_cad.stl
- constructioninfo

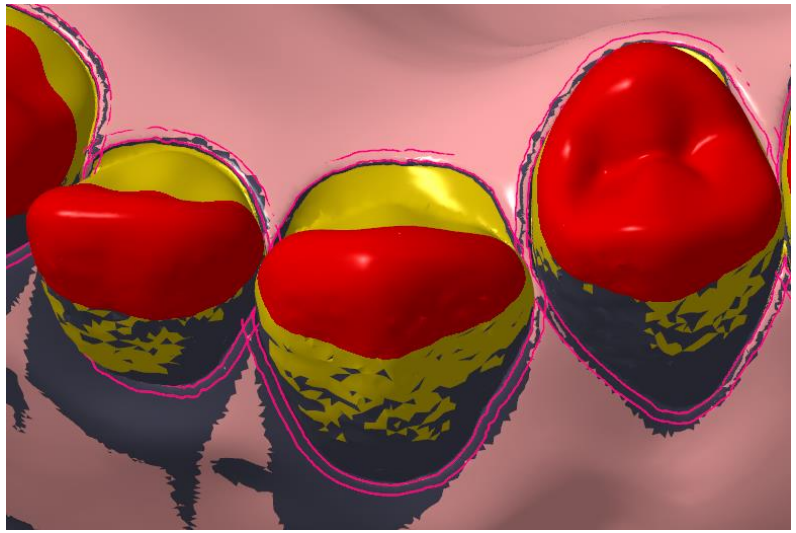
3shape
- stl
- monoblock.stl
- pts

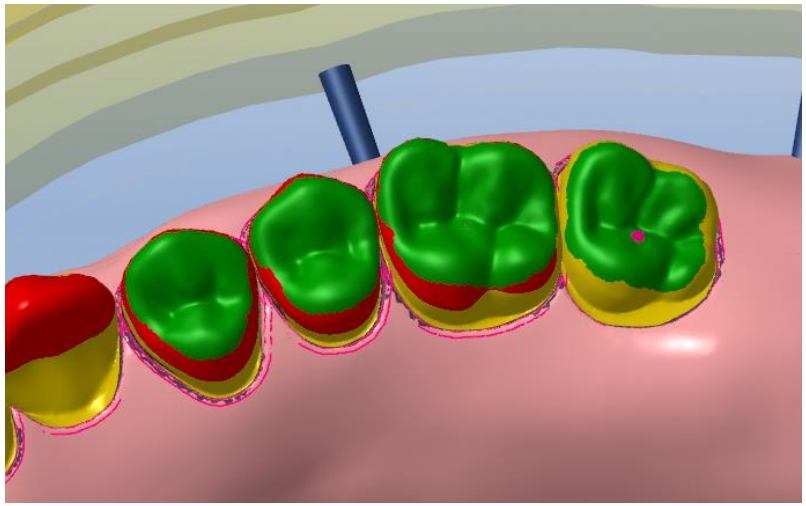
The prostheticbase_cad.stl/stl file is imported. The prostheticmonoblock_cad.stl/monoblock.stl will then be displayed. This calculates the height of the blank WITH teeth. However, only the prosthesis base needs to be in the blank when positioning (see pictures).





It is essential to ensure that all tooth pockets have a double pink outline curve, otherwise the strategy will not work properly. If the teeth are to be finished (reoccluded), they must be marked using ‘Milling Zone Designation’ (see Chapter 7.4).

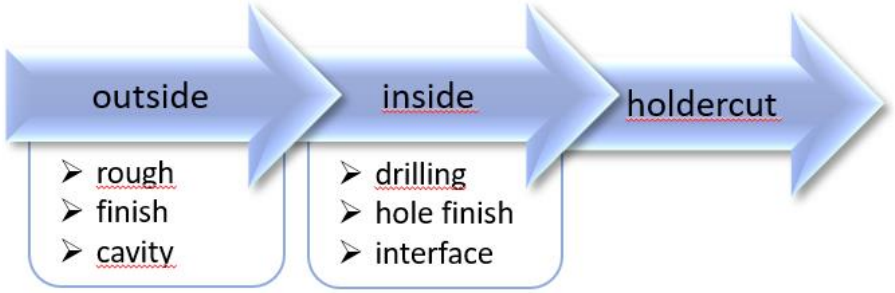




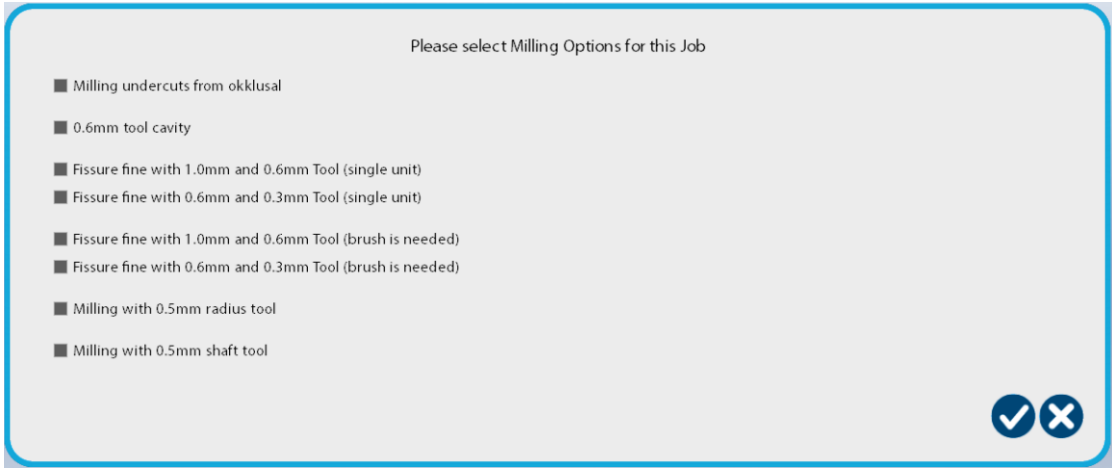
The calculation can then be started. The warning that objects are larger than the blank can be ignored if the stl was placed as described. The calculation will then generate two iso files. First the iso has to be milled, which only mills out the occlusal side with the pockets.
The holder can then be removed from the zero point clamping system in order to glue/polymerize the teeth. Then the holder is clamped back into the machine and the part2.iso started to mill the basal side and potentially reocclude the teeth.

11 Strategy options

Imes icore built unique strategies for its customers. These strategies are built in blocks and are divided by material and object types. Each material has its own milling parameters so that the tools perform optimal. Each object type has its own options so that different requirements can be met efficiently.
The base strategy is similar for every process and is basically:



The base strategy is not optimal for all stl. Some specifics might be missed by it (e.g. fissures). Strategy options are made specifically to detail these areas on certain object types. The list of possible options will appear once the calculation starts. Checking options will add them to the milling process. This chapter will show a brief explanation of how these options work.

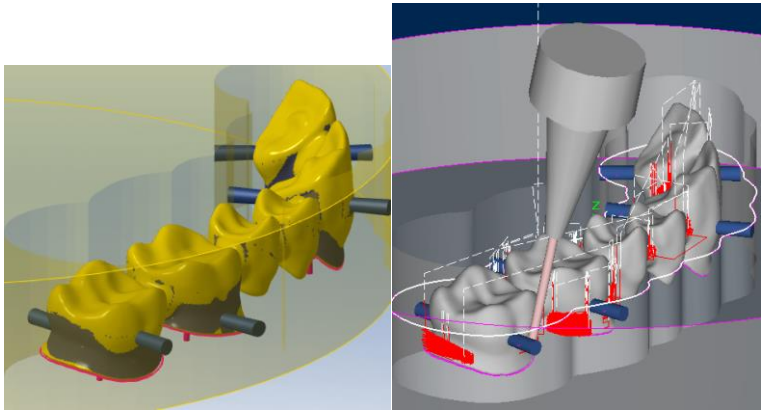


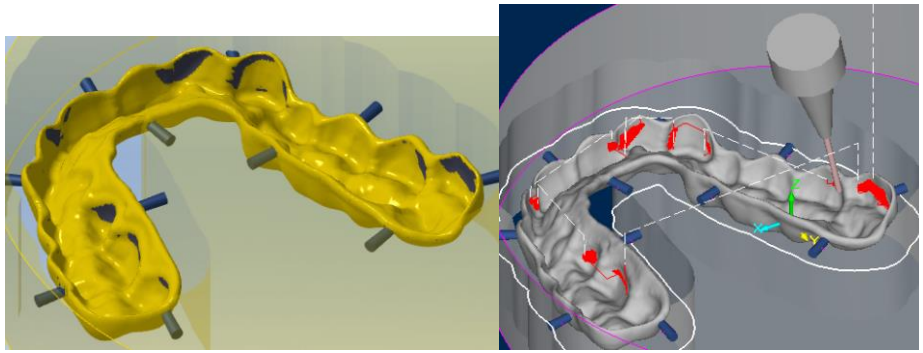
11.1 4mm Roughing with tool T30

This option is exclusive PMMA/PEEK/Wax and 6mm shaft machines.
The roughing process will start with T30 which is larger and can thus work faster. Afterwards T11 will rough the rest material.

11.2 Milling undercuts from occlusal/lower side

Undercuts are shown in dark blue on stl. These areas cannot be reached from the 0 direction which is used for most operations. To reduce the rest material in these areas it is necessary to angle the burr during the milling process. This option is a 5 axes simultaneous operation from the occlusal or basal side which angles the burr in A and B direction up to the machine limit automatically.





11.3 1,5mm tool cavity

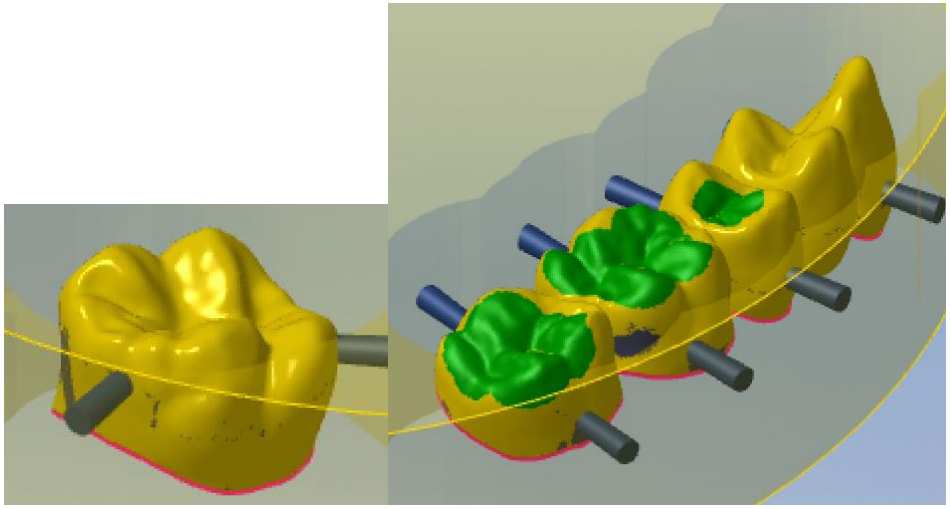
This option is exclusive for milling metal cavities. It activates the 1,5mm tool (T3/T8/T28) to prepare cavities for the 1mm tool. This option is enabled automatically since it enhances the tool life of the 1mm tool. It can be deselected though it is recommended.

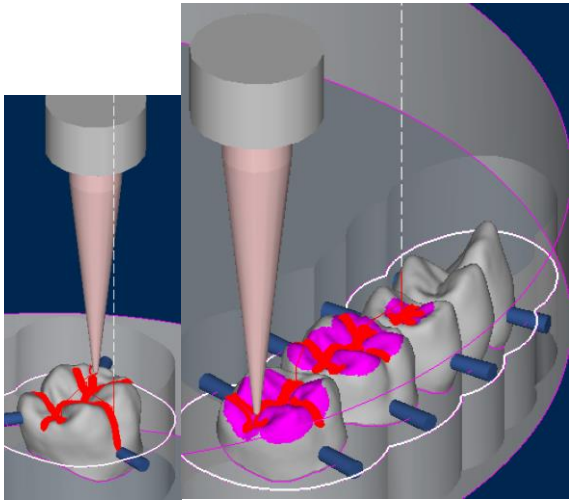
11.4 0,6mm tool cavity

Usually imes icore recommends to use an option in the CAD that prepares cavities for a 1mm tool. In some cases of anterior teeth this would mean a huge coping instead of an elegant one. In those cases it is possible to prepare the cavity for a 0,6mm tool. In combination with this option T32 will additionally be used for soft materials to mill cavities.

11.5 Fissure fine with 1mm and 0,6mm/0,6mm and 0,3mm tool

Usually the smallest tool used for the outside is either 1,5mm (metal) or 1mm (soft). If fissures are required to milled finer one of these options can be chosen. Two extra strategy steps will be activated to safely mill the fissures in the required quality. For multi-unit stl like hybrid abutments and bridges these steps need the area to be milled coloured by the ‘select’ tool (brush, see chapter 7.4)



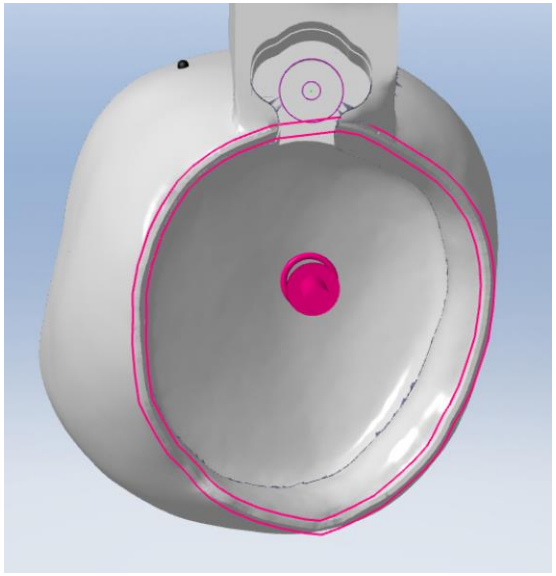


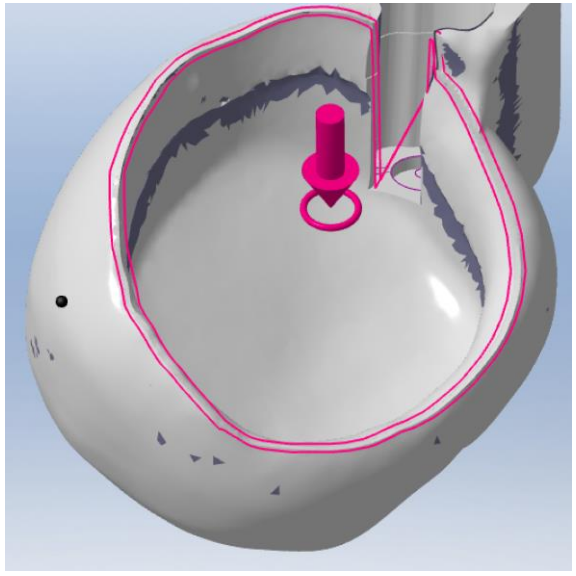
11.6 Superfinish outside

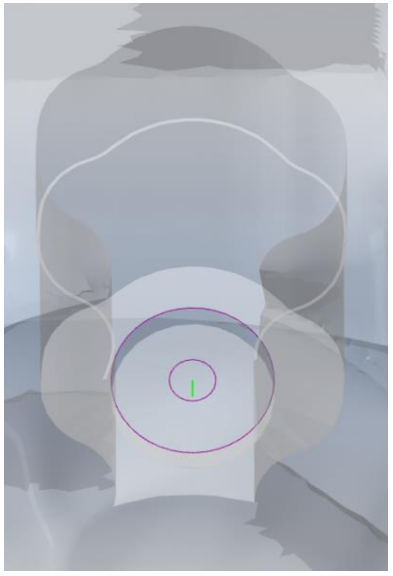
The base strategy for metal will finish the outside (occlusal and basal) with a 2mm burr and use a 1,5mm burr for finer details. The superfinish option will add a finishing step over the whole outside with a 1,5mm burr which will result in a nearly polished surface (extra step requires extra time).

11.7 TK soft

This step is meant for friction pieces of secondary telescopic stl. These friction elements are usually attached to cavities and cannot be reached with the standard steps. This option needs a special preparation of curves for for the stl:
The inside of the friction element needs an automatically detected drilling axis. It is also necessary that the cavity curve of the CAM only includes the cavity (NOT the friction element position). This may require a manual repositioning of the cavity curve (compare chapter 7.11.1.2). It does not matter if the curve falls into the gab. This option will then add strategy steps to successfully mill the friction position.

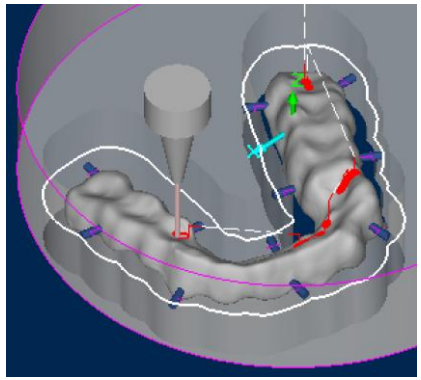






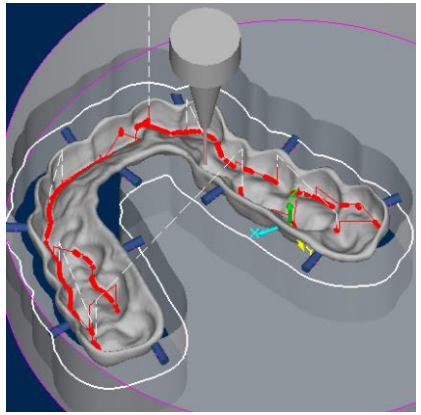
11.8 Upper side with 1mm tool

This option is available for surgical guides and bite splints. These object types are usually finished with a 2,5mm tool on the occlusal side. This option adds a rest material step with a 1mm tool. Thus finer details can be milled.



11.9 Lower side with 0,6mm tool

This option is available for surgical guides and bite splints. These object types are usually finished with a 1mm tool on the basal side. This option adds a rest material step with a 0,6mm tool. Thus finer details can be milled.

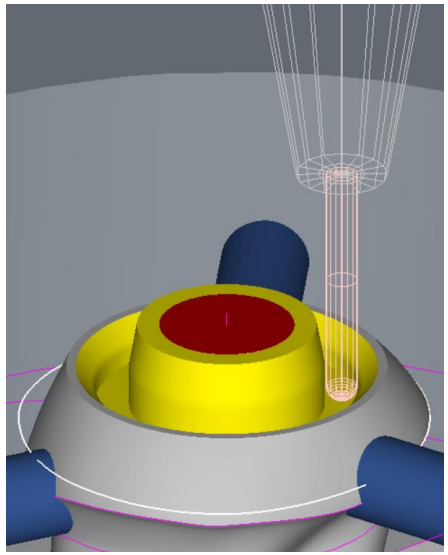


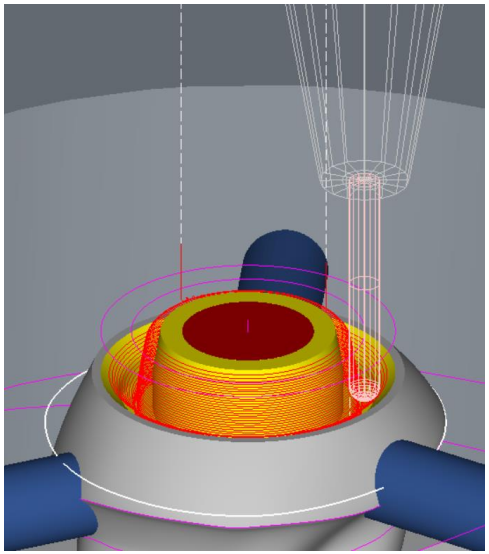
11.10 Drill tools

Usually holes will be milled (milled, not drilled) with 1,5mm and 1mm tools. If this option is activated the tools T80 and T81 will be used to drill the holes.

11.11 Milling with 0,5mm radius/shaft tool

This option adds the tools T18/T19 as a finishing step for interfaces. Since these tools cannot bear much load they will only work slowly. It is recommended to only activate them when interfaces require them since the option will mill on every available interface.





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