hyperDENT®

V8

INSTRUCTION MANUAL Version 8

FOLLOW-ME!
TECHNOLOGY GROUP

Contents

1 General information

1.1 Appropriate use

hyperDENT® is a CAM software used to manufacture dentures.

The range of parts includes:

- *Coping*
- *Coping bridge*
- *Crown*
- *Crown bridge, Maryland bridge*
- *Abutment*
- *Abutment crown*
- *Abutment bridge*
- *Abutment crown bridge*
- *Inlay/onlay*
- *Inlay/onlay bridge*
- *Anatomical inlay/onlay*
- *Inlay/onlay crown bridge*
- *Telescope*
- *Overpress*
- *Bitesplint*
- *Model*
- *Model stump*
- *Bridge (stump- and implant-supported)*
- *Crown bridge, using adhesive and screwed directly*
- *Abutment bridge, using adhesive and screwed directly*
- *User-defined parts*
- *Full lower dentures*
- *Full upper dentures*

hyperDENT® has an open software architecture that is able to load parts in STL format. The most diverse CAD systems, scanners and milling machines can thereby be combined – as well as blanks and tools from different manufacturers.

1.2 hyperDENT® instruction manual

This handbook contains the instruction manual for program versions and options.

The described functions are only available to you with the corresponding program version and the relevant options.

Even without restrictive reference to an option or version, the description in this handbook does not constitute a guarantee that the program function is available as part of your license.

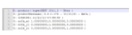
1.3 hyperDENT® program versions

hyperDENT® is available in two versions:

- *Compact*
- *Classic*

Information about the function scope of the different versions is available on the website.

1.4 Version number



The current version number of hyperDENT® can be found under Help -> About and is also shown in the following files:

- *Log file*
- *Project management file*
- *POF file*

1.5 User-specific profiles

The profiles (settings for blanks and building elements) are stored in the database. Only copies of the original profiles can be changed; the originals will remain unchanged. The new user-specific settings (profiles) are saved in a customer-specific location.

- *Connector profiles*
- *Sintering pin profiles*
- *Blank geometries*
- *Blank types*

1.6 Updating construction machines

Ensure that the control units of your construction machines have the current program version.

When using hyperDENT® with a Röders milling machine in combination with the automation for controlling the Denttrays, it must be ensured that the control unit has received an update since 16 November 2011.

2 Installation

2.1 Prerequisite

hyperDENT® has the following software requirements:

- *Microsoft® .NET Framework 4.6.1*
- *hyperVIEW®, Version 2015*

Microsoft .NET Framework Version 4.6.1

- *.NET Framework 4.6.1 must be installed and set up ready for operation before hyperDENT® is installed. If Framework is not yet installed on the computer, it is automatically installed from the CD.*

***hyper*VIEW®**

- *The installation of hyperDENT® also requires hyperVIEW®.*

Installing the program via Setup

1. Insert the installation DVD into the DVD drive.

The installation program starts up automatically after a few seconds.

Tip: Automatic start of CD
If the computer is configured so that DVDs do not automatically start, then you must start the installation program manually:
Click on <Start> and select <Execute>.
Enter the letter of the DVD drive followed by ":\setup" (e.g.: D:\setup) and then click on <OK>.

---

2. Follow the instructions of the installation program:
   Make sure to read the legal instructions.
   During installation, you will be asked to enter the path for the directory in which you want to install hyperDENT®:
   Default: C:\Programs\FOLLOW ME\hyperDENT.
   Confirm with Next or enter a different directory/drive.

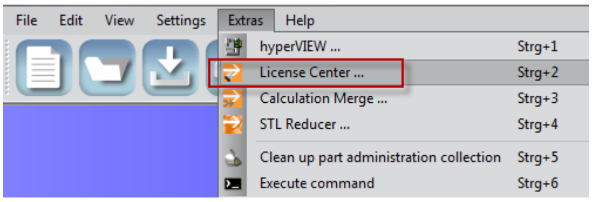
A link to the start of the program is automatically created by the installation routine.

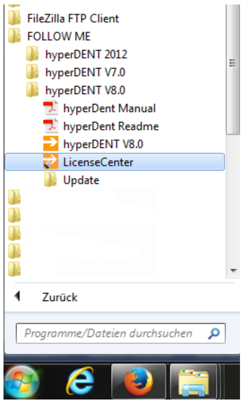
Observe the installation advice in the installation guide and the program's Readme file.

2.2 License Center

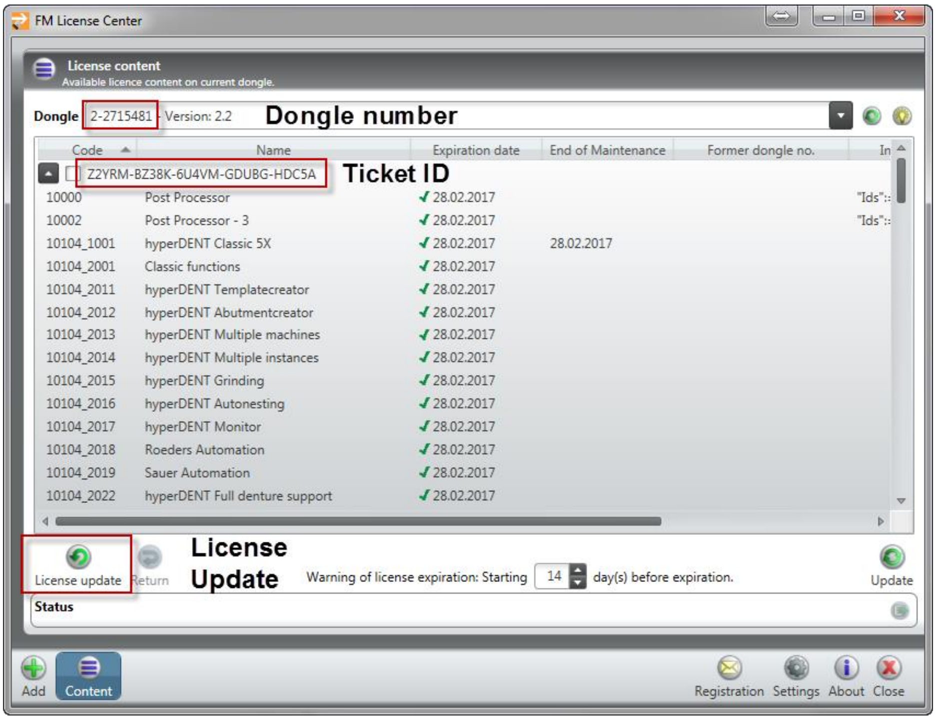
The License Center contains the license management function. Here, you can view all the held licenses and machines. You can also view the ticket ID and the dongle number. A license update is needed if any new functions/options are enabled or the maintenance period is extended.

The License Center can be opened via [Extras > License Center] or via the Windows start menu.



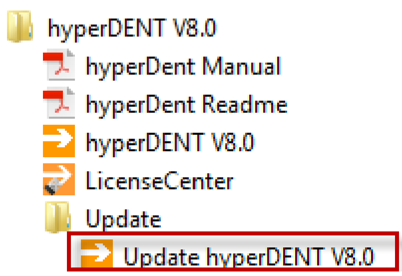


Click on License update to call up the changes.



2.3 Software – updates

If software updates of the current release are available, they can be downloaded and installed directly. To do so, click on hyperDENT update in the Windows start menu. If this does not work, please contact support.



You can also find the Readme PDF and the handbook as a PDF in the Windows start menu. The Readme file documents any changes, improvements, new developments, etc. in the intermediate releases.

3 Operating functions

3.1 Overview of program interface

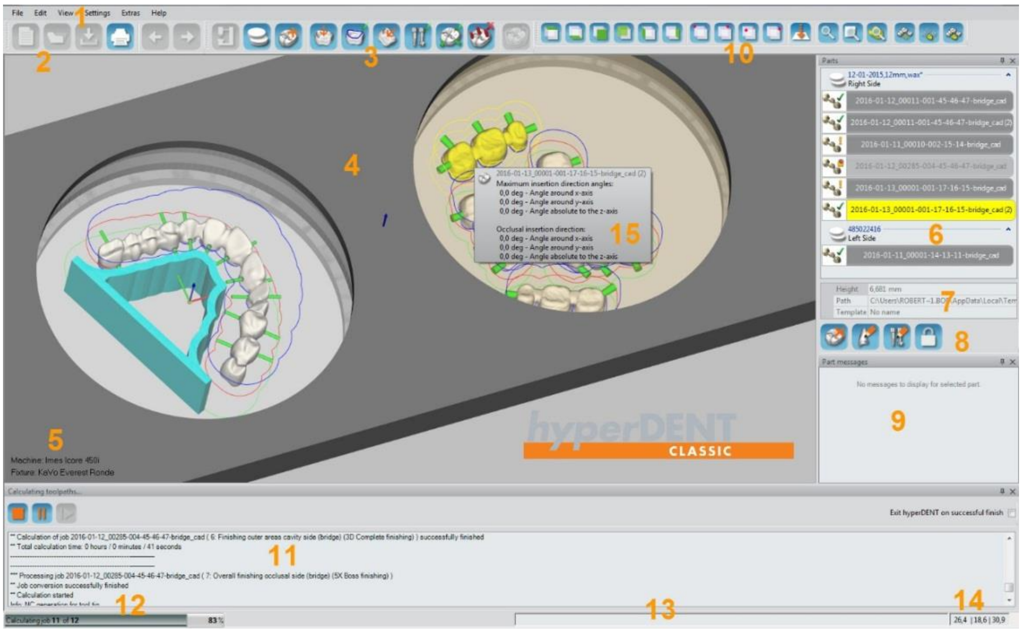


Figure 3-1

1. Menu bar
2. Toolbar – Project management
3. Toolbar – Process steps
4. Workspace
5. Project information – Machine and holders
6. Part browser with blank data
7. Part data
8. Toolbar – Part functions
9. Message screen
   Pictogram “Cross” = Calculation not possible
   Pictogram “Caution” = Machining is possible but may be faulty
10. Tool bar
11. Calculation of toolpaths
12. Status bar with progress bar for the calculation and information display (text)
13. Information row with details about necessary actions for the currently selected function
14. Angle specifications for the position for viewing the blank
15. Information window: depending on the cursor position, dynamic display of angle specifications for the part, the margin line or the screw channel

3.2 Toolbar – Process steps

The description of the process steps can be found under the relevant chapter headings.

Select milling unit	Select milling machine and fixture.
Load blank	Select blank and insert into the holder.
Load part	Load part data/tool data.
Set milling direction	Set insertion direction, main machining direction (alignment of the part to the tool), and adjust direction.
Identify part features	Highlight and select margin line and pontics, screw channel, and interface geometry, and define individual machining areas.
Tilt part in blank	Tilt part to minimize the height in the blank – inclination for optimized position of 3+1-axis machining.
Select template	Select milling strategy for machining.
Set connectors	Set support pins for machining.
Set sintering pins	Set sintering pins for the sintering process.
Calculate toolpaths	Calculate construction data with the toolpaths.

3.3 Menu bar, menu items, icons

Menu [File], context menu, toolbar

New project	Create new project.
Open project	Select and open existing project, close current project.
Save project	Save current project.

	Save copy of project	Save current project under a different name or in a different directory.
	Print page setup	Arrange print page, determine page size, alignment, and margins.
	Project print setting selection	Select project settings for printing.
	Print project preview	Create and display print preview with the selected settings.
	Print project	Print current project with the selected print setting selection.
	Exit	Exit hyperDENT®, close program.
	Menu [Edit], toolbar	
	Undo	Undo last action.
	Redo	Redo last action.
	Delete selected	Delete highlighted part.
	Blank	Call up submenu.
	> Rotate blank	Rotate blank in the fixture, e.g. for better use of the space left, greater distance between the part and fixture, adapt the tilt part to the swiveling axis of the fixture.
	Part	Call up submenu.
	> Edit template parameters	Change template for part. Only available in template generator module option.
	> Move part	Highlight selected part to be moved.
	> Change part type	e.g. from bridge to crown, if the system suggestion is to be changed.
	> Edit part name	Edit part name, if the system suggestion is to be changed.

> Nest parts in blank – Local	Automatically place the part in the ideal position in the blank close to its current location.
> Nest parts in blank – Global	Automatically place the part in the ideal position in the blank.
> Align part with screw channel axis	Align the part suitably in the prefabricated blank.
> Set part sintering pin top plane	Create part sintering pin top plane.
> Set occlusal insertion direction	Set current view direction to the part as occlusal insertion direction (= machining direction), rotate if necessary.
> Open output directory	Open the output directory for the NC files with the calculated toolpaths in the file system.
> Show toolpaths	Show toolpaths after successful calculation.
> Open in hyperDENT® Calculation Merge	After successful calculation, display toolpaths in the hyperDENT® Calculation Merge add-on module.
> Lock part	Lock selected part.
> Save part	Save selected part.
> Export part	Export selected part.
Margin line	
> Edit margin line	
> Change type of margin line	E.g. from coping to inlay/onlay, if the existing setting is to be changed.
> Change undercut property of coping	E.g. from “Cavity without undercuts” to “Cavity with undercuts”, if the existing setting is to be changed.
> Insertion direction from view direction	Set current view direction to the part as occlusal insertion direction (= machining direction), and rotate part if necessary.
Connector	
> Edit connectors	Change settings for connectors.

> Move connector	Highlight selected connector to be moved.
> Apply parameters from connector profile	Use default for connectors.
Sintering pin	
> Edit sintering pins	Change settings for sintering pins.
> Move sintering pin	Highlight selected sintering pin to be moved.
> Apply parameters from sintering pin profile	Use default for sintering pins.

Context menu [Edit]

The menu items displayed depend on the part selected.

- *General*

Delete selected	Delete highlighted part or element.
View	Call up submenu, select display setting.
Rotate view to surface normal from mesh	Rotate the view and set the view direction to the part at the highlighted point vertically in relation to the surface.
Rotate view to insertion direction	Rotate view to the insertion direction of the selected margin line.
Rotate view to opposite insertion direction	Rotate view to the opposite insertion direction of the selected margin line.

- *Blank*

Unload blanks	Unloading blanks from the current project.

- *Part*

Edit template parameters	Change template for part. Only available in template generator module option.
Change part type	e.g. from bridge to crown, if the system suggestion is to be changed.

Move part	Highlight selected part to be moved.
Move part into a different blank	Selected part is removed from the blank and inserted in a different one.
Nest parts in blank – Local	Automatically place the part in the ideal position in the blank close to its current location.
Nest parts in blank – Global	Automatically place the part in the ideal position in the blank.
Align part with screw channel axis	Align the part suitably in the prefabricated blank.
Set part sintering pin top plane	Create part sintering pin top plane.
Set occlusal insertion direction	Set current view direction to the part as occlusal insertion direction (= machining direction), rotate if necessary.
Calculate toolpaths	Calculate construction data with the toolpaths.
Print selected parts	Print parts with current print setting selection.
Lock part	Lock selected part.

- *Margin line*

Edit margin line	
Change type of margin line	E.g. from coping to inlay/onlay, if the existing setting is to be changed.
Change undercut property of coping	E.g. from “Cavity without undercuts” to “Cavity with undercuts”, if the existing setting is to be changed.
Insertion direction from view direction	Set current view direction to the part as occlusal insertion direction (= machining direction), and rotate part if necessary.

- *Connector*

Move connector	Highlight selected connector to be moved.
Edit connectors	Change settings for connectors.

- *Sintering pin*

Move sintering pin	Highlight selected sintering pin to be moved.
Edit sintering pins	Change settings for sintering pins.

Menu [View], submenu, tool bar

Show part messages	Show messages about the parts.
Top view	View according to machining alignment.
Bottom view	View according to alignment for the machining of the opposite side.
Left, right view Front, back view	
Front left, front right view Back left, back right view	
Rotate view to surface normal from mesh	Rotate the view and set the view direction to the part at the highlighted point vertically in relation to the surface, e.g. in the case of attachment primary parts, perfectly align the machining direction of the user-defined areas.
Rotate view to insertion direction	Set the view to the part according to the insertion direction.
Rotate view to opposite insertion direction	Set the view to the part according to the opposite insertion direction.

Zoom all	Center blank or non hidden parts on the workspace.
Draw and zoom to window	Set zoom area and center on the workspace.
Zoom selected	Center selected part on the workspace.
Hide selected	Hide selected part.
Hide all except selected	Show the selected part, and hide the rest.
Show all	Show all parts.
Menu [Settings]	
Machining	Call up submenu and set default.
> Machines	Select milling machine (postprocessor), set options for postprocessor, enter axis boundary, enter values for calculating milling times.
> Fixture	Select holder.
Blanks	Create and manage blank.
> Blank types	Create blank: Name, Material, Geometry.
> Blank administration settings	Set display and function of blank administration.
Building elements	Call up submenu and define defaults for connectors and sintering pins.
> Connectors	Define connectors, set default.
> Screw channel connectors	Define screw channel connectors, and set default.
> Sintering pins	Define sintering pins, set default.
Tools	Call up submenu and create and manage tools and tool holder.

> Tools	Define tools (template generator module option).
> Tool holder	Define tool holder (template generator module option).
Milling strategies	Call up submenu and create milling strategy: Name, Parameters, Allocation (template generator module option).
Parts	Call up submenu and create and manage part types and part information.
> Part types	Manage part types, create and manage user-defined part types.
> Part information	Enter designations for additional part information to be displayed in the part browser.
Importing database objects	Import data from another database: Tools, Tool holder, Material, Blank geometry, Blank type, Milling strategy.
General	Call up submenu, general program settings.
> Miscellaneous	Language, path specifications for directories, messages, information, accuracy.
> Load wizard	Wizard and wizard functions available when loading a part.
> Project management	Settings for project management.
> Part tracking	Settings for part administration.
> Consistency checks	Type and scope of the consistency check.
> Calculation	Output directory, settings for the calculation and behavior in the event of an error.
> Postprocessing	Output directory, file name for the NC file, settings for creating the NC file.
> Navigation	Allocation of the mouse buttons for zoom, rotate and move (pan).

> Display	Anti-aliasing, color allocations for holders, part, margin lines, workspace building elements.
> Connector behavior	Updating and behavior in the event of overlaps.

Menu [Extras]

hyperVIEW®	Call up simulation program (optional).
License Center	Function for license management and activation.
hyperDENT® Calculation Merge	Call up the add-on module, display the NC file after a successful calculation and combine the individual calculations.
Reduce mesh	Application for reducing and optimizing mesh data.
Clean up part administration collection	Clean up part group (optional).
Execute command	Select command from the selection menu and apply to the activated part.

> Save part
> Export part
> Delete part transformations
> Delete part nesting transformations
> Refresh licensing
> Log settings
> Set scaling correction of part
> Saving external geometry

3.4 Part browser

Parts

Display of the parts saved in the project and the loaded blanks, including relevant information.

Context menu

	Delete part	Delete selected part.
	Edit template parameters	Edit/change template for specific part (template generator module option).
	Change part type	e.g. from bridge to crown, if the system suggestion is to be changed.
	Edit part name	Edit part name, if the system suggestion is to be changed.
	Move part into a different blank	Selected part is removed from the blank and inserted in a different one.
	Nest parts in blank – Local	Automatically place the part in the ideal position in the blank close to its current location.
	Nest parts in blank – Global	Automatically place the part in the ideal position in the blank.
	Calculate toolpaths	Calculate construction data with the toolpaths.
	Print selected parts	Print parts with current print setting selection.
	Open calculation output folder	
	Lock part	Lock selected part. A part can be locked as follows: - Manual locking - Locking by the system during the calculation of the toolpaths - Locking after the toolpaths have been calculated, as identification for used blank material, with simultaneous display of the part as “Outline”.

3.5 Part data

Additional part information	Display and entry of additional, customer-specific part information. The information (designation) must first be entered under the menu item [Settings] > [Parts] > [Part information settings].
Height	Height of the selected part.
Path	Path for locating the part file in the file system.
Template	Assigned template.

3.6 Toolbar – Part functions

Delete part — Delete selected part.

Change part type — e.g. from bridge to crown, if the system suggestion is to be changed.

Edit template parameters — Edit/change template for specific part (template generator module option).

Lock part — Lock selected part.
A part can be locked as follows:

- Manual locking
- Locking by the system during the calculation of the toolpaths
- Locking after the toolpaths have been calculated, as identification for used blank material, with simultaneous display of the milled area of the part as “Outline”.

3.7 Information window

Depending on the cursor position, the information window for parts in blank shows the data about the elements to which the cursor is pointing: Part, margin line, screw channel, etc.

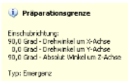
Part

Name/designation	Name of the part

Maximum insertion direction angles

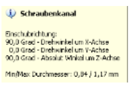
Rotation angle around X, Y-axis Angle absolute to the Z-axis	Maximum deviation of the machining direction on the respective axis.
Occlusal insertion direction Rotation angle around X, Y, Z-axis	Maximum deviation of the occlusal machining direction on the respective axis.

Margin line 



Insertion direction Rotation angle around X, Y-axis Angle absolute to the Z-axis	Maximum deviation of the coping insertion direction from the hyperDENT® coordinate system. The deviation depends on the set insertion direction and on the inclination of the part in blank.
Type Margin line Abutment base Emergence profile Inlay/onlay User-defined area Tooth pocket Alveolar ridge	Type of the margin line.

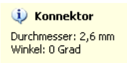
Screw channel



Insertion direction Rotation angle around X, Y-axis Angle absolute to the Z-axis	Deviation of the tilt of the screw channel.

Angled screw channel	Absolute angle from the occlusal side to the main axis
Min./max. diameter	Min. and max. diameter of the screw channel.

Connector



Diameter	Diameter at the part.
Angle 0 degree	Cylindrical

Sintering pin

Diameter	Diameter at the part.
Angle 0 degree	Cylindrical

3.8 General operating instructions

Mouse operation

Click	Click with the left mouse button.
Double-click	2x click with the left mouse button.
Right-click	Click with the right mouse button.
Drag	Drag and drop – select, drag, and drop. Highlight with the left mouse button, hold button down and move to new position with the cursor, release button.
Select part	Click on the part with the left mouse button: The part is highlighted in yellow; the following actions can be performed for the selected part.
Move part	Double-click on the part with the left mouse button: The part is highlighted in orange and can now be rotated or moved using the mouse.

Unlock part	Right-click on the part icon in the part browser: Call up the [Unlock] menu item.
Rotate view	(Dynamic rotation) Press right mouse button and move mouse.
Move view	(Pan) Press Ctrl key and right mouse button or both mouse buttons and move mouse.
Zoom	Move mouse wheel forward or backward.

Select

Select	Highlight selection: Click with the left mouse button on the part, text, display, entry.
Multiple selection	Press Ctrl key and highlight selection: Hold Ctrl key down and click with the left mouse button on the parts, texts, entries.
Multiple selection in part browser	Press Ctrl key and highlight selection: Hold Ctrl key down, press left mouse button and drag the displayed selection frame over the parts to be highlighted in the part browser.
Selection menu	Menu bar, dropdown menu: Open menu: Click on the menu bar. Select menu item: Click on the menu item.
Context menu	Open menu: Right-click on the workspace or part display. Select menu item: Click on the menu item.
Selection list	Dropdown list, listbox: Open menu: Click on the arrow symbol. Select entry: Click on the list entry.
Selection box	Mark selection, remove marker: Click on the selection. The selection is marked with a checkmark or the checkmark is removed.
Option field	Highlight selection: Click on the selection. Remove selection: Click on a different selection. The selection is marked with a dot.
Tab	Tab: Click on the tab.

Displays

The displays are dependent on different conditions:

General settings that are configured
Menu [Settings], menu item [General]

Blank administration settings that are configured
Menu [Settings], menu item [Blanks] > [Blank administration settings…]

selected object

current operational step

3.9 Space mouse

Alternatively, you can also control hyperDENT® with a space mouse.

Configuration of key assignment:

Key	Command	
Fill	Zoom all	Zoom All
T1	Draw and zoom to window	Top View + Zoom All
T2	Top view	Top View
T3	Bottom view	Bottom View
T4	Left view	Left
T5	Right view	Right
T6	Front view	Front
T7	Back view	Back
T8	Front right view	Right Front
T9	Front left view	Left Front
T10	Back right view	Right Back
T11	Back left view	Left Back

4 Quick guide

4.1 Starting the program

1. Double-click on the program icon <hyperDENT...>:

Figure 4-1 – Toolbar

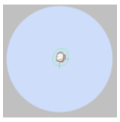
Once the program is running, the toolbar displays the icons for project management and the process steps that are required for machining.

We recommend that you work through the process steps using the icons from left to right.

- *Missing process steps are highlighted by a red cross, whilst completed process steps are identified by a green checkmark.*
- *You can also open other hyperDENT® instances and edit several projects at the same time (Classic version).*

4.2 The process steps in sequence

1. Either create a new project and save.
   Or open an existing project.
   Or start with the next step to construct an individual part and discard the details or save at a later time in a project.
2. Load part.
   Select and load the STL file with the part data for the workpiece. The file can be in any of the directories, the part selection shows a preview of the workpiece.



The loaded part is placed in the blank according to the data that is loaded.

For the subsequent process steps, a specific alignment of the parts to the tool axis is required.

3. Load blank.
   Select the blank from the part administration and load into the project. All parts are placed on the blank.

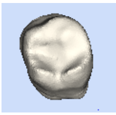
4. Select milling unit.
   Set the milling machine and the associated holder for the machining process. The details are incorporated in the program calculation.

The other icons are active only if a part is selected.

5. Set milling direction, adjust direction.

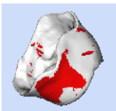
   Set the general alignment (= machining direction) of the part:

   Rotate part onto the side on which the undercuts are to be minimized. Then define this side as occlusal or cavity.



   Click on [Occlusal] if the occlusal side or chewing surface of the part is visible.

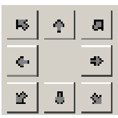
   Click on [Cavity] if the inner side of the coping is visible.



   Then adjust the direction if necessary:

   Click on [Update].
   Established undercuts are identified by a checkmark in the selection window and appear in red on the part.

   If necessary, make fine alignment adjustments to the part:

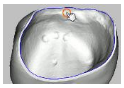


   Click on the [Arrow keys] in the selection window.
   The part is gradually rotated.

   Adjust direction again.

Repeat the undercut check and fine alignment, especially for 3-axis machining, until there are either no more undercuts in the coping or they are very small and are in a non-critical area.

6. Identify part features
   Highlight the margin line, other part characteristics and, depending on the part type, the pontics, the screw channel, or the interface geometry.

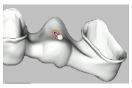


Select the [Margin line] tab and select Type.

Copings, inlay
Click on the part, or with copings click on the cavity and/or within the margin line:
The crown edge – margin line – is determined and highlighted in blue.

Abutment base, emergence profile
Click on the relevant boundary line:
The boundary line is identified and highlighted in color.

The type of margin line must be compatible with the part type, otherwise the calculation is not possible for safety reasons.



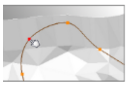
Select the [Pontics] tab and click on the center of each pontic or supporter:
The pontic or supporter is identified by a blue dot.

Select the [Implant interface] tab and click on the edge of the screw channel.
Define interface geometry or load model for interface geometry.

If necessary, determine the coping-specific alignment:
Close the selection window.
Align the coping so that there are no undercuts and then in the context menu, select menu item [Insertion direction from view direction].
The insertion direction is set and highlighted.

Set template correctly.

If necessary, set the undercut machining:
Highlight the margin line and, in the context menu, select the menu item [Change undercut property of coping] > [Coping has undercuts].



If necessary, determine the user-defined machining area:
Select the [Margin line] tab, select the [User-defined area] type. Call up the [Draw contour] function and draw the user-defined area. Further details --> "Identify part features" > "User-defined area".

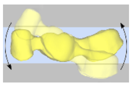
Set template correctly.
Set details correctly under "Tilt part in blank".

7. Tilt part in blank:
   Set the tilt angle of the part in the blank.

Click on [Center]:
If the part is at the top or bottom in the blank.

3+1 optimization:
Align part optimally to the rotation axis of the machine.

Rotate 180 degrees around Z:
Rotate part optimally by 180 degrees to the rotation axis of the machine.



Tilt part.

Minimize height.
If the part is askew in the blank and is therefore too high.

Minimize inclination (angle optimization)
If the part is at quite an angle and if this produces large undercuts with unfavorable tilt angles.

Enter degree value for maximum tilt angle.

Select axis.

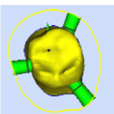
Click on [Tilt].
The part, together with the tool axis, is tilted.

When you enter a tilt angle, the part is tilted within the blank, the position to the tool axis remains unchanged. This requires clamping for the machining: 3+2-axes or better 5-axes.

8. Select milling strategy.
   Select and/or change the template for the part.

9. Set connectors.

[Automatic] mode, use default or select Type, Size, and Count.



Click on the part:
The connectors are set automatically.

If necessary, select [Manual] mode and set the additional connector manually.

If required, close the menu, highlight the pin and move or delete it.

Entering the connectors is the last required process step. The completion of all process steps is indicated in the part browser by a checkmark next to the part icon.

10. Set sintering pins (optional, only if required).

    Select Angle, Size, and Parameter profile.

    Click on the place on the part where the pin is to be set:
    The pin is set at the selected place.

    If required, close the menu, highlight the pin and move or delete it.

11. Save project and save the process setting: Optional, if you want to keep the data.
    Click on the icon and the project is saved.

12. Calculate toolpaths.
    Click on the icon:
    The construction data and toolpaths are calculated. The progress is shown in the log.

13. Save project: Optional, if you want to keep the data and continue to use the blank.
    Click on the icon and the project is saved.

14. Print project:
    Click on the icon:
    The project data and the current settings are printed, e.g. for documentation.

15. Create NC program.
    The NC program (postprocessing) can be created in different modes.

- Depending on the setting, the file is loaded either after the prompt or automatically, and postprocessing takes place in the background.
- The program for the simulation (hyperVIEW®) starts up automatically once the calculation has finished, depending on the mode selected in the general settings.

---

16. Simulation (optional).
    The simulation shows the representation of the toolpaths in accordance with the axis movements of the machining device. The simulation takes place based on the calculated NC file.

5 Project management

The project management gives you an overview of the projects and blanks that have been saved and the parts that have been placed and machined. Each project file also corresponds to a blank and can be loaded as a project via the project management or as a blank via the blank administration.

hyperDENT® saves all details of a project in project files on the hard drive. You can determine the directories under [Settings] > [General] > [Project management].

The project files contain all information about the parts to be machined. Make sure you back up the project files regularly, preferably onto an external data carrier.

The project management functions can be accessed via the [File] menu or toolbar.

New project	Open a new, empty project.
Open project	With the project management, open the selection window to select and open a project that has already been saved.
Save project	Save the project that is open. Projects that are saved using the current version of hyperDENT® can no longer be read by older versions of hyperDENT®.
Save copy of project	Save the project that is open under a different name or in a different directory.
Print page setup	Align page, set size, orientation, and margins.
Project print setting selection	Select project data for printing.
Print project preview	Print preview.
Print project	Print project data.

Exit	Exit hyperDENT®: close the project that is open, close the program.

- *A project can only be opened by one user. An open project is blocked to other users.*
- *A user can always only open one project.*
- *If another project is opened, the current project is closed. Classic version: You can start hyperDENT® several times in order to process several projects at the same time.*
- *If a project is closed that has already been changed, then a confirmation prompt appears:*

Save	Save the changes, close the project.
No	Discard the changes, do not save, close the project.
Cancel	Cancel the process, do not save the changes, do not close the project.

You can save hyperDENT® projects in any directory in the file system.

hyperDENT® projects are given the following file name extensions:

".hdproj"	*Project file and relative path to STL file of the part.*
".hdprojz"	*Project file and saved STL file.*

5.1 New project

The icon is active as soon as the program starts up.

Use this menu item to create a new project, clearing any selected machines, materials, and parts.)

5.2 Opening project

To construct additional parts from a blank, you must load the project and blank again. Then you can add additional parts and machine them.

The icon is active as soon as the program starts up.

Load a saved project for further machining from the project list of the project management or from the file system.

Once the project is loaded, the blank is displayed on the workspace, and details of the blank appear in the project information.

The selection window shows the project management with the list of projects, blanks, preview, and blank data for the selected project, along with the filter for selecting the blank.

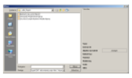
The display is dependent on the menu [Settings] > [Blanks] > [Blank administration...]. Further details --> “Settings” > “Blanks” > “Blank administration settings”.

Preview	Top view of the blank from the selected project for a quick assessment of the available space left: Gray display of parts not calculated yet, display of milling boundaries for parts that have been calculated.

To preview the blank, the project must be opened and saved using the current version of hyperDENT®.
It can then however no longer be opened using the previous version of hyperDENT®.

Name	Displays the name of the project/blank.
External ID	Displays the external number of the blank for administration purposes, e.g. storage place.
Charge number	Displays the external number of the blank for administration purposes, e.g. same production run.
Blank type	Displays the type of blank.
Material	Displays the material of the blank.
Color	Displays the color of the blank.
Height	Displays the height (thickness) of the blank. The height must be sufficient so that the dental restoration does not overlap the blank.
Fixture	Displays the fixture used.
Machine	Displays the machine used.
Last saved	Displays the date on which the project was last saved.
Path	Path name in the file system: drive, directory, file name.

Parts in blank	Displays the name of the parts that are placed on the blank and that are saved in the project.
Scaling	Scaling factor for X, Y, and Z-axis or uniform scaling for XYZ, material-specific. Depends on the details in the menu [Settings] > [Blanks] > [Blank types] > [Materials].
Filtering	Selection filter for the project by blank data: to quickly find suitable blanks. The fields displayed are dependent on the settings in the blank administration.
Show empty blanks	Show/hide blanks that have been created but not used.
Clear filter	Clear filter setting, show all.
File	Selection window for opening projects from the file system.
Open	Open selected project.
Cancel	Cancel process, do not open the project.



5.2.1 Opening project

Either

1. Click on the project in the project list or file system:
   The project is selected; preview and data are shown.
2. Click on [Open]:
   The project is loaded and is displayed on the workspace.

Or

1. Double-click on the project in the project list or file system:
   The project is loaded and is displayed on the workspace.

5.2.2 Adjusting table

Order of columns

Change

1. Click on the column heading, hold the mouse button down and drag column to new position.

2. Release mouse button:
   The column is inserted in the new position.

Reset

1. Call up context menu and select menu item [Reset order of columns to default]:
   Right-click on the column heading and click on the menu item.
   The column width is reset to the default setting.

Column width

Change

1. Click on the boundary line between the column headings, hold the mouse button down and set the column width.
2. Release mouse button:
   The column is inserted in the new position.

Or

1. Double-click on the boundary line between the column headings:
   The column width is set to the predefined width or the maximum width of the content or the heading.

Reset

1. Call up context menu and select menu item [Reset width of columns to default]:
   Right-click on the column heading and click on the menu item.
   The column width is reset to the default setting.

Sorting

1. Click on the column heading:
   The table is sorted by column contents in ascending or descending order – according to the arrow icon for the sort order.

Showing/hiding columns – Blank administration settings

The columns displayed in the table and the options for filtering the blanks can be set under [Settings] > [Blanks] > [Blank administration...]. Here you can show and hide the columns and filters for the display. Further details --> “Settings” > “Blanks” > “Blank administration settings”.

5.3 Already constructed parts

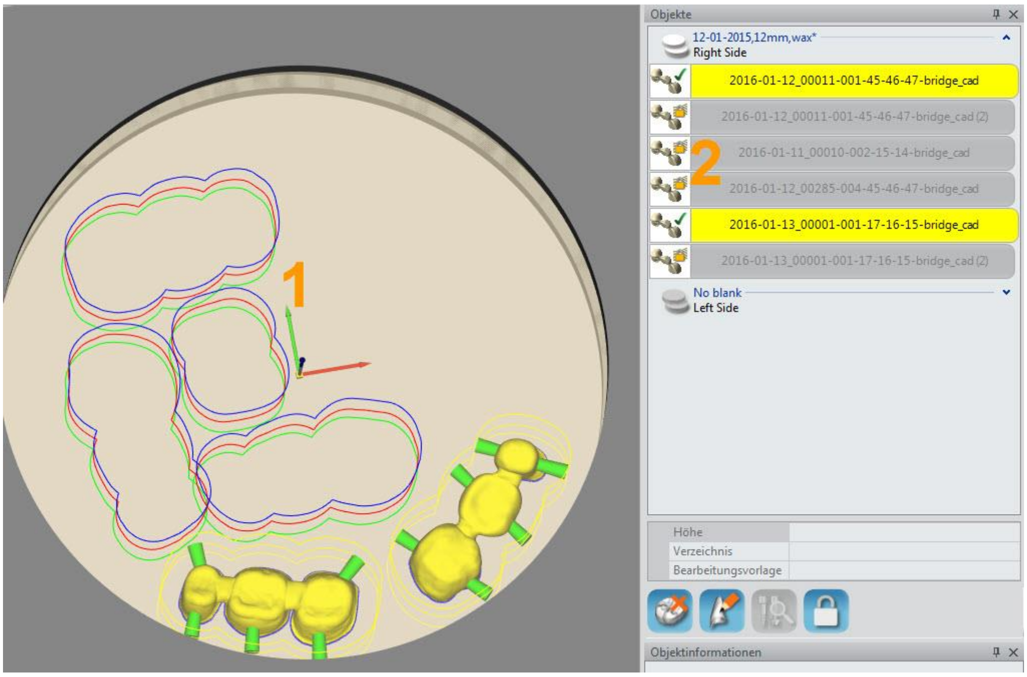


Figure 5-1

Already constructed parts are shown as “Outline” (1). “Constructed” is defined by the software as all parts for which a calculation has already been fully completed and for which you can therefore assume that they have been constructed and that the raw material has been used here.

Already constructed parts are also identified as locked and have a padlock icon (2) in the part browser. Right-click on the part to unlock it and reload it.

Saving the used material in the project means:

- *you always have an overview of space that is still available on the blank;*
- *you can move new parts in such a way that they are securely placed in the full material.*

5.4 Printing project

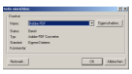
5.4.1 Setting up page, and printing project

Print page setup, Page layout

The selection window shows the options to format the print page: Size, Alignment, Margins.

Paper	Selection menus for setting the “size” and “source” of paper to be printed.
Size	Selection menu for the paper format. Take into account the formats of the printer to be used.
Source	Selection menu for the paper feed. The selection depends on the printer selected.
Orientation	Option field for setting the orientation of the print page as “Portrait” or “Landscape”.
Margins	Entry fields for margin width. The minimum width depends on the printable area of the printer to be used.
OK	Accept setting, close window.
Cancel	Cancel process, keep old settings, close window.

Printer

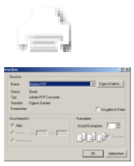


The selection window shows details about the selected printer, the selection menu for the printer, and the printer properties.

Name	Selection menu for the printer.
Properties	Call up printer properties.
Network	Select network printer.
OK	Accept setting, close window.
Cancel	Cancel process, keep old settings, close window.

Print project preview, Page view

The window displays the print preview with the selected data and the following icons: Print, Zoom, Page layout, Close, Page number.

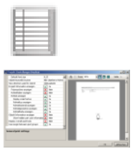


Printing project

The selection window displays details about the selected printer, the selection menu for the printer, and the printer properties, along with the input option for the pages to be printed (print range) and the number of printouts.

OK	Accept setting, start printout.

5.4.2 Project print setting selection



The selection window shows the selection for the data to be printed, the print preview and the icons: Print, Page setup, Zoom, Page display, Page number, Page navigation.

Font size	Default font size
Print part selection	Set print range.
Select all parts in blank	Print blank and part data.
Select all parts in blank with toolpaths	Print blank with part data and calculated toolpaths.
Select all parts in project with toolpaths	Print project with parts and calculated toolpaths.
View direction used for sketch	View direction to the blank for the printout:
Occlusal side	Display of occlusal side.
Cavity side	Display of cavity side.
Display mode	In 1:1 scale on the printout
No image	
Sketch	
Real image/screenshot	
Display project information	Enable/disable printout of project information.
Display machine	Select project information for the printout.

Display fixture	Select project information for the printout.
Display blanks	Select project information for the printout: Scaling factor, blank type, blank material, blank geometry, blank color.
Part information	Enable/disable printout of part information.
Display part type	Select project information for the printout.
Display part file name	Select project information for the printout.
Display part path	Select project information for the printout.
Display part template	Select project information for the printout.
Display part toolpaths exist	Select project information for the printout.
Show part user information	Select project information for the printout.
Show hidden user information	Select project information for the printout.
Display used tools for part	Select project information for the printout.
Display overall used tools	Enable/disable printout of tool information.
Use margin between part groups	Enable/disable insertion of empty row.

5.4.3 Automatically printing project after calculation

The data defined under "Print project setting selection" is sent automatically as a printout to the default printer, as soon as the calculation is completed and the NC file is created. As a result, you receive, for instance, the setup plan with the blank, part and tool details matching the NC file.

- *The settings for the data to be printed can be made in the [Print project setting selection] submenu of the [File] menu. Further details --> "Print project" > "Print project settings".*

- *The settings of the printer are made in the [File] menu, and the [Print project] submenu. Further details --> "Print project" > "Page setup, Print project".*
- *The automatic printout can be enabled and disabled in the [Settings] menu under the menu item [General settings] > [Postprocessing] > [Print project automatically after calculation].*

6 Loading part

The part represents the dental restoration. The part data is saved in an associated part file.

The icon is active as soon as the program starts up.

You can load the parts from any directory.

Several part files can be simultaneously selected and loaded; the preview is then inactive and placement takes place outside of the blank.

Once you have selected a part file, the part preview is shown. You can then determine the maximum dimensions, along with the type and construction for the dental restoration.

Type and construction control the automatic defaults for the following process steps.

If the CAD information is loaded via a defined interface, then many process parameters have already been determined. To do so, select the correct file type.

Once the part is loaded, it is added to the part browser and displayed in the center of the blank.

For optimum use of the blank, move the part to a suitable place in the blank manually or via the automatic placement.

Part tracing gives you a quick overview of the parts that have already been loaded into a project or have already been calculated.

6.1 Loading part

The selection window shows the currently selected directory with the list of files along with a window for the part preview.

Search in	Select directory and drive.
File name	File name of the file.
File type	File types for selection. It is important to have the correct file type when working with a defined interface.

.stl	Model files, standard setting for part data Wieland, DentalWings, Exocad, Zfx, 3Shape DentalDesigner Parts (with .3SFM-CAM output), 3OX.
.hdpart	hyperDENT® parts.
.hdpartz	hyperDENT® parts, hyperDENT® parts model, e.g. for stored interface geometry.
auftrag.ini	KaVo parts, KaVo original process.
scene.xml	Geomagic parts.
Open	Open file, load part.
Cancel	Cancel process, do not load part.
Preview	Enable/disable preview.
Status	Shows status of the selected part.
New file *Nested file* *Machined file*	
Display nested in blanks	Name of blank on which the part is already nested.
Measure	Select axis (X, Y, Z) for the measurement display.
Size	Max. dimension for the selected axis.
In blank	Max. blank dimension for the selected axis, if a blank with scaling factor is already loaded.
Type	Selection for the type of dental restoration. Automatic transfer via the defined interface.
Coping *Coping bridge* *Crown* *Crown bridge, Maryland bridge* *Abutment* *Abutment crown* *Abutment bridge*	

Abutment crown bridge

Inlay/onlay

Inlay/onlay bridge

Anatomical inlay/onlay

Inlay/onlay crown bridge

Telescope

Overpress

Bitesplint

Model

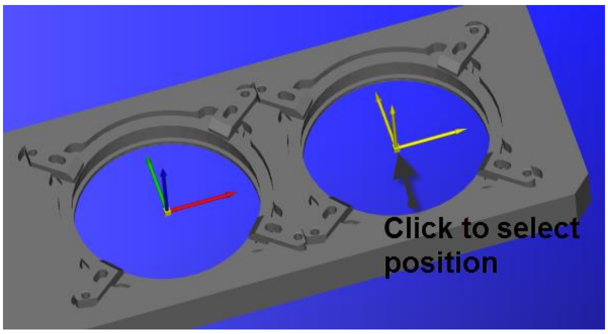
Model stump

User-defined part

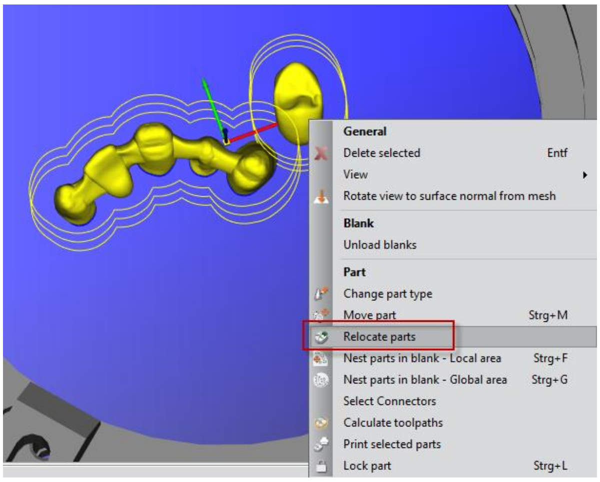
Full lower dentures

Full upper dentures

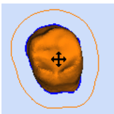
	Material	Selection field for the raw material.
	Position	Selection of the position of the blank in multi-fixtures. Possible positions are shown graphically in the dropdown menu for selection. As an alternative to selecting the position in the holder, it is also possible to select the desired position by clicking on the corresponding coordinate system before opening the “Load part” dialog.



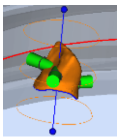
If you need to modify the position subsequently, you can do so via the context menu [Relocate parts]. If automatic nesting is used, the part(s) are nested in the blank again according to the respective holder specifications.



6.2 Placing part



1. Double-click on the part with the left mouse button:
   The part is highlighted in orange and can now be rotated or moved using the mouse.
   In this way, you can place the entire parts to be machined in the blank. In the case of a prefabricated blank, the part must be placed with great accuracy: --> “Load part” > “Place part precisely in prefabricated blank – align with screw channel”.



   The axis line of the part is additionally shown in the lateral view.
   The part can be placed in terms of height on the axis line.
   The part can be tilted by pressing the Ctrl key.
   In the main views, the part tilts around the screen view axis.

Fixture boundary

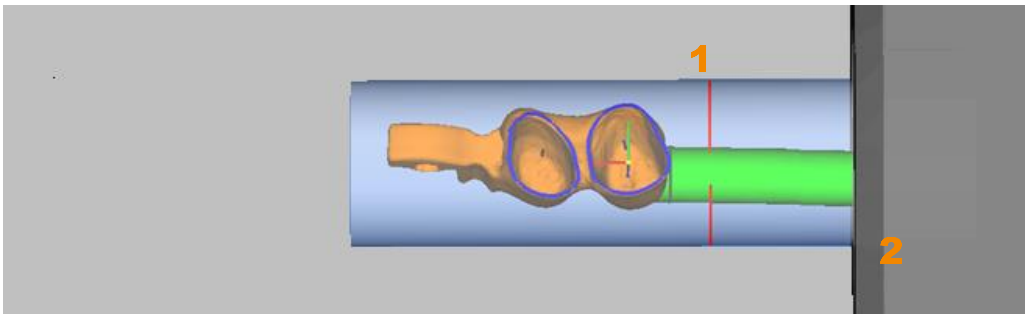
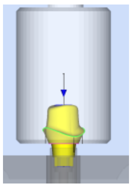


Figure 6-1

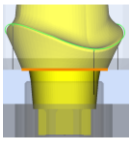
If the “Fixture” boundary is selected, the boundary line (1) (the prescribed distance to the holder (2)) is displayed when placing the parts in moving mode. --> “General job parameters” > “Bounding strategy”.

6.3 Placing part precisely in prefabricated blank



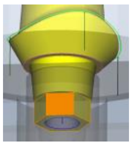
In the case of prefabricated blanks (prefabs), the screw channel, the screw fit, and the interface geometry are ready immediately. To ensure that the structure to be created will fit here perfectly, the part must be placed with great precision and rotated into the correct position.

- *Alignment of the screw channel axis*
  Align the screw channel axes of the part and the blank, i.e. so that the axes are positioned directly above each other and point in the same direction.



- *Height of the part – Moving*
  The reference surfaces are positioned directly above each other in the direction of the screw channel axis, e.g. the prefabricated interface surface on the blank and the corresponding interface surface on the part.
- *Orientation of the part – Rotation*

  The part is placed at a specific angle to the interface geometry.



  The interface geometries of the part and the blank are flush above each other, and so are precisely aligned with each other in their angled position around the screw channel axis, e.g. the prefabricated hexagon on the blank and the hexagon on the part.

As a rule, the part data is transferred from the CAD system to match the blank data. A manual adjustment is possible, if necessary: --> “Align part with screw channel”, “Move part in relation to screw channel” and “Rotate part in relation to screw channel”.

6.3.1 Aligning part with screw channel

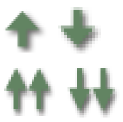
1. Select the part, call up the [Edit] menu, and select the following menu item: [Part] > [Align part with screw channel].

The window shows the available buttons for precisely aligning and rotating the part.

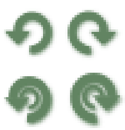
Move up/down arrow buttons	Move the part along the screw channel axis with an increment (offset) or multiple increments.
Rotate left/right arrow buttons	Move the part along the screw channel axis with an increment (angle) or multiple increments.
Undo	Cancel the operation, and undo the movement step by step.
Offset increment	0.01 to 10 mm Increment for moving along the screw channel axis
Angle increment	0.01 to 10° Degree of rotation around the screw channel axis.
Move part in relation to the reference surface	Move part along the screw channel axis until the reference surfaces coincide.
Rotate part in relation to highlighted point	Rotate part around the screw channel axis until the highlighted points coincide.
Rotate part in relation to the reference surface	Rotate part around the screw channel axis until the highlighted reference surfaces coincide.
Close	Cancel process, and close the window.

6.3.2 Moving and rotating manually

1. Select the part, call up the [Edit] menu, and select the following menu item: [Part] > [Align part with screw channel].
2. Enter the offset increment for the movement.

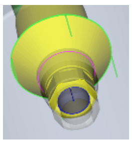


3. Press the arrow buttons and move the part along the screw channel into the required position.
4. Enter the angle increments for rotation.



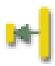
5. Press the arrow buttons and rotate the part around the screw channel axis into the correct position.

6.3.3 Moving part semiautomatically in relation to the reference surface

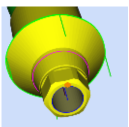


Move the part along the screw channel axis until the selected reference surfaces coincide.

1. Select the part, call up the [Edit] menu, and select the following menu item: [Part] > [Align part with screw channel].

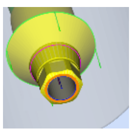


2. Call up the function [Move part in relation to reference surface].
   Click on the function key.
   The holder and the blank are hidden, and the part is displayed to let you select the reference surface.



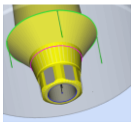
3. Select the reference surface on the part:
   Click on the reference surface on the part that coincides with the corresponding reference surface on the blank.
   The part is hidden and the blank is displayed.

4. Select the reference surface on the blank:
   Click on the reference surface on the blank that coincides with the corresponding reference surface on the part.



5. The part is moved automatically along the screw channel axis until the selected reference surfaces coincide.
   The new position is displayed on the screen.

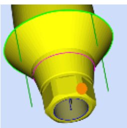
6.3.4 Rotating semiautomatically in relation to highlighted point



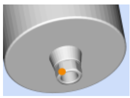
Rotate the part around the screw channel axis and align it in any desired position in relation to the prefabricated interface geometry.

1. Select the part, call up the [Edit] menu, and select the following menu item: [Part] > [Align part with screw channel].

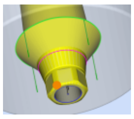
2. Call up the [Rotate part in relation to highlighted point] function.



3. Mark a point on the part:
   Click on a point on the part that you wish to coincide with the corresponding reference surface on the blank.

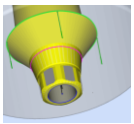


4. Mark a point on the blank:
   Click on a point on the blank that you wish to coincide with the corresponding reference surface on the part.



5. The part is rotated automatically around the screw channel axis until the marked points coincide.
   The new position is displayed on the screen. The angle information is displayed in the status bar.

6.3.5 Rotating part semiautomatically in relation to the reference surface

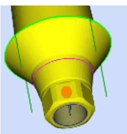


Interface geometries with an anti-rotation lock require the precise orientation of the part, so that the part is correctly aligned with the prefabricated interface surfaces of the blank.

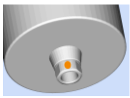
Rotate the part around the screw channel axis and align it with the prefabricated interface geometry so that, for example, the surfaces of the hexagon in the part coincide with the prefabricated surfaces on the blank.

1. Select the part, call up the [Edit] menu, and select the following menu item: [Part] > [Align part with screw channel].

2. Call up the function [Rotate part in relation to reference surface].

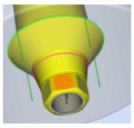


3. Mark a reference surface on the part:
   Click on a reference surface on the part that you wish to coincide with the corresponding reference surface on the blank.



4. Mark a reference surface on the blank:
   Click on a reference surface on the blank that you wish to coincide with the corresponding reference surface on the part.

If the corresponding reference surfaces on the blank are not shown in this view, it is sufficient to click in the vicinity of the assumed reference surface, e.g. a click in the same screen position as before when making a selection on the part.



5. The part is rotated automatically around the screw channel axis until the marked reference surfaces coincide.
   The new position is displayed on the screen. The angle information is displayed in the status bar.

6.4 Placing part automatically – autonesting

During loading, hyperDENT® can place the part (1) in the blank automatically, making the ideal use of the available space.

The settings are made for each holder (fixture): [Settings] menu > [Machining] submenu > [Fixtures] tab > [Additional properties].

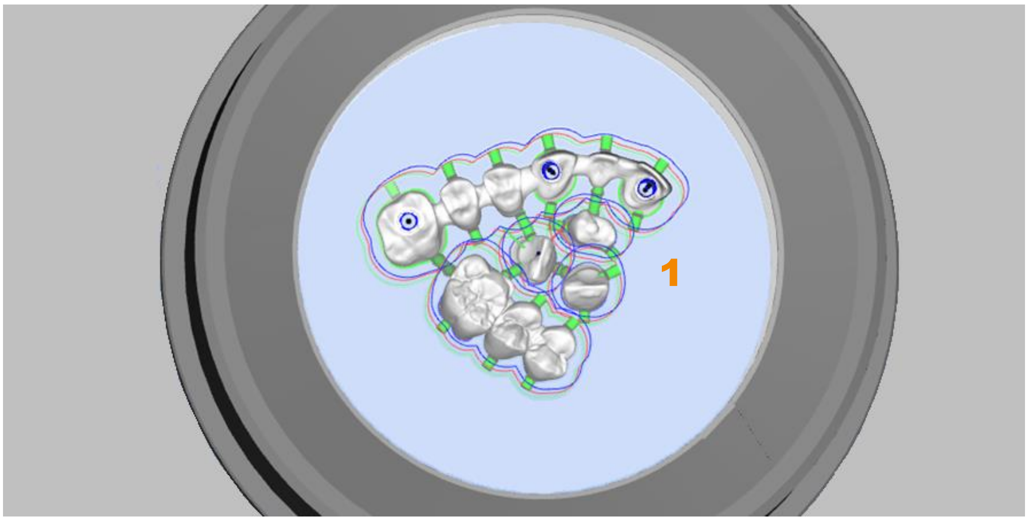


Figure 6-2

6.4.1 Setting automatic placement

1. Call up machining settings:
   Select [Settings] menu and call up the [Machining] menu item.
2. Call up the [Fixtures] tab.
3. Select the fixture (holder):
   Click on the row with the desired holder.
4. Call up the [Additional properties] function:
   Click on the button.

Switching on

5. Switch on [Object nesting].

   Set Use automatic nesting to “Yes”.
6. If necessary, choose the nesting direction:
   Various possibilities are available to choose from, depending on the holder type

   a. From inside to outside
   b. From outside to inside
   c. From left to right
   d. From right to left
   e. From top to down
   f. From down to top

   In addition, you can choose the “Position of 1st part”
7. Accept setting, and close window.
   The automatic placement is then enabled for this holder.

Switching off

8. Set Use automatic nesting to “No”.
9. Accept setting, and close window.
   The automatic placement is then disabled for this holder.

6.4.2 Placing part automatically during loading (nesting)

The automatic placement function must be enabled for the selected holder. The part is then placed in the blank automatically during loading, making the ideal use of the available space.

You can move the part manually at a later time.

6.4.3 Placing part automatically at a later time (nesting)

If automatic placement is enabled for the selected holder, you can automatically place a part that was already loaded (1) in the blank, making the ideal use of the available space, via the menu or the context menu.

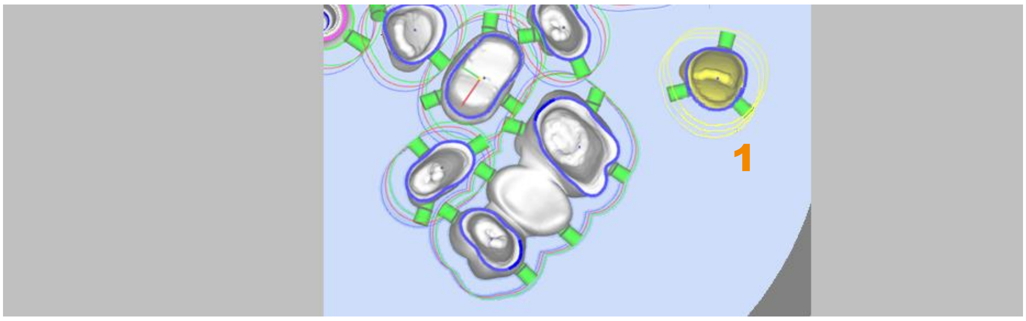


Figure 6-3

1. Selecting the part (1).
2. Call up the [Machining] menu, and select the menu item [Part] or call up the context menu.

Either

3. Select the menu item [Nest parts in blank – Local]:
   The part is automatically placed in the ideal position in the blank close to its current location (2).

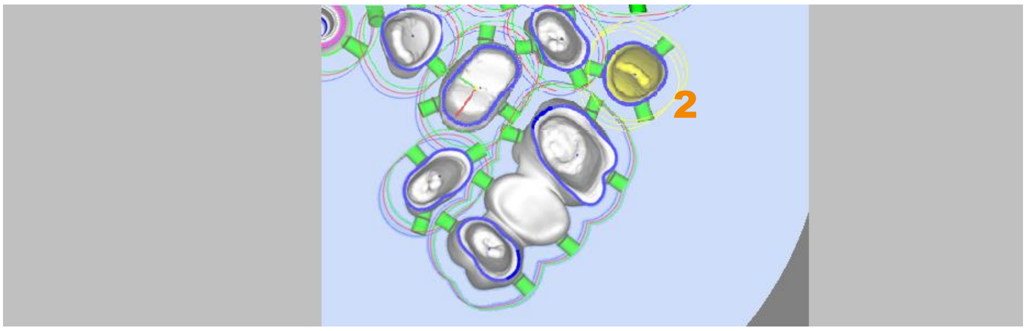


Figure 6-4

Or

4. Select the menu item [Nest parts in blank – Global]:
   The part is placed in the ideal position in the blank (3) according to the specifications in the holder settings.

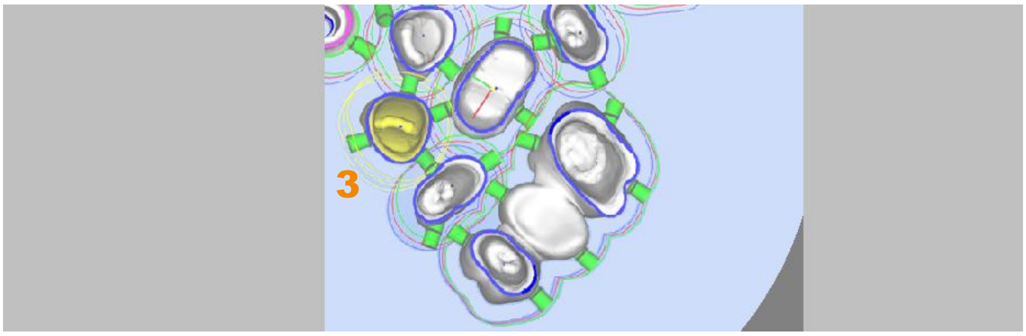


Figure 6-5

6.5 Consistency check

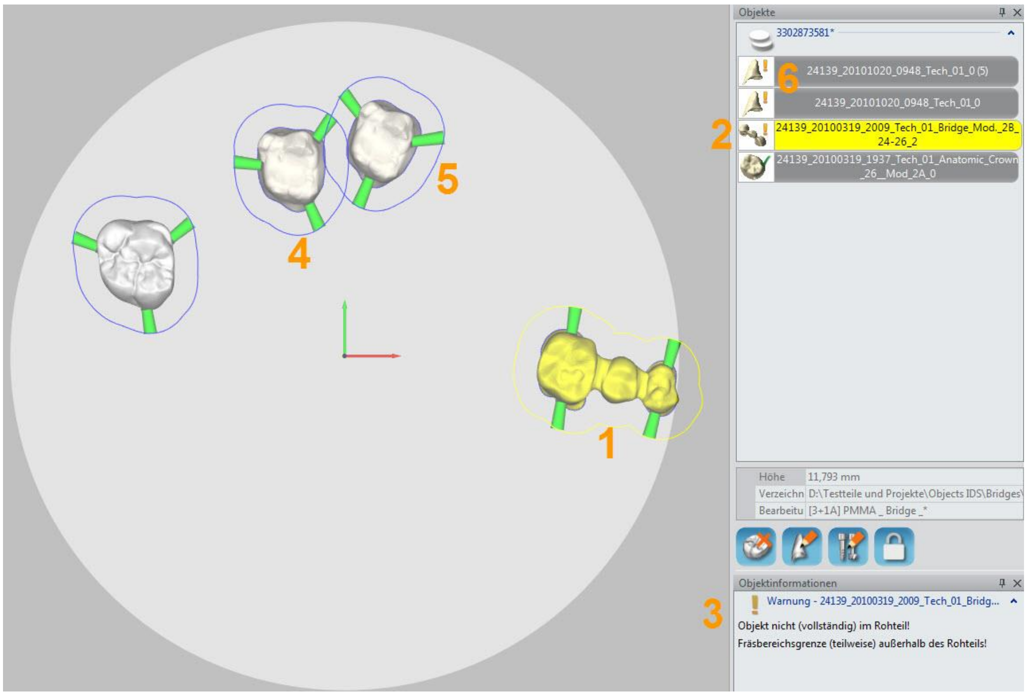


Figure 6-6

If a part is incorrectly placed in the blank, you get a warning message or a stop message, depending on the default under the menu item [Settings] > [General] > [Consistency checks].

The part (1) is outside of the blank, this is shown in the part browser (2) and highlighted by the plain text message (3).

Parts (4) and (5) overlap, this is also shown in the part browser (6).

6.6 Defined interface – Using the wizards

Use the [Load wizard] function to enter the process parameters much quicker since the dialog boxes for the next process steps are automatically opened in the sequence of the process chain.

1. Call up General settings:
   Select [Settings] menu and call up the [General] menu item.
2. Call up the [Load wizard] area.
3. Change the setting for the function [Use load wizard].
   Click on the row and select the setting in the selection menu.
4. Select the individual steps that the wizard is to perform.

Yes	Switch function on.
No	Switch function off.
Force	Force may be necessary if, when the part is loaded via a defined interface, the desired part information (e.g. the pontic position) has not been supplied as well.

Use load wizard	
Yes	Switch wizard on: The toolbar functions are automatically called up or skipped once the previous function has been confirmed.
Adjust direction	
No	If the tool alignment (insertion direction) is transferred.
Identify parts	
No	If the margin line is transferred.
Tilt part	
No	If the tool alignment (insertion direction) is transferred.

Select template		
	No	If the template is automatically selected correctly via the defaults and no changes are needed.
Nesting mode		
	Yes	The part is activated in moving mode directly after insertion and can be placed immediately.
Set connectors		
	Yes	If material information is transferred and the connectors are automatically set correctly via the default settings.

5. Save setting by clicking on [OK], exit window.

6.7 Direct interface – Working with original CAD information

The direct interface to CAD programs simplifies the import of CAD information for processing in hyperDENT®.

3Shape Dental Designer

raw stl/3OX interface
Use of the general 3Shape CAD output as a direct interface.
The 3sfm output is no longer necessary.

Settings for the 3Shape output

ID	Configuration_01_manuProcess30
Name	Milling R0.4mm
Manufacturing process method	CADOutputRawSTL.dll
Output margin line	Yes
Output outer margin line	No
Output 3OX file	Yes
Output INF file	No
Output abutment base curve	Yes
Compress files	No

Start Implant Direction Position file name with OrderID	No
Milling	Yes

6.8 Direct interface – Connection to the CAD system

The direct connection (CAD Connect) to CAD programs simplifies the import of CAD information for processing in hyperDENT®:

- *The CAD information is transferred automatically to hyperDENT®.*
- *The new parts appear in the list of parts still due for machining and no longer need to be loaded via the file system.*
- *Optionally, hyperDENT® can be started with the transferred parts from the CAD system.*

Available for CAD system

- *Exocad (not yet available)*
- *3Shape*

Activating function

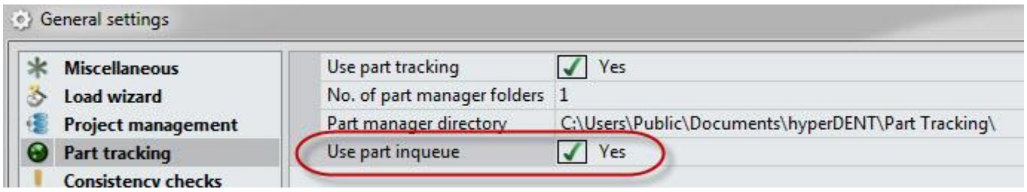
The aim of this function is to allow the transfer of restorations that are modeled and managed with a dental CAD system directly to *hyper*DENT.

Following a successful configuration, this application allows you to open designed parts directly from the user interface of the CAD and

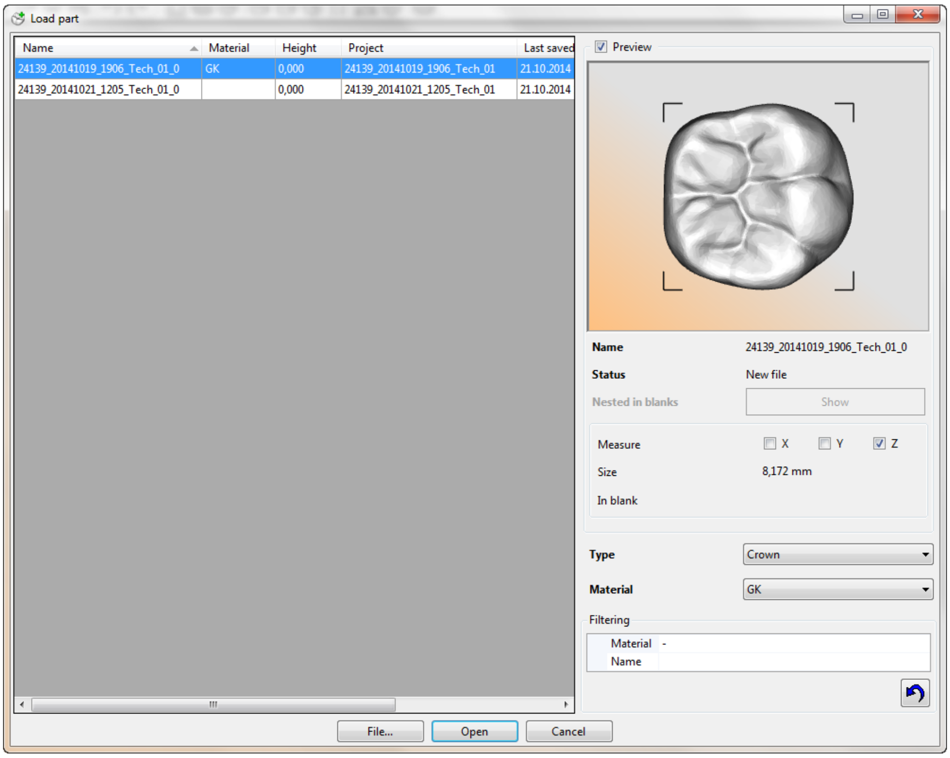
- To transfer the information about the parts to be produced to the *hyper*DENT parts management and
- Optionally, to open *hyper*DENT with the corresponding parts.

Caution: In order to view the parts to be produced in the form of a list for part selection, it is necessary to activate the corresponding function in hD:

You can activate the function via the [Settings] menu, submenu [General settings], menu item [Part tracking] under the parameter [Use part in queue]: --> "Settings" > "General" > "Part tracking" > "Use part inqueue".



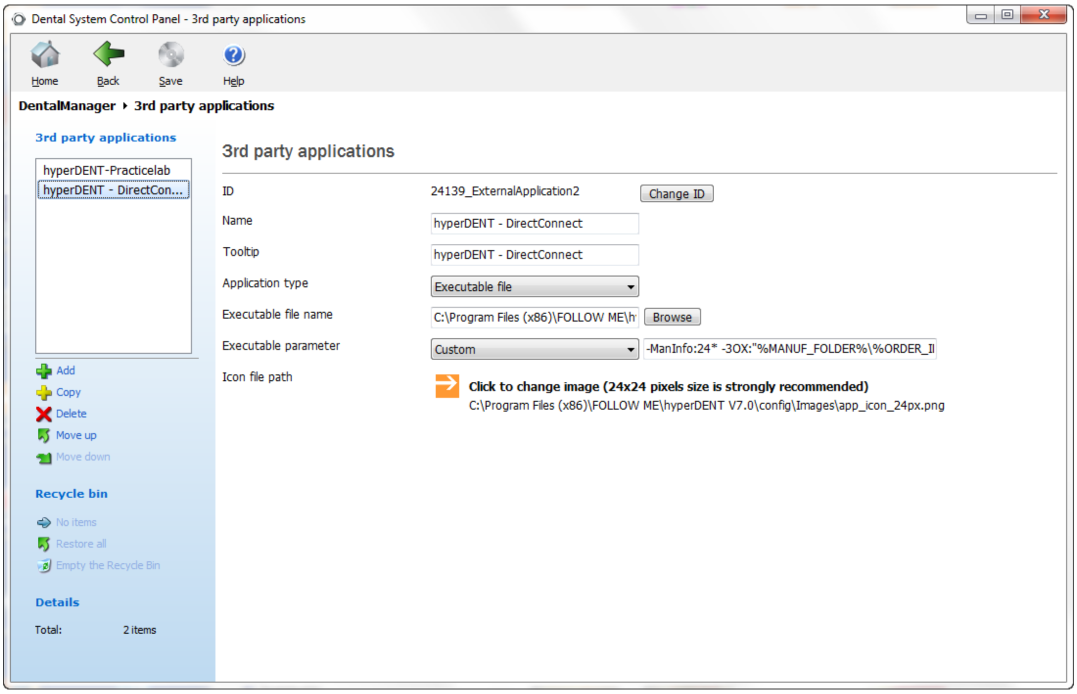
Effect in hD on the “LoadPart” dialog:



The list dialog above appears instead of the standard file selection dialog.

The new connection was developed to let you populate this list.

Settings in the 3Shape Dental System:
Configuration in the Dental System Control Panel -> 3rd party applications:



Please select the file “3sconnect.exe” from the win directory of the installation as an executable.

As a custom parameter, please specify the following: -3ox:“%MANUF_FOLDER%\%ORDER_ID%\%ORDER_ID%.3ox”

Optionally, you can also specify a parameter for filtering the parts to be transferred according to the “Manufacturer ID”.

-ManInfo:<ManufacturerID>

The manufacturer ID can be viewed in the 3ox file.

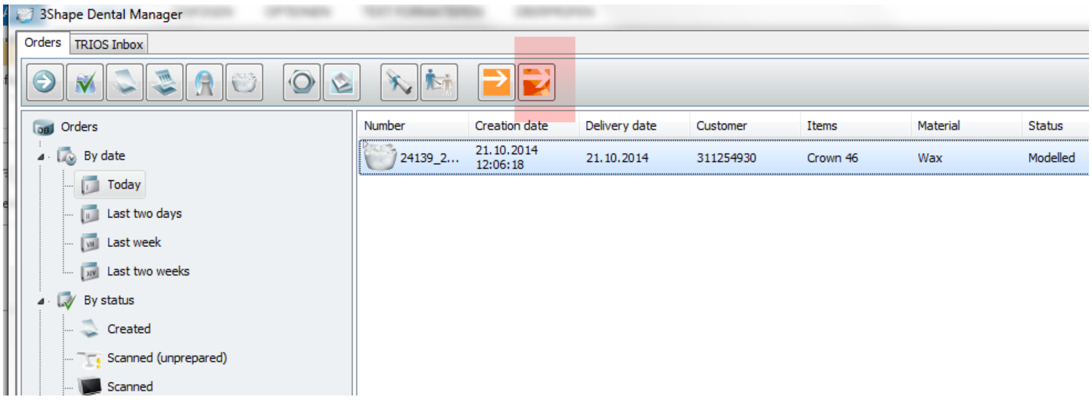
```
<AdditionalOrderInfo/>
<ModelElements>
  <ModelElement displayName="Crown 46">
    <TypeIDs>
      <TypeID uNN="30" typeID="Configuration_01_IDCrownWax08" anatomical="false"
        <InsertionDirUpper x="0" y="0" z="1"/>
        <InsertionDirLower x="0" y="0" z="-1"/>
        <ImplantPosition/>
      </TypeID>
    </TypeIDs>
    <ModelElementIndex>0</ModelElementIndex>
    <Color displayName="Red Wax">IDColorWax01</Color>
    <Material displayName="Wax">Configuration_01_Material3</Material>
    <NoOfUnits>1</NoOfUnits>
    <OrderDate>2014-10-21T12:06:18.000</OrderDate>
    <ShipmentDate>2014-10-21T00:00:00.000</ShipmentDate>
    <DeliveryDate>2014-10-21T00:00:00.000</DeliveryDate>
    <ReceiveDate>2014-10-21T00:00:00.000</ReceiveDate>
    <ScanModule>ScanItRestoration</ScanModule>
    <DesignModule>DentalDesigner</DesignModule>
    <DesignModuleVersion>2.9.9.5</DesignModuleVersion>
    <ScanFiles>
      <ScanFile fileType="AntagonistModel" uNN="0" path="c:\3shape\24139_20141021_
      <ScanFile fileType="Preparation" uNN="0" path="c:\3shape\24139_20141021_120
    </ScanFiles>
    <CAMFiles>
      <CAMFile path="c:\3Shape\ManufacturingDir\24139_20141021_1205_Tech_01\24139_
      <CAMFile path="c:\3Shape\ManufacturingDir\24139_20141021_1205_Tech_01\24139_
    </CAMFiles>
    <ManufacturerID>24139</ManufacturerID>
    <ManufacturingProcessID>24139_ManufacturingProcess6</ManufacturingProcessID>
    <IntermediateFiles/>
    <ProcessStatusID>Modelled</ProcessStatusID>
    <StatusCode>30</StatusCode>
    <ModelFileName>c:\3shape\24139_20141021_1205_Tech_01\CAD\24139_20141021_1205_
    <AdditionalModelElementInfo>
      <AdditionalInfoItem key="ElementId" value="ME9656482E167E4AE8B794E84625D5EA
      <AdditionalInfoItem key="ShaderMaterial" value="Zirconium" datatype="string
    </AdditionalModelElementInfo>
```


Icon:

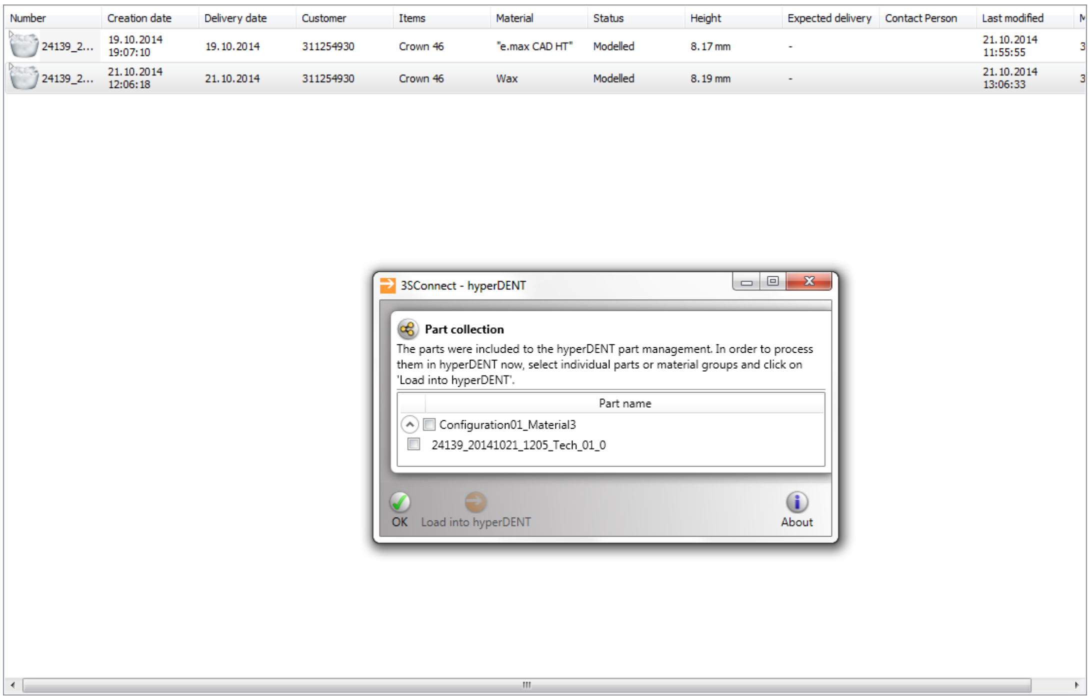
A 24x24 png (appConnect_24px.png) can be referenced from the config/Images directory of the hD installation.

Application:

An additional button appears in the Dental Manager.



If this button is pressed after selecting a project and at least one part is found that corresponds to the filter criterion, the dialog shown below appears.



This dialog can

- Be closed immediately (the hD InQueue was already fed with the relevant information) or

Alternatively, any parts from the list can be selected and you can launch *hyper*DENT with “Load into *hyper*DENT”. The selected parts are loaded immediately in hD.

The part manager directory must be configured accordingly in the CAD system and in hyperDENT®: --> “Settings” > “General” > “Part tracking” > “Part manager directory”.

7 Loading blank

The blank is the material from which the dental restoration is made. Material and shape can be saved as the blank type in the program database.

Apart from the standard blanks, you can also use prefabricated blanks (prefabs) with a finished screw channel, screw fit, and interface geometry. These prefabricated blanks are usually provided by the system supplier to match the corresponding holder.

7.1 Loading blank

The icon is active as soon as the program starts up.

Load the blank for the machining from the blanks saved in the blank administration or from the file system.

Once the blank is loaded, it is displayed on the workspace, and details of the blank appear in the project information.

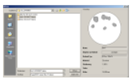
If necessary and for a new project, you can change the selection at a later stage prior to the calculation.

The selection window shows the blank management with the list of blanks, the preview, and data for the selected blank, along with filters for selecting the blank.
The display is dependent on the menu [Settings] > [Blanks] > [Blank administration…]. Further details --> “Settings” > “Blanks” > “Blank administration settings”.

Preview	Top view of the selected blank: Displays the outline of the placed and calculated parts as a screenshot for a quick assessment of the available space left.
Name	Displays the name of the blank.
External ID	Displays the external number of the blank for administration purposes, e.g. storage location.

Charge	Displays the external number of the blank for administration purposes, e.g. same production run.
Blank type	Displays the type of the blank.
Material	Displays the material of the blank.
Color	Displays the color of the blank.
Height	Displays the height (thickness) of the blank. The height must be sufficient so that the dental restoration does not overlap the blank.
Fixture	Displays the fixture used.
Machine	Displays the machine used.
Last saved	Displays the date on which the project was last saved.
Path	Path name in the file system.
Parts in blank	Displays the parts that are placed and saved in the selected blank.
Scaling	Scaling factor for X, Y, and Z-axes or uniform scaling for XYZ, material-specific. Depends on the details in the menu [Settings] > [Blanks] > [Blank types] > [Materials].
Filtering	Selection filter for the blank to quickly find suitable blanks. The fields displayed are dependent on the settings in the blank administration. The height filter is also adopted for the selection window [New blank].
Show empty blanks	Show/hide blanks that have been created but not used. Yes – Empty blanks appear on the list No – Empty blanks do not appear on the list

Environmental filters	Yes Already available information for the selection of blanks is used. Only possible blank types are shown in the list. Function can be activated in the selection window.
Show individual blocks	Blanks for individual elements are also shown in the list
Display locked blanks	Yes No – If blanks from a different user are being machined, they are not shown in the list.
Count	Count of blanks in the list
Clear filter	Clear filter setting and show all items.
Position	Selection of the position in multi-fixtures. Possible positions are shown graphically in the dropdown menu for selection.
File	Selection window for loading blanks from the file system.
New	Call up selection window for loading new blanks.
Open	Load selected blank.
Cancel	Cancel process, and do not load blank.



7.1.1 Loading blank

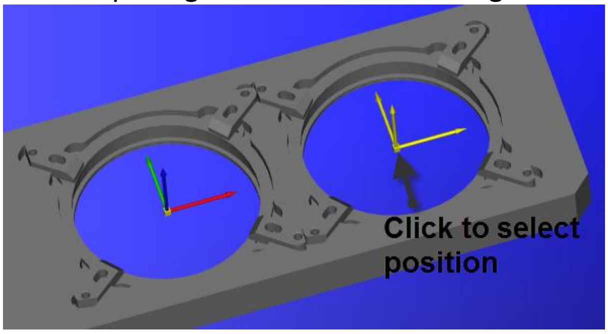
Either

1. Click on the blank in the blank list or the file system:
   The blank is selected; preview and data are shown.
2. Click on [Open]:
   The blank is loaded and is displayed on the workspace.

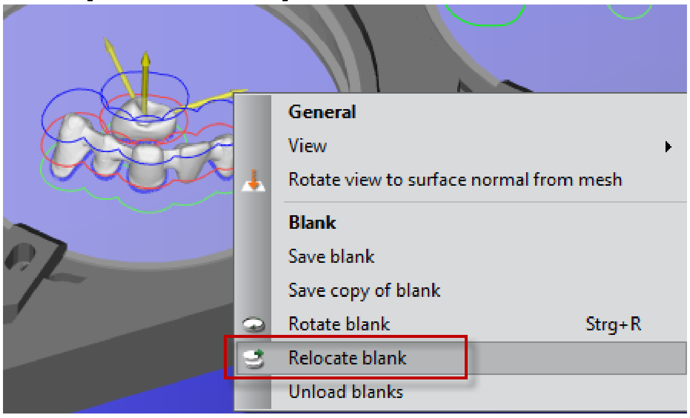
Or

1. Double-click on the blank in the blank list or the file system:
   The blank is loaded and is displayed on the workspace.

In the case of multi-fixtures, the position in the holder must be selected.
As an alternative to selecting the position in the holder, it is also possible to select the desired position by clicking on the corresponding coordinate system before opening the “Load blank” dialog.



If you need to modify the position subsequently, you can do so via the context menu [Relocate blank].



The entire blank, including any held parts, is then relocated.

7.1.2 Adjusting table

Order of columns

Change

1. Click on the column heading, hold the mouse button down and drag column to new position.
2. Release mouse button:
   The column is inserted in the new position.

Reset

1. Call up context menu and select menu item [Reset order of columns to default]:

Right-click on the column heading and click on the menu item.
The column width is reset to the default setting.

Column width

Change

1. Click on the boundary line between the column headings, hold the mouse button down and set the column width.
2. Release mouse button:
   The column is inserted in the new position.

Or

1. Double-click on the boundary line between the column headings:
   The column width is set to the predefined width or the maximum width of the content or the heading.

Reset

1. Call up context menu and select menu item [Reset width of columns to default]:
   Right-click on the column heading and click on the menu item.
   The column width is reset to the default setting.

Sorting

1. Click on the column heading:
   The table is sorted by column contents in ascending or descending order – according to the arrow icon for the sort order.

Showing/hiding columns – Blank administration settings

The columns displayed in the table and the options for filtering the blanks can be set under [Settings] > [Blanks] > [Blank administration...]. Here you can show and hide the columns and filters for the display. Further details --> “Settings” > “Blanks” > “Blank administration settings”.

7.2 New blank

New blanks for machining can only be loaded from the previously defined blank types:
--> Menu [Settings] > [Blanks] > [Blank types…].
Depending on the settings for the blank types, the display only shows blank types that match the project and the blank holder.

The selection window [New blank] shows the selection filters and data for the blank type along with the entry fields for the blank.
The display is dependent on the menu [Settings] > [Blanks] > [Blank administration…]. Further details --> “Settings” > “Blanks” > “Blank administration settings”.

Filter blank type selection	Selection filter for the blank type: Material, Minimum Height, Geometry, Color and Environment Filter. The displayed fields are dependent on the settings in the blank administration, the loaded part, the stored interface information, and the previous blank. The height filter is adopted from the selection window [Load blank]. If the height filter is deactivated and the part is loaded, the smallest sufficient height is preselected.
Type	Selection field for the blank type.
Scaling	Scaling factor for X, Y, and Z-axis or uniform scaling for XYZ, material-specific. Depends on the details in the menu [Settings] > [Blanks] > [Blank types] > [Materials].
Blank type data	Data of the selected blank type/blank.
Material	Displays the material of the blank.
Geometry	Displays the shape of the blank, which has been created in the menu [Settings] > [Blanks] > [Blank type…], and the [Geometries] tab.
Color	Displays the color of the blank.
Blank identification	Name and number of the blank.
Name	Name that can be freely selected for the blank. If nothing is entered, a number is automatically generated and entered as the name.
Generate	Generate number and enter as the name for the blank.
External ID	External number of the blank for administration purposes, e.g. storage place.

Charge number	Notes concerning the blank for administration purposes.
Position	Selection of the position in multi-fixtures. Possible positions are shown graphically in the dropdown menu for selection.

7.3 Creating, editing blank type

The details of the blank type describe the material, geometry, and color of the blanks and they are needed to calculate the toolpaths, machine movements, and collision check.

- Blanks for machining can only be loaded from the previously defined blank types.
- Blank types can only be created using the previously defined geometries and materials.

7.3.1 Creating, editing blank type

For a new blank type, enter a new name, select the material, geometry, and color, and save the entry.

If you select an existing name, then the data for this blank type is displayed. Changes can only be made via the “Edit” function.

1. In the menu [Settings], call up the menu item [Blanks] and then the menu item [Blank types…].
   The [Blank type settings] window is displayed.

2. Click on the [Blank types] tab.
3. Create new blank, change, save, delete, copy it.

Name	Name that can be freely selected for the blank type. Allocate a name, preferably with reference to Material and Geometry, e.g.: CoChr Disc 100x10 for Material, Shape, Diameter, Height. This gives you a better overview when new blanks are created.
Material	Selection field for the material of the blank.
Geometry	Selection field for the shape of the blank that has been created using the [Geometries] tab.

Top color	Selection field for the color of the blank.
Bottom color	Selection field for the bottom color of the blank. The details “Top/bottom color” are used to define a blank with a color gradient.
Transparency	Selection field for the transparency level of the blank.
Milling strategy profiles	Assign, create milling strategy (optional).

7.3.2 Creating, editing geometry

The geometry describes the shape of the blank:

- *Disc* — Disc shape
- *Cylinder* — Cylinder shape
- *Box* — Box shape
- *Extrusion* — Blanks in any form that are described by a closed contour and are defined by a height specification.
- *Freeform* — Blanks of any shape that are described by a model file (closed STL model).

  Prefabricated blanks (prefabs) — Prefabricated blanks of any shape with a finished screw channel, screw fit, and interface geometry that are described by a model file (closed STL model).

1. In the menu [Settings], call up the menu item [Blanks] > [Blank types...].
2. Click on the [Geometries] tab.
3. Create new geometry, edit, use, delete, copy it.

Name	Name that can be freely selected for the shape of the blank. Allocate a name, preferably with reference to the shape, e.g. “disc100-14” for Disc, Diameter 100 mm, Height 14 mm.
Type	Selection field for the shape of the blank. Disc, Cylinder, Box, Extrusion, Freeform.
Diameter	Diameter of the blank.

Height	Height of the blank.
Width	Width of the blank.
Depth	Depth of the blank.
+/-	Tolerance, production-related deviation of the blank from the specified value for diameter, height, width, depth. The blank value is taken into account when placing the parts, whilst the tolerance value is also taken into account for the calculation of the milling paths, e.g.: 14 mm + 0.2 mm = 14.2 mm.
Extrusion curve	Call up the selection window to select and open the file with the geometry data.
Freeform model	Call up the selection window to select and open the file with the geometry data. The freeform model must be available as an *.STL file.
Attachment model	Call up the selection window to select and open the file with the geometry data. Attachment models securely attached to blank geometries with a blank, that are to be included in the collision check. They are subject to the same transformations as the blank geometry (rotation of blanks) and are displayed in hyperDENT® as part of the holder geometry. The attachment model must be available as an *.STL file.
Individual element geometry	Yes Geometries needed for the production of individual elements such as preform abutments and hybrid ceramic blocks are referred to as individual element geometries. This parameter should be set to YES if you wish to use the filter options in the blank management.

Prefab abutment geometry	Call up the submenu, and specify the insertion point for the part if the interface geometry was fully machined already: --> "Prefabricated blanks – Prefabs". The freeform model with the interface geometry must be available as an *.STL file.
Zero point	Calls up the submenu for specifying the location of the zero point for inserting the part.
Screw channel axis	Calls up the submenu for specifying the location of the screw channel axis for inserting the part.
X-axis	Calls up the submenu for specifying the location of the X-axis for inserting the part.
Hole diameter	Diameter of the screw channel.
Alias settings	Calls up the submenu for further settings for the mapping function of the defined interface for inserting the part. Display, add, or delete. setting
Name	Name of the alias setting.
Rotation	Angle for the rotation around the screw channel axis.
Offset	Specification for moving along the screw channel axis.

- The freeform model and the attachment model must be available as an *.STL file.
- The coordinate systems of the fixture, part, and attachment model must have the same alignment.
- The coordinate systems of the prefabricated blank, the fixture, and the part must have the same alignment.

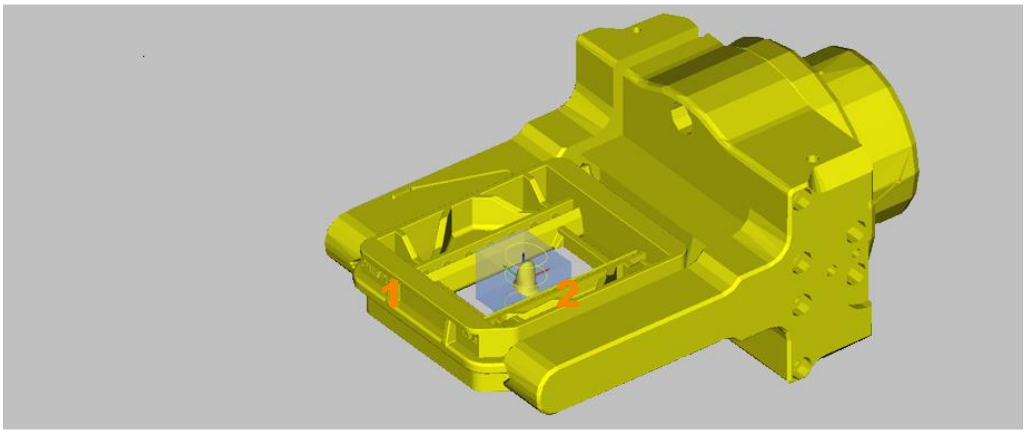


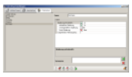
Figure 7-1

The attachment model can also be used with other blank shapes, for example to create a frame (1) around the blank (2).

7.3.3 Creating and editing new material

The materials that are determined here form the basis for the material selection when you create the blank types.

Suitable templates can be assigned by default to the material. This simplifies and speeds up the process when you set the process parameters at a later date.



1. In the menu [Settings], call up the menu item [Blanks] > [Blank types…].
2. Click on the [Materials] tab.
3. Create new material, edit, use, delete, copy it.

Name	Name that can be freely selected for the material. Assign a name, preferably with reference to the material, e.g. “CoCr” for cobalt chrome alloy.
Scaling	Scaling is required if the material needs postprocessing which results in the size being changed, e.g.: Shrinking when sintering zirconium oxide.
Uniform scaling	Resizing is the same in all axes.
Yes	
Default X scaling	Default value for uniform scaling.
No	

Default X, Y, Z scaling	Default values for variable scaling in the X, Y, and Z-directions.
Fix scaling	Set the resizing to the scaling set as default here. The scaling then cannot be changed when creating new materials and loading blanks.
Toolpath length factor	Factor for adjusting the toolpath to different materials for controlling tool change (OEM-specific special function).
Synonyms	Alternative designations (synonyms) for the material. Add, remove synonyms.

7.4 Prefabricated blanks – Prefabs

In the case of prefabricated blanks, the screw channel, the screw fit, and the interface geometry are ready immediately.

The suitable blanks, blank data, and holder data are supplied by the system manufacturer. You can request further information from the hyperDENT® support: --> “Contact”.

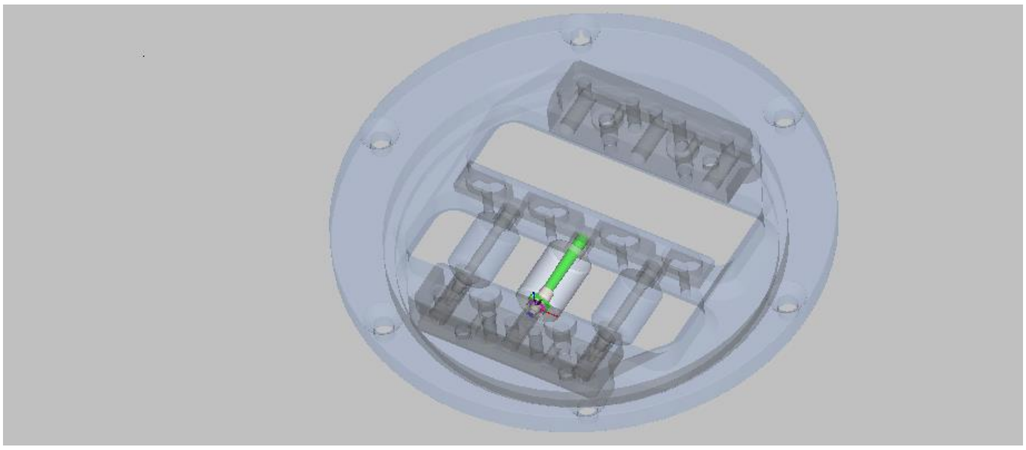


Figure 7-2

- The insertion point for the blank and the part must be coordinated with the existing holder definition.
- The coordinate systems of the holder, the blank, and the part must coincide.
- The STL models for the holder and the blank must be closed.

Insertion point for the part

The insertion point for the part is specified in the definition of the geometry of the blank: --> “Load blank” > “Create, machine blank type” > “Create, machine geometry”.

An additional necessary rotation or movement (Z-offset) can be specified as an alias setting for the mapping mechanism of the import function for the part: --> “Load blank” > “Create, machine blank type” > “Create, machine geometry” > “Alias settings”.

8 Selecting milling unit

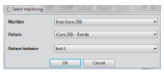
The milling unit is the machining device used for dental restoration. The data for the milling unit is postprocessed in hyperVIEW®.

The icon is active as soon as the program starts up.

To start the machining process, load the milling unit and fixture from the predefined machines and fixtures.

The project data is calculated for the selected machining device and the geometry of the fixture is included in the collision check.

If necessary, you can change the selection at a later stage prior to the calculation.



The selection window shows the list boxes for the machine and fixture.

Machine	Select machine. The machine determines the process for calculating the toolpaths. The details are set for each machine in the menu [Settings] > [Machining…].
Fixture	Select a fixture for the machining that is then displayed on the workspace.
Fixture instance	If several instances/items are available for a holder type, they may be selected explicitly. They must be entered in the holder definition file (.fmdf). On this subject, please consult the FOLLOW-ME! support.
OK	Accept selection, close window.
Cancel	Cancel process, close window.

If the “Fixture” boundary is selected, the boundary line (the distance to the holder) is displayed when placing the parts --> “General job parameters” > “Bounding strategy”.

8.1 Fixture

The fixture is the holder for the blank in the milling unit.

- *The geometry of the fixture is provided as a service by the software provider at the initial installation stage.*
- *For prefabricated blanks (prefabs), the geometries of the blanks are supplied by the system supplier to suit the geometry of the fixtures or they are entered as a service by the software supplier.*
- *For the automatic placement function for the parts in blank, the corresponding settings must be carried out for the fixture: --> “Settings” > “Machining” > “Fixtures” > “Additional properties” > “Object nesting”.*

9 Setting milling direction

The saved data not only determines the shape and size but also the alignment of the dental restoration in relation to the insertion direction and tool axis.

The icon is active only if a part is selected.

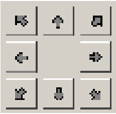
Depending on the data that is loaded, you must set the milling direction:

- *Insertion direction in Z-axis (tool axis)*
- *Occlusion top*
- *If possible no undercuts in the coping*

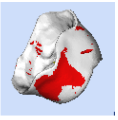
9.1 Setting milling direction

The selection window shows the buttons for the basic alignment, fine adjustment, and direction adjustment.

	Rotate	Rotates the view.
	Machining direction	View from the machining direction.
	Opposite machining direction	View from the opposite direction.
	Initialize	General alignment.
	Occlusal	Set side as occlusal side.
	Cavity	Set side as cavity side.
	Fine adjustment	Align part accurately.
	Degree	0.01 to 10 Degree value for fine adjustment.



Arrow keys	Rotate the part in the direction of arrow by the degree value.
Undercuts	Calculate undercuts in check direction.
Update	Adjust direction.



Show undercut areas

The undercut areas appear in red on the part.

Repeat the undercut check and fine alignment, especially for 3-axis machining, until there are either no more undercuts in the coping or they are very small and are in a non-critical area.

9.2 Coping-specific machining direction

For parts with several copings (bridge, blocking), you can set several machining directions (1, 2). This is necessary if the tooth stumps have significantly different insertion directions.

If the machining direction is not determined by the CAD, you can set the coping-specific alignment via the context menu.

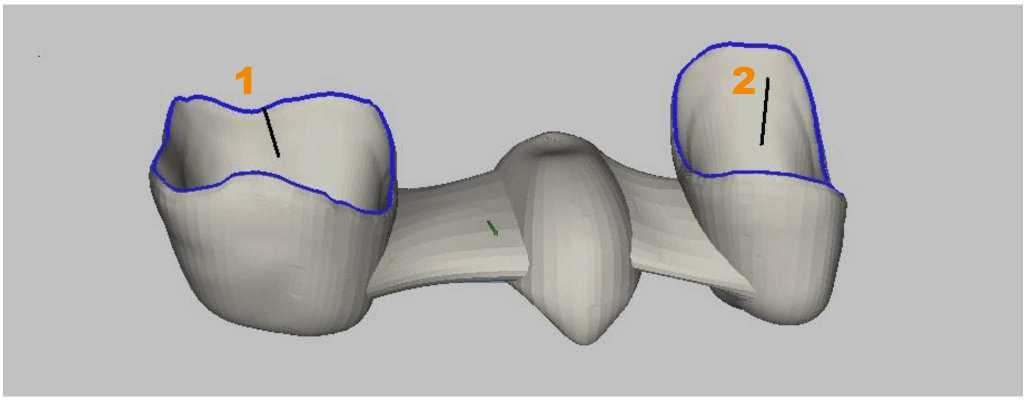


Figure 9-1

The function is only available once the process stage “Identify part features” has been completed.

Setting the specific alignment

1. Select the margin line of the desired coping (1, 2):
   Click on the margin line.
   The selected margin line is shown in a different color.
2. Define the new machining direction for the coping (1, 2).
   Using the right mouse button, rotate the part to the new position so that as far as possible no undercuts occur in this coping.
3. Call up the context menu and select the menu item [Insertion direction from view direction]:
   The new machining direction is shown by a line on the coping.
4. Repeat the process for the other copings of the part.

The correct settings must be configured in the template.

Deleting specific alignment

Select the margin line and delete it.

9.3 Occlusal machining direction (insertion direction)

An occlusal machining direction can be defined for individual copings.

The function is only available once the process stage “Identify part features” has been completed.

Setting occlusal machining direction

1. Define the new machining direction for the occlusal side of the coping:
   Keeping the right mouse button pressed, rotate the part to the new position so that as far as possible no undercuts occur on this coping.
2. Call up the context menu and select the menu item [Set occlusal insertion direction]:
   The new machining direction is shown by a line on the coping.

Deleting occlusal machining direction

Reload part.

9.4 Undercut machining in coping for 3+1 machines

Function used to machine undercuts in copings on 3+1 or 3+2 machines without simultaneously controlled axes. Machining is performed with two differently set jobs via a rotation axis (for 3+1).

If additional undercut machining is required, you can determine this via the context menu.

If necessary, set the undercut machining:
Highlight the margin line and, in the context menu, select the menu item [Change undercut property of coping] > [Coping has undercuts].

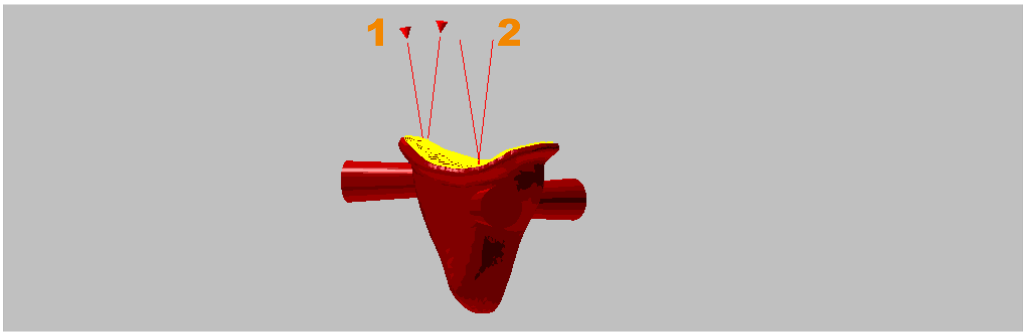


Figure 9-2

The function is only available once the process stage "Identify part features" has been completed.

Selecting undercut machining

1. Select the margin line of the coping:
   Click on the margin line.
   The selected margin line is shown in a different color.
2. Align the part to the rotation axis --> "Tilt part in blank" > "3 + 1 rotation optimization".
   The part must be aligned to the rotation axis in such a way to allow the necessary tilting via the rotation axis.
3. Call up the context menu and select the menu item [Change undercut property of coping] > [Cavity with undercuts]:
   Undercut machining is displayed through a tick in the menu item and a symbol in the coping.

The correct settings must be configured in the template.
The correct settings must be configured in the process step "Tilt part in blank".

Deleting undercut machining

1. Select the margin line of the coping.
2. Call up the context menu and select the menu item [Change undercut

property of coping] > [Cavity without undercuts]:
The undercut machining has been deselected and the symbol on the coping is deleted.

10 Identifying part features

The part features include the edge of the crown (= margin line), pontics, screw channel, and the interface elements of the implants. For subsequent calculations and automated machining, it is important that these areas are identified.

The icon is active only if a part is selected.

If the CAD information is loaded via a defined interface, then the part features have usually already been defined.

Failing that, you can identify parts using the program function, with at least one margin line per coping.

10.1 Margin line

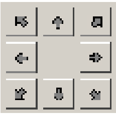
1. Select the [Margin line] tab:
   The selection window shows the settings for labeling the margin line.

Parameters	Define the type of the margin line and the creation type.
Type	Select the margin line. The margin line type must match the part type, otherwise the calculation is not possible. *Coping* *Inlay/onlay* *Abutment base* *Emergence* *User-defined area* *Tooth pocket*
Mode	
Manual	Manually label the area.
Automatic	Automatically determine the margin line.
Draw contour	Draw contour for user-defined area.

Curves	Connect contour points with curves.
Straight lines	Connect contour points with straight lines.
Freehand line	Create contour with a freehand line.
Angle	Angle range of coping edge.
Margin line	Specifications for the margin line.
Selection	Display and selection for margin line count and number.
Offset	Default value (width of coping edge) to move the margin line to the other edge of the coping edge.
Positive value	Move inwards.
Negative value	Move outwards.
Has undercuts	Mark undercuts for multiple machining with a fixed tilt. The correct settings must be configured in the template.
Category	Number of the relevant machining area.
Move	Move margin line by the offset value.
Back	Undo last move.

Insertion direction	Show, define the insertion direction.
Rotate view to insertion direction	Rotate the view and set the view direction:
Rotate to insertion direction	
Rotate to opposite insertion direction	
Alignment	Define coping-specific insertion direction (machining direction):
From view	Set current view direction to the part as insertion direction (= machining direction).

Fine-adjust direction	Align part accurately.
Degree	0.01 to 10 Degree value for fine adjustment.
Arrow keys	Rotate the part in the direction of arrow by the degree value.
X, Y	Rotation around the respective axis.
Z-angle	Deviation of the insertion direction from the hyperDENT® coordinate system.



The tilting of the part in relation to the height minimization is taken into account here already. Further angle deviations from the machining (template) still need to be added.

Calculate undercuts in check direction	
Include opposite direction	Undercut machining in view from both sides
Exclude opposite direction	Undercut machining in insertion direction only

Angle

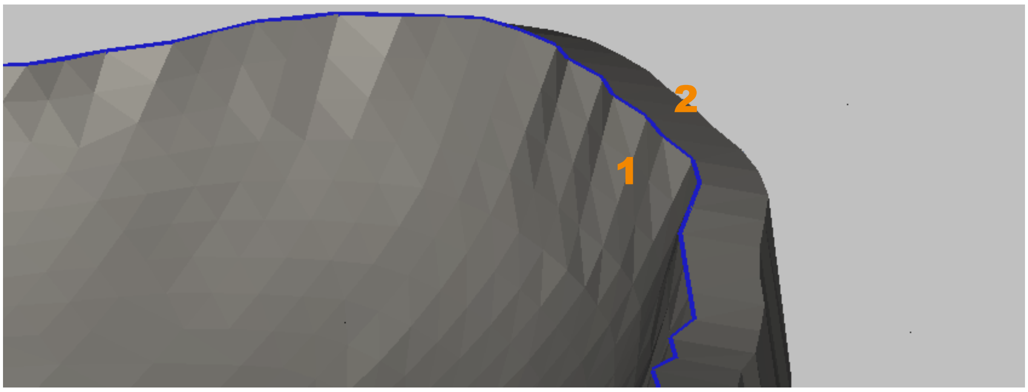


Figure 10-1

Angle range of coping edge = angle in which the system looks for a continuous edge, the margin line.

If, for example an angle of 30° is entered, then a continuous area between cavity (1) and coping edge (2) is looked for where the angle is ≥ 30°.

This ensures, amongst other things, that the margin line is on the inside of the coping edge.
If there is a malfunction, it may be useful to reduce the angle range to 20°.

10.1.1 Automatically determining the margin line

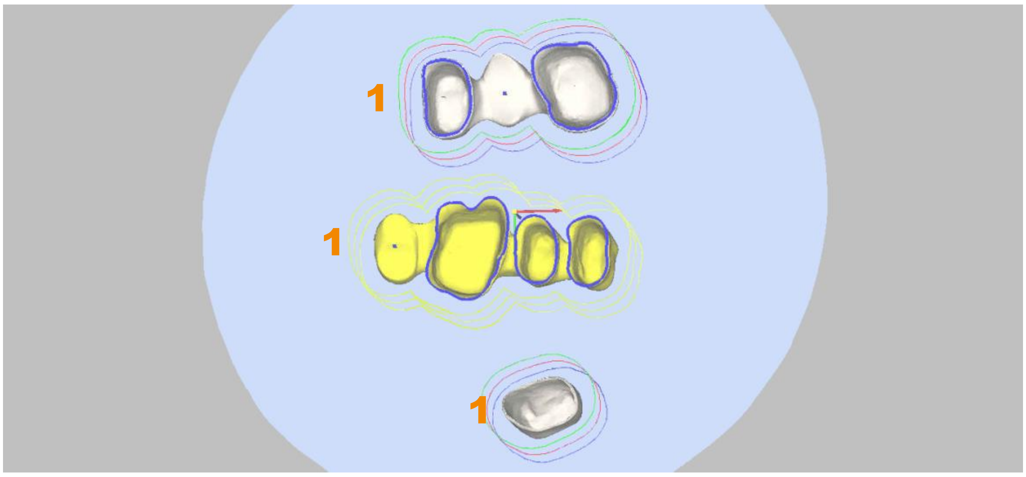


Figure 10-2

1. Select the [Margin lines] tab.
2. Select “Automatic” mode.
3. Select type.
4. Determine the margin line:
   Click into the part (1); for bridges, click into the cavities:
   The crown edge – margin line – is determined and highlighted in blue.

10.1.2 Manually determining margin line

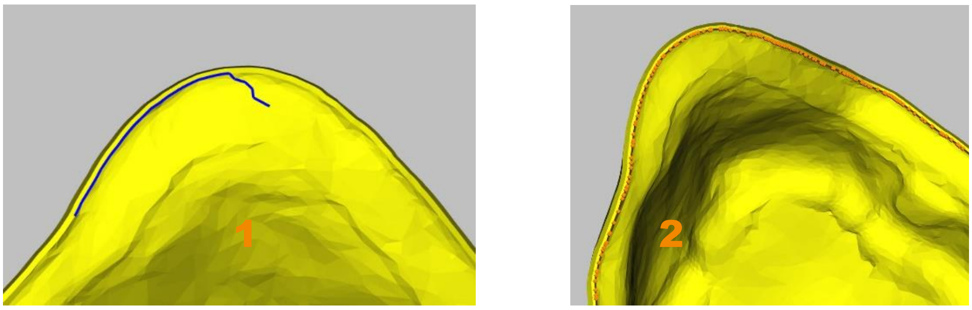


Figure 10-3

1. Select the [Margin lines] tab.
2. Select “Manual” mode.
3. Select type.
4. Define the starting point by clicking on the edge of the coping, and follow the edge with the cursor (1).
5. If necessary, set intermediate points on the margin line and complete the smooth curve (2).
6. If the curve is closed, individual points can be edited.

For a fine adjustment of the anchor points, use the zoom function and 3D view.

7. Delete anchor point:
   Click with the scroll wheel of the mouse onto the anchor point or drag the anchor point to a free area next to the part.

10.1.3 Determining abutment base

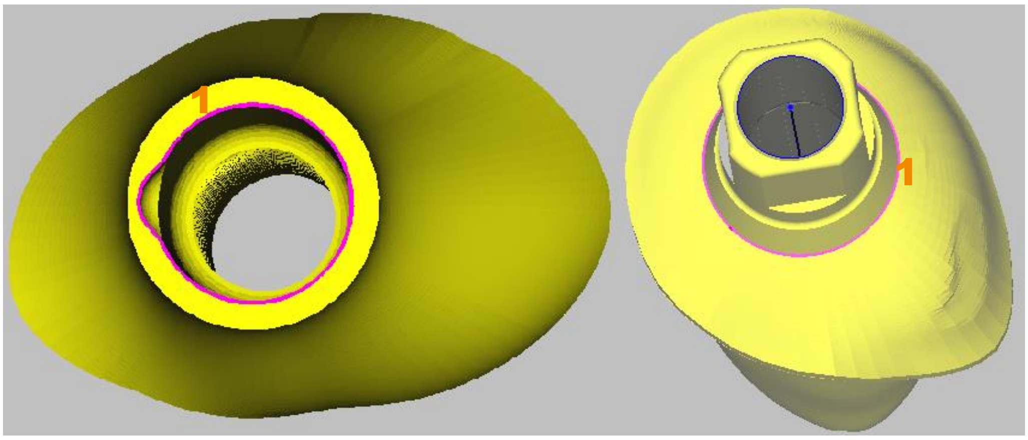


Figure 10-4

1. Select the [Margin lines] tab.
2. Select mode.
3. Select type “Abutment base”.
4. “Automatic” mode:
5. Click on the boundary of the abutment base (1):
   The boundary line is identified and highlighted in color.
6. “Manual” mode:
7. Proceed in the same way as for “Manually determining margin line” and place the anchor points onto the boundary line of the abutment base.

You can determine a separate machining direction for the machining area of the abutment base: --> “Coping-specific machining”.

10.1.4 Determining emergence profile

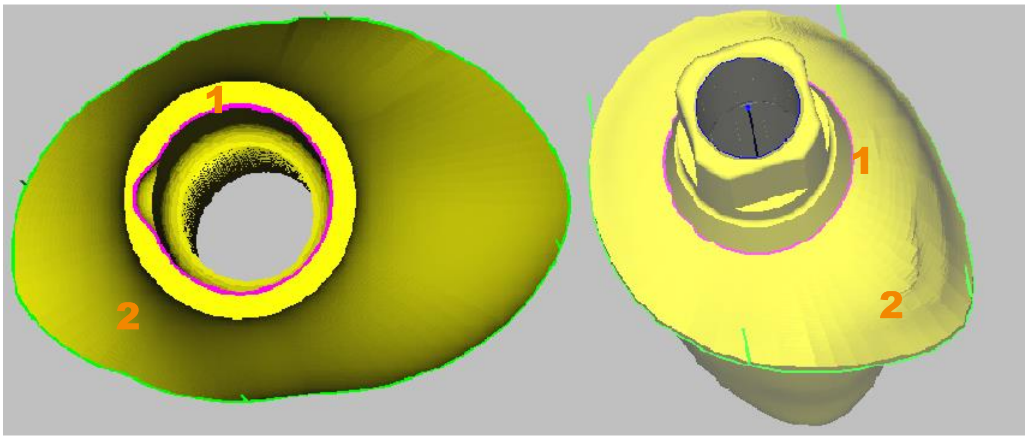


Figure 10-5

1. Select the [Margin lines] tab.
2. Select mode.
3. Select type “Emergence”.
4. “Automatic” mode:
5. Click on the boundary of the emergence curve (2):
   The boundary line is identified and highlighted in color.
6. “Manual” mode:
7. Proceed in the same way as for “Manually determining margin line” and place the anchor points onto the boundary line of the emergence profile.

You can determine a separate machining direction for the machining area emergence: --> “Coping-specific machining”.

If, for example, the abutment is steeply angled, you can also set an occlusal machining direction: --> “Set occlusal insertion direction”.

10.2 User-defined area

For the separate machining of special part areas, e.g. special contour geometries (1, 2) or undercut areas (3), you can define user-defined areas and allocate separate categories with their own machining parameters to these areas (Classic version).

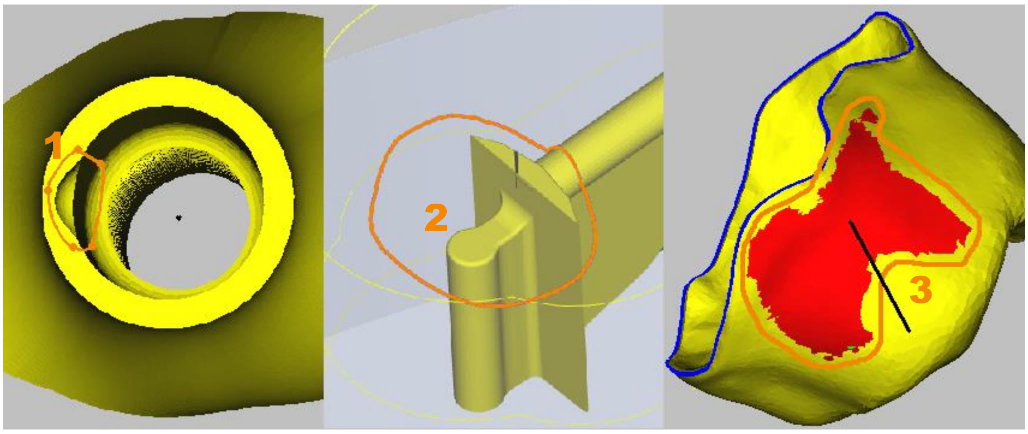


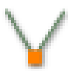
Figure 10-6

Drawing area in

1. Select the [Margin lines] tab.
2. Select mode.
3. Select the “User-defined area” type.
4. The mode is set to “Manual”.

5. Call up the [Draw contour] function.
6. Select the line type.

Curves	Connect contour points with curves



Straight lines	Connect contour points with straight lines

7. Draw contour line.
   Click 3 times on the edge of the area to create a triangle (1).
8. Create new anchor points by clicking on the sides of the triangle and then drag them to the position of the desired area by keeping your finger pressed on the mouse.

9. Manually map the area precisely by creating additional anchor points or by drawing a freehand line.
10. Select the line type.

11. Freehand line Create contour with a freehand line.
12. Draw freehand line:
    Keep the left mouse button pressed and manually map the area precisely with a freehand line (2, 3).
13. Apply line:
    Click on [Accept].
    The contour line is saved.
14. Draw additional contours, edit the contour line or close the window.
15. Allocate the category for the template:
    Enter the category number of the template that is to be used for this area.
16. Create categories of the template: --> “Milling strategies” > “Edit user-defined areas”.

Deleting area

1. Select the area and delete it.
   Click on the contour line of the area.
2. Call up context menu.
3. Select menu item [Delete selected].

10.3 Pontics

1. Select the [Pontics] tab:
   The selection window shows the settings for labeling the pontics.

Count	Number of markings for the pontics.

Marking the pontics

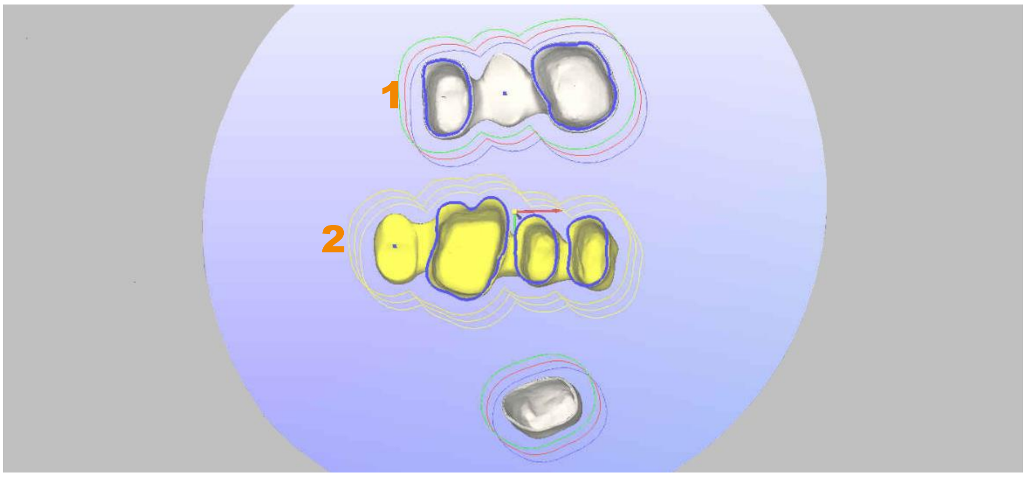


Figure 10-7

1. Select the [Pontics] tab.
2. Mark all pontics (1) and also supporters (2) by clicking in the center:
   The pontic or supporter is identified by a blue dot.

10.4 Editing inlay/onlay bridges

For bridges with different preparation methods for the abutment teeth, you can set the machining strategy separately for each coping (1) and inlay cementing area (2) within the margin line. This is necessary if for instance there is a coping preparation (1) and an inlay preparation (2).

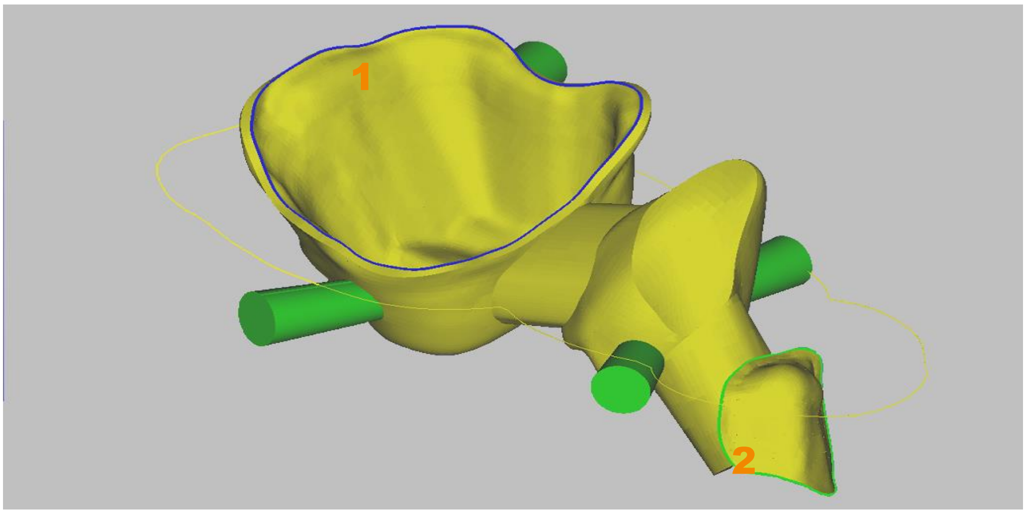


Figure 10-8

The function is only available once the process stage “Identify part features” has been completed.

Changing margin line type

1. Select the margin line of the desired coping (2):
   Click on the margin line.
   The selected margin line is shown in a different color.
2. Call up the context menu and select the function “Change type of margin line”:

Coping (1)	Cavity of the coping.
Inlay/onlay (2)	Inlay/onlay cementing area.

   A type that has been changed is shown in a different color.
3. Repeat the process for the other copings, inlays, or onlays of the part.

The correct settings must be configured in the template (template generator module option).

Deleting a type that has been changed

1. Select the margin line and delete it.

10.5 Interface geometry, screw channel

Directly connected abutments have some very complex interface geometries to the implant in addition to the screw channel.
You can subdivide them into individual areas (planes) for the machining. You can assign a separate category from a template to each of these areas for the individual machining.

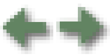
10.5.1 Entering implant interface

1. Select the [Implant interface] tab.
   The selection window shows the settings for highlighting the connection geometry and screw channel of implants and the corner machining for inner geometries.

Mesh roughness	Slider to set the mesh fineness (quality) of the present part data (STL data).
Selection	Display and selection for screw channel count and number.
CAD information	Details from the data.
Min./max. diameter	Diameter of the selected marking, e.g. selected section of the screw channel.
Type of geometry determination	Details of the geometry data.
Explicit	Define the interface geometry in hyperDENT®.
External	Load interface geometry from a file.

Explicit

Determining areas of the interface geometry in hyperDENT®.



Arrow	Move the display of the tabs.
Planes	Display and selection for count and number of machining sections of the machining area of the abutment base geometry, which is assigned to the marked screw channel.
Selection	Display and selection for count and number of the machining section of the abutment base geometry.

Abutment base geometry	Type of interface geometry.
Inner	Inner geometry
Outer	Outer geometry
XY allowance	Default value in XY. Extends for example the inner outline to compensate for the forcing back of thin tools for materials that are difficult to cut.
Category	Category number of the template that is to be used for this section.
Corners	Machining of the inner geometries of abutments by means of additional drill holes.
Selection	Display and selection for corner hole count and number.
Thread	Selection for thread cutting and thread type.
Thread	Display and selection of thread type.
Offset thread begin	Default value for the start of the thread boring operation.
Offset thread end	Default value for the end of the thread boring operation.

External

Interface geometry and defined areas from an external file – geometry replacement (optional).
The name of the geometry is displayed in the information row.

Delete	Delete external interface geometry.
Preview	Preview window for external interface geometry.
Open	Open file with external interface geometry.
Global XY allowance	Global allowance for all categories specified at the interface for the templates. The values can be stored on a part-specific basis.

Position	Buttons for the precise alignment and rotation of the interface geometry.
Move up/down arrow buttons	Move the part along the screw channel axis with an increment (offset) or multiple increments.
Rotate left/right arrow buttons	Move the part along the screw channel axis with an increment (angle) or multiple increments.
Offset increment	0.01 to 10 mm Increment for moving along the screw channel axis
Angle increment	0.01 to 10° Degree of rotation around the screw channel axis.
Move part in relation to the reference surface	Translate interface geometry on axis to adjust a surface with an interface from part.
Rotate part in relation to highlighted point	Rotate part around the screw channel axis until the highlighted points coincide. Use points to rotate interface geometry on axis to adjust with an interface from part
Rotate part in relation to the reference surface	Rotate part around the screw channel axis until the highlighted reference surfaces coincide. Use planes to rotate part on axis to adjust with blank border.

10.5.2 Saving global XY allowance on a part-specific basis

The global allowance applies to all categories of the interface geometry of a part. The values can be stored on a part-specific basis.

The file must have the same name as the part file (".hdpartz") and must be given the file extension ".hdpp".

The file must be created as a text file in XML format.

Example: file entry for global allowance

```
<?xml version="1.0" encoding="UTF-8"?>
<Part>
<partdata>
```

<xyoffset>0.001</xyoffset> [1]

</partdata>

1. Entry for global allowance

10.5.3 Saving external interface geometry, saving categories

You can read the external interface geometry out of an existing project and save it in a file. This means that the data is available to you as an external interface geometry for other projects.

You can also read out the categories of the integrated interface geometries and save them in a file.

Saving data

1. Select the screw channel or the desired interface geometry:
   Click on the screw channel or the interface geometry.

2. In the [Extras] menu, call up the [Execute command] menu item.
3. Enter and accept the command [Save external geometry]:
   Click on [Execute].
4. Select the file path, then the “.hdpartz” file type and accept the selection:
   Click on [OK].
   The file is saved in the specified directory.

Reading out category

5. Load the “.hdpartz” file in hyperDENT® and read out the category.

10.6 Determining screw channel

The screw channel is determined from the view of the cavity side.

The screw channel is closed automatically for all machining outside of the screw channel. Faulty STL facets around the screw channel are removed automatically.

For abutments with a full interface geometry, the screw fit and the different screw channel diameters are automatically identified.

Screw channel identification also works with parts for which the channels are not completely modeled or even closed. However, the base geometry must exist on each part.

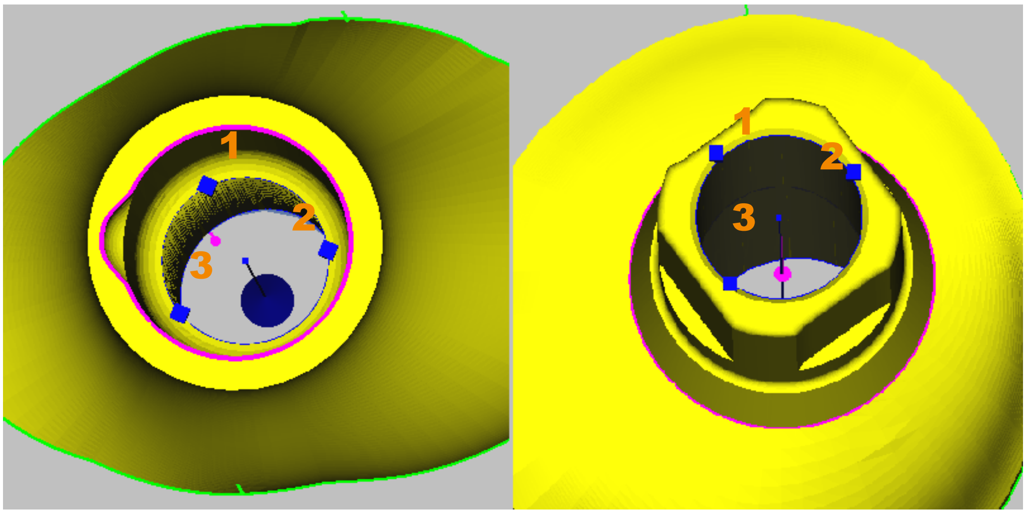


Figure 10-9

1. Select the [Implant interface] tab.
2. Set resolution to fine/rough:
   Set slider [Mesh roughness] according to the quality of the current STL data within the range of “Fine – Rough”.
3. 3 clicks on the section at the start of the screw channel (1, 2, 3):
   The screw channel is identified and highlighted in color; the center (rotation axis) is also highlighted in color.

The screw channel forms its own machining area.

10.7 Detecting the angled screw channel

Many geometries of angled screw channels are detected automatically. The procedure is the same as with normal screw channels.

If they are not detected, manual detection is also possible.

Procedure:

- *Select screw channel on the cavity side.*

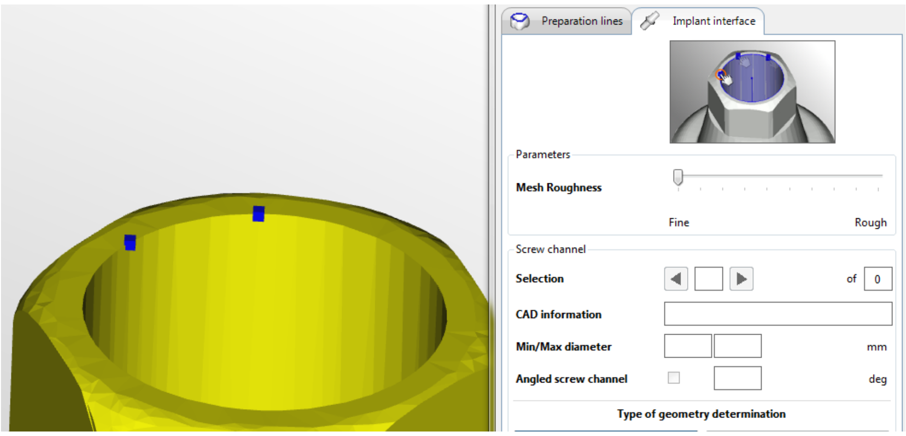


Figure 10-10 Selecting the screw channel on the cavity side

- *Select screw channel on the occlusal side. To do so, click on three point in the screw channel in the area of the angle.*

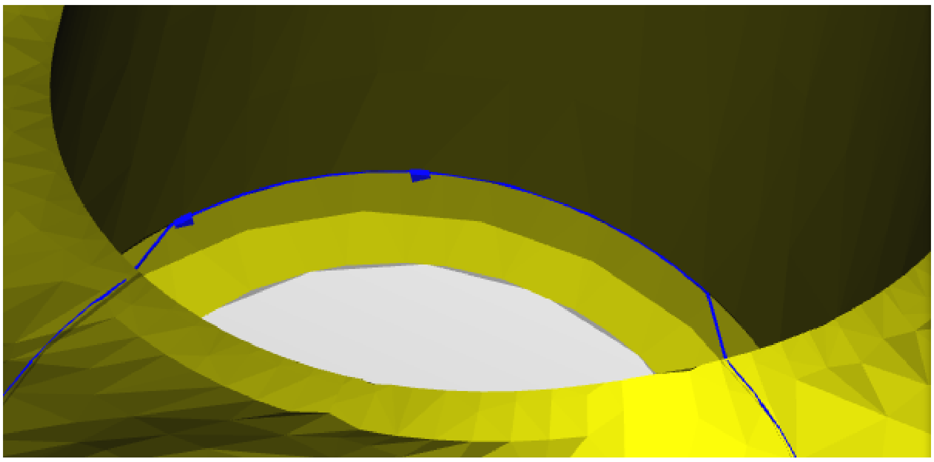


Figure 10-11 Selecting the screw channel on the occlusal side

- *If the occlusal screw channel is not detected properly, this may be because the cut edge is not a circle. If that is the case, you must proceed with the "Use ellipse" function (in the context menu, by clicking with the right mouse button).*

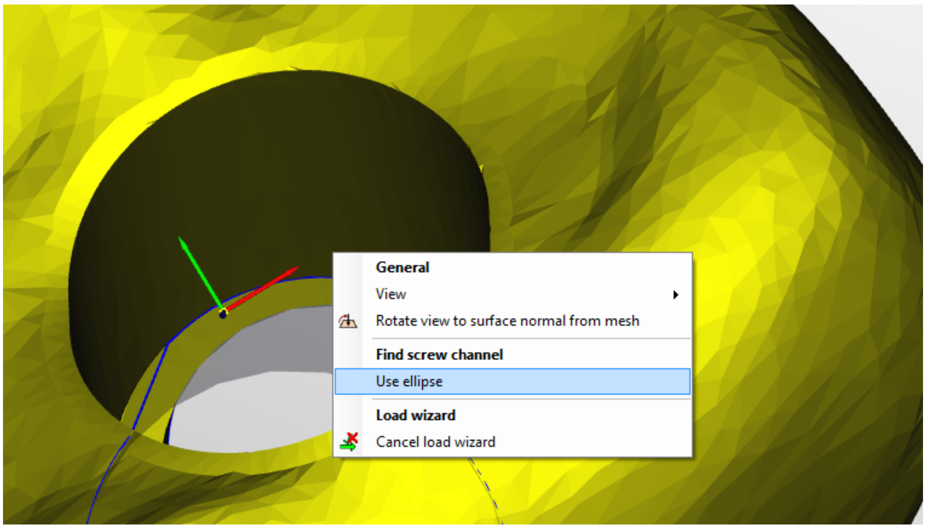


Figure 10-12 Use ellipse, occlusal side

- *Two screw channels should then be available*

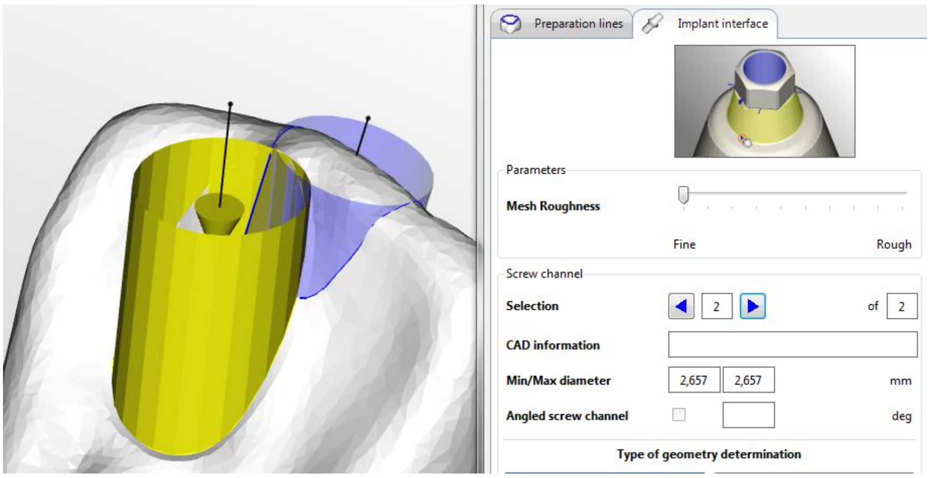


Figure 10-13

- *Combine the two screw channels with the "Combine screw channels" function (in the context menu, by clicking with the right mouse button). To do so, the screw channels must be selected.*

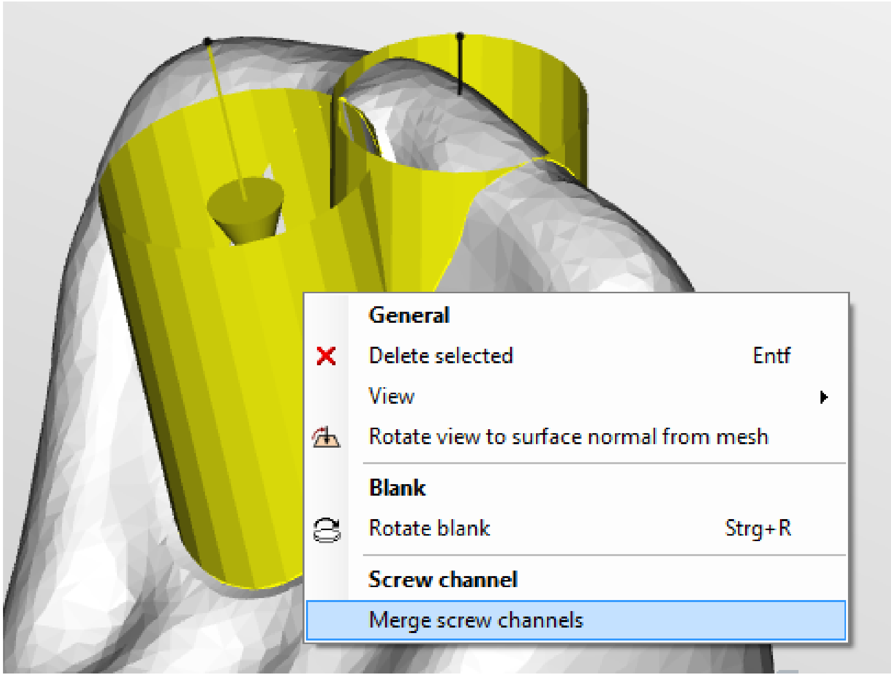


Figure 10-14 Combining screw channels

- *Combined screw channels*

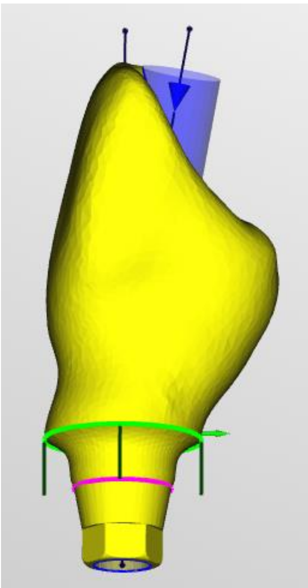
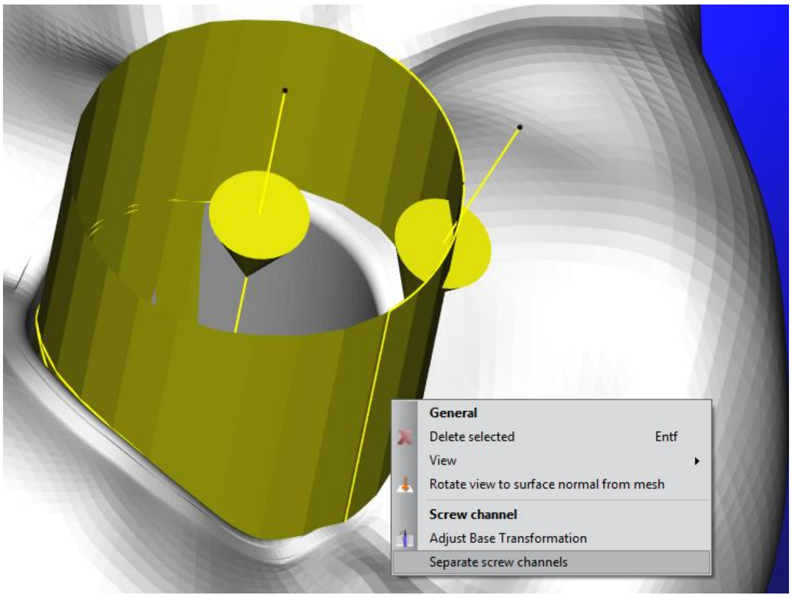


Figure 10-15 Combined, angled screw channel

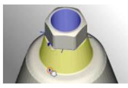
- *Separate faulty angled screw channels*

If angled screw channels are not detected correctly or are loaded from the CAD, they can be separated again.

To do so, click in the context menu on “Separate screw channels”.



10.8 Planes – setting sections for the machining area



You can subdivide complex interface geometries of the implant into individual sections (planes) for the machining.

You can assign a separate category from a template to each of these sections for the individual machining.

A single template can therefore produce very different interface geometries.

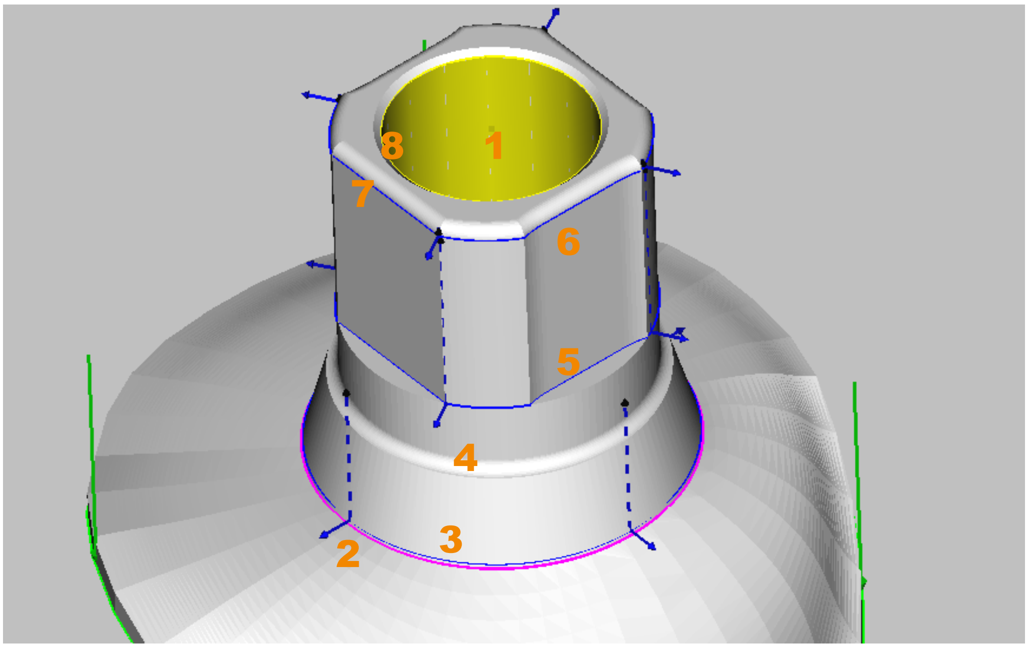


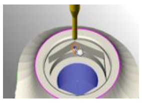
Figure 10-16 Interface geometry with machining areas

1. Select the [Implant interface] tab.
2. Set resolution to fine/rough:
   Set slider [Mesh roughness] according to the quality of the current STL data within the range of “Fine – Rough”.
3. Select screw channel:
   Click on marking of the screw channel (1).
4. Click on the [Explicit] button if it is not yet active.

5. Select the [Planes] tab.
6. Starting from the marking (2) (abutment base), mark the bottom machining section (3).
7. Select the boundary position:
   Abutment base geometry “Inner” for inner outline or “Outer” for outer outline.

8. If necessary, enter the offset value: XY offset.
9. Select the category for the template:
   Set the number for the category.
10. Then mark the top boundary (4) of the first machining section,
    e.g. transition to next level surface.
11. Select the boundary position:
    Abutment base geometry “Inner” or “Outer”.
12. If necessary, enter the offset value: XY offset.
13. Repeat the process for the other machining sections (5, 6, 7, 8).
    Enter the corresponding category number of the template.
14. Click on [Close]:
    The machining sections are saved.
15. Create categories of the template: --> “Abutment” > “Machining method for finishing implant interface geometry”

10.9 Corners – setting the machining of inner geometries



You can machine the corner radii of inner geometries with an optional drilling operation.

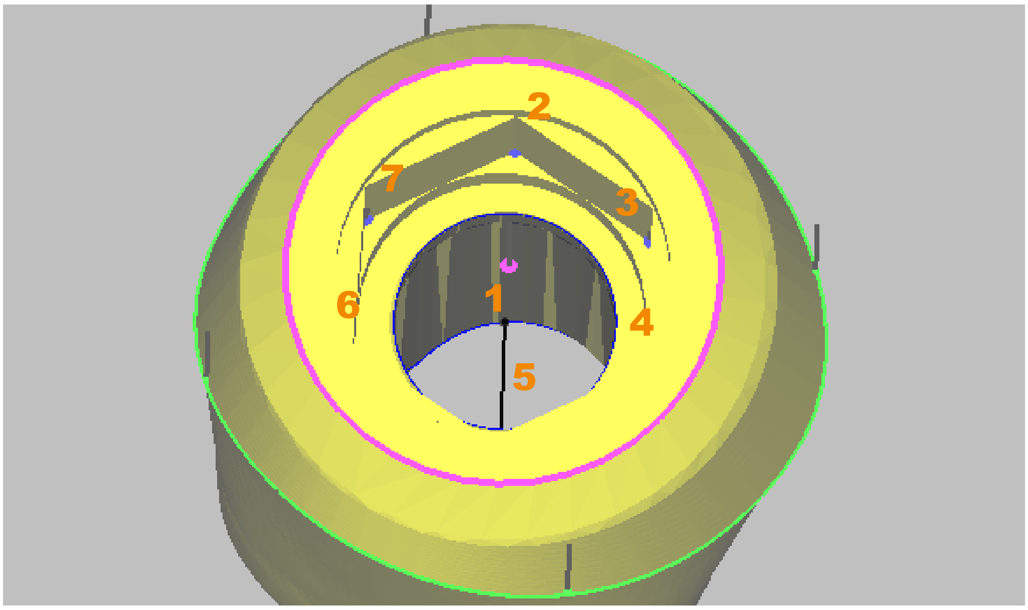
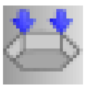


Figure 10-17 Corner machining of inner geometries

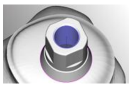
1. Select the [Implant interface] tab.
2. Select screw channel:
   Click on marking of the screw channel (1).
3. Click on the [Explicit] button if it is not yet active.



4. Select the [Corners] tab.
5. Mark corners (2, 3, 4, 5, 6, 7) for drilling.
6. Click on [Close]:
   The corner markings are saved.

A separate drilling operation is used for corner machining which should be defined in the template.

10.10 Thread cutting



Function for the manufacture of an internal thread (thread cutting) in the section of the screw channel area with the smaller diameter (1).
The threads comply with ISO and UNF standards up to approx. 4 mm

Before thread cutting, the screw channel must be predrilled with the appropriate core hole diameter. The core hole diameter is shown in the selection window for the thread types.

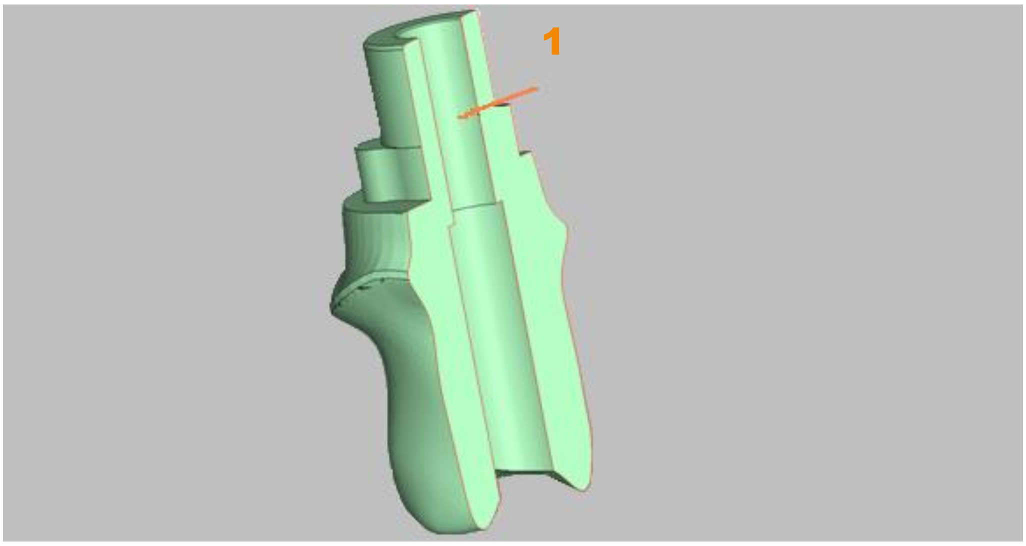
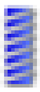


Figure 10-18 Thread milling

Predrill with core hole diameter!
Before thread cutting, the screw channel must be drilled with a core hole diameter appropriate for the thread type.

1. Select the [Implant interface] tab.
2. Select screw channel:
   Click on marking of the screw channel (1).
3. Click on the [Explicit] button if it is not yet active.



4. Select the [Thread] tab.
5. Mark threads and select thread type:
   The nominal diameter or the thread identifier and the core diameter of the thread type are shown in the dropdown menu.
6. If necessary, enter the offset: Start of thread and end of thread.

7. Click on [Close]:
   The corner markings are saved.

10.10.1 Thread types

The available thread types are shown in the selection.

- *ISO (metric) Thread diameter – core hole diameter*

  Type Thread Ø – core hole Ø

  ISO M 1 – 0.75

  ISO M 1.1 – 0.85

  ISO M 1.2 – 0.95

  ISO M 1.4 – 1.1

  ISO M 1.6 – 1.25

  ISO M 1.7 – 1.3

  ISO M 1.8 – 1.45

  ISO M 2 – 1.6

  ISO M 2.3 – 1.9

  ISO M 2.5 – 2.05

  ISO M 2.6 – 2.1

  ISO M 3 – 2.5

  ISO M 4 – 3.3

- *UNF Thread identifier – core hole diameter*

  Identifier – core hole Ø

  0 – 80 UNF – 1.25

  1 – 72 UNF – 1.55

  2 – 64 UNF – 1.9

  3 – 56 UNF – 2.15

  4 – 48 UNF – 2.4

  5 – 44 UNF – 2.7

  6 – 40 UNF – 2.95

  8 – 36 UNF – 3.5

Further details on the threads are available at
http://www.gewinde-normen.de/en/index.html

10.11 Individual machining directions for abutments

You can set individual machining directions for the machining areas of abutments – “Occlusal side”, “Emergence”, “Abutment base” without an interface geometry.

The function is only available once the process stage “Identify part features” has been completed.

Default for the machining areas:

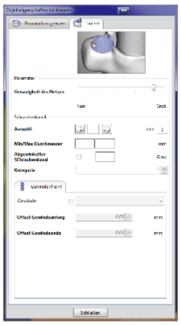
- *Occlusal side* — Parallel to the main insertion direction:
  --> “Set milling direction”.
- *Emergence, abutment base* — Parallel to the screw channel.
- *Import via a defined interface* — All alignments and boundary lines are accepted by CAD.

Determining individual machining direction

1. Select the margin line:
   Click on the margin line.
   The selected margin line/boundary line is shown in a different color.
2. Set the new machining direction for the machining area:
   Keeping the right mouse button pressed, rotate the part to the new position so that as far as possible no undercuts occur in this machining area.
3. Pull up the context menu and select the menu item [Insertion direction from view direction] or [Set occlusal insertion direction]:
   The new machining direction is shown by a line on the machining area.

Deleting individual machining direction

Select the margin line and delete it.

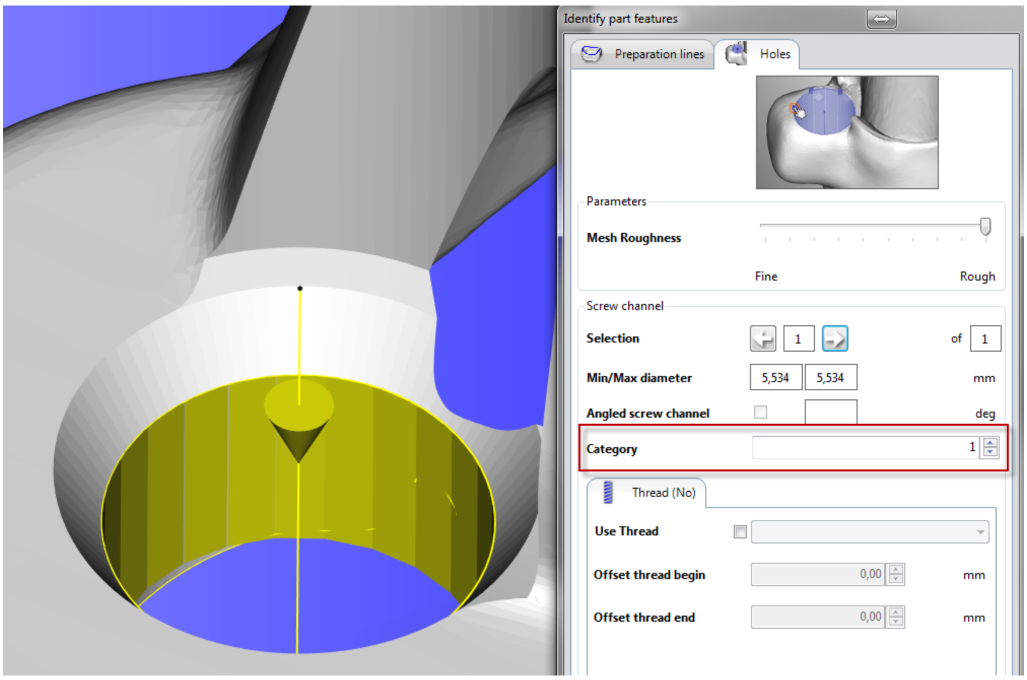


10.12 Holes

Bores or threads can be machined into parts with the help of holes. This function is available in all part types.

You can define such holes, like screw channels, by setting three points on the perimeter of the bore. Thread machining is carried out in a similar way to screw channels of abutments.

Category	To make it possible to differentiate between various bores, a category must be set. This category must be referenced in the template in the screw channel machining job.



11 Tilting part in blank

The insertion direction (= machining direction) of the aligned part is parallel to the tool axis, which prevents undercuts in the coping. However, this alignment may mean that the dental restoration is heavily tilted and therefore protrudes from the surface of the blank.

The icon is active only if a part is selected.

Minimizing height

You can change the tilt angle and thereby reduce the height.

The alignment of the insertion direction to the tool axis and undercut areas remains unchanged.

For machining purposes, the blank, together with the holder must be set at an angle (tilted machining). This requires at least a 3+1 machine.

- Risk of collision!
  The tilt can cause a collision between the blank, the fixture and the tool holder. Follow the program instructions when performing the calculation.
  The milling boundaries are extended for the tilted machining, depending on the settings in the template.

Minimizing inclination, angle optimization

In the case of steeply angled parts, e.g. abutments, the tilt of the machining directions must be aligned to an average value so as not to exceed the maximum tilt angle of the milling machine.

Undercut machining

You can also perform undercut machining for a 3+1-axis machining. To do so, you must enter the rotation axis and rotation optimization and set the respective parameters in the template.

- Risk of collision!
  The milling boundaries are extended for the undercut machining, depending on the settings in template.

The selection window shows the buttons that are used to rotate and tilt the part and minimize the height.

Machine	Details about the machine.
Rotation axis X, Y	Selection box for the rotation axis (X, Y) of the machine: Axis around which the tool (holder) can be pivoted. Essential for undercut machining
Position part	Details about the tilt part and alignment.
Center part	Center the part in the height in the blank.
3+1 rotation optimization	Optimally align the tilt part to the rotation axis. Essential for coping-specific machining on 3+1 machines. Also possible for tilted parts without insertion direction.
Rotate 180 degrees around Z	Align the tilt part by rotating it by 180°, e.g. for more favorable milling boundaries.
Lock placement	
	Rotation
	XY position
	Z position
Tilt part	
Minimizing	
Height	Minimize the height; the inclination can increase, e.g. for bridges in order to use thinner raw material.
Inclination	Minimize the inclination; the height can increase, e.g. for steeply angled abutments or copings with a slanting tooth edge and slanting margin line.
Max. tilt angle	Set the maximum tilt angle.
Lock tilt axis	Secure inclination.
Tilt	
Arbitrary axis	Create an inclination on an arbitrary axis.
Rotation axis	Inclination around the rotation axis.
To blank height	Tilt part to blank height.

Minimize	Minimize inclination.
Reset tilting	Reset the inclination.

Example of tilt part in blank

For a tilted workpiece, the rotation axis of the machine must be taken into account. If there is no 5-axis simultaneous machining, then the tilt axis of the dental restoration must be brought into line with the rotation axis of the machine.

Example of minimize height

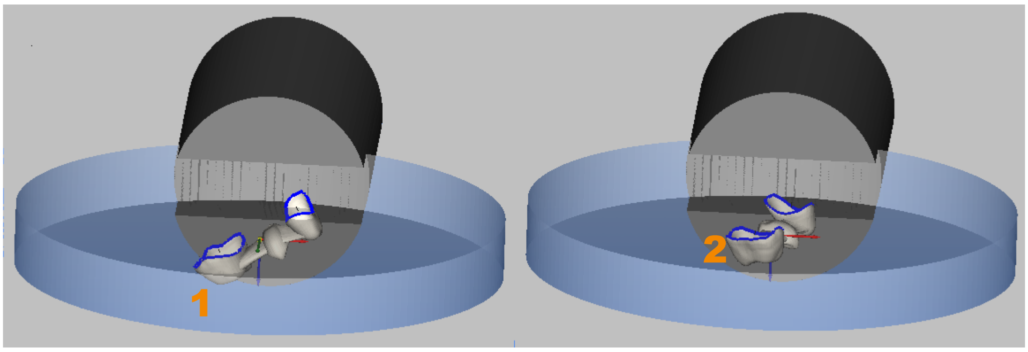


Figure 11-1

Part (1) has been manually placed. As a result of the direction adjustment setting, the part protrudes from the blank.

Due to the kinematic options of the milling unit, X was set as the rotation axis of the machine when the tilt part was determined.

Click on [Tilt] and the part is:

- *Rotated* — The part is aligned in relation to the rotation axis (2) of the machine, so that it can be tilted for machining.

and

- *Tilted:* — The part is tilted in accordance with the defined degree value and direction so that it is fully in the blank.

Example of angle optimization

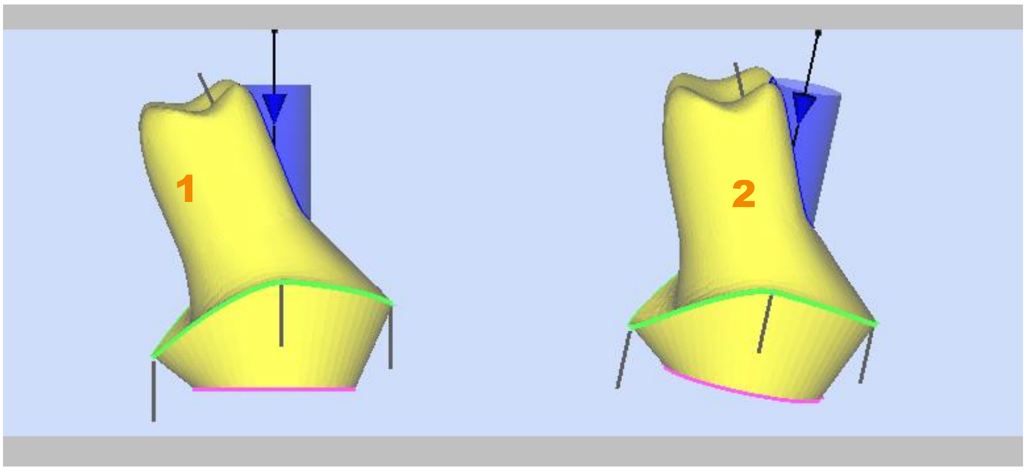


Figure 11-2

Part (1) has been manually placed and the occlusal insertion direction has been set --> "Occlusal machining direction". The part shows evidence of undercuts in this position which require considerable tilting. By tilting the part, the undercuts are reduced which creates more favorable tilt angles for the machining.

12 Selecting template

The templates contain the working plan (milling strategy) for machining on the milling unit and are available for different materials and part types.

Different templates can be selected for the different parts, depending on the details in the previous process steps.

The icon is active if at least one part is selected. You can also select several parts.

Select the template that you would consider from experience to be most suitable for the selected part.

The template can significantly influence the quality, accuracy, and runtime of the calculation and machining process.

The selected template must be compatible with the construction machine and must be suitable for the material.

Several parts with the same template or several parts with different templates can be calculated and machined together in a blank.

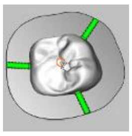
If a template is to be assigned to several parts, the parts must have the same part type.

The selection window shows the buttons used to select the templates.

Template profiles	Selection box for the template.
Accept	Accept selection.
Close	Close window.

13 Setting connectors, sinter frames

Connectors



The connectors or support pins are small connections that hold the part in the blank during the milling process. The dental restoration can thereby be machined from all sides. Separate the connectors after the milling stage and remove the pins.

The icon is active if at least one part is selected. The function works for all parts.

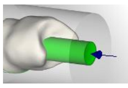
Set enough connectors so that the part is retained securely and accurately until machining has finished.

You can set the connectors automatically or manually and save these settings as defaults.

You can change each set connector individually, assign a cut to it, or delete it.

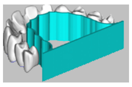
In the case of open fixture geometries, if the milling area boundary protrudes beyond the blank, connectors will only be set inside the blank.

Screw channel connectors



The screw channel connector is used for prefabricated blanks (prefabs). It runs along the prefabricated screw channel from the occlusal side of the part to the opposite holder, and surrounds the screw channel either in cylindrical form (0°) or conical form depending on the entered angle.

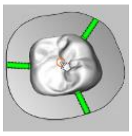
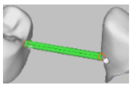
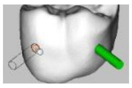
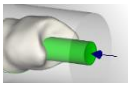
Sinter frames



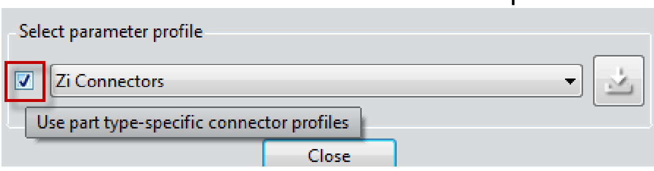
The sinter frame supports the dental restoration during the sinter process and avoids the risk of large arched parts becoming deformed.

13.1 Setting connectors

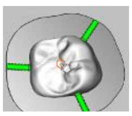
1. Select the [Connectors] tab:
   The selection window shows the settings for the connectors. The function [Set connectors] is active if the window is open.

Mode	Type of connector placement.
 Automatic	Click on the part: Automatically set all connectors in accordance with the settings.
 Traverse	Click on original and target part one after the other: Connect two parts with overlapping milling area using a traverse (consisting of two connectors) (optional).
 Manual	Click on the position for the connector: Set a connector in accordance with the settings at this location. Optional with cut.
 Screw channel connector	Click on the part: Set the screw channel connector in the course of the prefabricated screw channel according to the settings. Optional with cut. Apply the modified settings.

Take ALL parts into account	All parts that are not locked or calculated are incorporated in the calculation of the connector positions. For this purpose, no connectors should be present. One possibility to ensure this is to deactivate the automatic placement of connectors in the “Load wizard”. If the parts are selected, any existing connectors can be selected in the [SelectConnectors] context menu and can all be deleted there at once.
Angle	Taper angle of the connector: Cylinder = 0
diameter	Diameter at the part.
Wall thickness	0.1 mm Wall thickness of the screw channel connector. The diameter of the connector on the part is: Diameter of the prefabricated screw channel + 2x wall thickness.

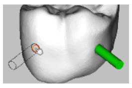
Distance to the margin line.	Distance to the margin line. If the safety distance is too small, this may damage the crown edge.
Count	Number of connectors for a crown that are automatically set.
Connectors at pontic position	Also set connector at the pontic.
Cut	Connectors that are automatically cut at the end of the machining. Manually set individual connectors with cut or assign cut at a later stage. The job must be stored in the template.
Cut depth	Percentage value for the cut:
0%, none	No cut.
100%, complete	Complete cut, the connector is completely cut.
Cut safety distance	Distance of the cut to the part. If the distance is too small, this may damage the part.
Parameter profile	Selection field for the saved connector profiles.  To make it possible to use the part-specific connector profiles, there must be a check in the selection box in front of the dropdown menu.
 Save	Call up selection window [Save profile]. Save settings, save under a new name, set as default.

13.1.1 Setting connectors automatically



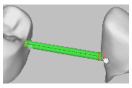
1. Click on the part:
   All connectors are set automatically according to the settings.

13.1.2 Setting connectors manually



1. Click on the position for the connector:
   A connector is set at this location according to the settings. Optional with cut.

13.1.3 Setting connectors as a traverse



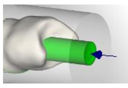
If the milling area of the two parts overlaps, you can connect the parts using a traverse.

1. Click on original part for the traverse:
   The auxiliary line for the traverse is displayed from the original part.
2. Drag auxiliary line for the traverse to the target part and click on the target part:
   The traverse is placed between the parts. The traverse consists of two connectors.

Traverses can be selected and edited with a click. If you wish to only edit one side of the selected traverse, the respective side can be deselected by pressing the CTRL key and the other one can be edited individually.

13.1.4 Setting screw channel connector

The function is active for prefabricated blanks with a prefabricated screw channel.



1. Click on the part:
   The screw channel connector is set automatically according to the settings.

13.2 Deleting connectors

1. Select connector.
2. Call up context menu.
3. Select menu item [Delete selected].

13.3 Editing connectors

You can change the settings for an existing connector, e.g. assign, change, or remove a cut.
Editing can be done across the parts.

Call-up via the context menu or the menu [Edit]

1. Select connector.
2. Call up context menu or menu [Edit] > [Connector].
3. Select menu item [Edit connectors].

The selection window shows the settings for the selected connector.

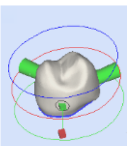
Edit	Activate settings for the machining.
Angle	Taper angle of the connector: Cylinder = 0
Diameter	Diameter at the part.

- *Caution!*
  In the case of screw channel connectors, the diameter must be greater than: screw channel diameter + double the wall thickness.

Milling cut	Assign or change the cut at a later stage. For connectors that are automatically cut at the end of the machining.
Cut depth	Percentage value for the cut:
0%, none	No cut.
100%, complete	Complete cut, the connector is completely cut.
Cut safety distance	Distance of the cut to the part. If the distance is too small, this may damage the part.
Apply	Apply changes.
Close	Close window, do not apply changes.

13.4 Moving connectors

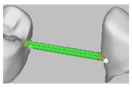
For an existing connector, you can change the pin on the part and the direction. You can therefore for example better adapt the automatically set connectors to the shape of the part and optimize the construction.



1. Select the connector by double-clicking on it:
   The axis of the connector and both end points are shown. The pin on the part is highlighted by a line.
2. Click on the end point or starting point with the left mouse button. Hold the mouse button down and move to the desired position.

3. Finish the selection by clicking on the workspace: The connector is displayed at the new position.

13.5 Autoconnecting the connectors



If the milling area of two or more parts overlaps, you can autoconnect the connectors of the parts (optional).
This function depends on the menu [Settings] > [General] > [Connector behavior]: --> "Edit connector settings" > "Connector behavior".

Reloading parts

1. Place objects:
   Ensure that the milling boundaries of the parts overlap, but do not protrude into other, unfinished parts. Automatic at "Automatic nesting" YES ("Settings" > "Machining" > "Fixtures" > "Additional properties" > "Use automatic nesting").
2. Set connectors in automatic mode: --> "Set connectors".
   The connectors are also placed between the parts.

Moving parts

1. Highlight part to be moved:
   Double-click on the part.
2. Move part so that the milling boundaries of the parts overlap, but do not protrude into other, unfinished parts. Connectors that overlap are connected to one another; connectors that intersect the milling boundary are connected to the other part.

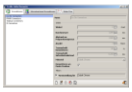
Removing connection

1. Move part so that the milling boundaries no longer overlap.
   The connection between the connectors is cut.

13.6 Editing connector settings

13.6.1 Settings for connectors and screw channel connectors

1. Call up the menu [Settings] and select the menu item [Building elements...].
2. Select the [Connectors] tab or the [Screw channel connectors] tab.

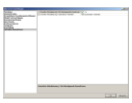


The selection window shows the list of connector types, the settings for the selected connector type, and the toolbar for editing the settings: New, edit, apply, delete, copy.

Name	Name for the connector type.
Angle	Taper angle of the connector: Cylinder = 0
Diameter	Diameter at the part.
Wall thickness	0.1 mm Wall thickness of the screw channel connector. The diameter of the connector on the part is: Diameter of the prefabricated screw channel + 2x wall thickness.
Distance to the margin line.	Distance of the connector to the margin line. If the distance is too small, this may damage the part (crown edge).
Count	Number of connectors for a crown that are automatically set.
Milling cut	Cut depth (percentage value) for connectors that are automatically cut at the end of the machining.
0 %	No cut.
100%	Complete cut, the connector is completely cut.
Cut safety distance	Distance of the cut to the part. If the distance is too small, this may damage the part.
Material	Displays the assigned material.
Connectors at pontic position	Also set connectors at the pontics
Default for	Determining entries as default for the displayed material.

13.6.2 Connector behavior

1. Call up the menu [Settings] and select the menu item [General].
2. Select [Connector behavior] area.



The selection window shows the setting options for updating the connectors and the toolbar for editing the settings: OK, cancel.

Connector update/connect overlapping connectors

Yes	Autoconnect existing connectors that overlap (1).

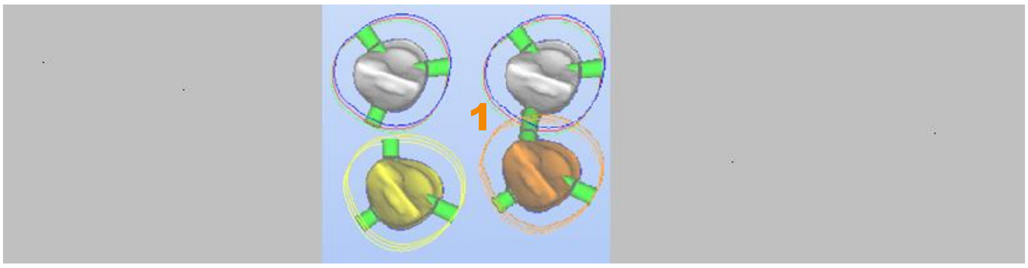


Figure 13-1

No	Do not autoconnect connectors.

Connector update/autoconnect	Connect existing connectors if the connectors overlap after manual positioning of the part or connector. If you move the part or the connector far enough that the milling boundaries no longer overlap, the connection between the connectors is removed.
No autoconnect	Do not autoconnect connectors.
Connect connectors of other parts	Connect connectors of a part (1) with the moved part (2), if the milling boundary of the moved part is pushed over the connector.

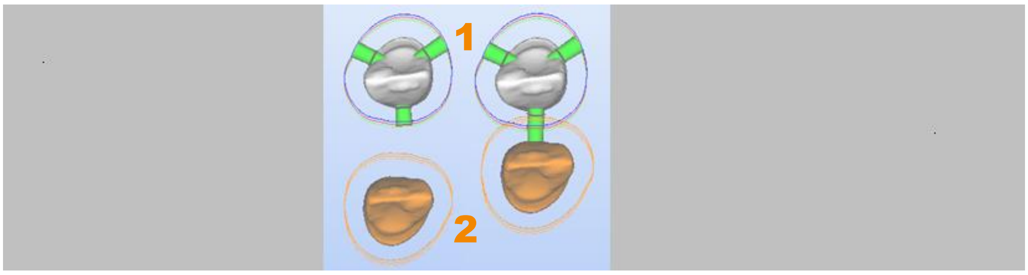


Figure 13-2

Connecting connectors of a moved part	Connect connectors of the moved part (2) with another part (1), if the connector of the moved part is pushed over the milling boundary of the other part.

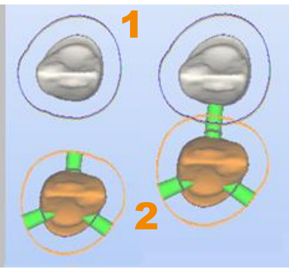


Figure 13-3

Connect all	Connect connectors of all parts if their milling boundaries (1) are pushed over one another.

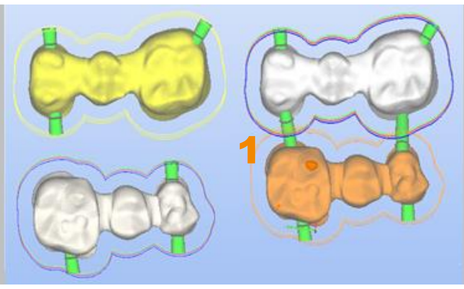
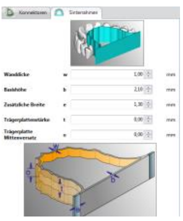


Figure 13-4



13.7 Creating sinter frames

1. Select the [Sintering pins] tab:
   The selection window shows the settings for the sinter frame. The function [Set sinter frame] is active if the window is open.

Wall thickness w	mm Wall thickness of the sinter frame in the area of the dental restoration.
Base height b	mm Wall thickness of the sinter frame at the base, the connection between the two struts.

Additional width e	mm	
	Extension of the base toward the outside	
Support plate thickness t	mm	
	Thickness of a plate in the core of the sinter frame	
Support plate center offset o	mm	
	Offset of the support plate from the center	

2. Click on the part:
   The sinter frame is created.
3. Connect sinter frame with the dental restoration using connectors.

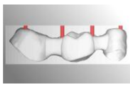
The inner area of the frame can be used to mill another part.
To do so, this part must be loaded before applying the sintering frame to the bridge part, or the parts must be calculated one after the other.
Optionally, the support plate can also be set to the material thickness.

By editing the connectors, you can specify whether the outer connectors are entirely disconnected and the inner ones are partly or entirely disconnected.

Detaching the inner area

With the appropriate template parameters, the inner area can be pushed out of the frame using light pressure.

14 Setting sintering pins



The sintering pins are small posts that support the part during the subsequent sintering process and provide a level plane for larger dental restorations. The sintering pins must be removed after sintering.

Sintering pins can be inserted as an option and are not essential for the completion of the process steps.

14.1 Setting a sintering pin

The icon is active if at least one part is selected. The function works for all parts.

You can save the settings for the sintering pins as defaults.

You can move, edit, or delete the set sintering pins individually.

The selection window shows the settings for the sintering pins. The function [Set sintering pins] is active if the window is open.

Angle	Taper angle of the sintering pin: Cylinder = 0 An angle of > 0° is required so that the sintering pins are not damaged during the milling process. The larger the angle, the larger the plane area on the part.
Diameter	Diameter at the boundary area. If the distance to the part is large, then the plane area on the part also becomes larger.
Parameter set	Selection field for saved sintering pin types.
Save	Call up selection window [Save profile]. Save settings, save under a new name, set as default.

1. Click on the position for the sintering pin:
   A sintering pin is set at this location in accordance with the settings.

14.2 Deleting a sintering pin

1. Select sintering pin.
2. Call up context menu.

3. Select menu item [Delete selected].

14.3 Editing a sintering pin

You can change the settings for an existing sintering pin.
Editing can be done across the parts.

Call-up via the context menu or the menu [Edit]

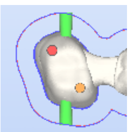
1. Select sintering pin.
2. Call up context menu or menu [Edit] > [Sintering pin].
3. Select the menu item [Edit sintering pins].

The selection window shows the settings for the selected sintering pin.

Edit	Activate settings for machining.
Angle	Taper angle of the sintering pin: Cylinder = 0 An angle of > 0° is required so that the sintering pins are not damaged during the milling process. The larger the angle, the larger the plane area on the part.
Diameter	Diameter at the boundary area. If the distance to the part is large, then the plane area on the part also becomes larger.

14.4 Moving a sintering pin

You can move an existing sintering pin to another position on the part or to another part in order to better adapt it to the shape of the part.

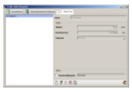


1. Select the sintering pin by double-clicking on it: The sintering pin is shown in orange.
2. Holding the left mouse button down, move the sintering pin to the desired position.
3. Finish the selection by clicking on the workspace or pressing [Esc].

14.5 Editing sintering pin settings

Call-up via the main menu

1. Call up the menu [Settings] and select the menu item [Building elements...].
2. Select the [Sintering pins] tab.



The selection window shows the list of sintering pin types, the settings for the selected sintering pin type and the toolbar for editing the settings: New, edit, apply, delete, copy.

Name	Name for the sintering pin type.
Angle	Taper angle of the sintering pin: Cylinder = 0
Diameter	Diameter at the boundary area.
Material	Displays the assigned material.
Default for	Determining entries as default for the displayed material.

14.6 Setting part sintering pin top plane

Especially where the surfaces of the blank are concave, we recommend that you determine a sintering pin boundary area for one or more parts.

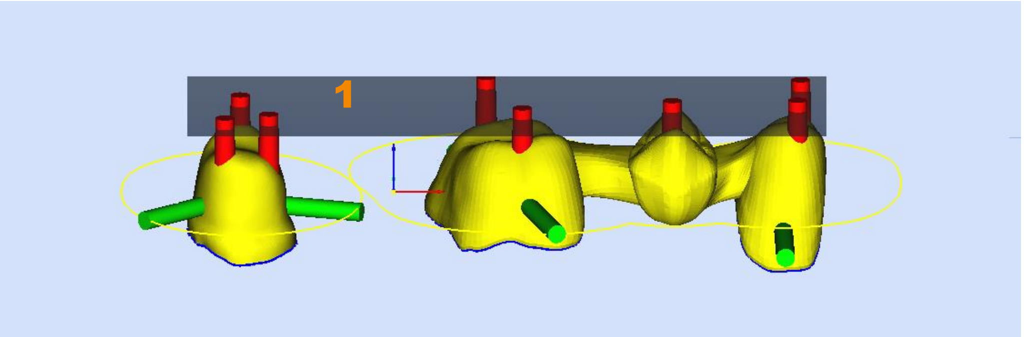


Figure 14-1

1. Select one or more parts.
2. Either call up the context menu or from the menu [Machining], select the menu item [Part] and menu item [Set part sintering pin top plane]:
   The boundary area (1) is shown in gray.
3. Using the mouse, drag the boundary area (1) to the desired position.

15 Calculating toolpaths

The toolpaths indicate the milling movements that the machining device must perform to mill the part, the dental restoration, out of the raw material.

The toolpaths are created based on details from the previous process steps and templates.

The icon is active if at least one part is selected for which all process steps have been completed. You can also select several parts – the function works for all parts that are selected.

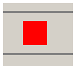
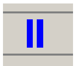
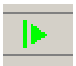
Click on the icon to start the calculation.

Once the calculation has started, the selected parts are locked for further machining, see icon in the part browser.

A message in the message window indicates when the calculation is finished. The part is locked and appears in the blank in gray or as an outline. The area in the blank is therefore considered to have been machined.

Depending on the selected settings, once the calculation has finished, hyperVIEW® is started, the NC file is created and the toolpaths are displayed.

The display screen shows the sequence of the calculations along with any possible error messages that may have occurred and the progress bar stating the job and the progress of the calculation as a percentage.

	Stop	Cancel calculation.
	Pause	Pause calculation.
	Resume	Resume calculation that has been paused.
	Exit hyperDENT® on successful finish of the calculation	Close hyperDENT® once the calculation has successfully finished.

The calculated data can be displayed in the machining directory with the file "blank.hv".

15.1 Notes concerning the calculation

- The type of margin line must be compatible with the part type, otherwise the calculation is not possible for safety reasons.
- If the direction selected or communicated for insertion or machining is incorrect to a significant degree, a warning message is displayed prior to the start of calculation.

15.2 Calculation Merge

In the case of fixtures with several parts, e.g. for prefabs, the part-specific toolpaths are determined in individual calculations in hyperDENT®.

With hyperDENT® Calculation Merge, you can merge the individual files with individual calculations into a joint file containing all of the calculations.

This joint file can then be converted in a postprocessor cycle in hyperVIEW® into the machine-specific NC file with an optimized tool change: --> “Create NC file, simulation (optional)”.

Load this NC file onto your machine and start the machining of all the blanks with an optimized tool change.

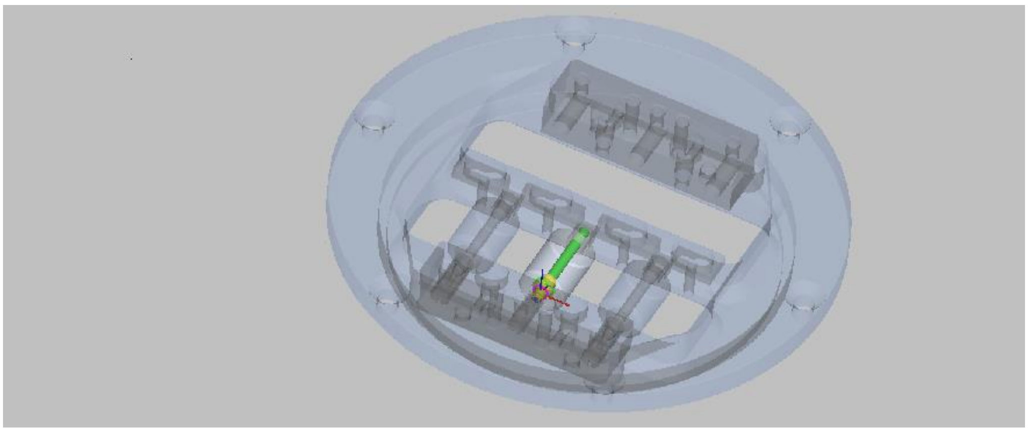


Figure 15-1 Holder for several blanks

Starting hyperDENT® Calculation Merge

Either

1. Call up the [Extras] menu and select the menu item [hyperDENT® Calculation Merge].

Or

1. After a calculation, call up the context menu and select the menu item [Part] > [Open with hyperDENT® Calculation Merge].

The selection window shows the management for the calculations, the list of calculations including the parts and tools, the merged parts and tools, the blank names, the preview for the fixture, and the function for updating the display and inserting new calculations.

Menu bar, menu items, icons

File	
Exit hyperDENT® Calculation Merge	
Edit	
Update	
Add calculation	
Open calculation output folder	
Show toolpaths	
Settings	
General	Call up submenu.
Miscellaneous	Call up area.
Filter calculations	Enable/disable filters for calculation time.
Filter after X hours	Specification of number of hours for filter.
Help	Call up help and information.

Display for parts and calculations

List with the calculations and specifications about the part, administrative	data, blank type, fixture (holder), and details about the selected part and its calculation.
Tick	Selection of the part for the merging of calculations.

- Caution!
  Select only those parts for merging that will fit in the joint holder and will not be superimposed on each other.
- Check the selection in the preview for the merge.

Preview	Preview for the selected part in the list.
Part	Name and symbol of the selected part.
Tools	Symbol, number, name of the tools of the selected part.

Display for merging

Merge preview

Preview of all parts that are selected (checked) in the list for merging.

- Caution!
  The selected parts (2, 3, 4) must fit the joint holder (1) and must not be superimposed on each other (5, 7).

Example

Permissible selection
Merging is possible.

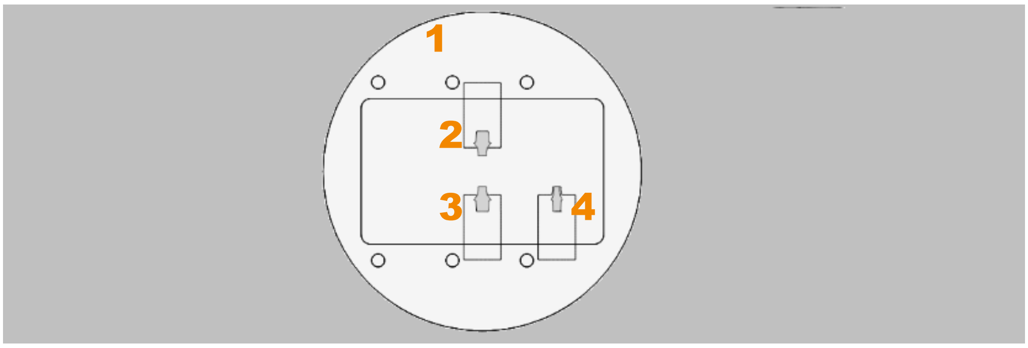


Figure 15-2 Preview of merge

Example

Impermissible selection
Merging is not possible.
The parts are superimposed on each other (6) or are calculated for different holders (5, 6, 7, 8).

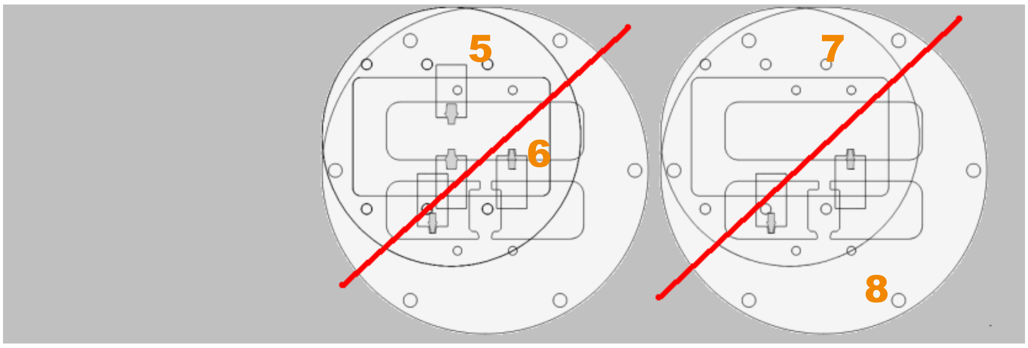


Figure 15-3 Preview of merge, selection impermissible

Combined parts

List of the combined parts.

Combined tools	List of the tools in the combined parts. For machining, these tools must be available on the machine.
Blank name	Name of the blank.
Start merge	
Output file	Output folder with the newly created file of the combined parts.

To make sure that the calculation merge can work, “**Calculate in NC coordinates**” must be set to **YES**. This must be set for every holder that you want to use with this function. This function can be invoked in Settings -> Edit -> Fixture -> Additional properties.

16 Creating NC file, simulation (optional)

In hyperVIEW®, the part-specific toolpaths from hyperDENT® are converted into a postprocessor cycle in the machine-specific NC file.

Load this NC file onto your machine and start the editing process.

You can also simulate the toolpaths in hyperVIEW® (optional).

The hyperVIEW® program starts automatically once the calculation has finished. The data is loaded automatically.
You can manually start hyperVIEW® by selecting the menu item [Extras] > [hyperVIEW®] or the context menu [Part] > [Show toolpaths].

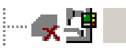
A detailed description of hyperVIEW® can be found via the menu item hyperVIEW® Help (F1), see: hyperVIEW®, Important processes, Run postprocessor cycle.

The key steps are detailed in the following description.

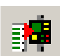
16.1 Postprocessor cycle, creating an NC file

The display window for the [NC jobs] tab shows the details and possible selections for the postprocessor cycle.

1. Select the NC jobs tab.
   The toolpaths and fixtures are set correctly by default in the [NC jobs] tab.



2. Select machine.



3. Create NC file:
   Click on the [Write NC files] icon at the top of the toolbar.
   This opens the window with the tools.
4. Check the tool entries in the window.
5. Start creation:
   Click on OK.

The NC file is stored in the file directory according to the predetermined path.

6. Load file onto the machining device.

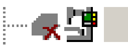
16.2 Simulation (optional)

The simulation is used to display the toolpaths with the tool, blank, fixture, and machine model.

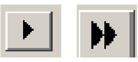
The simulation is run using the toolpaths that have been calculated in hyperDENT®.

The display window for the [Simulation] tab shows the setting options for the simulation.

1. Under the [NC jobs] tab, hide the fixture:
   Click on the lights icon in front of the [Fixture] field.



2. Select the [Simulation] tab.



3. Start simulation:
   Click on the icon in the recorder bar.
   > = Step by step
   >> = Continuous
4. Set the run rate using the slider.

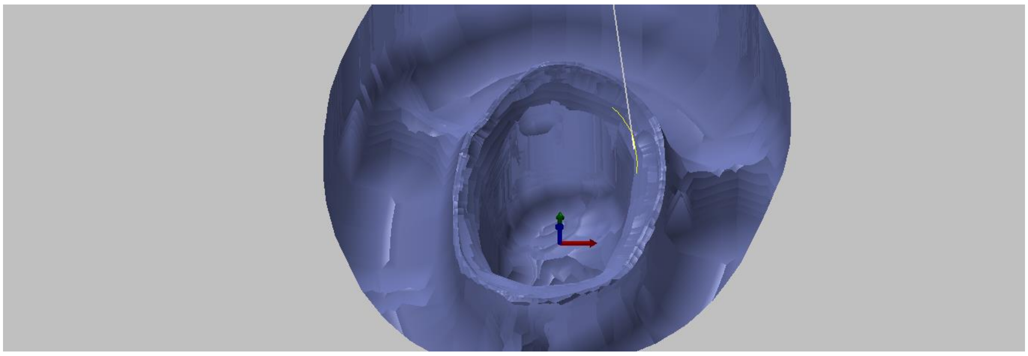


Figure 16-1

17 Settings

17.1 Machining

Settings for the machine and fixtures that you can select in the process step [Select milling unit] --> “Select milling unit”.

17.1.1 Configure machine

The machine determines the process for calculating the toolpaths. By using hyperDENT®, you can configure machines in such a way that axis limitations are already checked by hyperDENT® during the calculation.

You can therefore set different combinations of the rotation axes:

- *Pure table kinematics*
  All rotation axes are in the table (e.g. C-table on A-bridge).
- *Pure head kinematics*
  All rotation axes are in the head.
- *Mixed kinematics*
  One rotation axis is in the table and the other rotation axis is in the head.

Primary and secondary axis

In order to set the correct axis, you must determine the primary and secondary axis:

- *Pure table kinematics and pure head kinematics*
  The primary axis is always the axis which “carries” the other axis if there is a rotation. Therefore, if the primary axis is rotated, then the position of the secondary axis always changes too.
- *Mixed kinematics*
  The primary axis is always the table axis.
  The secondary axis is the head axis.
- *You can select A (rotation around X), B (rotation around Y), or C (rotation around Z)*
  as the primary axis.
- *Depending on which primary axis you choose, the secondary axis can either be an A or C (primary = B), a B or C (primary = A) or an A or B (primary = C).*

- *The direction of rotation for the axes is always indicated in a mathematically positive sense, i.e. counterclockwise.*
- *The designation of the axes corresponds to the “right-hand rule”.*

Right-hand rule

The thumb, index finger, and middle finger of your right hand define the coordinate system.

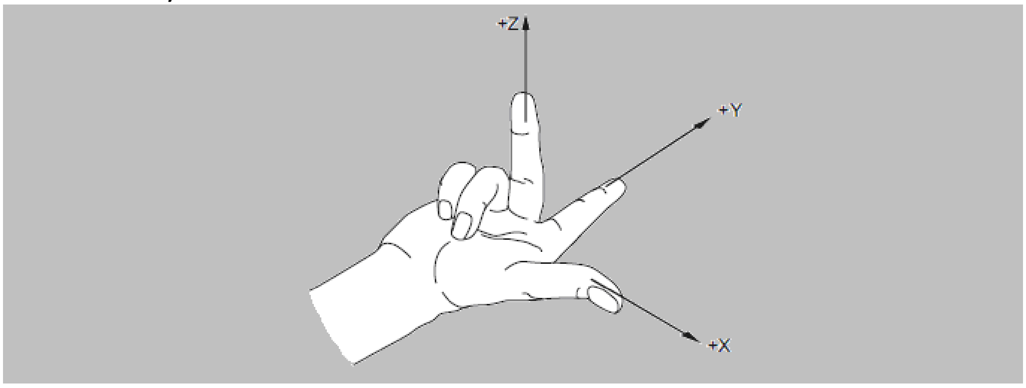


Figure 17-1

Rotational direction

The mathematically positive direction of rotation is obtained by (hypothetically) placing your hand around the desired axis so that your thumb points in the positive direction of the axis. Your other fingers indicate the positive direction of rotation.

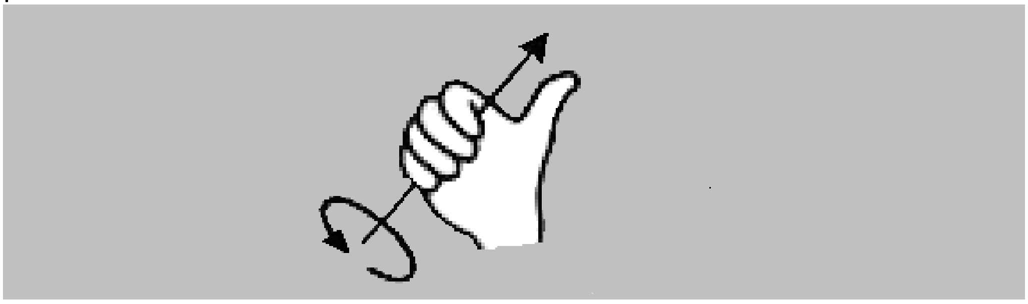


Figure 17-2

Relative tool movement

The entries for the rotational directions of the axes in a machine context are often very different, usually from the perspective of the movement, which the tool performs in relation to the piece.

Example of rotational directions in hyperDENT®

Rotational directions that are identified in hyperDENT® as positive or negative.

Example 1 Rotation around X (= red axis) in positive direction

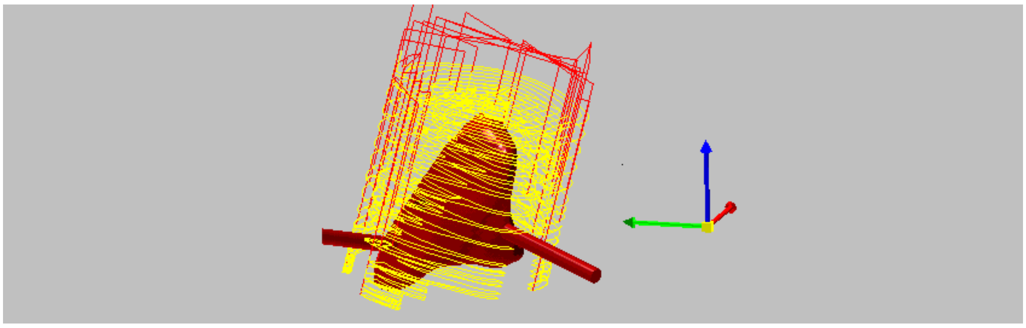


Figure 17-3

Example 2 Rotation around Y (= green axis) in positive direction

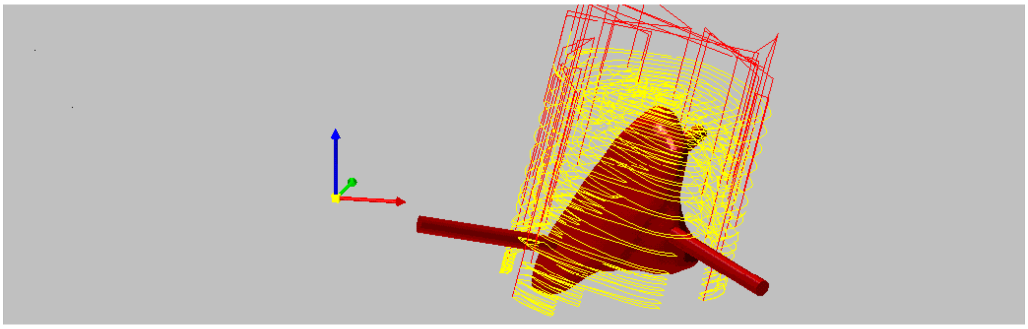


Figure 17-4

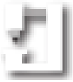
If the parameters for the rotational directions are defined in the menu, then it is important to take into account the rotational directions (mathematically positive) specified by hyperDENT®.

Example 3 Variant rotational direction

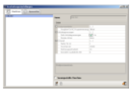
- *The machine does not rotate in the mathematically positive sense, but against the mathematically positive sense.*
- *The machine has a swiveling range of -10 to +120 in the A-axis.*

Since the rotational directions of the machine do not match the directions of hyperDENT®, the signs must be inverted:
i.e. the minimum angle is –120 and the maximum angle +10.

17.1.2 Machine



Select milling machine (postprocessor), set options for postprocessor, enter axis boundary, enter values for calculating milling times.



The selection window shows the [Machines] tab with the list of machines, the corresponding settings, and the machining options: Edit, Apply, Exit.

Machines	List of available machines DIN ISO = standard machine.
Name	Name for the machining device.

Postprocessing

Defaults for NC programs	
Global	Same for all machines, according to the general settings.
Individual	Separate for each machine, with the following machine-specific settings.
Postprocessing strategy	Standard Postprocessor type: Standard or user-specific (optional).
Post-processing mode	
	Settings for pulling up the postprocessor:
Start hyperVIEW®.	
Start hyperVIEW® and proceed after prompt.	
Immediate postprocessing (run process in the background).	
Tool-specific output	No
No	Output for all tools.
Yes	Tool-specific output.
PP Output directory	Settings for the directory for the NC file.
hyperDENT® calculation output directory	Same directory as for the hyperDENT® calculations.

hyperVIEW® configuration	Directory according to the hyperVIEW® configuration.
Fixed directory	Specify path name for the NC file.
Fixed directory	Path name: Drive, directory for the NC file.
Create subdirectory	
Yes/No	Create subdirectory for the NC file in the output directory.
PP Output file name	Settings for the name for the NC file:
hyperVIEW® configuration	Name according to the hyperVIEW® configuration.
Fixed name	Specify name for the NC file.
NC file name	Parameter for setting the name for the NC file: [STOCKNAME] (Blank name) [MODELNAME] (Name of the part) [TIMESTAMP] (Date, time) [FIXTURENAME] (Name of the blank holder) [MACHINENAME] (Name of the machine) [MATERIALNAME] (Name of the material) [BLANKTYPENAME] (Name of the blank type) [BLANKEXTERNALID] (External blank ID) [BLANKCHARGE] (Charge of the blank) [DATE] (Date) [TIME] (Time) [YYYY] (Specification of year, 4-digit) [YY] (Specification of year, 2-digit) [MM] (Month of the year) [DD] (Day of the month) [hh] (Hour of the day) [mm] (Minutes of the hour) [ss] (Seconds of the minute)

Create info file	Create file with program information. If the project file has already been saved, a reference to the project file is provided in the NC info file.

Axis limitations

By using hyperDENT®, you can configure machines in such a way --> "Settings" > "Machining" > "Configure machine", that axis limitations are already checked by hyperDENT® during the calculation.
To do so, specify the rotation axis and rotational direction according to the machine to be used.

Max. tilt angle for parts	
	This value corresponds to the maximum angle of the axis with the lowest possible deflection. The system attempts to tilt the part in such a way that the required height is achieved. The part height and the part-specific insertion directions are taken into account. In 5X simultaneous machining, you must make sure that sufficient axis leeway is still available for this machining. E.g. If the lowest axis can register 20 degrees and the system works with a 5-degree axis deflection in 5X machining, the max. tilt angle for parts may be 15 degrees at most.
Check axis limitations	
Yes	Check the axes during the calculation to see if they exceed the axis limitations.
No	No checks during the calculation.
Primary axis	
None	No rotation axis.
A	Machine has an A-axis, i.e. rotation around X.
B	Machine has a B-axis, i.e. rotation around Y.
Rotation direction	
Positive/Negative	Rotation direction of the axis.

Limitations	Scope of the limitation.
Unlimited	The axis has no angle limitation.
One area	The axis has an area to which it is limited.
Two areas	The axis has two areas to which it is limited.
Minimum angle 1	Smallest angle in area 1.
Maximum angle 1	Largest angle in area 1.
Minimum angle 2	Smallest angle in area 2.
Maximum angle 2	Largest angle in area 2.
Secondary axis	Additional, 2nd rotation axis:
None	No secondary rotation axis.
A	The machine also has an A-axis, i.e. rotation around X.
B	The machine also has a B-axis, i.e. rotation around Y.
C	The machine also has a C-axis, i.e. rotation around Z.
Rotation direction	
Positive/Negative	Rotation direction of the axis.
Limitations	Scope of the limitation.
Unlimited	The axis has no angle limitation.
One area	The axis has an area to which it is limited.
Two areas	The axis has two areas to which it is limited.
Minimum angle 1	Smallest angle in area 1.
Maximum angle 1	Largest angle in area 1.
Minimum angle 2	Smallest angle in area 2.
Maximum angle 2	Largest angle in area 2.

Milling time

Details for calculating the machining time.

Factor G1	1 - 2 Time factor for the milling time with feedrate.

1	Milling time for G1 calculated according to defined feedrate.	
>1 - 2	Since the actual feedrate also depends on the machine dynamics and number of axes moved, it may be smaller, which results in longer machining times. Use the factor (practical value) to take into account the deviation when calculating the machining time.	
Feedrate G0	Rapid traverse rate for calculating the time.	mm/min
Tool change time for tool change	Time for the tool change.	s
Constant additional times	Additional time that is added to the calculated machining time, e.g. for tool change.	
Default machine	Use selected machine as default.	

17.1.3 Fixture

Select the fixture, holder.

The selection window shows the [Fixture] tab with the list of available fixtures.

Fixture	List of available fixtures.

Filtering

Name	Name of fixture.
Machines	Allocation to all or only specific machines.
Blank geometry	Allocation to all or only specific blank geometries.
Default for	Use selected fixture as a default for all or only specific combinations from the allocations for machines and blank geometries.

Additional properties

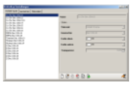
Part inclination	Setting of the preferred tilt axis for the automatic part tilt when loading the part. The maximum tilt angle can be set under [Settings > Machining > Machines > Axis limitations]. To be able to use the automatic Tilt part function, this setting must be made under [Settings > General > Load wizard].
Tilt axes	
Only X-axis	Tilt only on X-axis, suitable for 4-axis machines
Prefer X-axis	Tilt on both axes, X-axis preferred
Only Y-axis	Tilt only on X-axis, suitable for 4-axis machines
Prefer Y-axis	Tilt on both axes, Y-axis preferred
Any	Tilt on both axes without preference
Calculate in NC coordinates	
Yes	Perform calculation with identical NC coordinate system Necessary for processing the NC data with hyperDENT® Calculation Merge.
No	Use the present coordinate system. It is not possible to merge differing NC data with hyperDENT® Calculation Merge.
Rotation axis	Preferred rotation axis for the display on the screen.
Object nesting	Settings for the automatic placement of the parts in blank (autonesting). Further details --> “Load part” > “Place part automatically – autonesting”.
Use automatic nesting	
Yes	Load part and place it automatically.
No	Load part. The part needs to be placed manually.
Choose nesting direction	Arrangement of the parts according to the placement sequence.
From inside to outside	
From outside to inside	

	From left to right	
	From right to left	
	From down to top	
	From top to down	
Position of 1st part / zero point		The position depends on the selected nesting direction. Useful with block materials.
	Top left	
	Top middle	
	Top right	
	Middle left	
	Middle right	
	Low left	
	Low middle	
	Low right	
Performance		Selection of the placement mode according to performance criteria:
	Super speed – low precise	
	High speed – medium precise	
	Medium speed – high precise	
	Low speed – very high precise	
Allow overlapping milling boundaries		Specification about milling boundary.
	Yes	Milling boundaries overlap, saving material.
	No	No overlap of the milling boundaries.
Connectors adding to part		
	Yes	Include connectors in the calculation.
	No	Do not include connectors in the calculation.
Screwchannel adding to part		
	Yes	Include screw channel in the calculation.
	No	Do not include screw channel in the calculation.
Connectors offset [mm]		

17.2 Blanks

Create and administer blank types, blank administration settings

17.2.1 Blank types



Here, you can define the blank types from which you can load the blanks for the machining. Further details --> “Load blanks” > “Create, edit blank type”.

Blanks for machining can only be loaded from the blank types that are defined here.

17.2.2 Blank administration settings

Use the blank administration settings to influence the function for naming blanks and to set the display in the selection windows [New blank], [Load blank], and [Load project].

Here, you can show or hide the columns displayed in the table and the options for filtering the blanks.

1. Call up Blank administration settings:
   In the menu [Settings], call up the menu item [Blanks] > [Blank administration settings…].
   The window [Blank administration] with the [New blanks] and [Saved blanks] tabs and the selection boxes for the display, is displayed.

17.2.3 Blank administration – New blanks

The window shows the settings for displaying new blanks.

Filter blank type selection	Enable/disable the filter function for blank type. Enable/disable filter options.
Use environmental filters	Enable/disable filter option. Yes – Already available information for the selection of blanks is used. Only possible blank types are shown in the list. Function can be activated in the selection window.
Display blank type data	Enable/disable display options for new blanks.
Automatic naming parameters	Determine the details for the automatic creation of blank names.
Indexing	Use automatic number (default).

Date	Enable/disable “Also use current date”.
Blank type	Enable/disable “Also use”.
Material	Enable/disable “Also use”.
Geometry	Enable/disable “Also use”.
Blank identification parameters	Details for identifying the blank.
Name	(Default)
External ID	Enable/disable “Also use”.
Charge number	Enable/disable “Also use”.

17.2.4 Blank administration – Saved blanks

The window shows the settings for displaying existing and saved blanks.

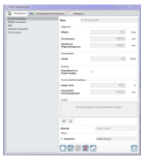
Activate blank administration	Enable/disable the call-up of the blank administration via the selection window [Load blank].
Activated	Select blank via selection window [Load blank].
Deactivated	Select blank via selection window [New blank].
Display used blank data	Enable/disable columns in the table for blanks already used.
Project preview, type	Screenshot Sketch
Image preview	
Image size	10-500
Blank type	
Material	
Color	
Height	

Machine	
Fixture	
Filter blank type selection	Enable/disable the filter function for blank type. Enable/disable filter options.
Blank type	
Material	
Color	
Height	
Name	
Show empty blanks	Yes – Empty blanks appear on the list No – Empty blanks do not appear on the list Only – Only empty blanks are shown
Show individual blocks	
	Enable/disable filter option.
Display locked blanks Yes No –	If blanks from a different user are being machined, they are not shown in the list

17.3 Building elements

Define default settings for connectors, screw channel connectors, and sintering pins.

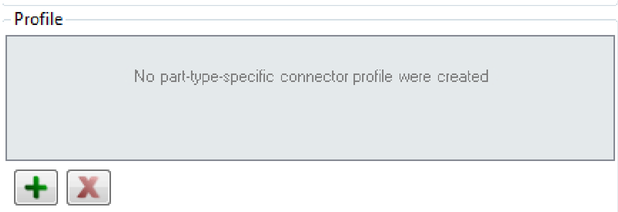
Connectors



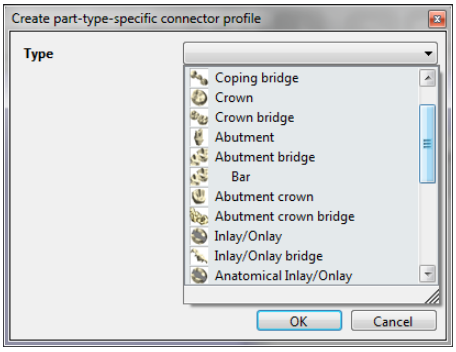
Create, delete, copy connector types and edit the settings for the selected connector type. Further details --> “Set connectors” > “Edit connector settings” > “Connector settings”.

To make it possible to analyze the various requirements of the part types better, it is possible to create part-type-specific connector profiles.

To add, select and edit the basic profile. Add with “+”



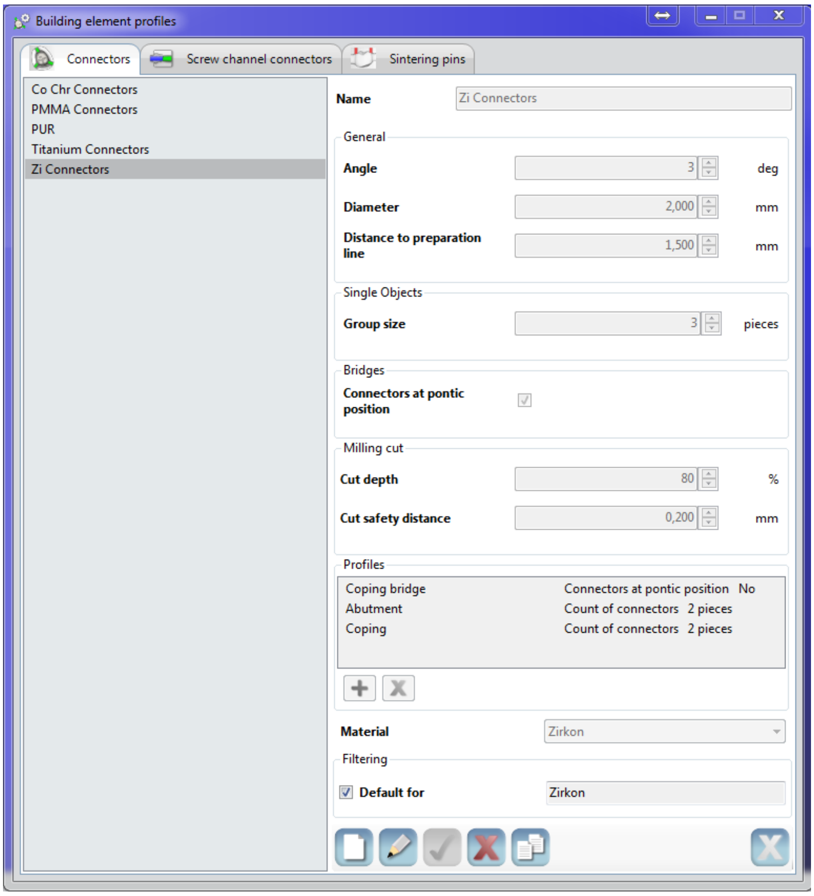
Select part type



Create part-type-specific connector profiles

Type	
Angle	
Diameter	
Connectors below emergence profile	
Connectors at pontic position	
Arrangement strategy	Count of connectors Distance between connectors

Confirm input



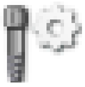
Screw channel connectors

Create, delete, copy screw channel connector types and edit the settings for the selected connector type. Further details --> "Set connectors" > "Edit connector settings" > "Connector settings".

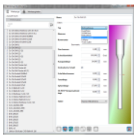
Sintering pins

Create, delete, copy sintering pin types and edit the settings for the selected sintering pin type. Further details --> "Set the sintering pins" > "Edit sintering pin settings".

17.4 Tools



Create and manage tools and tool holder.



List of tools and tool holders that you can select for machining.

Show for Tool type	Selection filters for the tools.

17.4.1 Entering tool data (optional)

The tool data describes the tool that is to be used to perform the relevant operation on the milling unit and is needed to calculate the toolpaths, machine movements, and collision check.

The following tools can be configured:

- *Ballmill, endmill, bullnose, drill tools.*
- *Tapered tools, tools with a thick shank.*
- *Lollipop cutter and T-slot cutter*

Tools can only be saved if a tool holder has been assigned.

1. Click on the [Tools] tab.
   The selection window shows a list of tools, the settings of the selected tool, and the machining functions: Create new tool, edit, use, delete, copy, export, print it.

Name	Name for the tool.
Type	Ballmill, endmill, bullnose, drill tool, lollipop, and T-slot cutter
Number	Unique number for the tool.

- *Caution!*
  A number can be allocated several times: Risk of mix-up, incorrect machining, breakage of tool, damage to machine.
  Make sure that you only use a tool number once within a project.

Comment	Comment about the tool.

Geometry

	Details of the tool shape.
Diameter	Nominal diameter of the tool at the cutting edge.
Collar diameter	Diameter of the shank above the ball/blade.
Blade height	Height of the blade on T-slot cutters.
Corner radius	Radius of the tool blade.
Corner radius shaft side	Radius of the blade at the top on T-slot cutters.
Length	Length from tool holder.
Tapered	Tapered tool shape.
Thick shank	Tools with a larger shank diameter.
Cone angle	Angle for tapered tools.
Shank diameter	Diameter of tool shank.
Chamfer length	Length of crossover to the tool shank.
Tip length	Length of the cutting area with the nominal diameter of the tool.
5X compensation length	Distance (pivot length) from the rotation axis (pivot point) of the milling head to the tool tip. Essential accurate details for the 5X-machining on machines with no RTCP, so that the rotation point is moved to the tool tip.

RTCP – Rotation Tool Center Point

In special cases, this function is also used for 5X-machining on machines with no RTCP.

The postprocessor converts the NC data through the necessary compensation movements in the X, Y, and Z-axis.

For this, you must enter the exact and reproducible distance between the rotation axis (1) of the milling head (pivot point) and the tool tip (2): the compensation length (pivot length).

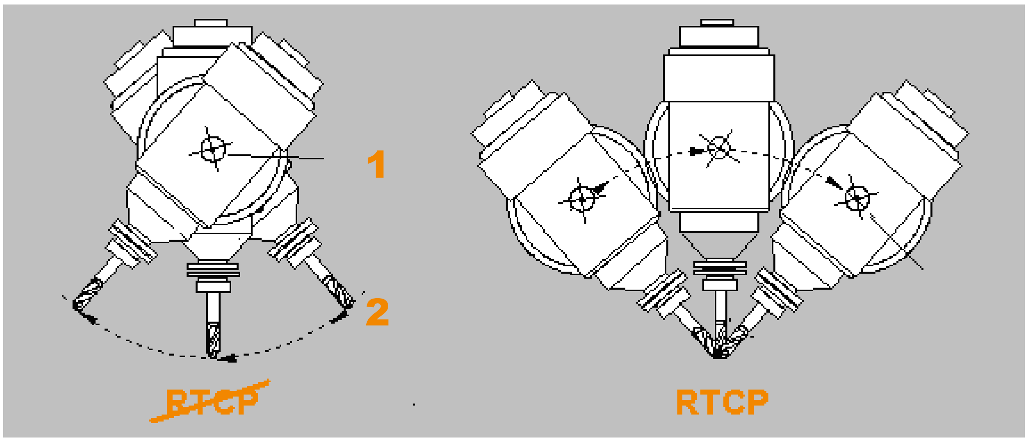


Figure 17-5

Holder	Designation of the holder.

17.4.2 Entering the tool holder (optional)

The data for the tool holder describes the geometry of the fixture device for the tool and is needed for the collision check.

The tool holders can be configured via the parameters, you cannot however currently enter a free geometry.

Tools can only be saved if a tool holder has been assigned.

1. Click on the [Tool holders] tab.
   The selection window shows a list of tool holders, the settings of the selected tool holder, and the machining functions: Create new tool holder, edit, use, delete, copy, export, print it.

Name	Name of the tool holder.
Data	Details about the shape of the tool holder.

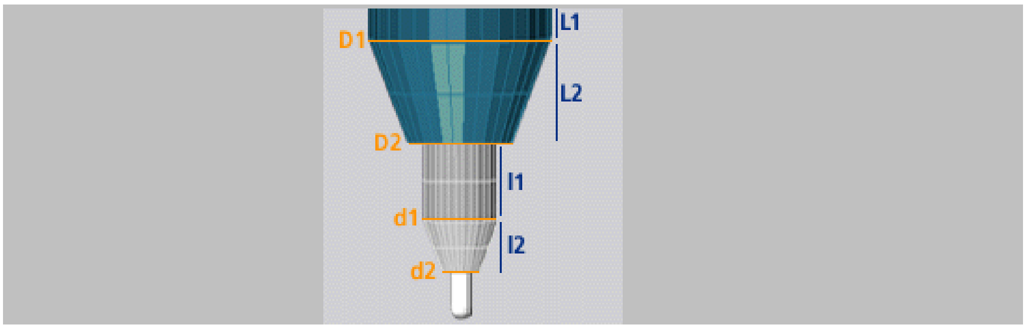
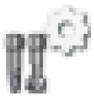


Figure 17-6

L1	Head length 1
D1	Head diameter 1
D2	Head diameter 2
L2	Head length 2
l1	Length 1
d1	Diameter 1
d2	Diameter 2
l2	Length 2

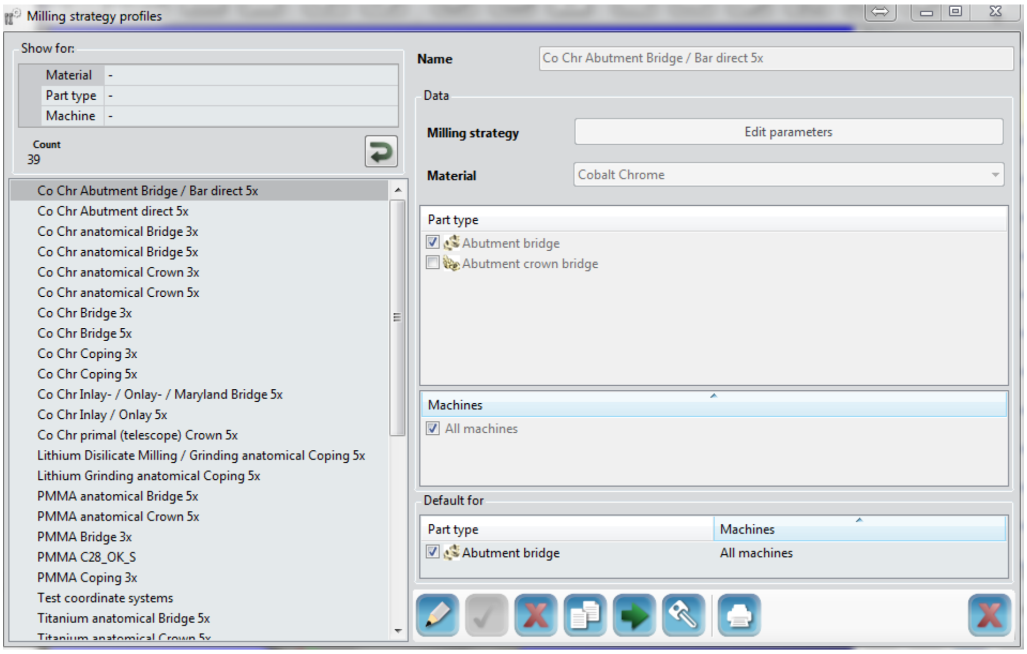
17.5 Milling strategies



List from which you can select the milling strategy (template) for the machining.

Template generator module option: Create and edit template, other details --> “Milling strategy” > “Edit template”.

Show for	You can also filter the templates by material, part type, and machine.
Data	
Milling strategy	Edit parameters
Material	Dropdown menu for the selection of the relevant material
Machines	Select the machine(s) for which you want to use this template
Default for	Selection option to allocate specific templates to the machines and/or part types



Multiple selection of templates is possible, to allocation material and machine.

17.6 Parts

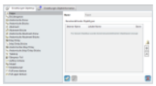
Create and manage part types and part information.

The parts, the dental restorations, are classified according to their specific features into part types, which are then allocated to the corresponding, suitable templates.

17.6.1 Creating and editing user-defined part types

You can derive your own, user-defined part types from the existing part types, and then save them under their own name. This new part type is then available in hyperDENT® for all other steps.

The new part type possesses the specific properties of the original part type, but may also be given, for example, its own, specially adapted templates (template generator module option).



1. Click on the [Part type settings] tab:
   The selection window shows the available part types, the related, user-defined part types, and the machining functions: edit, add, delete.

List	List of the available, original part types.
Name	Name of the selected part type.
User-defined part types	List of the user-defined part types created from the selected part type.
Internal name	
Local name	
Icon	Icon of the user-defined part type.

Adding a user-defined part type

2. Select the part type and call up the [Edit] function:
   Click on the part type and then click on [Edit].

3. Call up the [Add] function:
   The input window is displayed.
4. Enter the name for the part type.
5. If necessary, select its own icon.
   Select an icon from the file system; otherwise, the icon of the original part type will be used.

6. Confirm your input or cancel the process.

Editing a user-defined part type

7. Double-click on the name or the icon of the user-defined part type and change the name of select a new icon.

8. Confirm your input or cancel the process.

Deleting a user-defined part type

9. Click on the user-defined part type and delete it:
   Click on the part type and then click on [Delete].
10. Confirm the security question.

11. Confirm your input or cancel the process.

17.6.2 Part information

Determine name and designation for additional part information. These are displayed in the part data. Here, you can enter part-specific details.

The selection window shows the list of part information and the input and machining functions: new, edit, apply, delete, copy.

Name	Name of the part information in the list.
Designation	Designation of the part data display.
Hide	Hide display.
Mandatory	???

17.7 Importing database objects

Different settings and part types are saved by hyperDENT® in configuration databases and can be transferred between these databases using the export and import functions.

Material

Blank geometry

Blank type

Tool holder

Tool

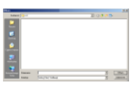
Templates/milling strategies

Export

1. Select the menu item or icon in the relevant windows:
   All currently selected parts and their referenced parts are copied to the export file.

Import

1. Select the menu item [Settings] > [Import database objects]:



The selection window shows the selection option for drive, directory, and export file along with the data to be imported.

2. Import file:
   Double-click on the file.
   The file is loaded.

The data is checked for identical parts (same internal ID).

3. If necessary, select [Overwrite existing objects?]:

Yes	If the internal ID is identical, the already existing parts are replaced with the imported objects.
No	Already existing parts are retained.

4. If necessary, select [Copy existing modified objects?]:

Yes	If the internal ID is identical, the imported objects are imported as a copy with a new internal ID. If the names of the parts are identical: Supplement the name of the imported object with an index, e.g.: [3x-Coping-CoChr] --> [3x-Coping-CoChr (2)].
No	Do not import parts with an identical internal ID.

Once imported, a window shows the imported objects and the changes to the database.

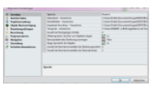
During the import process, the imported objects are given a new internal ID that differs from the previous ID. As a result, hyperDENT® cannot detect that a previously imported object is being imported again (multiple times).

17.8 General

The general settings determine the program behavior and display. The settings are subdivided thematically into several areas.

Path names can be changed optionally.

1. Call up General settings:
   Select [Settings] menu and call up the [General] menu item.
2. Call up the area:
   Click on the area in the left column.
3. Change the value:
   Click on the right column next to the parameter and enter the value, select the path or select the entry via the selection menu.



The selection window displays the list with the areas and the list with the parameters and values for the selected area.

OK Save changes, exit menu.

Cancel	Do not save, exit menu.

Changing the column width

1. Click on the right-hand boundary line of the column, hold the mouse button down, and set the width.

Or

1. Double-click on the right-hand boundary line of the column:
   The column width is set to the predefined width or the maximum width of the content or the heading.

17.8.1 Miscellaneous

Language	Select
Database directory	Path name for the directory with the databases for blanks, tools, and templates that are individually adapted to your machines.

- *If a change is made, hyperDENT® must be restarted.*
- *Notes concerning data backup*
  Save your individual settings: Regularly create a backup copy from the database directory onto another data carrier.

Fixtures directory	Path name for the directory with the fixtures.

- *If a change is made, hyperDENT® must be restarted.*

Implant interfaces directory	Path name for the directory with the interface geometries for the implants.
Temporary directory	Path name for temporary files.
Number of undo steps	Number of steps that can be undone.
Show question on deleting object	
Yes	Prompt to confirm deletion process.
No	Delete without further inquiry.

Force user scaling input	
Yes	Input of scaling required when loading a new blank.
No	Adoption of the predefined scaling for the stored material.
Show tooltip information for objects	
	Display information window.
No. of digits in material scaling values	
	Digits (“4”) for scaling values.
No. of digits in general decimal values	
	Digits (“3”) for other decimal values.

17.8.2 Loading wizard

Use load wizard Yes / No	Use wizard.
Adjust direction	Yes No Force – Always appears, even when CAD data is available.
Identify parts	Yes No Force – Always appears, even when CAD data is available.
Tilt part	Yes No Automatic – For using the automatic part tilt when loading parts.
Select template	
Nesting mode	
Set connectors	
Yes / No	Preselection for the individual steps that the wizard is to perform.

Force	May be necessary if, when the part is loaded via a defined interface, the desired part information (e.g. the pontic position) has not been supplied as well.

17.8.3 Project management

Settings for the project management which are important for calling up and saving operations.

The data needed for the management is saved on the hard drive in one or more directories.
The directories are created relative to where the administration directory is stored.
For example, once the loaded files and administration directory have been stored on an external hard drive, if you want to be able to access the full information, we recommend that you create an administration directory on each partition on which the project files can also be stored.

Automatically generate name for projects	
Off	No automatic saving.
hyperDENT® work directory	The project files are saved in the work directory of hyperDENT®.
Fixed directory	The project files are saved in the specified directory.
Fixed directory	Path name for the project files.
Save project automatically	
Never	No automatic saving.
After calculation	Save project files after calculation.
Before and after Calculation	Save project files before and after calculation.
Create project info file	
	Info file for workflow management systems (V7, SAP).

Activate blank administration

Yes	Use blank administration, select blanks from the selection window [Load blank].
No	Disable blank administration, select blanks from the selection window [New blank] or from the file system.

No. of project manager folders	1 - 3

Project manager directory

File path for the directory with the data for the project management.
Initial directory to which the other directories of the project management relate.

- *If a change is made, hyperDENT® must be restarted.*

17.8.4 Part tracking

Settings for part tracking to determine which part has been placed and machined in which blank and project.
The data needed for the management is saved on the hard drive in one or more directories.
The directories are created relative to where the administration directory is stored.
For example, once the loaded files and administration directory have been stored on an external hard drive, if you want to be able to access the full information, we recommend that you create an administration directory on each partition on which the project files can also be stored.

Use part tracking

Yes	Work with tracking, save assignment of parts to project and blank.

No. of part manager folders	1 - 3

Part manager directory	File path for the directory with the data for the part management. Initial directory to which the other directories of the part management relate.

- *If a change is made, hyperDENT® must be restarted.*

Use part inqueue	For the direct CAD connection (CAD Connect), activate the list for new parts that are still to be machined.

17.8.5 Consistency checks

Settings for messages in the message window for different processes when parts are placed (nesting):
no check/warning/error = stop message.

Part outside blank	
Parts overlap	
Part outside the fixture boundary	
Security distance to part	Security allowance so that parts located close together are not damaged if the blank is not in exactly the same position when it is reclamped in the machine.
Security distance to fixture	
Milling boundary outside blank	
Milling boundary outside fixture boundary	
Connector ends outside blank	
Milling boundary cuts other parts connector	
Connectors exist	
Screwchannel outlet opening ends outside blank	
Millboundary cuts other parts screwchannel outlet opening	

17.8.6 Calculation

Settings for the calculation of the project data.

Calculation output directory	
	Path name for all of the calculation data. The file “blank.hv” contains all relevant data. If hyperView® does not start automatically after the calculation, then the file “blank.hv” must be imported into hyperView® for the postprocessor cycle: Drag the file to the program window of hyperView® using the drag & drop function. For additional editing, see chapter “Create NC file, simulation”.
Max. number of parallel calculations	
	The calculation time can be reduced considerably with parallel calculations, depending on the hardware. The number of parallel calculations should be slightly higher than the number of available computation cores of the processor.
Use 64-bit cycles	These cycles should be used on 64-bit systems to shorten the calculation time.
Yes	64-bit cycles are used for the milling track calculation.
No	32-bit cycles are used for the milling track calculation.
Force calculation for one part only	
Yes	The calculation is only possible for a single selected part. Calculation is not possible for multiple selection.
No	The calculation is also possible for multiple selections and is performed for all selected parts.
Tool change optimization Yes	
	Process all parts that have been calculated together, for a quicker and more economical milling process with optimized toolchange, i.e. first of all process all milling paths across all parts using tool “A”, then tool “B”, etc.
Preserve temporary data No	
View cycle parameters	No

Check toolpath points against boundary	Yes
Yes	The toolpaths in the feedrate must lie within the milling boundary. Movements in rapid traverse are not checked.
No	No check; the toolpaths may exceed the milling boundary. If the parts are placed very closely, the adjacent parts may be damaged.
Stop all calculations on error	
Yes	Stop the calculation, do not transfer any data to the machine.
No	Stop the calculation for the part where the error occurred; continue calculating the other parts and release for milling.
Delete calculation data automatically	No
No	Retain data once the calculation has finished.
Yes	Delete data once the calculation has finished.
Delete after X hours	Number of hours The directory is deleted after the specified number of hours.
Also delete NC folder	Yes
Yes	The content of the NC output directory is deleted.

17.8.7 Postprocessing

Settings for creating the NC file with hyperVIEW®.
In order to use the automatic postprocessing, the calculation output directory and temporary directory should not have a deep directory structure, e.g.

c:/hyperDent	Calculation output directory
c:/tmp	Temporary directory

Postprocessing strategy	Standard Postprocessor type: Standard or user-specific (optional).
Post-processing mode	Settings for pulling up the postprocessor:

Start hyperVIEW®	
Start hyperVIEW® and proceed after prompt.	
Immediate postprocessing	Run process in the background.
Tool-specific output	No
No	Output for all tools.
Yes	Tool-specific output of an NC file after each tool change.
PP Output directory	
hyperDENT® calculation output directory	NC files are written in the calculation output directory in the subfolder NC.
hyperVIEW® configuration	The NC files are written machine-specifically in the output directory configured in hyperVIEW®.
Fixed directory	The NC files are written to the specified directory in the subfolder NC.
Fixed directory	Path name: Drive, directory for the NC file.
Create subdirectory	
Yes	Create subdirectory for the NC file in the output directory.
PP output file name	Name for the NC file Specify or create according to program-specific configuration.
hyperVIEW® configuration	The NC files are created machine-specifically using the settings configured in hyperVIEW®.
Fixed file name	An NC file (job mode not possible) is created for each calculation according to the default [NC file name].
NC file name	Default and parameter for creating the “Fixed name” for the NC file. Parameters can be used. Other characters can be inserted between the parameters.

[STOCKNAME]	Parameter for name of the blank.
[Modelname]	Parameter for model name of the first part.
[NCIDX:1]	Parameter for counter with the starting value (1) for the consecutive counting of the NC files, e.g.: "Crown_1.nc", "Crown_2.nc", "Crown_3.nc".
[TIMESTAMP]	Parameter for start time of the calculation (date, time).

[STOCKNAME]	(Blank name)
[MODELNAME]	(Name of the part)
[TIMESTAMP]	(Date, time)
[FIXTURENAME]	(Name of the blank holder)
[MACHINENAME]	(Name of the machine)
[MATERIALNAME]	(Name of the material)
[BLANKTYPENAME]	(Name of the blank type)
[BLANKEXTERNALID]	(External blank ID)
[BLANKCHARGE]	(Charge of the blank)
[DATE]	(Date)
[TIME]	(Time)
[YYYY]	(Specification of year, 4-digit)
[YY]	(Specification of year, 2-digit)
[MM]	(Month of the year)
[DD]	(Day of the month)
[hh]	(Hour of the day)
[mm]	(Minutes of the hour)
[ss]	(Seconds of the minute)

The parameters [Modelname] and [NcIDX] make it possible to incorporate hyperDENT® into an automated process:

- *The NC files are output on a tool-specific basis.*
- *The name of the NC file is identical to the part name, and only one part is calculated.*
- *The individual NC files are numbered consecutively, starting with "1".*

NC file name, multi-holder

	If a multi-holder is used, this name is used as the NC file name. Here, the same parameters are possible as with the NC file name.	
Create info file	Create XML file with essential information such as blank and calculated parts for the NC file and save under the same name in the same directory. The info file also contains details on the G0 and G1 toolpaths and times. If the project file has already been saved, a reference to the project file is provided in the NC info file.	
Contents of the info file		
<statistic>	Part of the info file with the G0 and G1 details.	
<toolchanges>	Number of tool changes	
<G0Length>	m	Toolpath G0
<G1Length>	m	Toolpath G1
<G0Time>	h:min	Time G0
<G1Time>	h:min	Time G1
<TotalTime>	h:min	Total time
Create screenshot before calculation		
Yes	Create screenshot and save in the NC output directory.	
Image view direction	View for the screenshot.	
Current view		
Top, bottom, left, right, front, back view		
Front right, front left, back right, back left view.		
Show names of parts to be calculated		
Yes	Show part names.	
Image resolution width		
	Width in pixels.	
Image resolution height		
	Height in pixels.	
Print project automatically after calculation		

Yes [Print project settings...]	Print according to the settings in [File] ->
Acoustic signal after calculation	
Yes	Different signal tones for successful and suspended calculations.
No	

17.8.8 Navigation

Zoom	Zoom behavior when you move the mouse wheel.
Rotate	Key combination to rotate the blank.
Pan	Key combination to move the blank.
Use part selection triangle	
Yes	Automatic part selection triangle for the multiple selection of parts
No	Part selection as customary; CTRL + left mouse button

17.8.9 Display

Settings for the individual color selection and presentation of the hyperDENT® displays.

Anti-aliasing		None – 32x
Application theme color		
	Background color of the application and the dialog window	
Icon resolution main		24-64
	Toolbars for project management and the process steps	
Icon resolution ancillary toolbar		24-64
	Tool bar	
Icon resolution sub		24-64
	Toolbar – Part functions	
Holder color		
Holder transparency		0 - 0.9

Part color	
Part color (new)	(Color after loading the part)
Margin line color	
Color other margin lines	
Color abutment base lines	
Color emergence profile	
Color user defined areas Color pockets	
Outline color	
Outline top color	
Outline bottom color	
Connector color	
Sintering pin	
Sinter frames	
Use development view (shows fixture elements without holder-specific view rotation)	
Force display of insertion directions	Display of the insertion direction defined or transferred via the interface:
Yes	Always show insertion direction.
No	Only show insertion direction if it deviates from the main insertion direction.

Display – graphic screen colors

Settings for the individual color selection of the hyperDENT® displays.

Background lower left
Background lower right
Background upper left

Background upper right	
Foreground text color	
Foreground text size Scaling User interface	
Selected part color	
Modify selected part color	
Use background image	Yes / No
Background image transparency	0 - 0.9
Background image sizing Adapt image size to fit background Keep background image size	
Background image alignment Center, bottom left, bottom right, top left, top right	

17.8.10 Connector behavior

Edit settings for autoconnecting the connectors. Further details are available in the chapter “Set connectors” > “Edit connector settings” > “Connector behavior”.

18 Milling strategies

The milling strategies set the working plan (template) for machining on the milling unit and are available for different materials and part types.

Different templates can be selected for the different parts, depending on the details in the previous process steps.

The selection window shows the list from which you can select the milling strategy (template) for the machining.

With the template generator module option, you can create and edit templates: --> “Edit template”.

The template generator module option provides you with additional functions and templates so that you can freely configure different settings and adapt them to specific tasks:

- *Freely configurable tools.*
- *Freely configurable tool holders.*
- *Templates with freely configurable milling strategies (material, tool) for adaptation to the requirements of the part type in use.*
- *Supplied templates with standard parameterization for zirconium oxide, cobalt chrome, PMMA, and titanium.*
- *Supplied test cycle for setting up the postprocessor and milling unit.*

The milling strategy can only be changed in the template generator module option.

18.1 Editing the template (optional)

The template contains the predefined milling strategies and machining parameters that are dependent on the material and part type. The templates contain a series of different jobs, the predefined machining methods and the machining cycles that are needed in order to produce an optimum operational result.

In the template generator module option, you can change the templates and adapt them to the requirements of the part type and material in use.

The predefined jobs are subdivided into groups according to the cycles being used, and are then used in turn:
1. Roughing
2. Rest machining
3. Finishing
4. Connector machining
5. Special

18.1.1 Calling up template via the part browser or the context menu

Call-up via the part browser or the context menu ensures that the template is acceptable for the selected part type:
Only templates and jobs that are suitable for the part type and material type are offered.

You can call up, change, and temporarily change the templates that are suitable for the part type and material type, and you can also save them as a new template under a different name.

For a part that has already been calculated, you cannot change the settings, only the parameters can be displayed.

1. Select a part from the part browser.

The icon is active if a part is selected.

2. Click on the icon or call up the context menu and select the menu item [Edit template parameters].

The window [Template parameters] shows the job list with the associated parameters.

Joblist	List of individual jobs that are completed in succession: --> “Edit job list”.
Parameters	List of parameters for the selected job: --> “Edit parameters”.
Parameter profile	Selection field for the saved templates that are compatible with the part and material.

Save	Call up the selection window [Save profile], save template, save under a new name, assign to part types, set as default.
Print	Print joblist with the parameters.
OK	Save changes, exit menu.
Cancel	Do not save, exit menu.

Changing the column width

1. Click on the right-hand boundary line of the column, hold the mouse button down, and set the width.

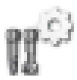
Or

1. Double-click on the right-hand boundary line of the column:
   The column width is set to the predefined width or the maximum width of the content or the heading.

18.1.2 Calling up template via the main menu

Call up the main menu to select and change or delete any template.

Make sure that you only use jobs that are acceptable for the part type and material type so that you can assign the part type to the template.



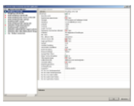
1. Call up the menu [Settings] and select the menu item [Milling strategies…].

The selection window shows the list with the templates, details about the selected template, and functions for the machining: edit, apply, delete, copy, export, password protect, print.

Show for	Selection filters for the templates: Material, Part Type, Permissible Machines.
Name	Name of the template. For a better overview, enter the name with a reference to the material and part type.
Milling strategy Edit parameters	Call up the parameters of the template, change and save the job list and parameters.

Material	Display, selection box for the material for which the template is valid.
Part type	Part type for which this template is valid.
Machines	Machines for which this template is valid. All machines or only for specific machines.
Default for	Use the template default for combinations resulting from the allocations of valid part types and machines.
Password protect	A password is required for this template.
License missing	A license is required for this template. The license is not available.
License available	A license is required and available for this template.

2. Select template.
   Click on the name of the template in the list.
3. Call up the [Edit] function:
   Click on the icon.
4. If necessary, enter the password.
   In the case of password-protected templates, enter the password; in the case of license-protected templates, the license must be available on the same computer.
5. Change name, material, part types, default or call up the function [Edit parameters].



The window [Template parameters] shows the job list with the associated parameters and the template names in the title bar.

Joblist	List of individual jobs that are completed in succession: --> “Edit job list”.
Parameters	List of parameters for the selected job: --> “Edit parameters”.
OK Save changes, exit menu.	
Cancel	Do not save, exit menu.

Tip:
To change the machine or the material for several templates, you can also select them all together and then edit them.

Changing the column width

1. Click on the right-hand boundary line of the column, hold the mouse button down, and set the width.

Or

1. Double-click on the right-hand boundary line of the column:
   The column width is set to the predefined width or the maximum width of the content or the heading.

18.1.3 Protected templates – Password, license

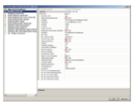
Password-protected templates can only be changed or called up via a part once the password has been entered.

License-protected templates can only be called up via a part if the valid license is available on the computer.

Setting, deleting password protect

1. Select template.
2. Call up password protect, click on the icon:
   The input window for the password is displayed with the name of the template.
3. Enter the password and enter it again to confirm.
   Without an input, the password protect for this template is canceled.
4. Save password by clicking on [OK].

18.1.4 Editing job list



The first entry [General settings] cannot be deleted or moved.

Then follows the list of individual jobs that are completed in turn in the order that is shown, along with the associated parameters.

The display screen on the right shows the parameters of the selected job.

Changing the column width

1. Click on the right-hand boundary line of the column, hold the mouse button down, and set the width.

Or

1. Double-click on the right-hand boundary line of the column:
   The column width is set to the predefined width or the maximum width of the content or the heading.

Enabling/disabling job

During the calculation, the system only works through the jobs that are enabled. Disabled jobs are not processed.

1. Select the job and enable or disable it:
   Double-click on the marking in front of the selected job:

On	The job is enabled and will be processed.
Off	The job is disabled and will not be processed. The job will remain in the list with its parameters.

Adding job

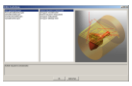
1. Click on the job list, call up the context menu and select the menu item [New job].

Either

2. Call up the group (Roughing, etc.) and select Job.

Or

3. Call up the menu item [Show job template selection dialog].



The window shows the groups and jobs, as well as a preview of the application type and toolpaths.

4. Select the new job:
   Click on the group, then double-click on the job.

The selected job is displayed at the end of the job list.

5. If necessary, move the job to the correct position in the job list.
6. If necessary, edit parameters and select cycle.

Moving job

1. Select the job and, holding the left mouse button down, drag it to the new position in the job list:
   The job is displayed at the new position.

Deleting job

1. Select one or more jobs, call up the context menu, and select the menu item [Delete job]:
   The job is deleted from the list.

Copying job

1. Select the job, call up the context menu and select the menu item [Copy job].
   The job is copied and entered with an index below the selected job in the job list.

Exchanging job

1. Select the job, call up the context menu and select the menu item [Exchange job template].
   The menu for the new jobs is displayed.
2. Either call up the group and select job or call up the menu item [Show job template selection dialog], call up the group, and select job.
   The parameters of the new job are displayed in the job list instead of the parameters of the selected job; the job name remains the same.
3. Adjust job name:
   If necessary, adapt the displayed job name to the actual job (--> hyperDENT® job name) to avoid mix-ups and incorrect machining.

Editing job name

1. Select job and change the job name:
   Double-click on the job name and change the name.

In assigning a job name, make a reference to the actual job in order to avoid mix-ups and incorrect machining.

18.1.5 Editing parameters

The parameters of the selected job are displayed in the display screen on the right.

Multiple selection

You can select several jobs and edit the common parameters at the same time. Only parameters that exist in all jobs are shown.

Changing parameters

1. Select job in the job list:
   The parameters are displayed in the display screen on the right. For a multiple selection, only the common parameters are displayed.
2. Change the value:
   Click on the right column next to the parameter and enter the value, select the path or select the entry via the selection menu.
   In the case of a multiple selection, the changes apply to all selected jobs.
3. Confirm the change:
   Click on OK. The change is not saved, it only applies to the selected part and the selected job.
   The change is shown in the part data as "Template modified".
4. Save the change:
   Click on the [Save] icon.
5. If necessary, cancel the multiple selection, select one job, and set the parameters that are suitable for the relevant job.

18.2 General settings – template

The general settings apply for all template jobs.

Margin line thickness	0.2 Minimum thickness of edge of coping (margin line) that machining will not go below. If the part is thicker at the margin line, the geometry of the edge of the coping is not changed. If the margin line is made thicker, it may also be necessary to extend the milling boundary.

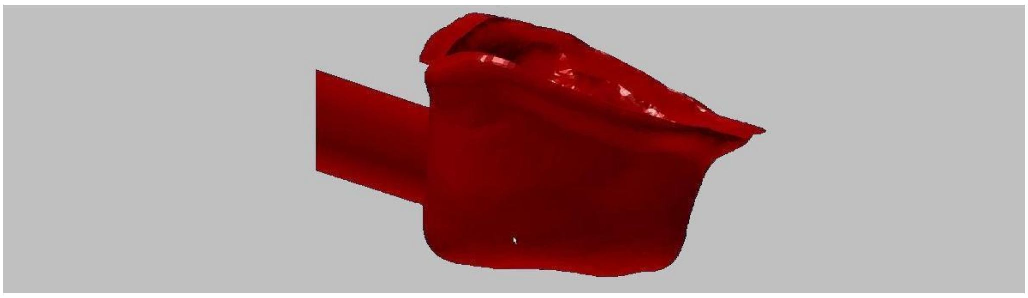


Figure 18-1

Milling boundary

For producing the dental restoration, the tool must be able to machine all areas of the part. In order to do so, the material must be removed in a sufficiently large area, the milling area, around the part.

The width of the milling area is dependent on the tool diameter and the slope for a coping-specific tilted machining process. The milling area must be large enough for the current machining process, otherwise the milling boundary must be extended.

The milling boundary prevents the risk of collision between the tool and the fixture, reduces the machining time, and enables several parts to be placed on one blank (e.g. disc).

Bounding strategy	Offset
Offset	Machining (roughing) only within a milling boundary. Help geometries (milling boundary 1 around the part) are created in accordance with a blank that is to be equipped with several parts (e.g. disc).

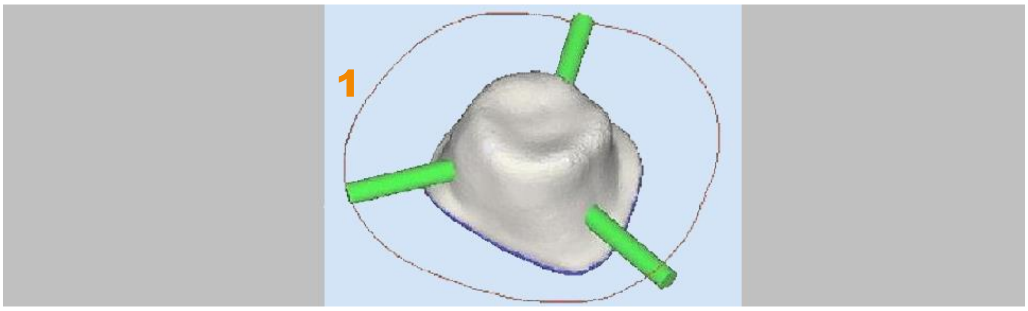


Figure 18-2

Fixture	Some special fixtures/blank geometries require help geometries to be created in such a way that the part can be accessed from all sides during the machining process. Setting required for section-wise machining. Connectors are set in the correct direction on a holder-specific basis.

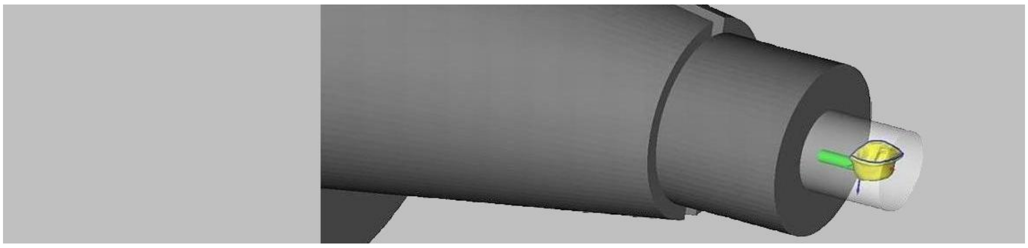


Figure 18-3

Offset milling boundary	2.5
	Width of the milling boundary around the part. Dependent on the milling diameter used and the slope for a coping-specific tilted machining process.
Calculation	Diameter of roughing mill + approx. 0.7 mm If the margin line is made thicker, it may also be necessary to extend the milling boundary.
Min. distance: Part – fixture	
	Corresponds with the boundary offset for limiting the fixture. Depends on the milling diameter.
Caclulation	Diameter of roughing mill + approx. 0.3 mm.
Boundary angle	3
	Angle of the milling boundary that is orthogonal to the blank surface. Important in order to protect the tool in the case of material that is difficult to cut.

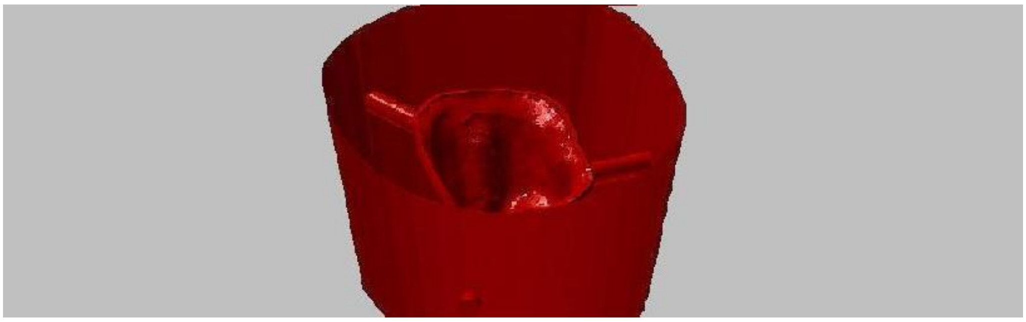


Figure 18-4

Boundary angle 2	3	
	Angle for the milling boundary that is used once a certain blank thickness is reached.	
Blank thickness bigger for boundary angle 2	mm	
	Once the specified blank thickness is reached, the angle for the second milling boundary is used.	

Start height of opening angle

Bottom of equator.

Bottom of equator

Default for work with milling boundary. The start height for the opening angle of the milling boundary is at the bottom of the equator (1) of the part.

Bottom of part

For better access, you can set the start height deeper onto the bottom (2) of the part (+ offset). This extends the milling boundary.

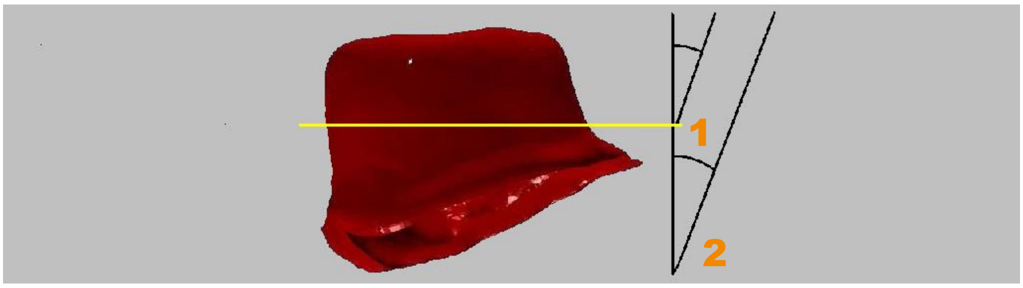


Figure 18-5

Offset height — mm

Offset for the start height of the opening angle across the bottom of the part.

Outer machining orthogonal to blank		No
Yes	Roughing also takes place when the main machining direction is tilted (height optimization) orthogonally to the blank.	
No	Tilted machining, all milling planes are tilted toward the blank surface.	

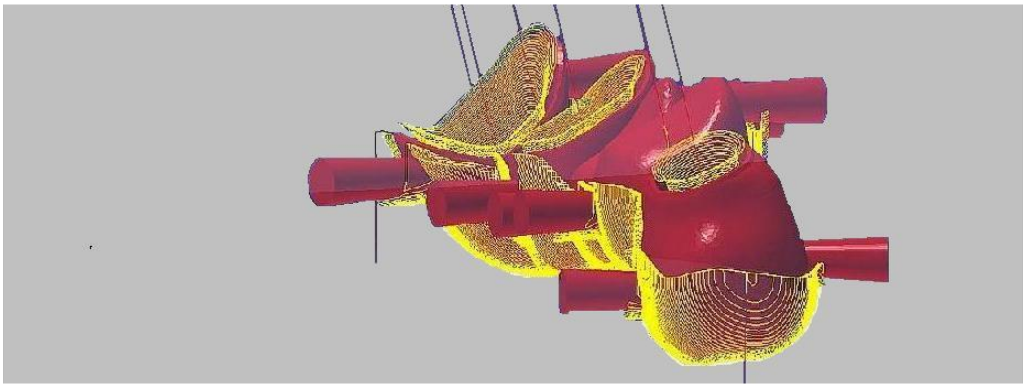


Figure 18-6

Coolant		1
1 / 0	Cooling lubricant on (1)/off (0)	
2 / 1 + 2		
3 / 1 + 3 / 2 + 3		
4 / 1 + 4 / 2 + 4 / 3 + 4		
	Other variants for switching the cooling lubricant make it possible to perform more complex processes, e.g. automation or ultrasonic machining. You have the option to set the cooling lubricant globally or to define it for each job. Follow instructions of the machine manufacturer!	

Tool reference		Center
Center / Tip	Machine-specific details to calculate the toolpath based on the center/tip of the tool.	

Clearance above stock	4	(absolute measurement)
	Clearance distance (1) above the blank, for rapid traverse movements on all 3 axes. The clearance refers to the Z-axis of the current milling process (frame).	

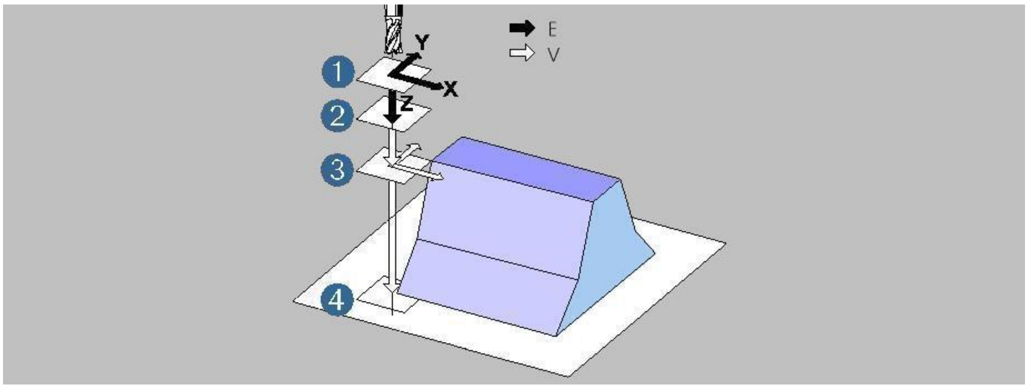


Figure 18-7

1. Clearance level
2. Clearance distance
3. Top of blank
4. Bottom of blank

Clearance distance		1
	Clearance distance (2) above the part, for the depth setting on the Z-axis: Above for rapid traverse, below for Z-feedrate.	
Check fixture for collisions in 3X jobs		No
Yes / No	Enable/disable collision check for the fixture in 3X jobs. The calculation of the toolpaths with collision check results in longer calculation times.	
Check fixture against collisions in 5X jobs		No
Yes / No	Enable/disable collision check for the fixture in 5X jobs. The calculation of the toolpaths with collision check results in longer calculation times.	
Holder allowance	Global allowance on the holder for roughing	
Reduce factor		0.9
	Factor for reducing the STL facets to speed up the calculation time. The factor refers to the machining tolerance:	
0 =	No reduction	
1 =	Reduction equal to machining tolerance	

Split model		Yes
Yes	Separate STL data is produced for 3D jobs with no facets in the undercut areas. The calculation time can hereby be reduced.	
No	STL facets are transferred completely to the cycle.	
Max. angle increment in 5X jobs		0.05
	Machine-specific value, limits the changes to the tool tilt that can be made between two points. The maximum G1 length and the value for the maximum angle increment depend on the control unit (RTCP) and machine. Follow information from the machine manufacturer!	
Comment	Text entry Entry of comments that are written in the NC info file and in some cases in the NC header (e.g. for Datron D5).	

Segment-wise (section-wise) machining

For raw materials that come as rods, it can be helpful to machine the part in sections.

Machining by segment can be enabled or disabled separately for each job via the parameters in all roughing, finishing, and rest machining jobs. It is therefore for example possible to completely machine the cavity side first, but machine the occlusal side in segments. That is, the occlusal side is divided into individual sections and each section is always fully machined.

There is an additional offset value for roughing jobs for the roughing mill.

The parameter [General settings] > [Bounding strategy] > [Fixture] must be set for the template.

- Caution! Risk of collision!
  The blank is not included in the collision check.

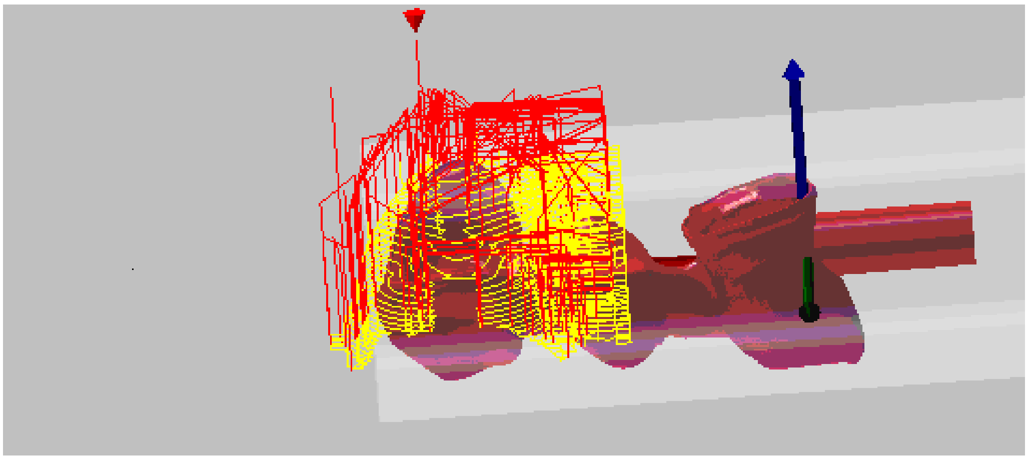


Figure 18-8

Min. segment distance to element border	mm	
	Minimum distance of the machining section to the element borders, e.g. boundary lines of margin line, emergence profile, abutment base.	
Min. segment width	mm	
	Minimum width of the machining section.	

18.3 Overview of 3D cycles

The predefined jobs are subdivided into base areas in accordance with the cycles in use: roughing, finishing, rest machining, drilling.

Arbitrary blank roughing

Z-constant removal of randomly formed blanks with option of blank tracking.

Suitable for rough preparatory work such as stripping all necessary areas in order to expose the part for the next finishing steps.

Blank tracking allows you to obtain an exact situation of the blank after the machining process.

Suitable for rest machining. In particular for materials that are difficult to cut (cobalt chrome), this can prevent breaking a tool.

Requires a longer calculation time due to the elaborate calculation for the blank tracking.

Profile finishing

Surface-overlapping, collision-free milling with different milling strategies using guide curves; optional slope-dependent machining.

*For the machining profiles X-axis and Y-axis,
XY-optimized machining is possible.*

Suitable for finishing the part surface of the occlusal side and the cavity side outside of the cavities and for machining the inlay/onlays within the margin lines.

Complete finishing

Z-constant finishing with automatic pocket-shaped machining of flat areas.

Suitable for finishing the part surface of the occlusal side and the cavity side outside of the cavities.

Enables low-risk standard machining of the part surfaces, in particular for material that is difficult to cut.

Equidistant finishing

Finishing with constant area step down; suitable above all for high-speed milling.
The machining is equidistant within a closed guide curve or flowing between two guide curves.

Ideal for fine finishing of areas around the margin line and the inner areas of the copings for 3X machining.

Z-level finishing

Z-constant finishing with optional slope-dependent machining.

Suitable for finishing the part surface of the occlusal side and the cavity side outside of the cavities.

Less suitable for finishing of plane surfaces.

Automatic rest machining

Targeted rework-machining of individual areas of space left after the finishing process.

Relates to rest machining of the preceding tool, but without blank tracking.

Less accurate but sufficient for the machining of softer materials, relatively short calculation time.

3D axis parallel peeling

Suitable for roughing with grinding tools, as the tool can also operate with shank parts – depending on the shape and definition of the tool. Machining is performed orthogonally to the blank.

3D U peeling

For roughing with grinding tools. The machining is carried out on three sides of the part.

Caution:

It is necessary to use a holder that is open on three sides, e.g. single-block holders.

The connector must be set in such a way that its direction roughly corresponds to the machining direction.

The boundary mode in the template must be set to “fixture”.

Segmented machining

This machining is suitable for large parts such as models, full dentures, large bridges, and bite splints.

The machining is divided into segments and follows the central curve and/or the alveolar ridge. The system machines in 3+2 axes.

A strategy in connection with profile finishing and equidistant finishing is possible.

18.4 Overview of 5X cycles

5X boss finishing

Finishing in the coping areas and occlusal areas with undercut machining. Machining is performed with constant step down.

5X profile finishing

Surface-overlapping 5X machining in X, XY optimized or ruled.

Undercut machining partly possible.

5X automatic rest machining

Targeted rework-machining of areas of space left, suitable for reducing the tool length.

Relates to rest machining of the preceding tool, but without blank tracking.

Undercut machining is not possible.

5X peeling

Suitable for roughing in the coping with grinding tools, as the tool can also operate with shank parts – depending on the shape and definition of the tool.

18.5 Overview of cycles for connector machining

Cut sinter frames

Cut sinter frames completely or partially.

Cut/reduce connectors

Cut connectors completely or partially.

Cut (reduce) connectors partially.

Cut according to the outline shape.

Cut parallel to contour.

Cut without residue.

Reductions and cuts are also possible from the cavity side.

Order of cuts is from the outside working in.

Additional milling cut, cavity side

Make cut from the cavity side.

Break connectors

Break parts completely out of the blank.

Requires reducing the connectors.

Using a special tool, the part is broken out of the blank by applying pressure to the occlusal side.

18.6 Overview of drilling cycles

Simple drilling The drill hole is created in a step, e.g. for center drilling, predrilling.	
Drilling with chip break The drill hole is created in several steps (drilling strokes). Retracting the tool after each drilling stroke breaks the chip and makes it easier to remove. *Retraction with rapid traverse around the retract value after each drilling stroke.* *Reduction of the step by the reduce value after each drilling stroke.*	
Drilling with pecking The drill hole is created in several steps (drilling strokes). Retracting the tool after each drilling stroke to the clearance distance breaks the chip and clears fine chips (drilling dust) out of the drill hole. *Retraction with rapid traverse to the clearance distance after each drilling stroke.*	
Drilling inside copings Predrilling for grinding of the coping and machining with upward tool movement only, no full cut.	
Drilling implant interface geometry Drilling corners in the interface geometry.	
Thread machining Creating an interface thread.	
Screw channel machining	

18.7 General job parameters

The following parameters exist in almost all jobs and are therefore only described here.

hyperDENT® job name	Original job name predefined by hyperDENT®.
Job name	Displayed in the job list, can be freely edited, preferably in accordance with the operational step and cycle, e.g.: “Roughing cavity”.

Calculate		Yes
	Yes	Calculate job
	No	Disable calculation, the job stays in the list, but is not included in the calculation.
Calculate if		Off Conditional calculation. This means you can calculate two jobs that are the same depending on the groove depth of tools of different lengths. ▪ Where the groove depth is smaller, shorter tools are better: They are more stable and produce better surfaces, greater accuracy and chipping performance, but cannot be used for larger groove depths because of the risk of collision and not being long enough. ▪ Longer tools are needed for bigger groove depths in order to reach the required depth.
	Off	Calculation always performed, e.g. for only one tool length.
	Groove depth bigger	Calculation only performed when the groove depth is bigger than the default value, e.g. for the longer tool.
	Groove depth smaller	Calculation only performed when the groove depth is smaller than the default value, e.g. for the shorter tool.

- Caution!
  Use tools with the same diameter and the same allowance.

Groove depth		Value from top of blank Reference plane for the conditional calculation.
	Bottom of blank	
	Mid of blank	
	Value from top of blank	The Z-level to the depth limit is made parallel to the blank.

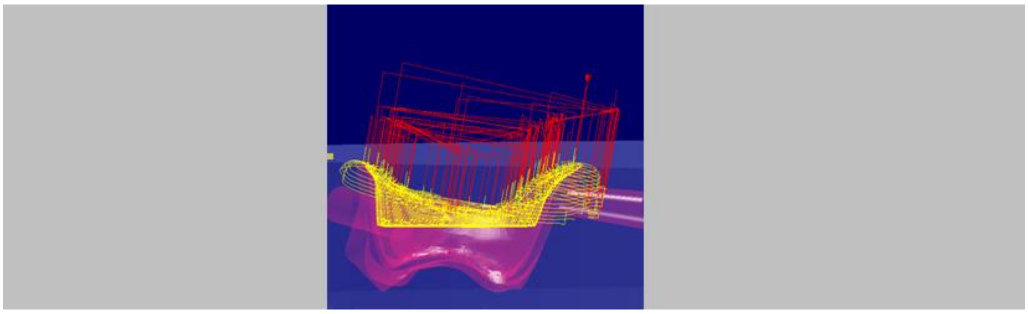


Figure 18-9

Value from top of machining area	Top of machining area refers to the highest point of the machined areas. The Z-level to the depth limit is made perpendicular to the machining direction.

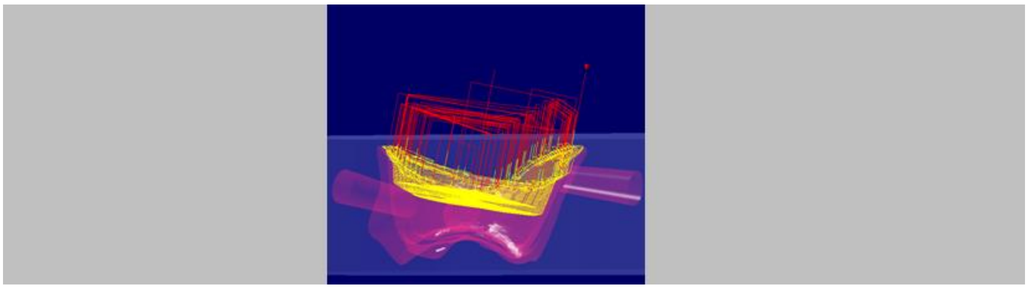


Figure 18-10

Depth value	Groove depth in mm, from top of blank or top of machining area.

Example 1: Conditional calculation

Roughing on arbitrary blank with successive job with longer tool.

1st Job	Roughing cavity side
	Parameter setting
Calculate if	= Off ...
Max. groove depth	= 5 (for a shorter tool)
2nd Job	Roughing cavity side
	Parameter setting

Calculate if	= groove depth bigger
Groove depth	= value from top of blank
Depth value	= 5 (for a longer tool)
	...
Max. groove depth	= unlimited

For a groove depth < 5 mm, only the first job is calculated, for a groove depth > 5 mm both toolpaths are calculated.

Example 2: Conditional calculation

Optional finishing where, depending on the depth value (e.g. 5) and position of the Z-level boundary area, the finishing job is automatically generated with the correct tool (short/long).

1st Job	Overall finishing, occlusal side, optional for coping depth < 5:
	Parameter setting:
Calculate if	= groove depth smaller
Groove depth	= value from top of blank
Depth value	= 5
	...
Max. groove depth	= unlimited
2nd Job	Overall finishing, occlusal side, optional for coping depth > 5:
	Parameter setting:
Calculate if	= groove depth bigger
Groove depth	= value from top of blank
Depth value	= 5
Max. groove depth	= Unlimited (limitation possible if required).

If the maximum required groove depth from the blank surface to the deepest point is smaller than 5 mm, then Job 1 is calculated with the shorter tool, otherwise Job 2 is calculated with the longer tool.

Strategy	Cycle for machining. Selection menu with a list of available cycles.

Checking the milling boundary

For producing the dental restoration, the material must be removed in a sufficiently large area, the milling area, around the part so that the tool can machine all areas of the part.

The milling boundary determines the width of the milling area and is dependent on:

the tool diameter

the inclination for tilted machining

the inclination for a coping-specific tilted machining

Using the boundary check, hyperDENT® can determine before the calculation whether the milling boundaries are large enough for the current machining process. This avoids abortion of the calculation and incorrect machining.

Non-tilted machining

- No automatic widening of the milling boundaries in the case of a main machining direction without tilting and orthogonal alignment of the part to the blank.

If necessary, the milling boundaries must be manually extended, e.g. if a tool with a widened shank and short tip collides with the blank if the groove depth is large.

Tilted machining

For tilted machining with active boundary check, a check is performed to establish whether the milling boundaries are large enough for the current machining process. If necessary, the milling boundaries are automatically extended.

With the "Tool tip" setting, the milling boundary is only extended according to the diameter of the tool tip, the tool shank is not checked.

If tools with a larger shank diameter are used at greater depths, a collision message is issued if the collision check is activated, otherwise there will be a collision between the tool shank and the blank.

Boundary check	Check (default for tilted machining) to establish whether the milling boundaries are large enough for the current machining process. If not, a message appears and the milling boundary must either be extended or, in the case of tilted machining, it is automatically extended.
Off	No check, a collision message may be issued if the collision check is activated, otherwise there could be a collision or unmachined areas.
Tool tip	Only the tool tip is checked. For tools with a widened shank and short tip, a collision message is issued if the collision check is activated, otherwise there will be a collision with the blank if the groove depth is large.
Tool shank	The tool shank is checked; it can be inserted into the blank. Larger milling boundaries mean a higher consumption of material and longer machining times. Older templates are automatically set to the tool shank for safety reasons.

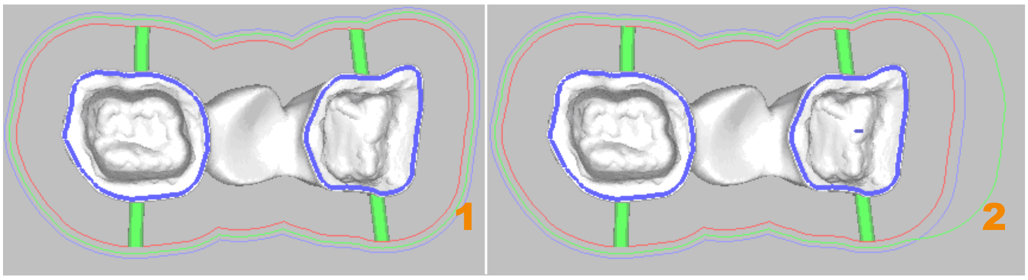


Figure 18-11

1. Milling boundary
2. Enlarged milling boundary with a large tilt

If a collision is detected when the milling boundaries for tilted machining are checked, the program automatically extends the milling boundary (2).

Errors may occur in isolated cases of extreme tilting.

- Caution!
  Select the check carefully to avoid the risk of collision.
  For a very large machining depth and short tool tip length, use the tool shank check.

Example of coping-specific tilting, milling boundaries

Coping machining with a steeply angled tool

Boundary check of tool tip

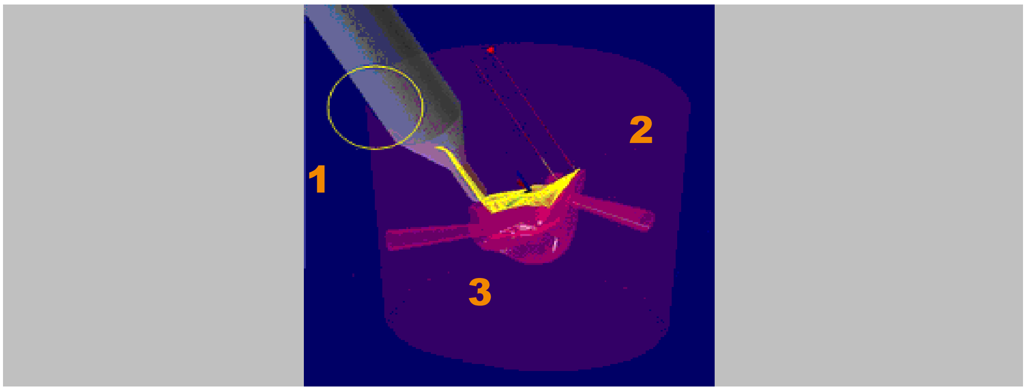


Figure 18-12

There is the risk of a collision (1) between the tool shank and the milling boundary (2) if the tool plunges into the unmachined area (3).

Example of boundary check of tool tip

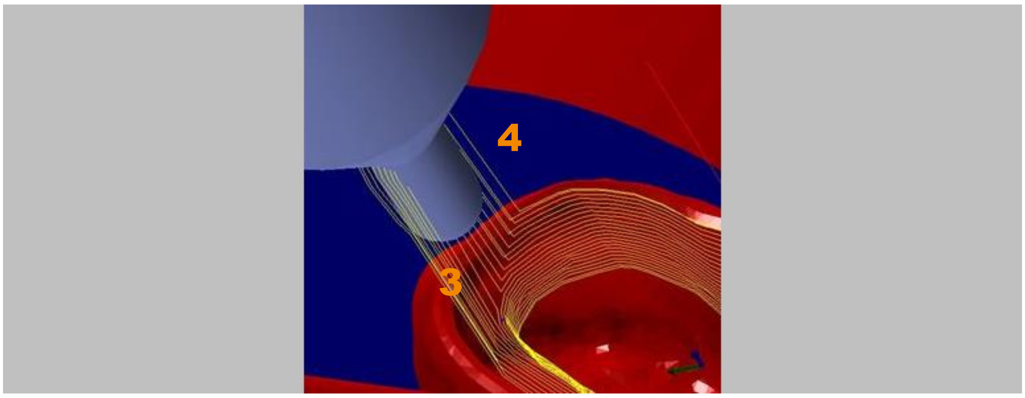


Figure 18-13

The “Tool tip” setting ensures that the tool retracts to the clearance level (4) during the calculation, to prevent a collision (1) with the milling boundary. This leaves unmachined areas (3). Collision message if groove depth is too large.

Example of boundary check of tool shank

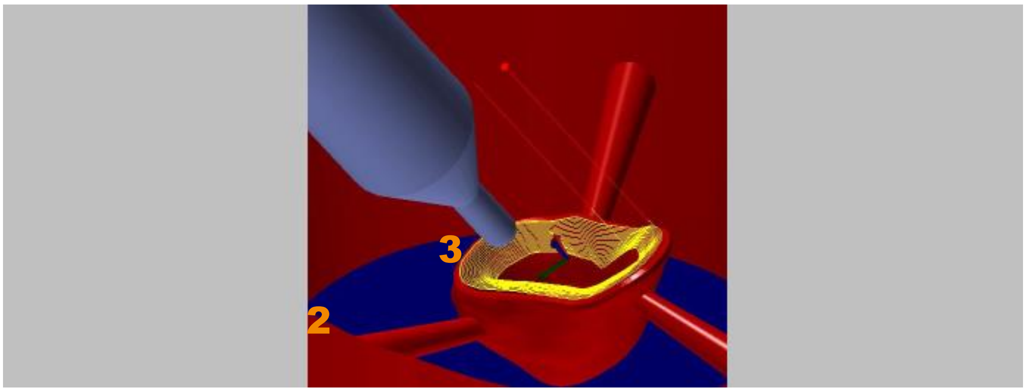


Figure 18-14

The "Shank" setting is used by hyperDENT® to identify that the milling boundary is not large enough, issue a message, and automatically extend the milling boundaries in the case of tilted machining.

After the milling boundary (2) has been extended, the area (3) can also be machined correctly.

Tool	Tool for this job, Selection menu with a list of preconfigured tools.
Spindle speed	Spindle speed of the tool in rotations/min. Follow instructions of the machine manufacturer (maximum speed)!
Feedrate	mm/min. Rate with which the tool processes the calculated tracks. Follow instructions of the machine manufacturer (maximum feedrate).
Feedrate axial	mm/min. Rate with which the tool processes the calculated tracks in Z-direction. Follow instructions of the machine manufacturer (maximum feedrate).
Reduced feedrate on full cut	Yes / No
Yes	Reduce feedrate to protect the tools when using material that is difficult to cut. Can result in an extended calculation and machining time.

Max. groove depth	Unlimited Depth limit of the toolpath for a job, according to the tool length.
Unlimited	No depth limit.
Bottom of blank	
Mid of blank	
Value from top of blank	The Z-level to the depth limit is made parallel to the blank.

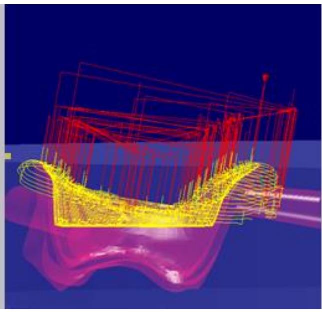


Figure 18-15

Value from top of machining area	Top of machining area refers to the highest point of the machined areas. The Z-level to the depth limit is made perpendicular to the machining direction.

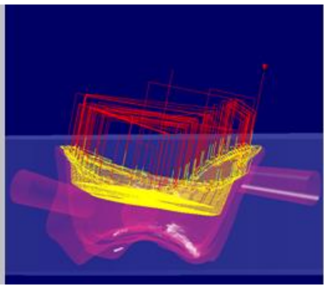


Figure 18-16

Depth value	Maximum groove depth in mm, from top of blank or top of machining area.
Use coping-specific tilting No	
	Details of the rotation axis for the coping-specific tilting. If the machining direction is not determined by the CAD, you can set it via the context menu: --> “Set milling direction” > “Coping-specific alignment”.
No	No coping-specific alignment/tilting.
X+Y rotation	Rotation axes for 5X machines.
X rotation	X rotation axis, for 3+1 machines.
Y rotation	Y rotation axis, for 3+1 machines.
Allowance	0.1
	Space left (in mm) that should be left over on the part surface after this stage for further machining (finishing, fine finishing).
Step down	Path in mm with which the tool is delivered to maximum in Z-direction. Corresponds to the clearance of the machining planes and determines the number:
One level	Step down > (surface – depth)
Several levels	Step down > (surface – depth)

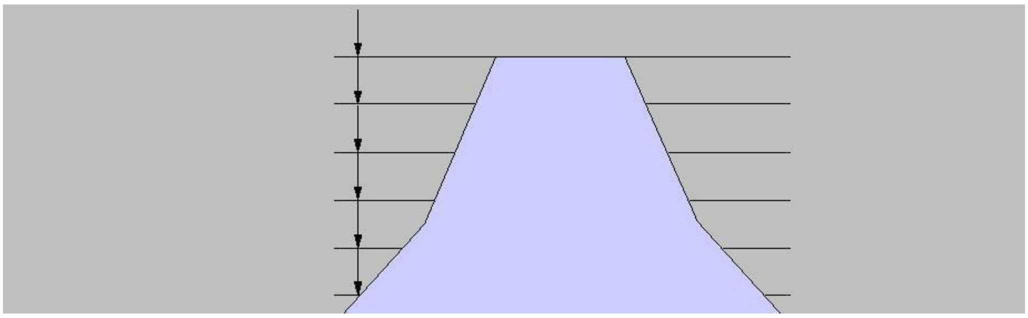


Figure 18-17

Step factor	0.33 Factor of tool diameter for the maximum horizontal step of the tool: "0.5" corresponds to a step of 50 % of the mill diameter.

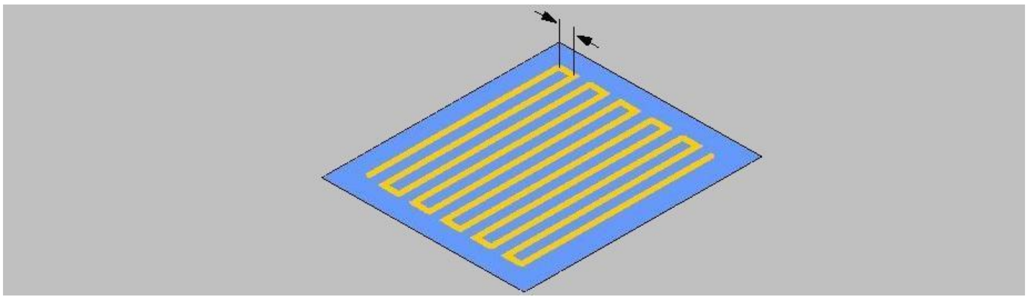


Figure 18-18

Fillet corners	No
Yes / No	Inside corners are rounded for all roughing and rest machining to protect the tool when using hard materials. The machining tolerance is hereby ignored and the calculation time increases.
Plunge strategy	Ramp
Ramp / helix	Method by which the tool plunges into the material in a Z-direction.
Planar mode	Method by which the tool processes the material in an XY-direction.
Optimized in	Processing from outside to inside
Inside out	Processing from inside to outside
Ramps/helix angle	45
	Angle in degrees with which the tool plunges into the material in a Z-direction: Input "90" corresponds to a direct step in the Z-direction.
Skip small pockets	Yes
Yes	Skip pockets, do not machine.
No	Machine pocket, set close to the tool diameter only for very narrow coping areas.

Collision check	Tool and holder Includes the tool or tool and holder in the calculation.
Tool only	Collision check for the tool only.
Tool and holder	Collision check for tool and tool holder.
Off	No collision check, quicker calculation. Only for non-critical parts.
Avoid collision	Yes
Yes	Calculation not canceled if there is a collision but alternative milling tracks found independently through the cycle.
No	Disable “Avoid collision”, for shorter calculation time.
Machining tolerance	0.05 Deviation (resolution) in mm, when calculating the toolpaths. In a template (job list for full machining), where possible only common values should be used as the calculation time increases if there are many different values. Optimum approach: E.g. one value for roughing cycles, second value for finishing cycles.
Stock model tolerance	0.1 Resolution of blank tracking in mm. For material that is difficult to cut, a finer resolution should be selected.
NC text before tool change NC text after tool change NC text before first position NC text after first position NC text after last position	Enter free text modules (job-specific NC data) such as control instructions, etc.

- *Caution! Risk of collision!*
  hyperDENT® does not perform a syntax or collision check of the NC texts

entered. Only use this function if the machine being used can evaluate the data correctly.

Separator for multi-line instructions in the NC program (line break).

Example of multi-line instruction

H123=28\nH124=4\nM22	Input of NC text
H123=28	Output
H124=4	
M22	

18.8 Cycle-specific job parameters

The following cycle-specific job parameters are structured in accordance with the machining methods and the associated cycles.

18.8.1 Reducing the blank on cavity and occlusal side

Special cycle for material removal for cylinder-shaped blanks. Endmills can be used.

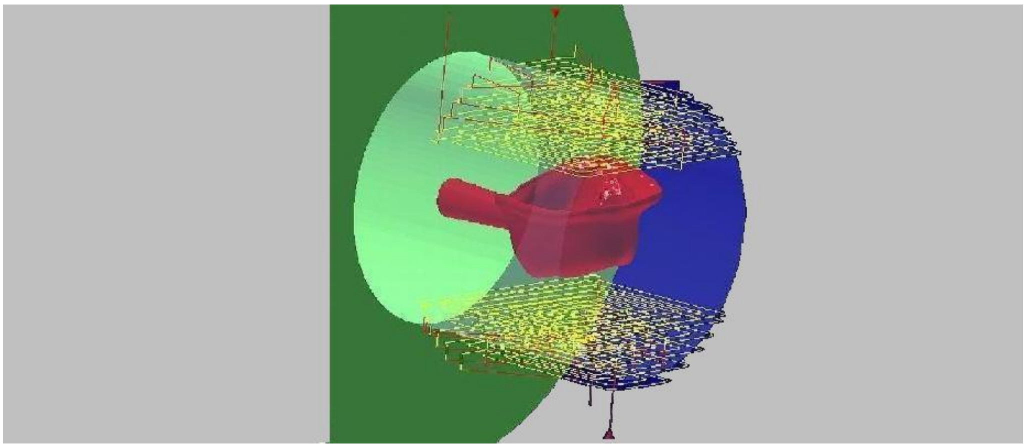


Figure 18-19

Boundary mode

	Area that is to be reduced
Fixture	Machining of the entire blank
Offset	Machining within a milling boundary

Min. depth	
	Depth that machining will reach in all cases
Top of part	Machining up to highest point on the part
Mid of part	Machining up to middle of the part
Bottom of part	Machining up to lowest point on the part
Top of blank	Machining up to the top edge of the blank
Mid of blank	Machining up to the middle of the blank
Bottom of blank	Machining up to the bottom edge of the blank
Z-offset	
> 0	Translate machining boundary up
< 0	Translate machining boundary down

18.8.2 Roughing on cavity, occlusal, and any side

Rough removal of material between the blank and part on the cavity side, occlusal side, or any side.

Side	Front
Front/Back/Left/Right	Enter the side to be machined; corresponds to the view direction for setting [Front view].
Example	Machining side
	In terms of the view direction the tool approaches the part from the back, i.e. actually approaches the front of the part.

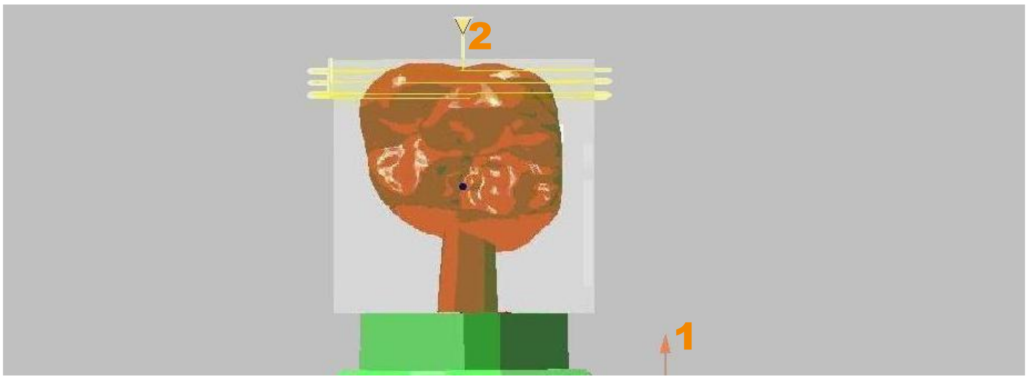


Figure 18-20

1. View direction: Setting [Front view].
2. Tool: Seen from the view direction, the machining area is on the "back" of the blank.

Machining depth	Parameters for the machining depth on the blank.
Max. of bounding box	Machining area from "back" view on the blank (bounding box). Machining from the start of the blank to the start of the part.
Example	Max. of bounding box (with additional offset)

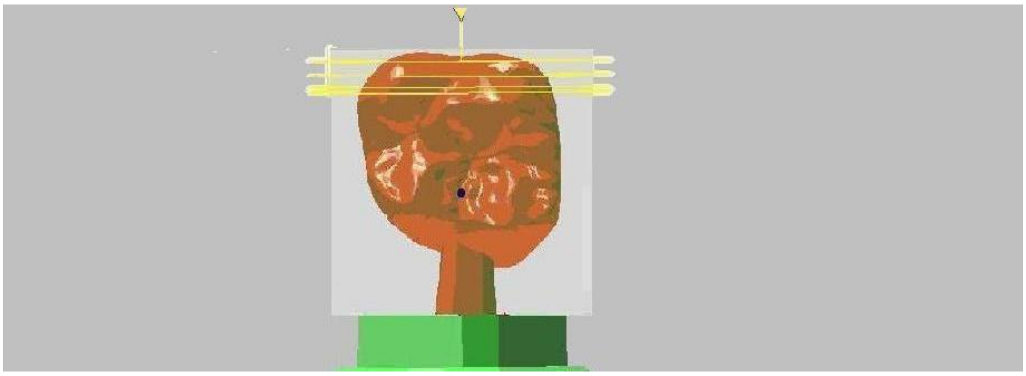


Figure 18-21

Min. of bounding box	Machining area from "from back to front" view on the blank (bounding box). Machining from the start of the blank to the end of the part.
Example	Min. of bounding box

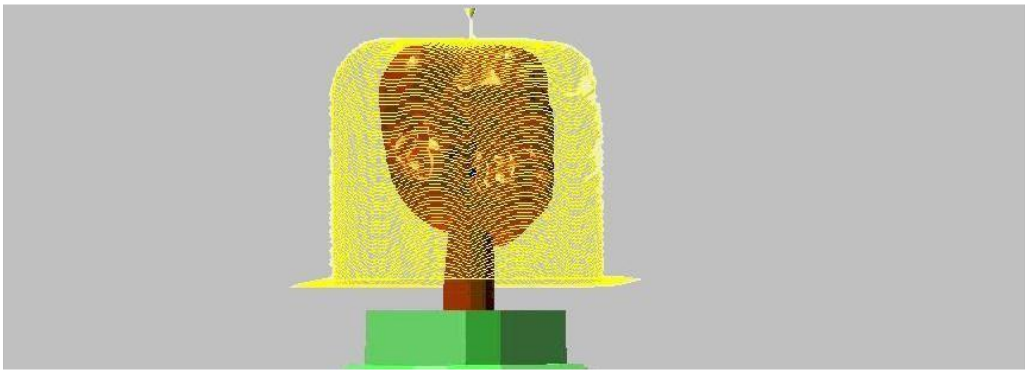


Figure 18-22

Center of bounding box	Machining area from “from back to center” view on the blank (bounding box). Machining from the start of the blank to the middle of the part.
Area	Determine work area.
Outer areas	Viewed from the Z-direction, the part is machined up to the equator.
All areas	The whole part within the confines of the bounding box is machined.
Depth offset	mm
	Additional value for the machining depth. A larger value results in deeper machining.
Protected area within the abutment base	
Yes / No	Activate/deactivate protection of the abutment base.
Use/transform fixture boundary	
Yes	When machining a side of the blank, use milling boundaries for fixture.
No	When machining the front side (from the view of the fixture).
Close copings	No
Yes	“Yes” must be entered if the setting --> “Special function coping-specific alignment” has been set for the part. “Yes” must also be entered for the following finishing cycles of the cavity.

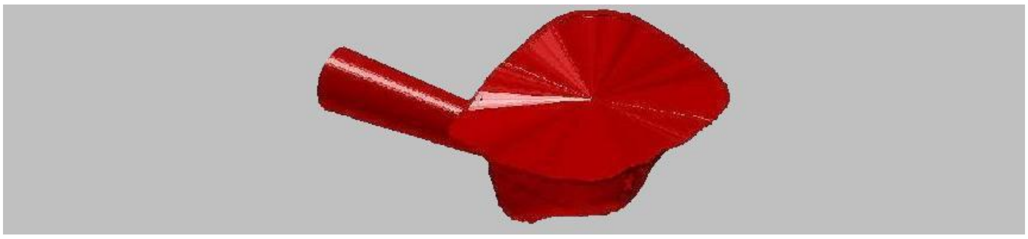


Figure 18-23

Min. depth	None
None	Roughing depth arises from pathway of equator.

	Mid of boundmesh	By setting the parameters “Milling boundary angle” and “Start height of opening angle” (see 18.4), a cone is produced on both sides for the machining. The cutting point of both cones defines the area “Mid of boundmesh”
	T	Machining up to highest point on the part
	Mid of part	Machining up to middle of the part
	Bottom of part	Machining up to lowest point on the part
	Top of blank	Machining up to the top edge of the blank
	Mid of blank	Machining up to the middle of the blank
	Bottom of blank	Machining up to the bottom edge of the blank

Finish pass — Yes

Smoothen roughing depth that the roughing cycle has left behind in order to protect the following tools during subsequent machining: Yes/No.

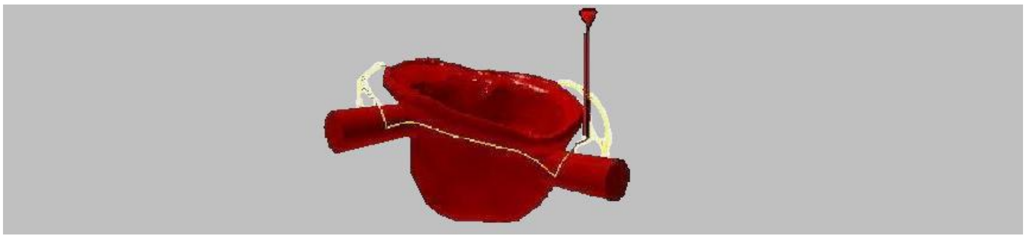


Figure 18-24

Plane mode		Tool movement for roughing.
	Rapid in	Tool’s full cut is suppressed, suitable for grinding strategies.
	Optimized in	Standard setting for roughing, unsuitable for grinding strategies.
	Inside out	Roughing from inside to outside

- Caution!
  Tool’s full cut possible. Therefore the setting [Optimized in] is unsuitable for grinding strategies.

18.8.3 Finishing/fine finishing the margin line

Targeted pre/fine finishing in the margin line area

Cycles	3D equidistant finishing 5X boss finishing

Outer machining happens automatically in synchronization when the job is set in such a way that, starting from the margin line, only the outer area is machined: [Inner offset] from margin line is smaller than or equal to the value for the stepover. Machining starts at the margin line.

- If this area is divided into two jobs – from the margin line in and from the margin line out – then all machining takes place in synchronization.

Max. angle for red. feedrate

If an uninterrupted full feedrate is desired, then this value should be set to “0”, otherwise the feedrate is reduced during plunge movements.

Machining direction

To reduce the risk of chipping, especially during grinding, you can set the machining direction.

Outside in / Top down

Inside out / Bottom up

Inner offset	Area (1) that is machined inside the coping, starting from the margin line.
Outer offset	Area (2) that is machined outside the coping, starting from the margin line.

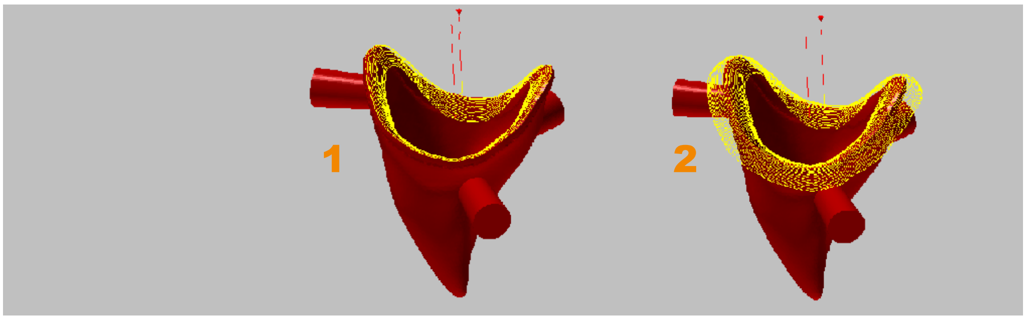


Figure 18-25

1. Inner offset
2. Inner and outer offset

18.8.4 Finishing/fine finishing inside the coping areas

Targeted pre/fine finishing in the area inside the coping from the margin line.

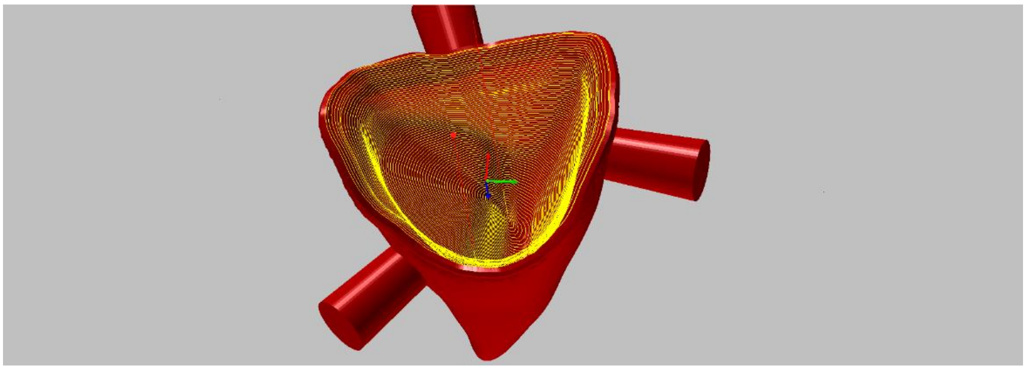


Figure 18-26

Cycles:	3D equidistant finishing 3D complete finishing 5X boss finishing 5X peeling

Max. angle for red. feedrate:

Value 0	Full feedrate; with a different value the feedrate is reduced during the plunge movements.
Offset	When set to “0”, the margin line is protected by an automatically generated help geometry. This means that this sensitive area is skipped during the coping machining process.
Value < or > 0	The protection mechanism is not effective. An offset function is called up.
Values > 0	The offset is generated outside the margin line.
Values < 0	The offset is generated inside the margin line.

- *Take care with undercut areas that start directly at the margin line:*
  If an offset < 0 is entered, the machining inside the coping may fail.

Coping-specific tilting with undercut machining for 3+1 machines

Undercut machining requires settings for the part: --> “Set milling direction” > “Undercut machining”:

Rotation axis

Alignment to rotation axis

and settings for the machining parameters:

Rotation axis

Multi axis machining

Differential angle for finishing

The part is rotated in a plus and in a minus direction depending on the rotation axis entered, and the toolpath is recalculated. Up to 4 jobs are created.

Multi axis machining	No
Yes/No.	Perform undercut machining for 3+1 machines.
Add angle	Differential angle for finishing.

5X boss finishing

Undercut machining	
Yes/No.	Perform undercut machining, even in 5X mode.
Desired tilt angle	Tilt angle of the mill for 5X machining. The value should be adapted to the machine kinematics and should not be too large.
Max. tilt angle	Maximum tilt angle of the mill for 5X machining. The value should be adapted to the machine kinematics and should not be too large.

5X peeling

Desired tilt angle	Tilt angle of the mill for 5X machining. The value should be adapted to the machine kinematics and should not be too large.
Max. tilt angle	Maximum tilt angle of the mill for 5X machining. The value should be adapted to the machine kinematics and should not be too large.

Position	According to drilling job Position in the coping for the plunging of the tool, coordinated with the previous drilling job [Drilling inside copings].
Automatic	
Center of coping	
Deepest point in coping	
Plunging angle	Tool track distance for the spiral plunging of the tool into the coping.
Plunge radius	Radius (half diameter) of the machining area for plunging of the tool.

18.8.5 Finishing within the margin line for bridges with inlay/onlay parts or Maryland bridges

If the margin line has been realized via the --> “Special function machining of inlay/onlay bridges”, then a special kind of machining can be determined for this area since the geometry of an inlay is considerably different to that of a coping.
To machine undercut-free areas, the cycle “3D equidistant finishing” is extremely suitable.

For undercut machining, the cycle “5X boss finishing” can be used as an option depending on the geometry of the part.

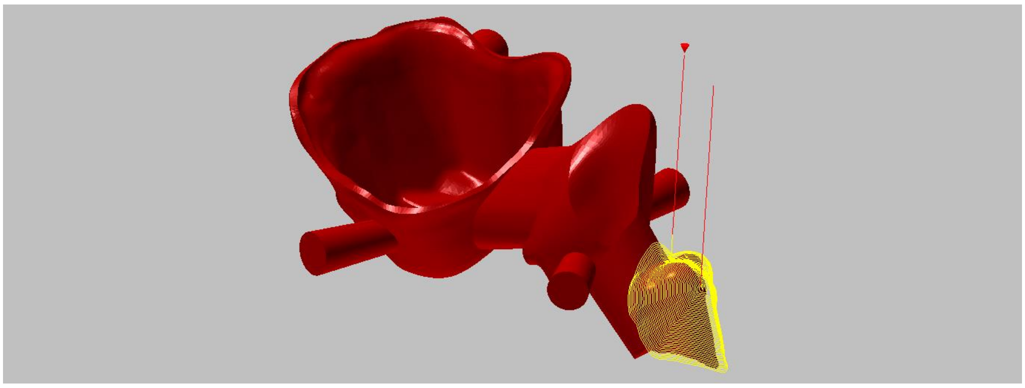


Figure 18-27

Cycles	3D equidistant finishing 3D complete finishing 3D profile finishing X-direction 3D profile finishing in XY-direction 5X boss finishing
Offset	In most cases ≥ half tool diameter, as otherwise unmachined areas may be left. No other function of this offset; no protection of the margin line when value = 0.
Max. angle for red. Feedrate	0
Value 0	Full feedrate; with a different value the feedrate is reduced during the plunge movements.
Coping-specific tilting	
Yes	The setting --> “Special function coping-specific alignment” has been used for the part; this area should therefore be machined with a separate tilt: Yes/No.

5X boss finishing

This cycle can only be used in isolated cases since the quality of the milling tracks depends heavily on the geometry of the milling area.

Undercut machining	
Yes	Undercut machining to be realized: Yes/No.
Desired tilt angle	Approx. ≤ 5°
Max. tilt angle	Approx. ≤ 10°

Maryland bridges

Here both margin lines should be identified with the --> “Special function machining of inlay/onlay bridges”. They can then be parameterized with the cycles from this section.

To avoid an undercut situation, the tilted machining often requires the setting --> “Special function coping-specific alignment”.

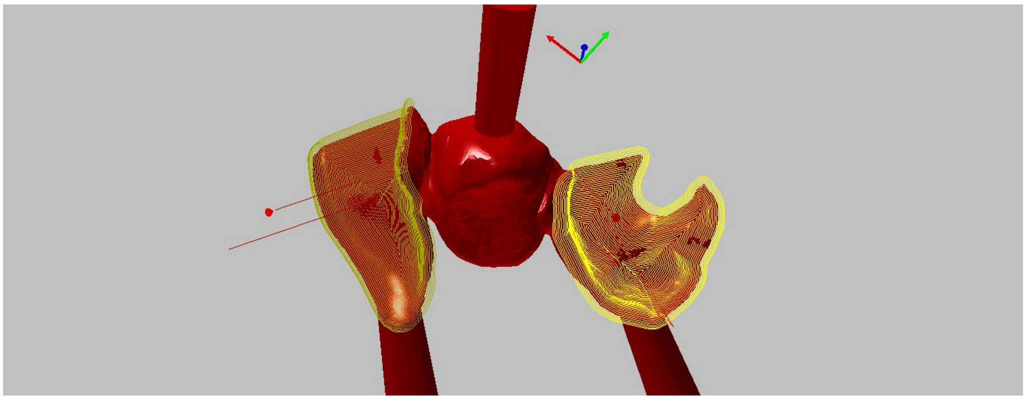


Figure 18-28

18.8.6 Rest machining

The machining of the space left should always be interposed if a mill is changed on a tool with a smaller diameter during machining.

The cycles for space left can identify and machine those areas in which a greater allowance has been left by the previous cycle, since the tool was unable to optimally remove the material due to the larger diameter.

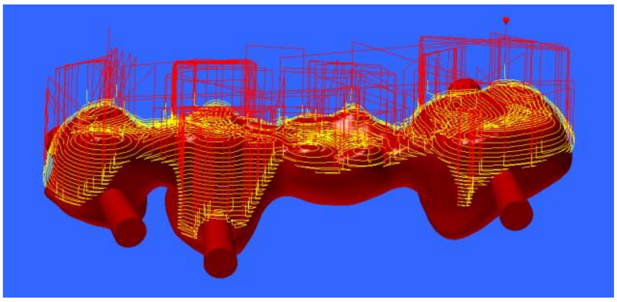
- Consistent use of this machining method is absolutely essential for materials that are difficult to cut, in order to prevent the tool from breaking and to assure precise production.
- Care must be taken to assure that the same allowance is entered as for the previous cycle.

Cycles	Automatic space left in 3X and 5X mode Roughing on arbitrary blank (with blank tracking).

3D arbitrary stock roughing

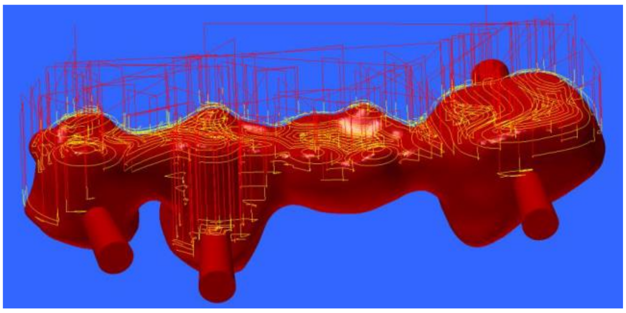
Finish pass all levels

Yes	**To prevent material from not being detected in the case of excessive stock model tolerance, leading to excessive Z-steps, especially in steep areas. The rest machining is removed plane by plane.**



No

Normal rest machining



Plane mode

Machining direction for rest machining. If you wish to create extremely short milling tracks, the planar mode should be adjusted.

Optimized in

Inside out

Caution!

"Optimized in" and "Inside out" can be set for the following jobs only:

- Rest machining inside pockets (toothpockets)
- Rest machining user-defined areas
- Rest machining inside abutment bases
- Rest machining inside copings

In the case of the following jobs:

- Overall rest machining cavity side
- Overall rest machining occlusal frame

- Rest machining outside copings, cavity side
- Rest machining outside abutment base, cavity side,

the machining is always “Optimized in”, as in these cases preroughing has always been carried out.

Automatic space left in 3X and 5X mode for rest machining inside copings

Diameter referenced tool	
	Enter the diameter of the tool from the job that has just been completed. Since the cycle has no blank tracking, the theoretical rest machining only refers to the referenced tool.
Boundary offset	Value to protect the margin line for zirconium oxide. This prevents the cycle from identifying and machining space left at the margin line.
Values < 0	The offset is generated inside the margin line. Keep the value as low as possible, up to approx. 0.5. As far as possible, use the offset in combination with an activated “Clip boundary”.
Z-level optimization	
Yes	Machining in narrow areas (pockets) is done plane by plane with depth step. Suitable for clearing out deep pockets that are deeper than the current tool diameter.

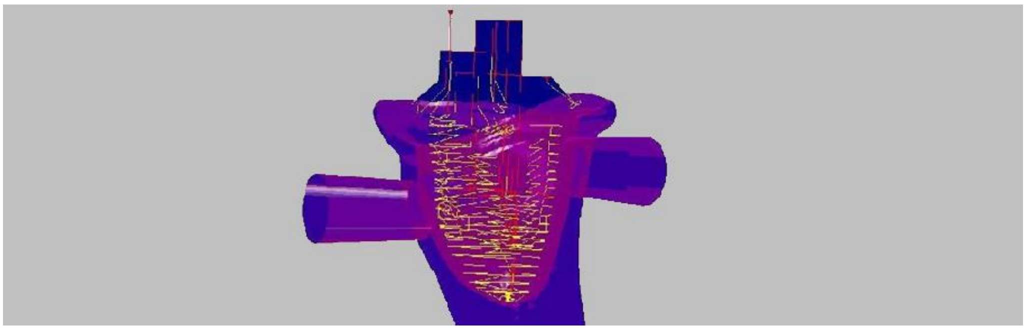


Figure 18-29

No	Z-level optimization is disabled.

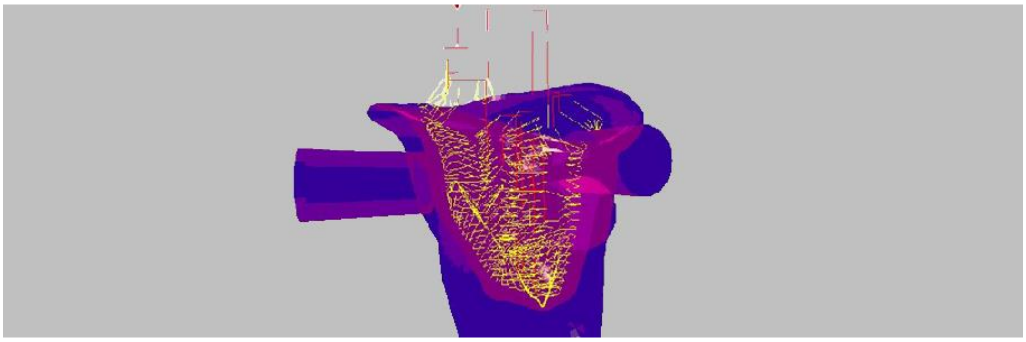


Figure 18-30

Clip boundary:

Yes	Also set the boundary of the margin line as the stop area so as to better protect the margin line from unintentional machining. Where possible, set to “Yes” if a “Boundary offset” is entered.

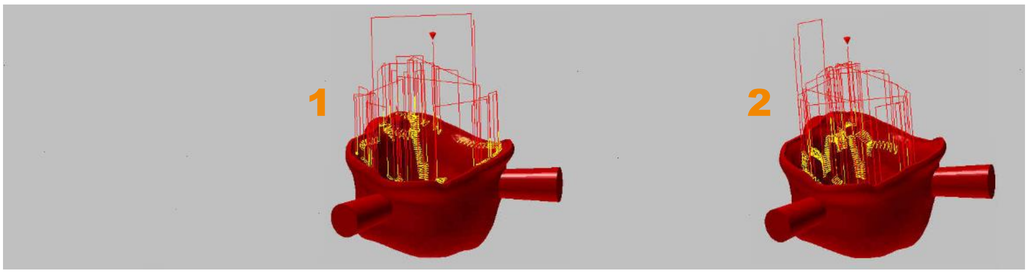


Figure 18-31

1. Toolpath without set parameter “Clip boundary”
2. Toolpath with set parameter “Clip boundary”

Strategy flat areas

Parallel	The toolpaths run in flat areas (1) parallel to the surface of the workpiece.
Normal	The toolpaths run in flat areas (2) helical to the surface of the workpiece. For safety reasons, “Normal” is suitable for material that is difficult to cut.

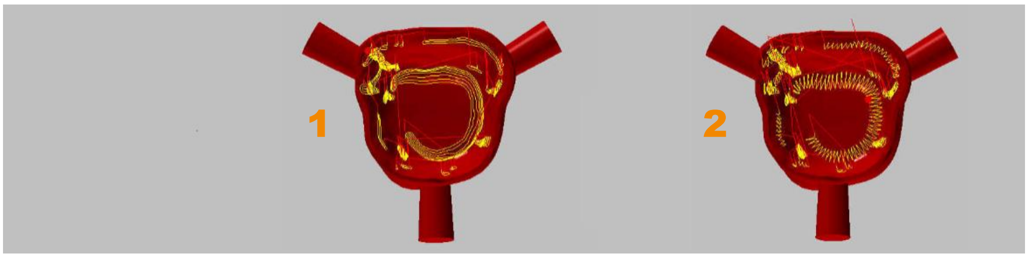


Figure 18-32

All areas		Yes
Yes	Machine all areas.	
No	Reduce milling boundary so that not all areas are machined, e.g. connectors: shorter machining time.	
Cutting depth flat areas		
	Maximum processing depth in flat areas. This value can be measured liberally if the "Normal" option is set in the parameter "Strategy flat areas" so that the existing space left is also removed.	
Slope dependent machining		Yes
	Slope dependent machining with "3D and 5X automatic restmachining" to allow for machining the space left with parallel strategies. The aim is to achieve better toolpaths when grinding.	
Yes	Proceed as before	
No	Rest machining in parallel strategy	

Automatic space left only in 5X variant

The 5X mode is suitable for shortening the tool length when machining materials that are difficult to cut.

This cycle offers no undercut machining.

Desired tilt angle		Max. approx. 5°
Tilt angle in area		Max. approx. 5°
Max. tilt angle		Max. approx. 10°
	As a rule, do not exceed the angle of approx. 10°	

18.8.7 Finishing cavity side outside of the coping areas/overall finishing, finishing occlusal side (for bridges overall)

Machining after roughing until the desired surface quality of the part is obtained and all space left has been removed.

The cycle “3D complete finishing” can be used in many cases as the universal solution.

Cycles on cavity side/occlusal side/copings/bridges/overall

3D Profile finishing normal to centercurve
5X profile finishing normal to centercurve

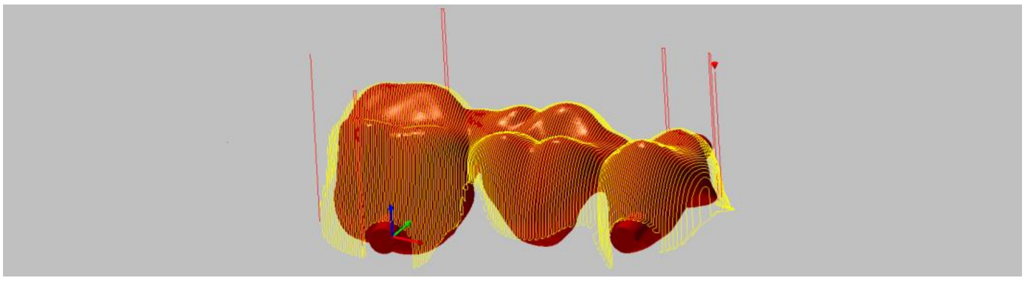


Figure 18-33

3D profile finishing strategy ruled (copings)
5X profile finishing strategy ruled (copings)

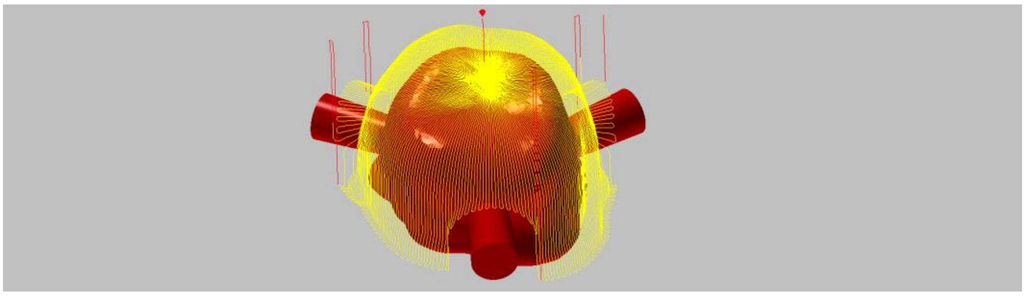


Figure 18-34

3D profile finishing X-direction (copings)
5X profile finishing X-direction (copings)

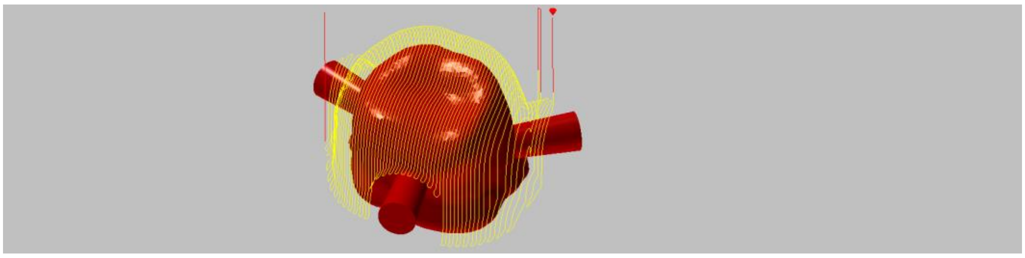


Figure 18-35

3D profile finishing XY-direction (copings)
5X profile finishing XY-direction (copings)

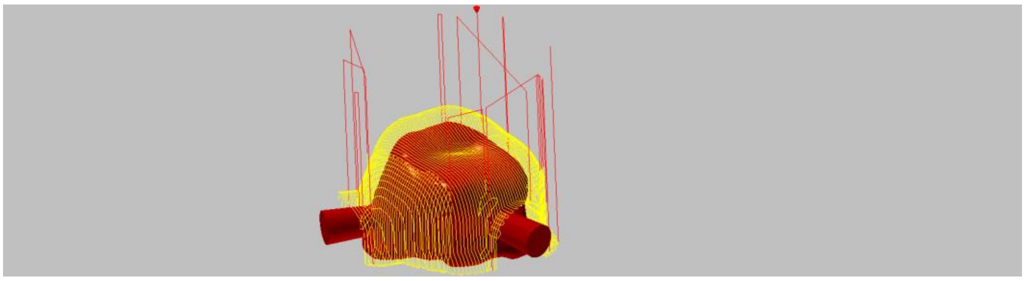


Figure 18-36

3D Z-level finishing

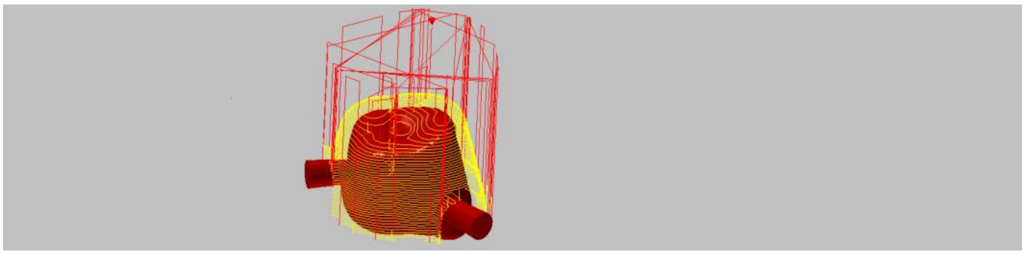


Figure 18-37

3D complete finishing

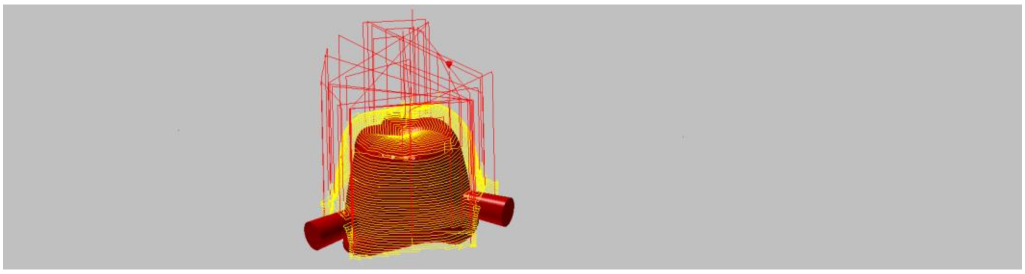


Figure 18-38

3D profile finishing pocket mode (bridges cavity side)
5X profile finishing pocket mode (bridges cavity side)

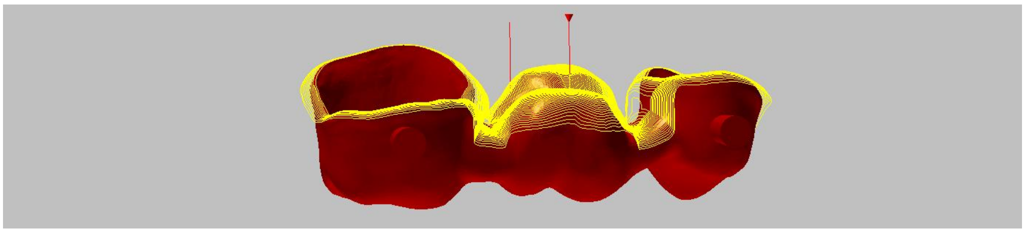


Figure 18-39

3D equidistant finishing

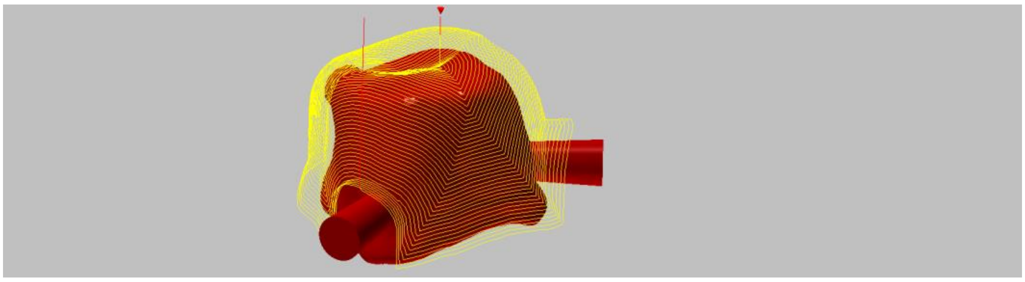


Figure 18-40

3D equidistant finishing flow (without rework)

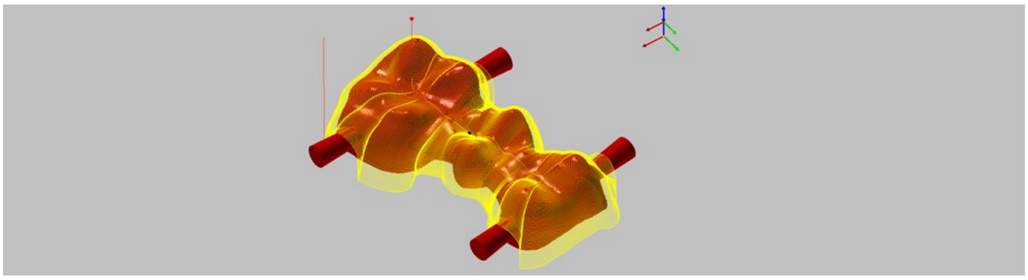


Figure 18-41

5X boss finishing (occlusal side)

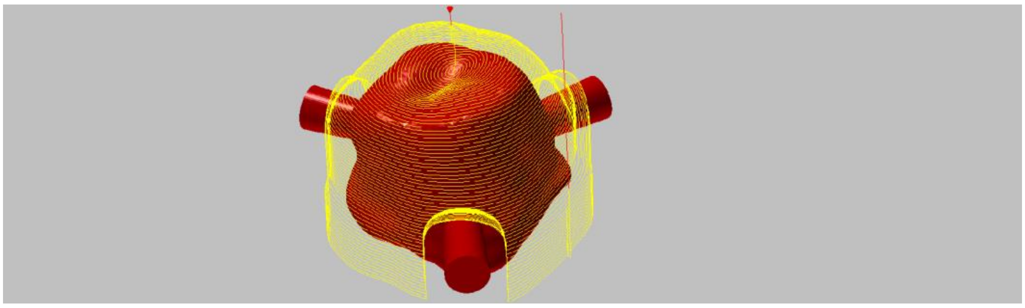


Figure 18-42

For all cycles

Max. angle for red. Feedrate	0
Value 0	Full feedrate; with a different value the feedrate is reduced during the plunge movements.

For 3D equidistant finishing and 3D equidistant finishing flow for occlusal side with bridges.

For equidistant finishing of the emergence profile and finishing outside of the coping, the setting “Add rework step” creates optimized toolpaths and increases safety in the margin line/milling boundary area.

Add rework step	Yes
Yes	Optimum surfaces thanks to equal intervals between tracks throughout the entire milling area.

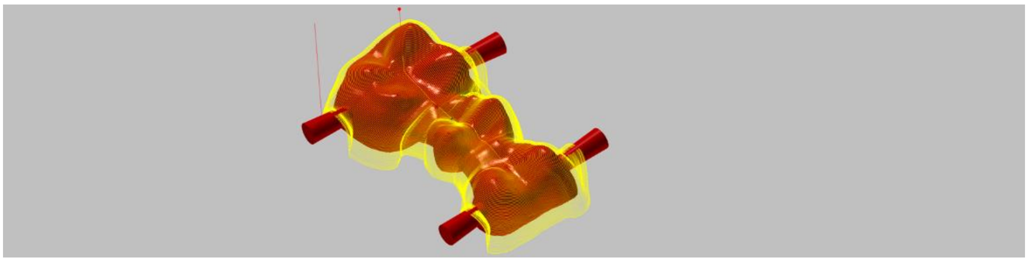


Figure 18-43

No	The return to the area of the connectors causes the intervals (1) between the milling tracks to change, which can negatively affect the surface quality.

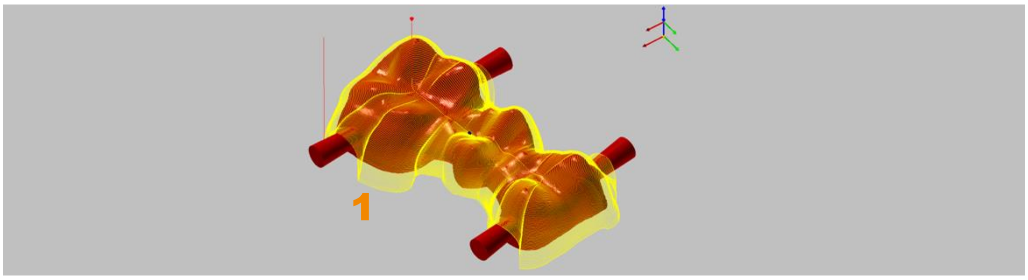


Figure 18-44

For 3D equidistant finishing

The shortest toolpaths are created for equidistant finishing.

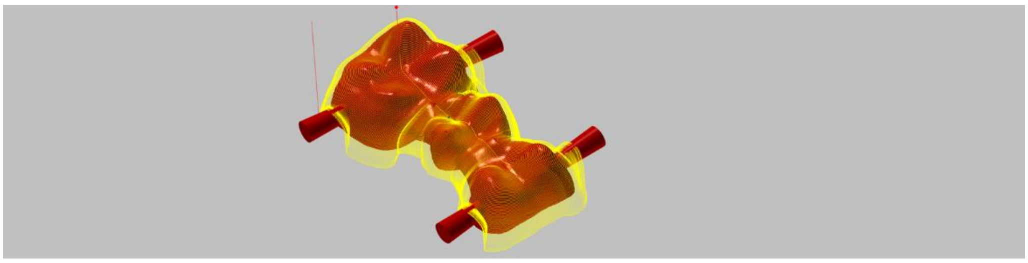


Figure 18-45

Machining direction		
From inside to outside		
From outside to inside		
Inner offset		mm
	Default value for machining from outside to inside.	

For all 5X cycles without undercut machining option

Desired tilt angle		Max. approx. 3°
Tilt angle in area		Max. approx. 4°
Max. tilt angle		Max. approx. 5°
	Set angle graduated in the area up to max. approx. 5°. In the case of 5X profile finishing for very jagged parts especially, the calculation may otherwise be aborted.	

For all 5X profile finishing cycles with undercut machining option

The option can be identified on the “Undercut machining” switch.

Desired tilt angle		≥ 10°
Tilt angle in area		≥ 10°
Max. tilt angle		≥ 10°
	Liberally measure the angle, set ≥ 10°. If necessary, also extend the milling boundary.	
Max. undercut depth	Maximum possible area for undercut machining.	
Calculation	Tool radius – 0.05 mm	

For all cycles machining occlusal side

Occlusal tilt with undercut machining for 3+1/3+2 machines with settings for the machining parameters:

Rotation axis

Multi axis machining

Differential angle for finishing

The part is rotated in a plus and in a minus direction depending on the rotation axis entered, and the toolpath is recalculated. Up to 4 jobs are created.

Use occlusal direction	
Yes	Required if --> “Occlusal insertion direction” has been set so that the milling process also takes place.
No	For machining without “Occlusal insertion direction”.
Multi axis machining	No
Yes / No	Perform undercut machining for 3+1 machines.
Add. angle	Differential angle for finishing.
Rotation axis	Rotation axis for the tilt.
X+Y	Rotation axes for 3+2/5X machines.
Y	X rotation axis, for 3+1 machines.
Y	Y rotation axis, for 3+1 machines.

The milling process takes place with the corresponding tilt if the axis entered is suitable for the machine being used.
Important, e.g. for angled abutments, to avoid unmachined areas in undercuts.
The function is only available for individual copings.

Example of multi axis machining

Multi axis machining	*Yes*
Add. angle	*10*
Rotation axis	*X*

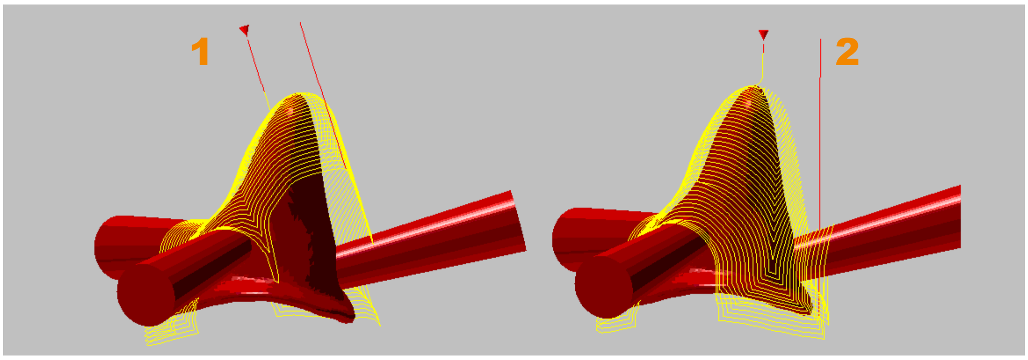


Figure 18-46

The first toolpath (1) is calculated with a tilt of 10 degrees and the second toolpath (2) with a tilt of -10 degrees.

Clip sintering pins Yes

Yes	Do not machine the optional sintering pins with this job. As a rule, it is enough to only machine the sintering pins using the roughing cycle. This can reduce the machine runtime without affecting the quality.

Add. margin line allowance

Allowance to protect the margin line that is added to the value --> “General setting” > “Margin line thickness”.
Important when finishing zirconium oxide so that the margin line area is not machined any further as it generally has already been fully machined.

Overall finishing occlusal side (bridge)

Tooth pockets closed

In dental prosthetics or models, the defined tooth pockets are closed and so not machined.

Yes	Tooth pockets remain closed for the finishing.
No	Tooth pockets are also machined.

For all cycles machining cavity side

Protect margin line	
Yes	Protection of margin line, important when finishing zirconium oxide so that the margin line area which generally has already been fully machined, is not machined any further (is omitted during machining or the mill is lifted out of the material intrusion beforehand).
Offset for margin line protection network	
	Standard setting 0.05 mm
	Adjustable value for a plane above the margin line for its protection during machining, if separate machining of the margin lines is activated.

For 3D complete finishing

Slope dependent machining

Yes — Slope dependent machining so that the tool is lifted less often from the part: Yes/No.

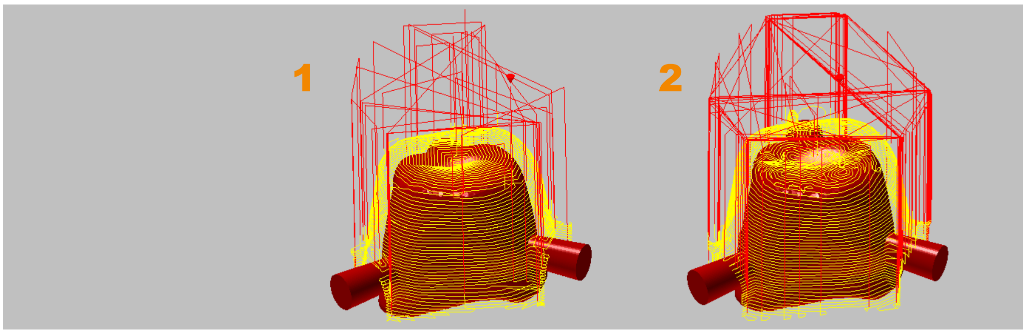


Figure 18-47

1. Yes: Slope dependent machining enabled
2. No: Slope dependent machining disabled

Machining method	Limit machining in a job to steep or flat machining areas. Only available for machining method “Finishing inlay/onlay”.

Split angle	degrees Angle for split into steep and flat machining areas. Only available for machining method “Finishing inlay/onlay”.
Step down	mm The step down for the steep areas can be set regardless of the horizontal step. The groove depth must be coordinated with the length of the cutting part of the tool. The depth step is repeated until the part is reached.
Work order flat areas	
From inside to outside	
Outside in	Pathway of milling tracks on the occlusal side.
Strategy steep areas	
Oneway	Machining in synchronization only: Better surface quality, longer runtime.
Zigzag	Machining in synchronization and in opposite direction: Shorter runtime, poorer surface quality.

For 3D profile finishing pocket mode (bridges cavity side), 5X profile finishing pocket mode (bridges cavity side)

Add. outer Z-level finishing	
Yes	Recommendation is “Yes” since the profile finishing cycle generally does not create workable milling tracks in steep areas. The cycle is used so that all areas are machined up to the Z-level boundary area (general boundary area for the machining of the occlusal and cavity side).
Advantage	Cycle calculates quickly. Nice milling tracks are created with (in the combination) relatively few lifting movements.
Disadvantage	The steep areas in the interdental space are not optimally machined.

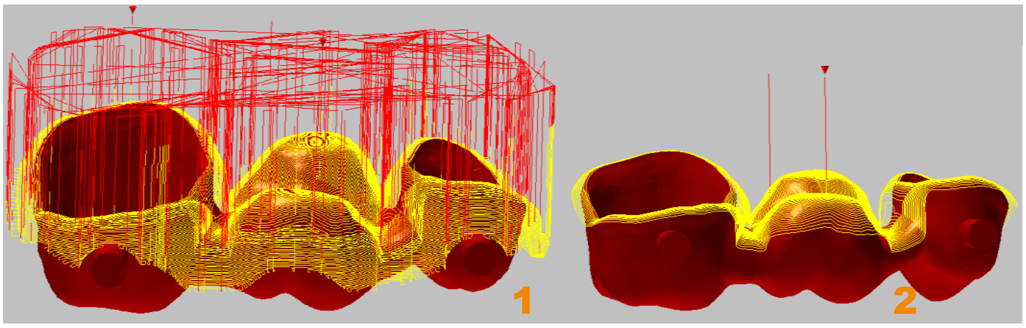


Figure 18-48

1. Yes: Add. outer Z-level finishing is enabled
2. No: Add. outer Z-level finishing is disabled

18.8.8 Finishing any side

Machining after roughing on any side until the desired surface quality of the part is obtained and all space left has been removed.

Cycles	3D Z-level finishing 3D complete finishing 3D equidistant finishing, suitable for finishing on the front side of Blue Blocks. 5X Z-level finishing Radial 5X equidistant finishing, possible to radially retract and omit/skip (clip) the connector, e.g. for grinding on the connector side of Blue Blocks
Side	Front
Front / Back / Left / Right / Top / Bottom	Enter the side to be machined; corresponds to the view direction for setting [Front view].
Machining depth	Parameters for the machining depth on the blank.
Max. of bounding box	Machining area from “back” view on the blank (bounding box).
Min. of bounding box	Machining area from “from back to front” view on the blank (bounding box).
Center of bounding box	Machining area from “from back to center” view on the blank (bounding box).

Depth offset	mm Additional value for the machining depth. A larger value results in deeper machining.
Use/transform fixture boundary	
Yes	When machining a side of the blank, use milling boundaries for fixture.
No	When machining the front side (from the view of the fixture).
Tilt strategy	
Automatic	The lead angle is automatically calculated.
Radial Z	Enter lead angle manually.

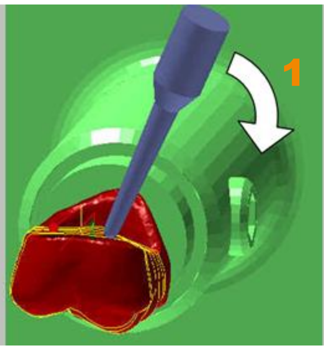


Figure 18-49

1. Lead angle

Lead angle	Value for the lead angle.

Desired tilt angle/max. tilt angle

Tilt angle in relation to the Z-axis.

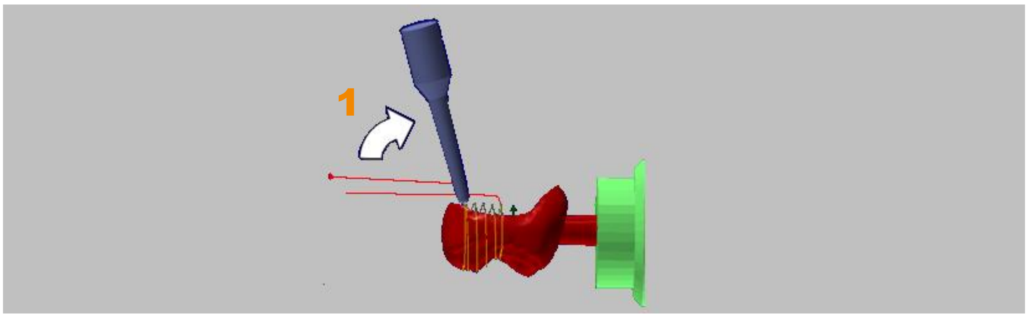


Figure 18-50

1. Tilt angle

Max. avoidance angle A/B	Maximum angle to avoid collision with regard to the A/B-axis.
Center at connector position	Yes
	To gain clean milling tracks around the connector.
Yes	The rotation point for the milling tracks around the connector is the center of the connector.
No	The rotation point for the milling tracks around the connector is the center of the part.
Invert infeed direction	Invert the machining direction.
Close copings	No
Yes	"Yes" must be entered if the setting --> "Special function coping-specific alignment" has been set for the part. "Yes" must also be entered for the following finishing cycles of the cavity.

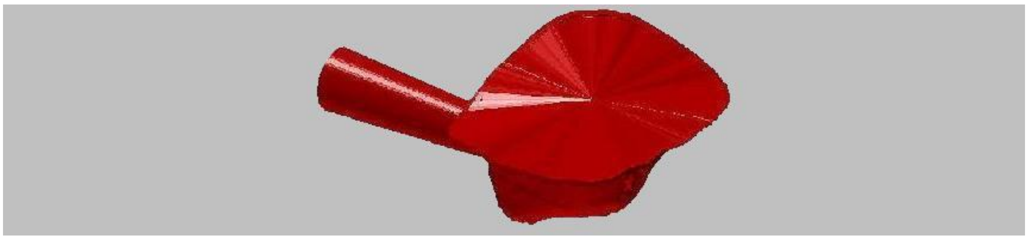


Figure 18-51

18.8.9 Roughing, finishing with grinding (optional)

Axis parallel roughing of glass ceramics and lithium disilicate with diamond-coated tools. Particularly well suited for use with 3+1 machines for which other roughing strategies are not possible for kinematic reasons.

The cycle can be configured in such a way that the toolpath only runs in an ascending direction.

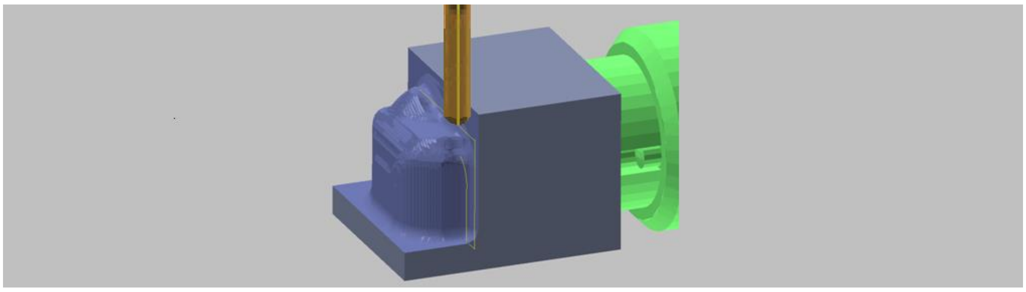


Figure 18-52

Cycles	3D axis parallel peeling Drilling inside copings (preparation for machining with 5X peeling). 5X peeling 3D U peeling
Machining direction	X/Y, positive/negative Entry of machining direction in X/Y positive or negative direction.
From inside to outside	
From outside to inside	
Type of cut	Way in which the tool movement is to take place.
Ascending	The tool moves only in an ascending direction during machining. Recommended machining for glass ceramics and lithium disilicate. A full cut is not permitted. For internal machining of copings, for example, the area must be predrilled, so that the tool can plunge.
Ascending + descending	The tool moves in an ascending and descending direction during machining. A full cut can take place.

- Risk of tool breakage due to full cut with grinding tools and with tools that are not designed for plunging into the material on the front side.

Step down	mm Step per cut. The maximum step is dependent on the tool and the blank. As a Z-limit boundary, a plane is always created at the bottom of the equator. The machining always occurs orthogonally to the blank – irrespective of the setting in the template. If the length of cut of the selected tool is sufficient, the step down can correspond to the value of the blank depth for the purposes of optimal calculation and milling times.
Begin of peeling	Start of machining on the blank.
Automatic	
Max. of bounding box	Start of front side, from “back” view on the blank (bounding box).
Center of bounding box	Start of center of blank (bounding box).
Min. of bounding box	Start of holder side, from “back” view on the blank (bounding box).
End of peeling	End of machining on the blank.
Automatic	
Max. of bounding box	Machining area from “back” view on the blank (bounding box).
Center of bounding box	End of center of blank (bounding box).
Min. of bounding box	End of holder side, from “back” view on the blank (bounding box).

3X peeling, roughing every side

Convex shell

Yes	Generate a convex shell and position it around the part for calculating the toolpaths.

The convex shell avoids excessive axial steps (2), e.g. during 5X Z-level finishing, and provides greater security during the grinding process.

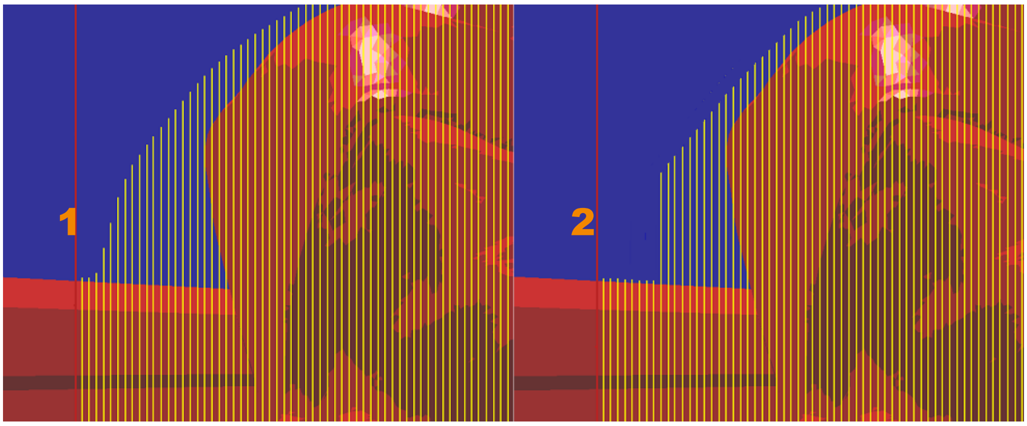


Figure 18-53

1. Toolpaths with convex shell
2. Toolpaths without convex shell

As the highest point on the margin line (1) coincides with the convex shell, and the milling path when finishing the margin line and when finishing inside copings/inlay always begins at the highest point, this also ensures greater security in the grinding process.

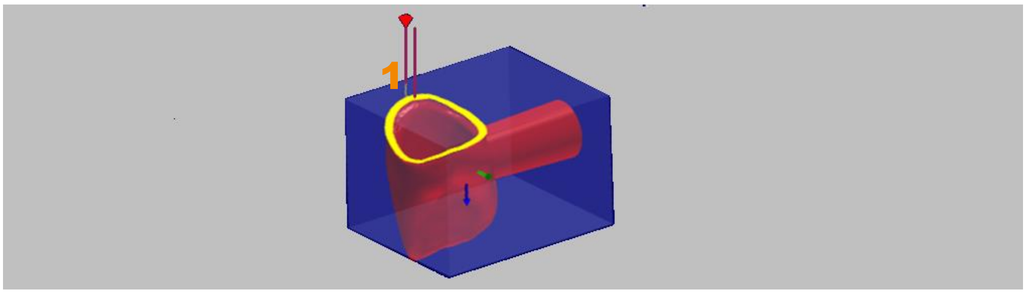


Figure 18-54

Extrude margin	Shift the beginning of the milling paths beyond the margin line (1) to ensure that you can even reliably machine complex inlays/onlays with a simultaneous offset for milling radius + allowance + tolerance (steep areas). The application is possible during roughing and finishing.

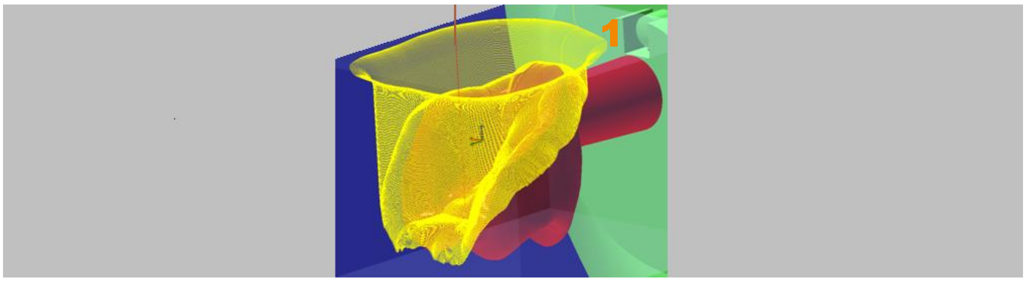


Figure 18-55

No	No extrusion of the machining margin.
Maximum of part	Extrusion up to the maximum machining margin of the part.
Maximum of blank	Extrusion up to highest point on the blank end.
Blank surface	Extrusion up to the blank surface.
Automatic	Automatic extrusion by the program.

Distance to connector interface point mm

The distance between the interface point of the connector on the part (2) and the end of the convex shell in the middle of the connector (3) determines the progress of the convex shell (4).

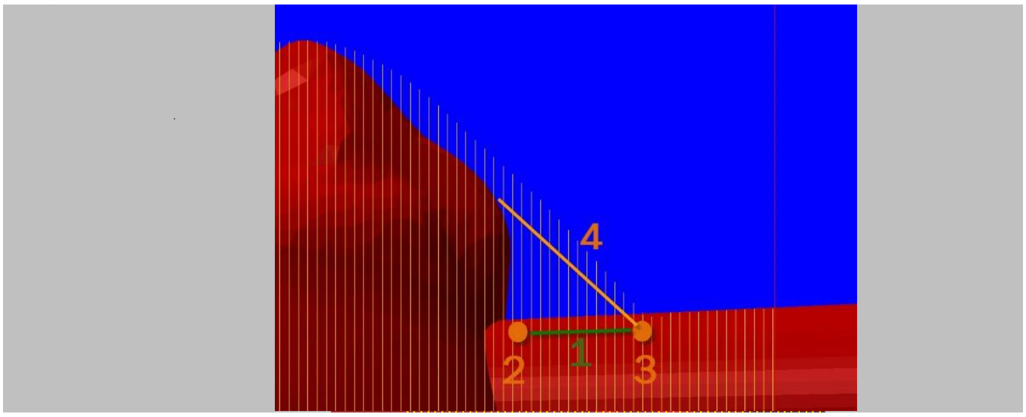


Figure 18-56

1. Distance to connector interface point
2. Connector interface point on part
3. End point of convex shell
4. Convex shell

Spiral equidistant finishing	The toolpaths run in spiral form at equal distances from the inside to the outside.

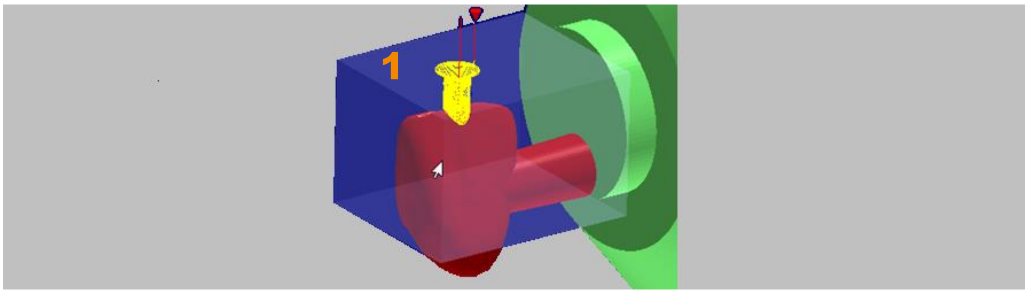


Figure 18-57

Machining direction

From inside to outside

From outside to inside

Offset	mm Default value for machining from outside to inside.
Offset inner boundary (margin line)	mm Default value for inner machining margin (margin line).
Offset outer boundary	1.25 mm Default value for outer machining margin.

18.8.10 Fissure machining

Only for full wax-up crowns and bridges.

Cycles	3D automatic space left 3D arbitrary stock roughing

The cycle “3D arbitrary stock roughing” at the end of a job list can result in the calculation time being significantly extended due to the blank tracking.

Once the occlusal side has been fine finished, it may be necessary to rework the fissures using a finer tool.

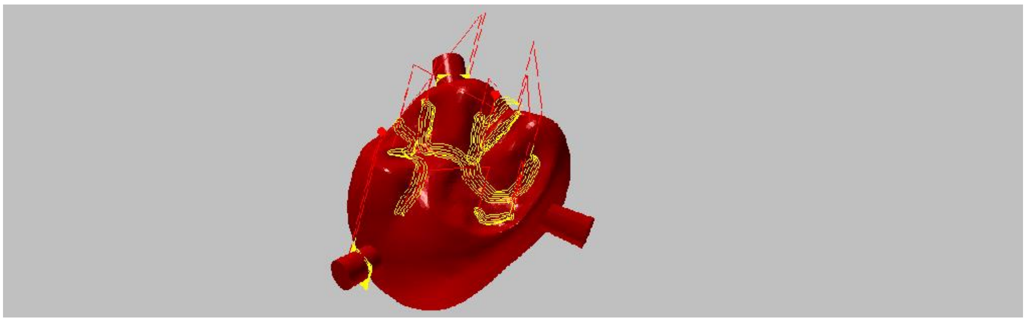


Figure 18-58

Diameter referenced tool		
	Enter the diameter of the tool from the job that has just been completed. Since the cycle has no blank tracking, the theoretical rest machining only refers to the referenced tool.	
Max. angle for red. feedrate		0
Value 0	Full feedrate; with a different value the feedrate is reduced during the plunge movements.	

18.8.11 Cutting, reducing, breaking connectors

Once the actual machining has been completed, the connectors are cut or reduced, for which a --> “Cut” has been set in the process step --> “Set connectors”.
Cuts can be set for no or all connectors, either completely or in part --> “Set connectors”, “Cut”.

Cycles	Cut/reduce connectors Reduce or completely cut connectors from one or both sides.

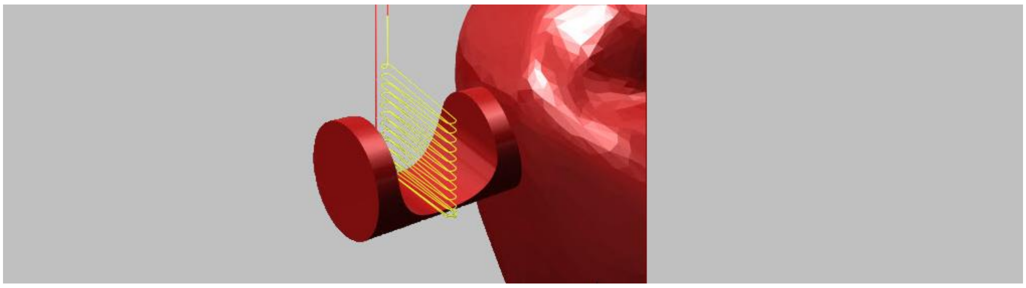


Figure 18-59

3D freepath milling
The cut follows the outline (viewed in the machining direction).

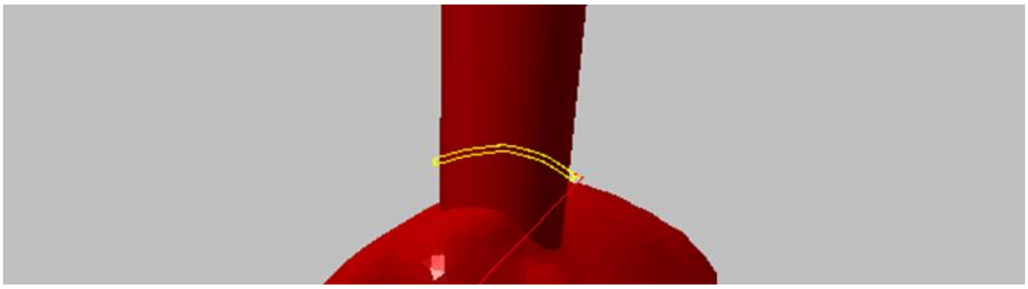


Figure 18-60

3D arbitrary stock roughing
The cut follows the surface (viewed in the machining direction). The connector is cut from the part without residue.

If 3D arbitrary stock roughing is used for the connector machining, the fixture allowance must not cover the connectors! The fixture allowance is not detected and this may lead to a collision!

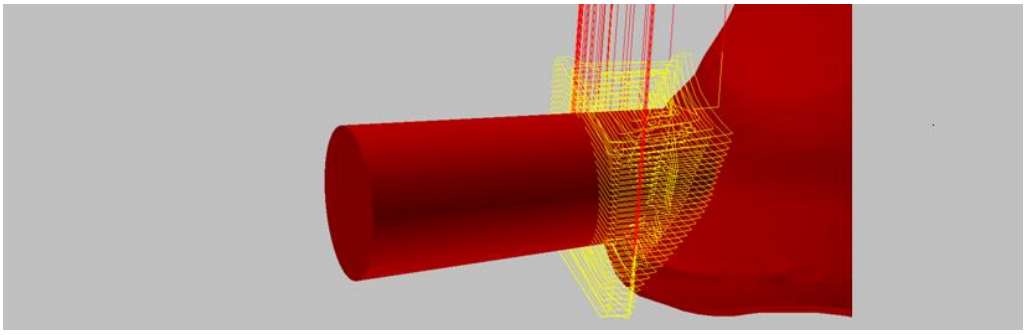


Figure 18-61

Additional milling cut on cavity side
Break connectors.

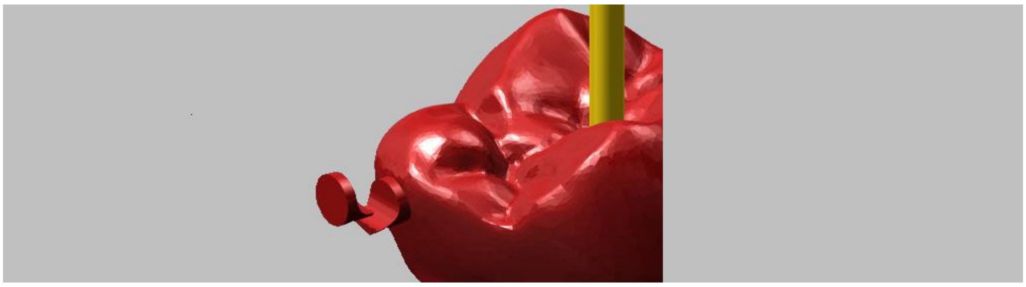


Figure 18-62

Cutting/reducing connectors

The order of the cuts for bridges is always from outside working in.

Connectors	Selection of connectors that are to be machined with this job.
Connectors to reduce	Only for connectors that are to be reduced: Cut setting < 100%.
Connectors to cut	Only for connectors that are to be cut: Cut setting = 100%.
All connectors	

Side	Occlusal side/cavity side Predefined (primary) side for machining. Cuts can be set from both sides. The required offset to the equator (viewed in the machining direction) is generated automatically.

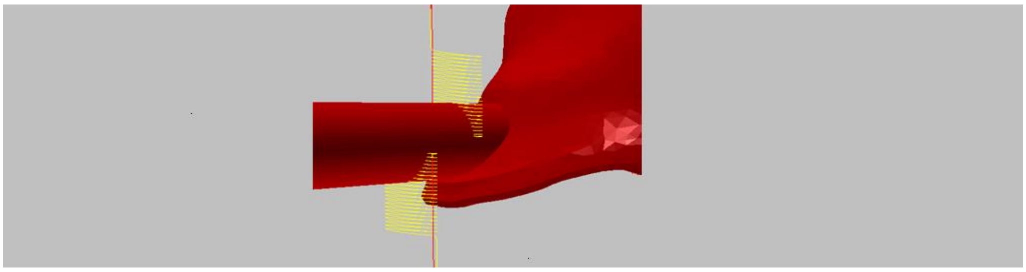


Figure 18-63

From both sides	
Yes	Generates cuts from the occlusal and cavity sides. Caution! Do not place cuts from the cavity side in undercut areas.
No	Only cuts from the occlusal or cavity side.

Reduce percentage — 0 - 100 %

Specifies for the connector reduction how much of the predefined cut depth is to be machined: --> setting [Milling cut] > [Cut depth].

Allowance part — mm

Additional distance (1) to the part for the cut.

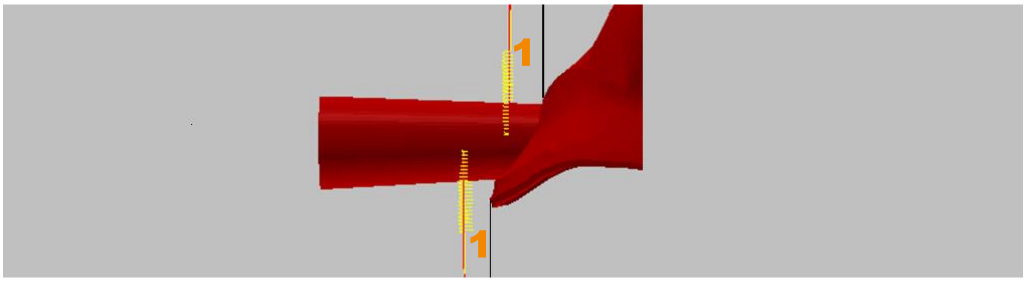


Figure 18-64

Contour parallel — No

Yes — The cut follows the outline of the part.

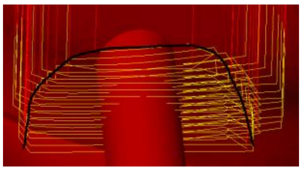
Additional cutting height — mm

Sets the start of the machining above the connectors in order to take into account the allowance after the roughing.

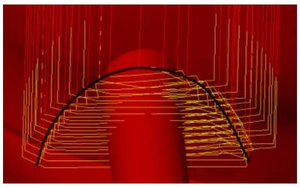
Rectangular box — No

For shortening the milling tracks/processing time in the case of connectors.

Yes — A rectangular box is placed around the connector for the calculation of the milling track.



No — The milling track calculation is carried out directly on the connector.



NC text (sequence)	
	Free NC texts for the entire connector machining. For the use of part collectors. Advance before the first connector and retract after the last connector.
Additional milling cut on cavity side	
Yes	Additional cut, e.g. for automation solutions to assure complete cutting out.
No	No additional cut.
Reduce percentage	Depth for the additional cut. The global settings for the cut are not copied here.
Breaking connectors	
Speed 0	Set spindle speed to 0.
Feedrate	Adapt to the material.
Machining depth	Penetration depth of the tool on the part.

18.8.12 Cutting the sinter frame

With the appropriate template parameters, the inner area can be pushed out of the frame using light pressure.

Cycles	3D freepath milling The cut follows the outline (viewed in the machining direction).

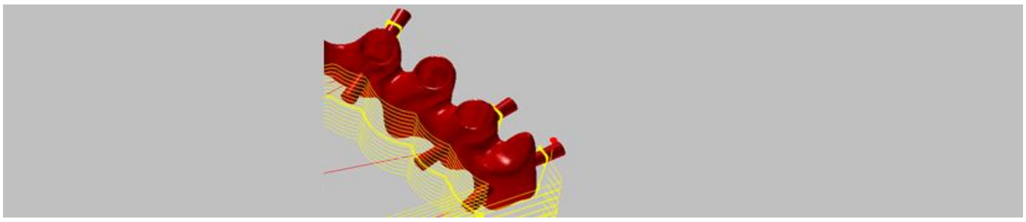


Figure 18-65

Side	Side for machining.
Occlusal side/cavity side/both sides	

Remaining material height, inner frame 0.1 - 0.05 mm

Remaining material thickness after the cut, so that the inner section (1) of the frame can be pressed out.

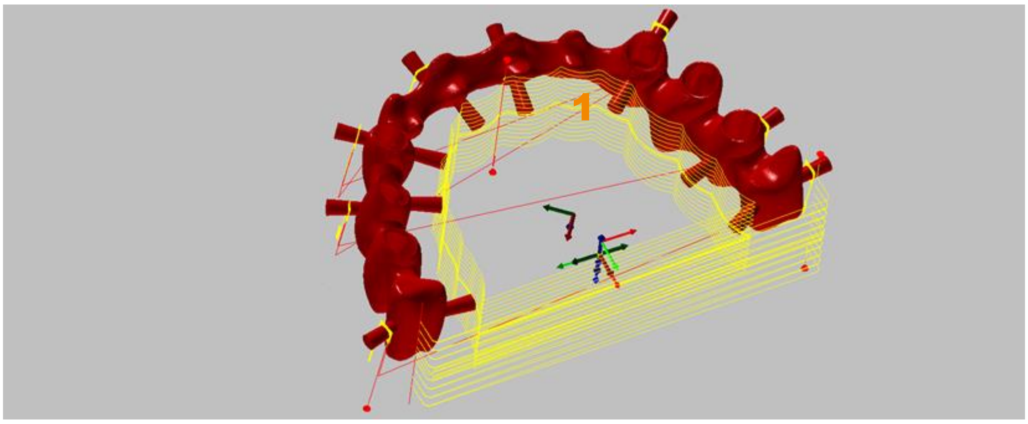


Figure 18-66

Remaining material height, base connectors 1 mm

Material thickness for an additional connector (1) in the area at the base of the sinter frame.

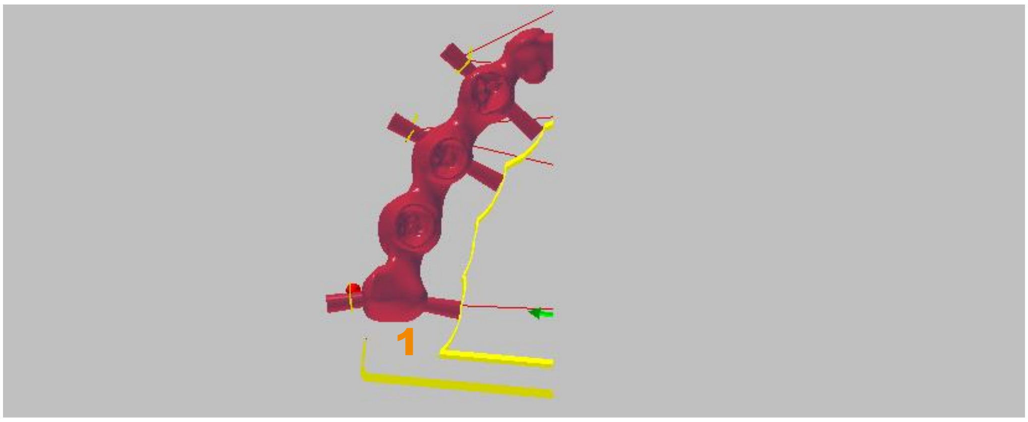


Figure 18-67

18.8.13 Machine splints

Machining of long cavities for the production of bite splints.

The splints are positioned as protection over the existing teeth and must surround them down to below the maximum tooth thickness to ensure a secure fit. That is why the splints have undercuts. The undercuts extend over the entire area of the cavity and feature opposite angles for the insertion and machining directions on both sides of the cavity.

Depending on the settings in the template, hyperDENT® may automatically detect these long cavities on the splint and divide them up into individual segments for machining on 3+2 machines.

- *Automatic division into segments for the machining of the long cavities.*
- *Separate machining of the end areas.*
- *Machining of a splint for an entire tooth sector with gaps.*

Example: splint with 3 segments (1, 2, 3)

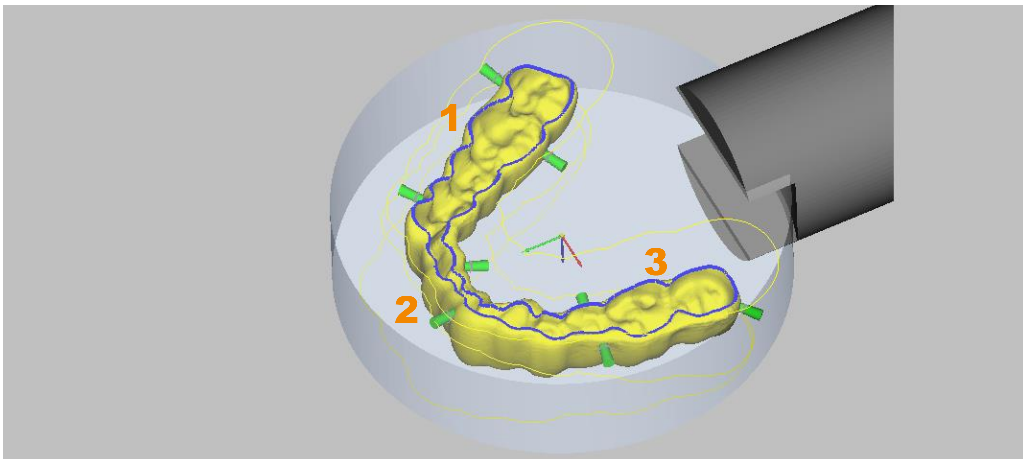


Figure 18-68

Machining takes place in six or more steps, for example:
3 segments with 2 different angles each for the tilts.

Cycles	Segmented profile finishing Segmented profile finishing, XY-optimized Segmented equidistant finishing

Finishing on the inside of long cavities

Angle difference of the segments

Angle deviation of the center line of the cavity as an automatic differentiation in part sections (segments). The smaller the value, the more segments will be created.

Angle of end segment

Angle for the size of the end segment.

0	No end segment.
>0	End segment is being created.

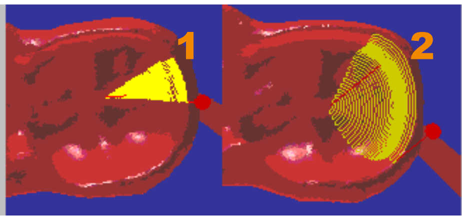


Figure 18-69

1. End segment angle of 30°
2. End segment angle of 120°

Desired tilt angle — Tilt angle for the machining of the segment.

Finishing inside coping

Within a splint for an entire tooth sector with gaps, there may be short cavities for individual teeth that can be machined with this job.

Calculate if — Short cavity

Short cavity	Calculation is performed only if there is a short cavity. Necessary for the machining of individual teeth within a splint, e.g. In the event of tooth gaps.

18.8.14 Drilling inside copings

Predrilling for grinding of the coping and machining with upward tool movement only; no full cut for 5X peeling.

Figure 18-70

Cycles

Simple drilling
The drill hole is created in a step, e.g. for center drilling, predrilling.

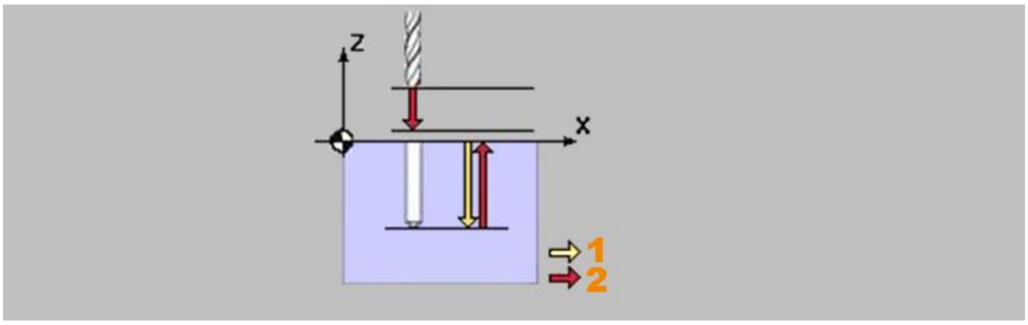


Figure 18-71

1. Feedrate
2. Rapid traverse

Drilling with chip break
The drilled hole is created in several steps (Z1, Z2, etc.).
After each drilling stroke, the tool retracts in rapid traverse mode around the retract value (3): Shorter chips, better removal of chips.
The step is reduced by the reduce value after each drilling stroke.

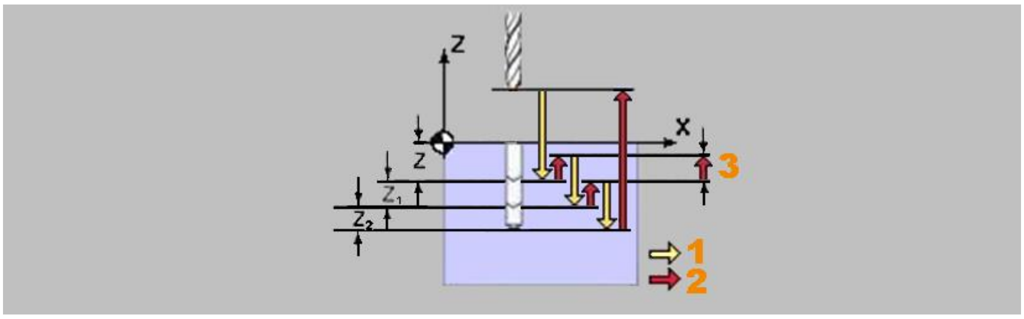


Figure 18-72

1. Feedrate
2. Rapid traverse
3. Retract value

Drilling with pecking The drill hole is created in several steps (Z1, Z2, etc.).
After each drilling stroke, the tool retracts in rapid traverse mode to the clearance distance (3): Clear the drill hole.
The step is reduced by the reduce value after each drilling stroke.

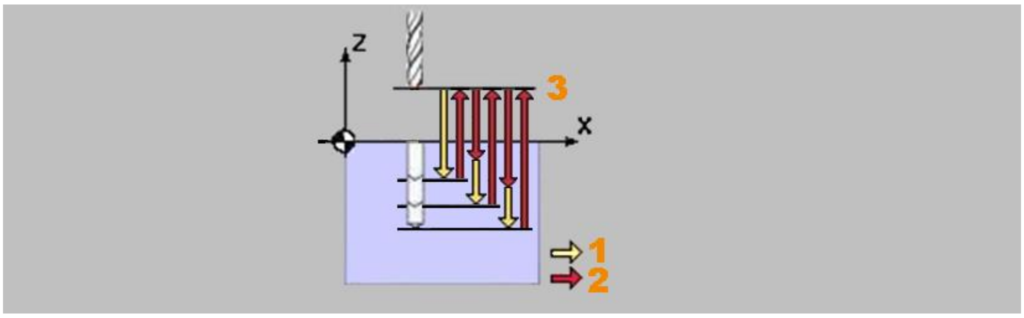
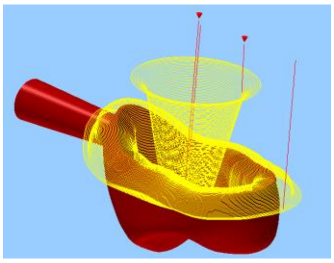


Figure 18-73

1. Feedrate
2. Rapid traverse
3. Clearance distance

Spiral milling
Cut the precise final diameter of the drill hole in a spiral milling procedure.

The combination of predrilling with a drill and spiral milling of any diameter permits quick, flexible production with just a few tools.

Hole diameter	mm Final diameter of the drill hole with spiral milling.
Funnel angle	(With 5X boss finishing only) Angle for opening the drill hole 
Side Front Back Left Right Top Bottom Inside of copings Occlusal insertion direction	Side where machining takes place.
Position	Position for the drill hole. This information is important for the position of subsequent machining.
Mid of coping	The drill hole is made in the center of the coping.
Deepest point in coping	The drill hole is made at the deepest point of the coping.

Caution! Synchronization with the predrilling job of the step point in the center during “Overall finishing cavity side (coping) – 3D equidistant finishing flow”

- Prerequisites:
  - Predrilling job => Side => Bottom
  - General settings in the job => Machining outside the copings, orthogonally to the blank => Yes

Top	Blank Determination of the start of the drilling.
Blank	Blank surface.
Convex shell	Highest point of the convex shell.
Automatic	The program determines the starting point.
Offset top	mm Additional offset for the start of the drilling.
Pecking depth	Stepdown of tool in mm, in the first drill stroke.
Reduce value	Reduction of the stepdown after each additional drilling stroke.
Stepover	mm Stepover in spiral milling.

18.8.15 Machining user-defined areas

For particular part areas, you can define user-defined areas and allocate separate jobs to them, which you then adapt to the required machining.

Cycles	Arbitrary stock roughing (space left) 3D automatic space left 5X automatic space left 3D Z-level finishing 3D complete finishing 3D profile finishing X-direction 3d profile finishing XY-direction
Category	- 0 - 9 - n Number for this machining process. Use this number to assign this machining process to a user-defined area. The calculation only happens if an area is given the corresponding category number: --> “Identify part features” > “User-defined area” and “Split the machining area into sections”. In this way, it is possible to machine particular part areas with the ideal template.

Complete machining for a category
Space left
Finishing

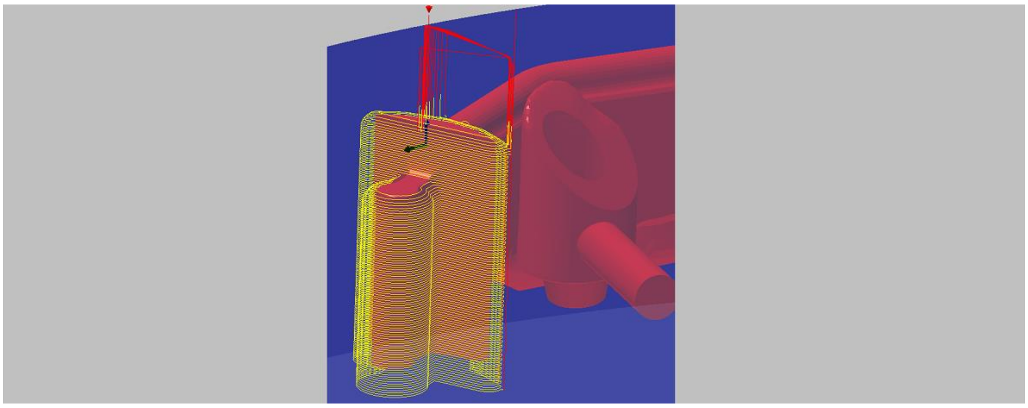
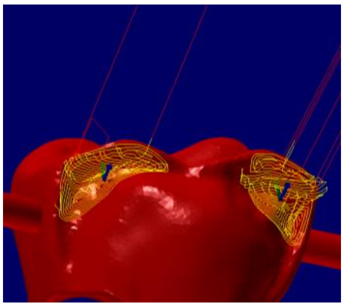


Figure 18-74

It is possible to machine more than one user-defined area with just one job. Every area to be machined must be entered in the machining categories of the job. Separate the machining categories with “/”, e.g. 1/2

Job template	Finishing userdefined areas
Jobname	Finishing userdefined areas
Calculate	☑ Yes
Strategy	3D Complete finishing
Boundary check	Shank
Tool	Zi Ballmill d1,0x14
Spindlespeed	30000
Feedrate	1000
Reduced feedrate	500
Max. angle for red. feedrate	90
Coolant	From general settings
Machining categories	1/2
Machining direction	Insertion direction
Machining depth	Max. of bounding box
Depth offset	0



Machining direction

This makes it possible to even machine the opposite side of the user-defined area.

Insertion direction

Opposite insertion direction

Machining depth

Max. of bounding box

Min. of bounding box

Center of bounding box

Equator of part

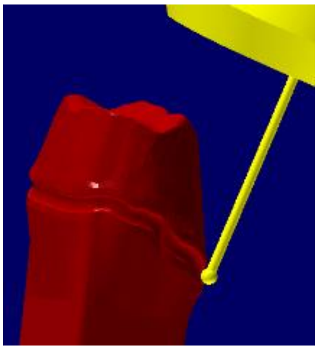
Depth offset		
	>0	Machining deeper
	<0	Machining higher

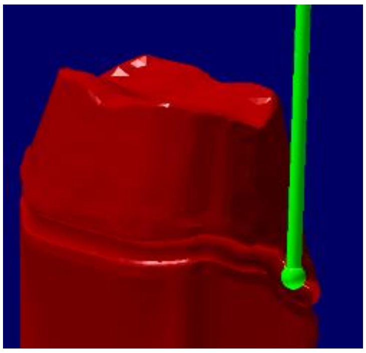
18.8.16 Concave molding

This job makes it possible to mill concave moldings on model stumps.

Cycles	3D freepath milling 5X freepath milling
Max. undercut depth	mm Value of the maximum undercut depth of the concave molding

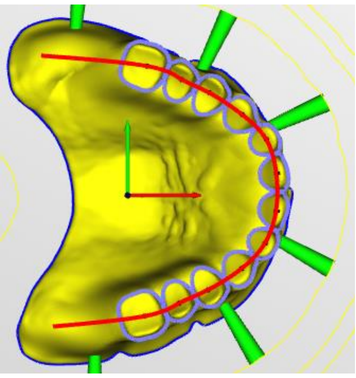
Caution!!! No collision check is available for these jobs.
The margin line must touch the upper edge of the concave molding.



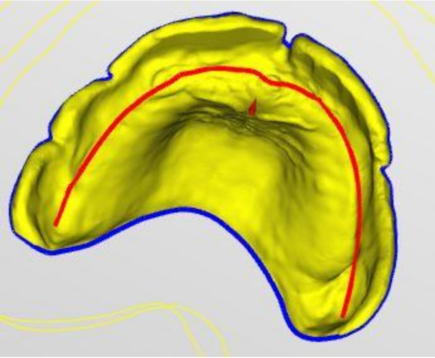


18.8.17 Full denture

Special jobs were developed for machining full dentures. They are based on the margin-line types Tooth pocket and Alveolar ridge (alveolar ridge line and tooth center line).



Tooth center line + tooth pockets



Alveolar ridge line

These machinings work only with a valid license for the full denture option.

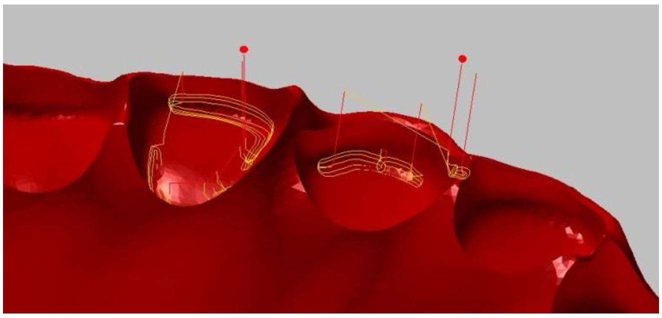
At present, CAD interfaces are available for the transfer of the tooth center line, the alveolar ridge line and tooth pockets from Dental Wings, 3Shape, and the Merz Dental Baltic Denture System.

Rest machining within tooth pockets

Rest machining of the tooth pockets, similar to the rest machining of copings. This job is based exclusively on the “Tooth pocket” margin-line type.

Available strategies:

- 3D arbitrary stock roughing
- 3D automatic rest machining
- 5X automatic rest machining



Rest machining inside the tooth pocket – 5X automatic rest machining

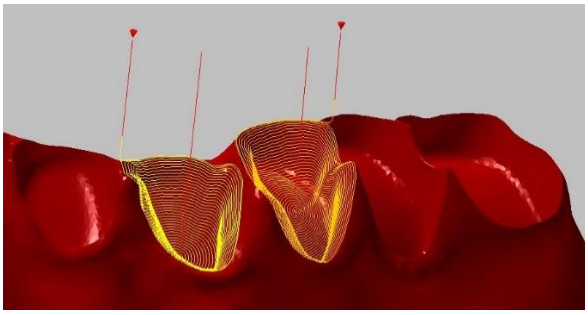
Finishing inside pockets

Finishing inside the pocket, similar to finishing in copings.

This job is based exclusively on the “Tooth pocket” margin-line type.

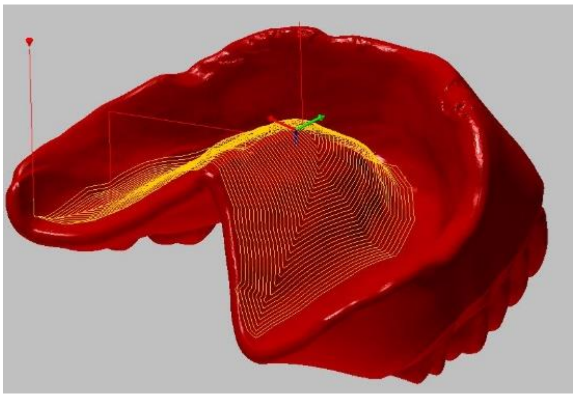
Available strategies:

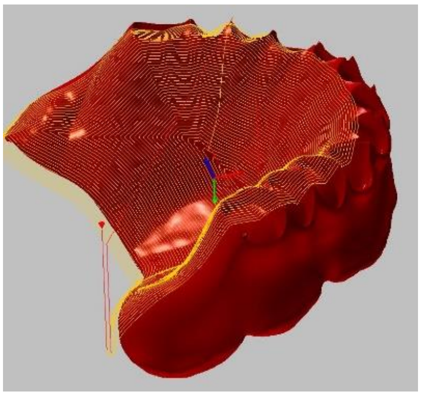
- 3D equidistant finishing
- 3D complete finishing
- 5X boss finishing



Finishing inside the pocket – 3D equidistant finishing

Finishing inner areas full denture





Finishing inner areas cavity side   Finishing inner areas occlusal side

hyperDENT Jobname	Finishing inner areas upper denture
Jobname	Finishing inner areas upper denture
Berechnen	☑ Ja
Berechne wenn	Aus
Strategie	3D - Profilschlichten - X-Richtung
Boundary-Prüfung	Werkzeugschaft
Werkzeug	PMMA Ballmill D2,0x40
Drehzahl	30000
Vorschub	3500
Reduzierter Vorschub	2000
Max. Winkel für red. Vorschub	90
Kühlmittel	Aus allgemeinen Einstellungen
side	Cavity side
Max. Eintauchtiefe	Unbegrenzt
Offset alveolarridge	2
Kappenspezifische Anstellung verwenden	X+Y Drehung
Aufmaß	0
Seitliche Zustellung	0,1

Parameters for finishing inner areas

Available strategies:

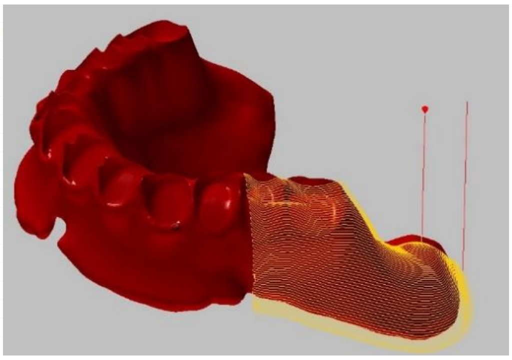
- 3D equidistant finishing
- 3D profile finishing X-direction
- 3D profile finishing XY-direction

Side: For the selection of the machining direction. Occlusal or cavity side.

Offset on alveolar ridge: The offset is based on the alveolar ridge line on the cavity side and on the tooth center line on the occlusal side. It shifts the machining toward the outside (+) or the inside (-).

Finishing outer undercut areas

hyperDENT Jobname	Finishing outer undercut areas full denture
Jobname	Schlichten hinterschnittbereiche außen Gaumenseite
Berechnen	☑ Ja
Berechne wenn	Aus
Strategie	Abschnittweises äquidistantes Schlichten
Boundary-Prüfung	Werkzeugschaft
Werkzeug	58-US20-K-2,0R1,0 Zecha
Drehzahl	40000
Vorschub	4000
Reduzierter Vorschub	2500
Max. Winkel für red. Vorschub	90
Kühlmittel	Aus allgemeinen Einstellungen
Rework hinzufügen	☑ Ja
Segment-Winkelunterschied	33
Soll-Anstellwinkel	10
Offset alveolarridge	0
Offset	0
Aufmaß	0
Seitliche Zustellung	0,15



Finishing outer undercut areas

Available strategies:

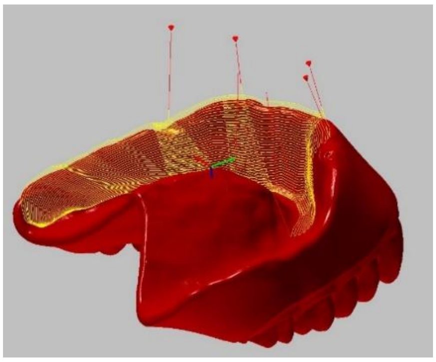
- Segmented profile finishing
- Segmented profile finishing, XY-optimized
- Segmented equidistant finishing (only advisable in connection with “Add. rework step”! Increased calculation time!)

Offset on alveolar ridge: Should remain at “0” in the outer area. **Based on the <u>tooth center line</u>.**

Desired tilt angle: Angle of the tool tilt for 3+2 axis machining.

Angle difference segments: Similar to finishing in long cavities. Defines the size of the individual segments.

Finishing inner undercut areas



hyperDENT Jobname	Finishing inner undercut areas upper denture
Jobname	Finishing inner undercut areas upper denture
Berechnen	Ja
Berechne wenn	Aus
Strategie	Abschnittweises äquidistantes Schlichten
Boundary-Prüfung	Werkzeugschaft
Werkzeug	PMMA Ballmill D2,0x40
Drehzahl	30000
Vorschub	3500
Reduzierter Vorschub	2000
Max. Winkel für red. Vorschub	90
Kühlmittel	Aus allgemeinen Einstellungen
Rework hinzufügen	Nein
Segment-Winkelunterschied	27
Soll-Anstellwinkel	25
Offset alveolarridge	2,5
Offset	0
Aufmaß	0,1
Seitliche Zustellung	0,1

Finishing inner undercut areas

Available strategies:

- Segmented profile finishing
- Segmented profile finishing, XY-optimized
- Segmented equidistant finishing

Offset on alveolar ridge: The offset is based on the alveolar ridge line and moves the machining toward the outside (+) or the inside (-).

Desired tilt angle: Angle of the tool tilt for 3+2 axis machining.

Angle difference segments: Similar to finishing in long cavities. Defines the size of the individual segments.

19 Abutment

Dental restorations that are placed directly on the implants mostly have complex interface geometries that require special machining.

The abutment module option provides you with additional functions and templates so that you can freely configure different settings, adapt them to specific tasks, and reuse them:

- *Prefabricated partial solutions for different interface geometries with the assigned templates.*
- *Creation of your own, optimized partial solutions for interface geometries and storage of the distribution and assignment of the templates for reuse.*
- *Geometry replacement, the exchange of imported interface geometries with your own appropriate, fully defined geometries with the relevant templates. Geometry replacement saves considerable time when creating templates, as the existing, fully-defined solutions can be used for the interface geometry.*

The milling strategy can only be changed in the template generator module option.

19.1 Milling strategies – Editing the template (optional)

The milling strategies set the working plan (template) for machining on the milling unit and are available for different materials and part types.

Different templates can be selected for the different parts, depending on the details in the previous process steps.

With the geometry replacement (optional), you can replace imported interface geometries with saved and already fully defined geometries complete with the relevant templates.

The selection window shows the list from which you can select the milling strategy (template) for the machining.

In the template generator module option, you can change the templates and adapt them to the requirements of the part type and material in use.

A description of how to do this is available in the chapter “Milling strategies” > “Editing the template (optional)”. There you will also find a description of the general and all cycle-specific job parameters.

19.2 Overview of milling strategies for abutments

The predefined milling strategies for abutments are subdivided in accordance with the cycles used in the areas: drilling, rest machining, finishing.

Drilling	Drilling of the interface geometry *Screw channel machining* *Drilling the screw channel.* *Drilling implant interface geometry* *Drilling corners in the interface geometry.* *Thread machining* *Creating an interface thread.*
Space left	Targeted rework-machining of remaining space left areas. *Rest machining outside abutment bases* *Rest machining inside abutment bases*
Finishing	Final machining of interface geometries and adjacent areas. *Finishing inside abutment bases* *Finishing outside abutment bases* *Plane finishing inside abutment bases* *Finishing emergence profile* *Overall finishing occlusal side (abutments)* *Finishing implant interface geometries* *5X boss finishing*

19.3 General job parameters for abutments

The following parameters exist in almost all jobs for abutments and are therefore only described here. You can find the description of the other general job parameters under --> “Milling strategies” > “General job parameters”.

Calculate		Yes
	Yes	Calculate job
	No	Disable calculation, the job stays in the list, but is not included in the calculation.
Calculate if		Off Conditional calculation, dependent on the screw channel diameter. This means you can calculate two jobs that are the same depending on the screw channel diameter for different tools.
	Off	Calculation is always performed, regardless of the screw channel diameter.
	Screw channel diameter bigger	Calculation only performed when the screw channel diameter is bigger than the default value.
	Screw channel diameter smaller	Calculation only performed when the screw channel diameter is smaller than the default value.
	Screw channel diameter between	Calculation only performed when the screw channel diameter is bigger than the first default value and smaller than the second default value.
Calculate if		Off Conditional calculation to permit the machining of one- and two-part abutments in a template.
	Off	Calculation is always performed.
	Screw fit exists	Calculation is performed only if a screw fit is available.
	Screw fit does not exist	Calculation is performed only if no screw fit is available.
Calculate if		Off Conditional calculation for the machining of straight and angled screw channels of abutment bridges and bridges in a single template.
	Off	Calculation is always performed.

Screw channel is angulated	Calculation is performed only if the screw channel is angulated.
Screw channel is not angulated	Calculation is performed only if the screw channel is not angulated.

- Caution!
  Use tools with the same diameter and the same allowance.

Groove depth — Value from top of blank
Reference plane for the conditional calculation.

Bottom of blank

19.4 Cycle-specific job parameters for abutments

19.4.1 Screw channel machining

The screw channel forms its own machining area.

Cycles — 3D arbitrary stock roughing

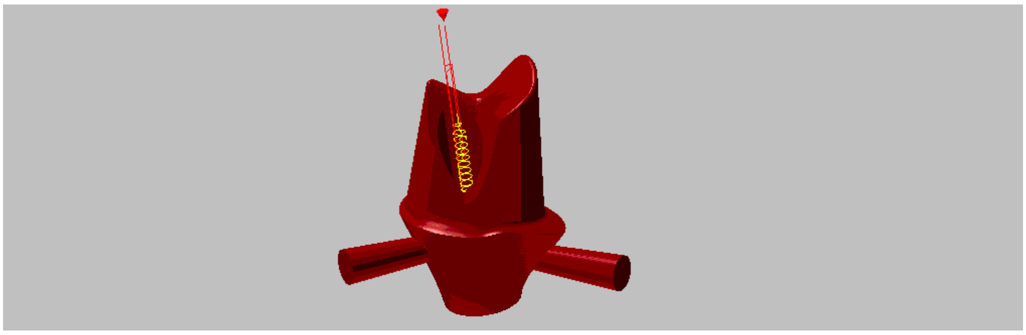


Figure 19-1

5X boss finishing

3D Z-level finishing

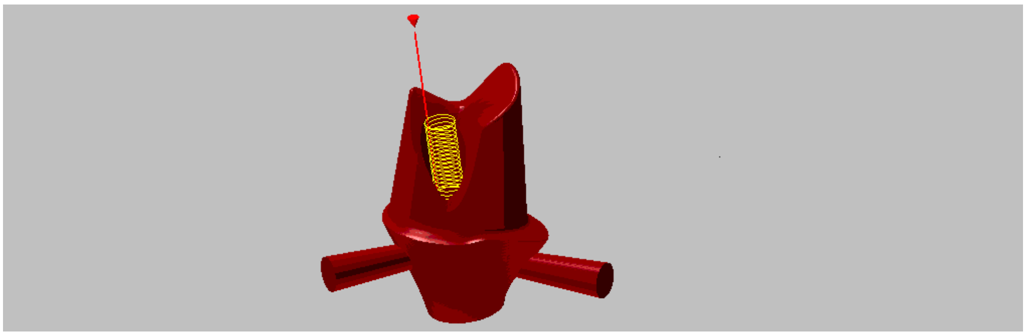


Figure 19-2

Simple drilling

Drilling with chip break

Drilling with pecking

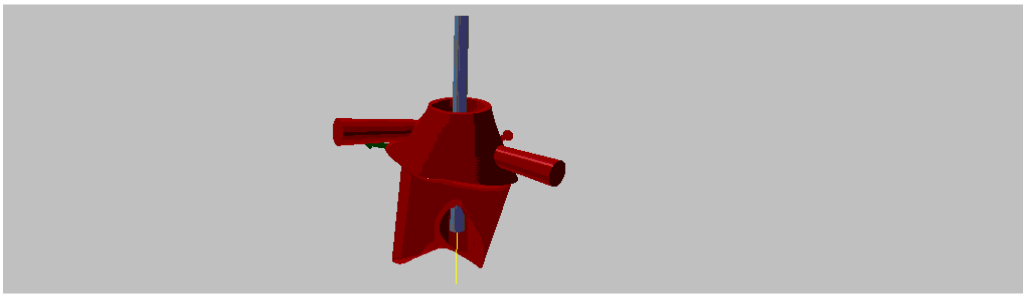


Figure 19-3

General job parameters for screw channel machining

The following parameters exist in almost all jobs for screw channel machining and are therefore only described here.

You can find the description of the other general job parameters under
--> “Milling strategies” > “General job parameters”.

Calculate if	Off Conditional calculation, dependent on the screw channel diameter. In this way, you can select the right tool for the present screw channel diameter.
Off	Calculation always performed.
Screw channel diameter bigger	Calculation only performed when the screw channel diameter is bigger than the default value.
Screw channel diameter smaller	Calculation only performed when the screw channel diameter is smaller than the default value.
Screw channel diameter between	Calculation only performed when the screw channel diameter is bigger than the first default value and smaller than the second default value.
Calculate if	Off Conditional calculation depending on whether a thread exists (is to be created). This means you can calculate the jobs depending on the thread for different tools, e.g.: calculate mill, thread cutter.

Off	Calculation always performed.
Thread exists	Calculation is performed only if a thread is available, e.g. for thread cutters.
Thread does not exist	Calculation is performed only if no thread exists.
Type of channels	To differentiate between different types
Implant interface	
	Screw channel in abutments
Other holes	
	For machining locator holes or other drill holes
Any type	
Machining category	
	Category of the hole in [Determine part properties > Holes]

For 3D arbitrary stock roughing, 3D Z-level finishing, drilling, drilling with chip break, drilling with pecking

Side	Side that is to be machined by this cycle.
Occlusal side/cavity side	
Offset for screw channel radius	mm Radius for the screw channel that deviates from the CAD information.
Top	Automatic Determination of the start of the screw channel machining.
Automatic	Use the current allowance from the blank tracking for the start of machining.
Maximum inside abutment area	
Blank	Blank surface
Begin of screw channel	
Mid of screw channel	
Begin of screw fit area	
End of screw fit area	

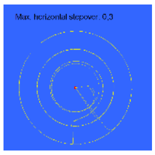
End of screw channel
Through
Begin of thread
End of thread
End of angulation

- Caution!
  Do not use the “Automatic” setting if the drill hole has already been predrilled with a screw channel machining operation.

Offset top		mm
	Additional offset for the start of the drilling.	
Machining depth		
	End point of the machining, based on the tool diameter.	
	Determination of the start of the screw channel machining.	
Value of top		
	Use the current allowance from the blank tracking for the start of the machining.	
Maximum inside abutment area		
Blank	Blank surface	
Beginning of screw channel		
Mid of screw channel		
Begin of screw fit area		
End of screw fit area		
End of screw channel		
Through		
Begin of thread		
End of thread		
End of angulation (lowest point)		
Depth offset		mm
	Additional offset for the end of the drilling.	

Tool tip compensation	Yes
Yes	Compensation for the height of the tool tip. The drill hole is much deeper so that in spite of the cone shape of the tool tip, the complete diameter is achieved at the required depth.
No	Depth is based on tool tip.

For 3D Z-level finishing

Spiral stepdown	Yes
Yes	Spiral machining of the screw channel
No	Z-constant stepdown. The stepdown is in the middle of the screw channel.
Machining priority	
Zigzag	Machining in synchronization and in opposite direction: Shorter runtime, poorer surface quality.
Plane	Machining in synchronization only: Better surface quality, longer runtime.
Max. horizontal increment	0-100
<0	 

For drilling, drilling with chip break, drilling with pecking

Simple drilling	The drill hole is created in a step, e.g. for center drilling, predrilling.

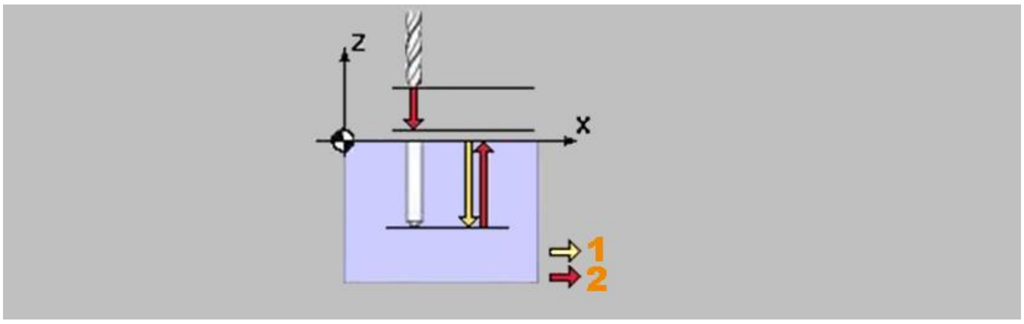


Figure 19-4

1. Feedrate
2. Rapid traverse

Drilling with chip break	The drill hole is created in several steps (Z1, Z2, etc.). After each drilling stroke, the tool retracts in rapid traverse mode around the retract value (3): Shorter chips, better removal of chips. The step is reduced by the reduce value after each drilling stroke.

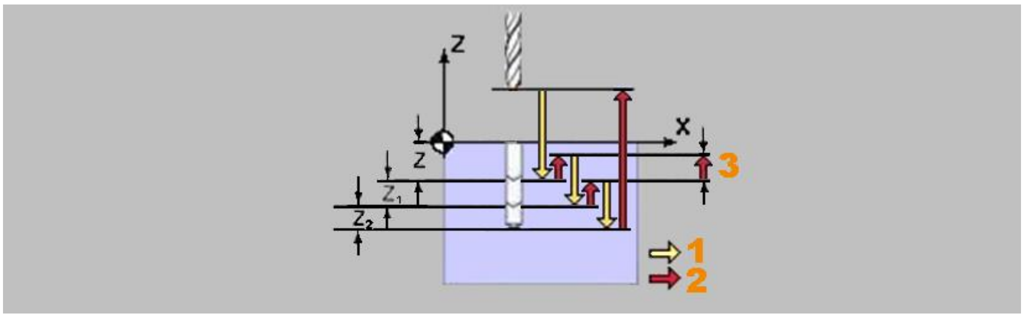


Figure 19-5

1. Feedrate
2. Rapid traverse
3. Retract value

Drilling with pecking	The drill hole is created in several steps (Z1, Z2, etc.). After each drilling stroke, the tool retracts in rapid traverse mode to the clearance distance (3): Clear the drill hole. The step is reduced by the reduce value after each drilling stroke.

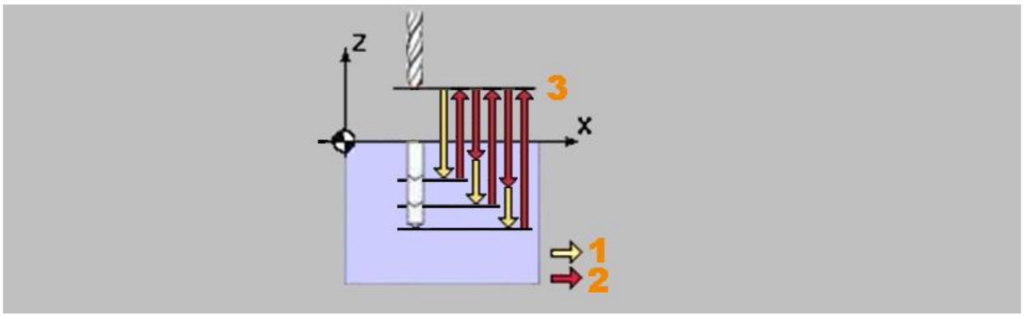


Figure 19-6

1. Feedrate
2. Rapid traverse
3. Clearance distance

Pecking depth	Stepdown of tool in mm, in the first drill stroke.
Reduce value	Reduction of the stepdown after each additional drilling stroke.

19.4.2 Thread machining

Create an interface thread. Machining takes place in several steps with stepover and repetitions for creation of the thread pitch.

For tool configuration, you can use any tool available in hyperDENT®, as only the tool diameter is included in the calculation.

Cycles	Thread machining

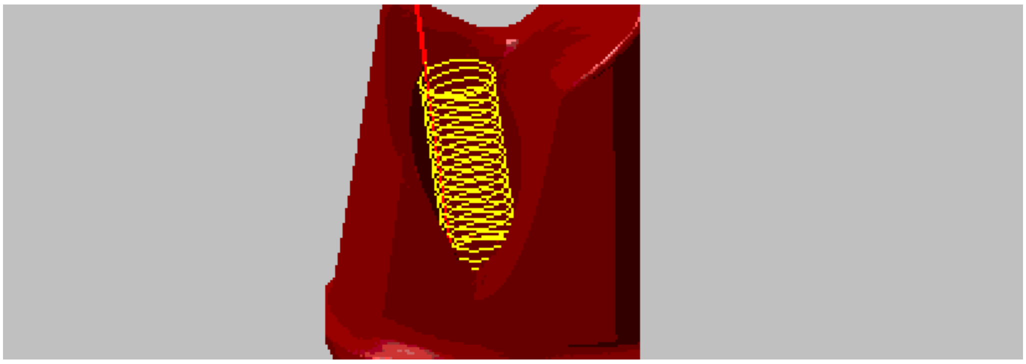


Figure 19-7

Side	Side that is to be machined by this cycle.	
Occlusal side/cavity side		
Offset for screw channel radius		mm
	Radius for the screw channel that deviates from the CAD information.	
Top		Begin of thread
	Determination of the start of the screw channel machining.	
Automatic	Use the current allowance from the blank tracking for the start of machining.	
Maximum inside abutment area		
Blank	Blank surface	
Beginning of screw channel		
Begin of screw fit area		
End of screw fit area		
Begin of thread		
End of thread		

- Caution!
  Do not use the “Automatic” setting if the drill hole has already been predrilled with a screw channel machining operation.

Offset top		mm
	Additional offset for the start of the drilling.	

Machining depth		End of thread
	End point of the machining, based on the tool diameter. Determination of the start of the screw channel machining.	
Value of top		
	Use the current allowance from the blank tracking for the start of the machining.	
Maximum inside abutment area		
Blank	Blank surface	
Begin of screw channel		
Mid of screw channel		
Begin of screw fit area		
End of screw fit area		
End of screw channel		
Begin of thread		
End of thread		
Depth offset		mm
	Additional offset for the end of the drilling.	
Tool tip compensation		Yes
Yes	Compensation for the height of the tool tip. The drill hole is much deeper so that in spite of the cone shape of the tool tip, the complete diameter is achieved at the required depth.	
No	Depth is based on tool tip.	
Machining direction	Upward	
Upward/Downward	Direction in which machining is to take place.	
No. of steps	Number of steps (stepovers) for the production of a thread.	

No. of spring cuts per step

Number of repeat (or empty) steps used to create the thread pitch.

Figure 19-8

19.4.3 Drilling corners of implant interfaces

You can machine the corner radii of inner geometries with an optional drilling operation.

Cycles — Simple drilling

Drilling with chip break

Drilling with pecking

Drilling ends at the level defined by the markings (2 - 7).

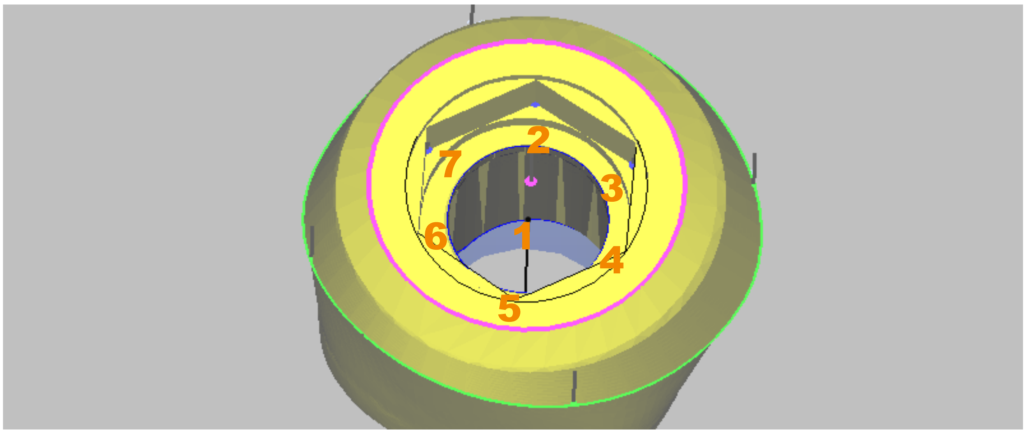


Figure 19-9

19.4.4 Plane finishing inside abutment bases

Machining to create an absolute plane surface at the abutment base.

Cycles — For 3D equidistant finishing

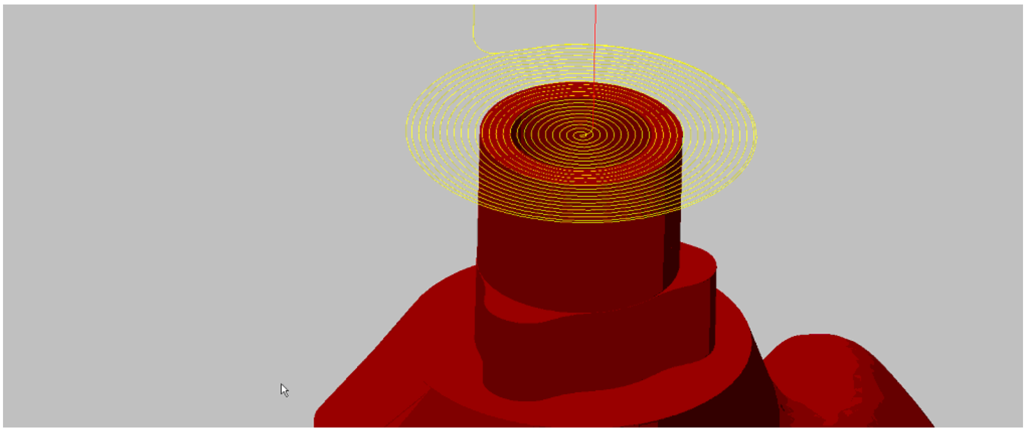


Figure 19-10

Offset	mm
0	Milling boundary as for abutment base.
>0	Milling boundary is extended.
<0	Milling boundary is reduced.
Machining direction	
From inside to outside	
Outside in	
	Pathway of milling tracks.

19.4.5 Rest machining inside abutment bases

Machining after roughing until the desired surface quality of the part is obtained and all space left has been removed.

Cycles	3D roughing on arbitrary blank 3D automatic rest machining 5X automatic rest machining
Offset	mm Offset for the boundary line abutment base.
–	Move boundary line abutment base in.
+	Move boundary line abutment base out.

Add. outer depth	mm
Depth	If offset is set, toolpath follows profile (in offset area) up to "outer depth".

19.4.6 Rest machining outside abutment bases, finishing outside abutment bases

Machining of areas (1) that are located in the undercut area in the main machining direction.

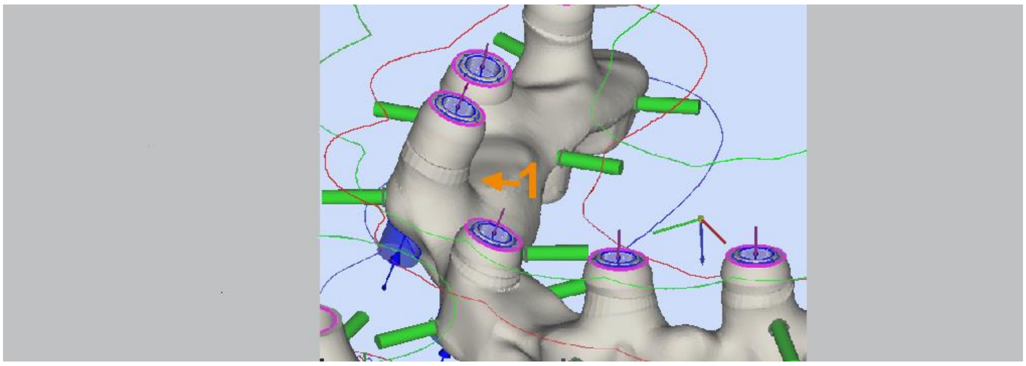


Figure 19-11

Rest machining outside abutment bases

Cycles	3D arbitrary blank roughing (with blank tracking)

Finishing outside abutment bases

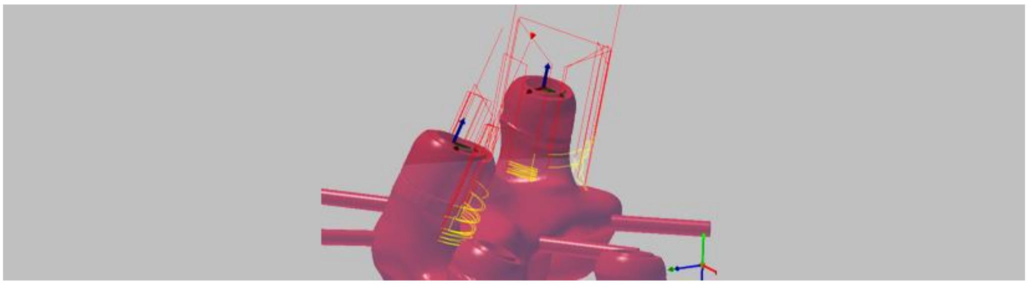


Figure 19-12

Cycles	3D Z-level finishing
Side	Side that is to be machined by this cycle.
Occlusal side/cavity side	

Use coping specific directions		No
	Details of the rotation axis for the coping-specific tilting. If the machining direction is not determined by the CAD, you can set it via the context menu: --> "Set milling direction" > "Coping-specific alignment".	
No	No coping-specific alignment/tilting.	
X+Y rotation	Rotation axes for 5X machines. Should only be used if the machining during bridge construction occurs in parallel to the screw channel direction.	
X rotation	X rotation axis, for 3+1 machines.	
Y rotation	Y rotation axis, for 3+1 machines.	
Offset		mm
	Outer milling boundary for this machining process. The value should be equal to at least the tool radius + allowance + security area (approx. 0.2 - 0.5). The inner milling boundary is the abutment base or the emergence profile.	

19.4.7 Finishing of abutment base

Cycles	3D equidistant finishing 3D complete finishing 3D Z-level finishing 5X boss finishing

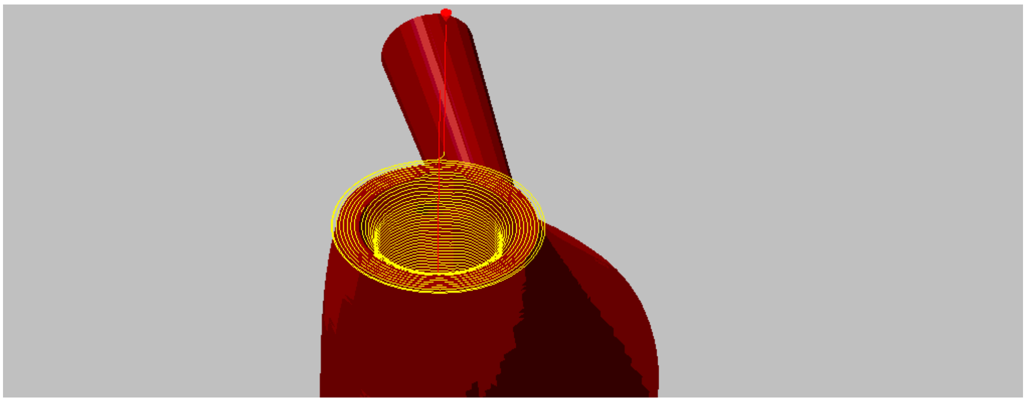


Figure 19-13

Exclude geometry inside	
Yes/No	Defined abutment base areas, e.g. exclude geometry from machining.
Offset	mm Distance to abutment base. Additional offset for the boundary line around the abutment base geometry, so that clean milling tracks can be created and additional tool tracks are not created on the geometry boundary line.
Add. outer depth	mm
Depth	If offset is set, toolpath follows profile (in offset area) up to “outer depth”.
Machining direction	
From outside to inside	
From inside to outside	
	Pathway of milling tracks.

19.4.8 Finishing emergence profile

Cycles

3D profile finishing ruled
3D complete finishing
3D Z-level finishing
3D equidistant finishing
5X profile finishing ruled
5X equidistant finishing

Equidistant finishing is usually the best cycle for the emergence profile, as this area requires a high surface quality.

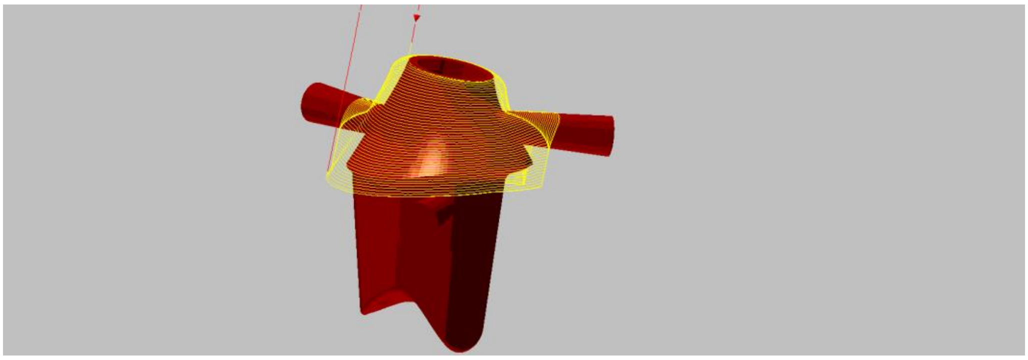


Figure 19-14

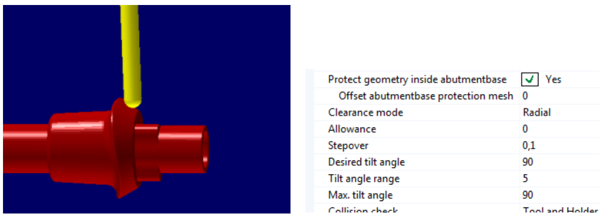
5X equidistant finishing

Clearance mode

Type of retraction movement after machining

Plane — Retraction in axis direction, for axial machining

Radial — Radial retraction, for radial machining



19.4.9 Finishing of occlusal for implant parts

Individual parts

Cycles	Overall finishing of abutment

Bridges

Cycles	Overall finishing of bridges

Equidistant finishing is usually the best cycle for the emergence profile, as this area requires a high surface quality.

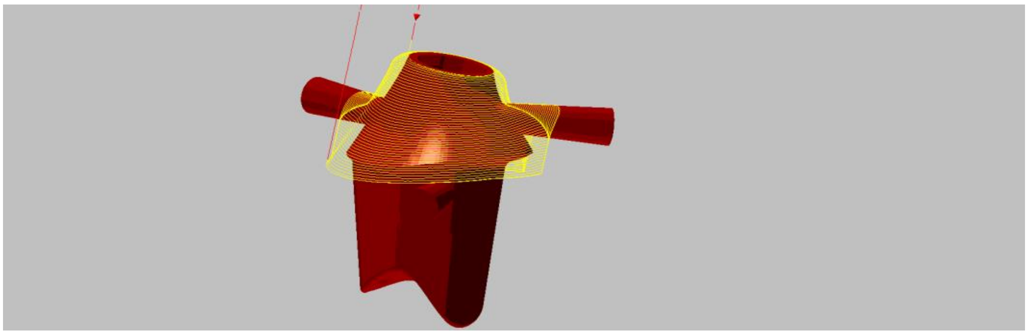


Figure 19-15

Offset abutment base	mm
	Offset for the boundary line abutment base.
–	Move boundary line abutment base in.
+	Move boundary line abutment base out.
Limit up to emergence profile	Yes
Yes	The Z-level boundary area to occlusal is always generated at the emergence profile and limits the cycle in the occlusal direction.
Overall finishing	Yes
Yes	Finish the whole area of the abutment base. The boundary line abutment base is not taken into account.

19.4.10 Machining method for finishing implant interface geometry

You can split the machining of the interface geometry into separate sections and assign unique jobs to these sections which you then adapt to the required machining.

Cycles	3D Z-level finishing
Side	
Inner walls	Inner geometry
Outer walls	Outer geometry
Category	- 0 to - 9 - n Number for this machining process. Use this number to assign this machining process to a section in the machining area of the abutment base. The calculation only happens if an area is given the corresponding category number: --> "Identify part features" > "Split the machining area into sections". This means you can machine different interface geometries with only one (comprehensive) template.
Machining direction	
From top to down	
From down to top	
	Pathways of milling tracks.
Machining height	
Complete	Machine complete machining area.
Partial	Machine partial area.
Machining height	mm Machining height for partial machining.
Add. allowance XY	mm Additional allowance in X and Y-direction.
Step strategy	
Pocket	Individual pockets are machined successively
Plane	Planar machining

Spiral stepdown

Available for pocket strategy

Partial machining for a category

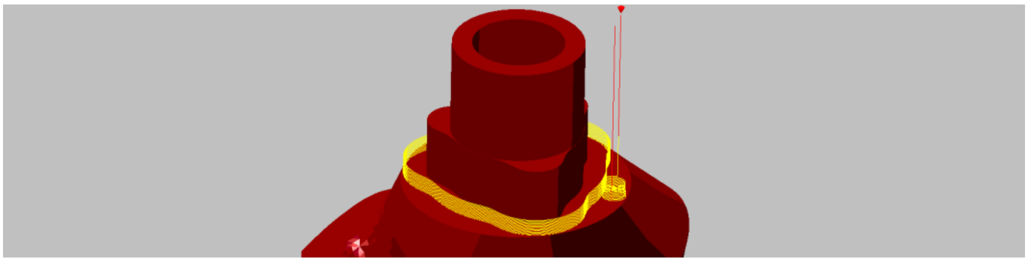


Figure 19-16

Complete machining for a category

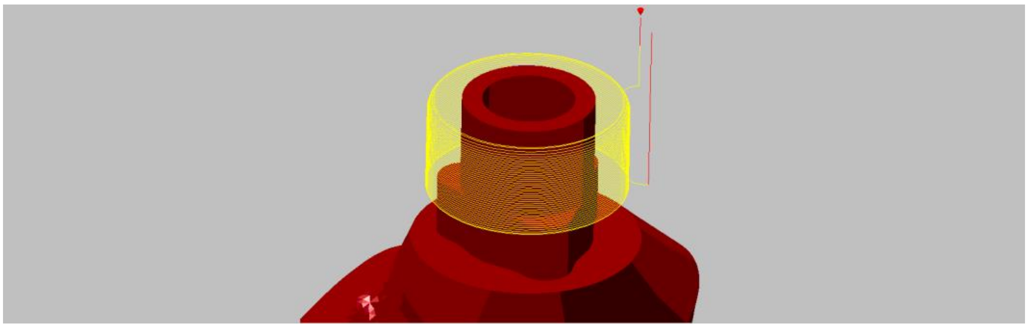


Figure 19-17

19.5 Geometry replacement for abutment connection geometries

Using the automated or manual geometry replacement (optional), you can ensure high-quality, reproducible milling results in hyperDENT®:

*Link *.STL data for the connection geometry with a template.*

Save the fully predefined connection geometry in the hyperDENT® database.

Replace the current connection geometry from the CAD file with the stored connection geometry.

The geometry replacement can take place automatically or manually: --> “Automatic geometry replacement”, “Manual geometry replacement”.

Predefined connection geometries (optional) are stored encrypted in the hyperDENT® database and are available for a user-friendly workflow: --> “Predefined geometries”.
The screw seat area of these geometries is compatible with the screws of the original manufacturers.

Geometries can also be manually applied to parts created for adhesive base parts.

19.5.1 Automatic geometry replacement

The automatic replacement of geometry data is performed during the “Load part” process using the data that is transferred via the defined interface.

Condition for automatic geometry replacement:

- *There is a stored geometry.*

  Either the stored geometry has the same designation specified as in the .STL file of the part.

  Or the designation of the stored geometry is allocated to the designation in the .STL file of the part via a mapping file.

- *The coordinate systems of the part and the stored geometry correspond.*

1. Check whether there is a stored geometry with the corresponding designation or a mapping file with corresponding entries.
2. Load part: --> “Load part”
   The part data and the data of the allocated, stored connection geometries are loaded.

The stored connection geometry is displayed in the same place as the original geometry and aligned to the screw channel and the abutment base.

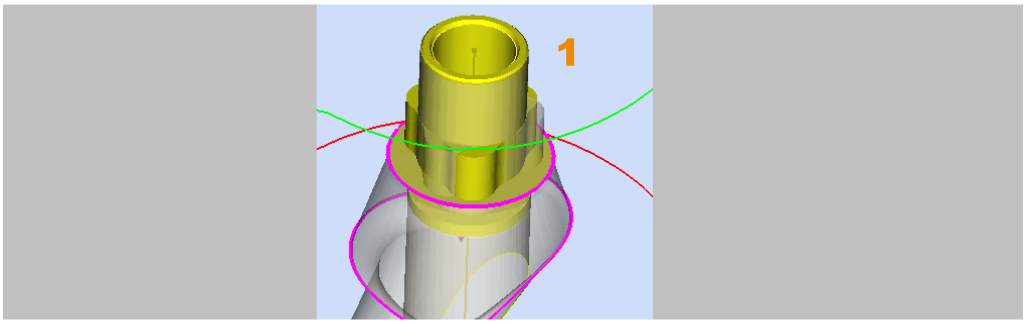


Figure 19-18

3. Check the exact alignment of the new connection geometry.
4. If necessary, align the connection geometry manually: --> “Manual geometry replacement” > “Align connection geometry”.
5. Save project.

For high accuracy of fit of the dental restoration, the connection geometry must be aligned very precisely in terms of height and angle.

19.5.2 Mapping file for geometry replacement

The mapping file is used to allocate the different designations of the original connection geometry from the CAD system to the predefined connection geometry from the hyperDENT® database during automatic geometry replacement.

In the mapping file, save the list of allocations:

Between the designations of the connection geometries that appear in the data in CAD information

To the designations of the predefined connection geometries that are stored in the hyperDENT® database and are to be used for the geometry replacement

To the desired movement in terms of height

To the desired rotation around the Z-axis

Creating, amending the mapping file

1. Determine the original designation of the part's connection geometry:
   Open CAD file, copy designation onto the clipboard.
   Close CAD file.
2. Open the mapping file of the geometry type to be used.
3. Enter the original designation from the CAD file at the start of the next free row:
   Paste in content from the clipboard.
4. Then enter a space and the equals sign (=).
5. Next, enter the designation of the stored geometry that you wish to use for the geometry replacement.
   You can enter a predefined geometry or your own stored geometry.
6. Repeat the procedure for any other allocations:
   You can allocate different designations from the CAD system to different, or also – if suitable – to the same saved geometry.
   Each allocation should be given its own row.
7. Save the mapping file.

The stored allocation is used for all processes in which you load parts via the defined interface and the appropriate data is available.

Example of mapping file

Row	Designation in CAD	=	Designation in the database
1	abcdef_unlocked3.5 = FM_NB_RP_UN_3.5		
2	xyz...._locked4.3 = FM_NB_RP_L_4.3		
3	uvw..._locked4.3 = FM_NB_RP_L_4.3		

Movements

As well as mapping to a stored geometry, you can also specify a move in the Z-direction.

Add the identifier "_MV" after the original name.

After the "=" sign comes the value (mm) for the move.

Positive value	*Translate up.*
Negative value	*Translate down.*

Example of mapping file with movement

Row	Designation in CAD = Designation in the database
1	abcdef_unlocked3.5 = FM_NB_RP_UN_3.5
2	abcdef_unlocked3.5_MV = 3
3	xyz...._locked4.3 = FM_NB_RP_L_4.3
4	xyz...._locked4.3_MV = -2.4

1. Mapping
2. Translate up by 3 mm
3. Mapping
4. Translate down by 2.4 mm

Rotation

As well as mapping to a stored geometry, you can also specify a rotation around the Z-axis.

Add the identifier "_Rot" after the original name.

After the "=" sign comes the value (°) for the rotation.

Positive value	*Rotate counterclockwise.*
Negative value	*Rotate clockwise.*

Example of mapping file with rotation

Row	Designation in CAD = Designation in the database
1	abcdef_unlocked3.5 = FM_NB_RP_UN_3.5
2	abcdef_unlocked3.5_Rot = 60
3	xyz...._locked4.3 = FM_NB_RP_L_4.3
4	xyz...._locked4.3_MV = -2.4
5	xyz...._locked4.3_Rot = -45

1. Mapping
2. Rotate 60° counterclockwise
3. Mapping
4. Translate down by 2.4 mm
5. Rotate 45° clockwise

Location of the geometry type

Exocad File: constructioninfo

Example


```
<MatrixImplantGeometry>
  <_00>0.90469139814376831</_00>
  <_01>-0.42536669969558716</_01>
  <_02>-0.024428809061646461</_02>
  <_03>0</_03>
  :
  <_31>-27.674861907958984</_31>
  <_32>11.338641166687012</_32>
  <_33>1</_33>
</MatrixImplantGeometry>
<FilenameImplantGeometry>C:\Program Files\Zfx Dental Design\ZfxCAD\library\implant\zfx-e-int\geo_Etec_30.sdfa</FilenameImplantGeometry>
<Center>
  <x>9.3247531027181374</x>
  <y>-27.611514435003819</y>
  <z>14.795235227917255</z>
</Center>
<ZRotationMatrix>
  <_00>0.99999911380667694</_00>
```


Figure 19-19

hyperDENT® searches the configured folder for a geometry with the file name: "geo_Etec_30.hdpartz" to be used for the automatic replacement of the restoration geometry.

- Caution!
  Geometry replacement only works for Exocad versions with a release date of October 1, 2010 or later. In earlier versions, errors will be encountered during placement.

3Shape Example

File: ImplantDirectionPosition.xml

ImplantDirectionPosition_0.xml


```xml
<root>
    <ToothIndex unn="3">
        <Kit>
            <System value="FM Nobel Biocare Replace Lock"/>
            <Name value="FM_NB_RP_L_5,0"/>
            <ID value="24139_AbutmentKit70"/>
        </Kit>
        <Position x="-0.04446099" y="-0.04008244" z="-3.89527726"/>
        <Direction x="0.00000000" y="0.00000000" z="1.00000000"/>
        <Rotation x="1.57079637"/>
        <Rotation y="-0.00000001"/>
        <Rotation z="1.17036057"/>
        <TransformMatrix m00="0.38981968" m01="0.00000000" m02="0.92089117" m03="-0.04446099"  m10
        <Xaxis x="0.38981968" y="0.92089117" z="0.00000001"/>
        <Yaxis x="0.00000000" y="0.00000000" z="1.00000000"/>
        <Zaxis x="0.92089117" y="-0.38981965" z="-0.00000000"/>
    </ToothIndex>
</root>
```

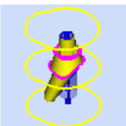

Figure 19-20

hyperDENT® searches the configured folder for a geometry with the file name: "FM_NB_RP_L_5,0.hdpartz" to be used for the automatic replacement of the reconstruction geometry.

19.5.3 Manual geometry replacement

The manual replacement process can also be performed without the data of the defined interface.

Before the replacement process, all part features – such as the abutment base, screw channel, etc. – must be determined manually if they are not already available.



1. Select the part:
   Click on the part.

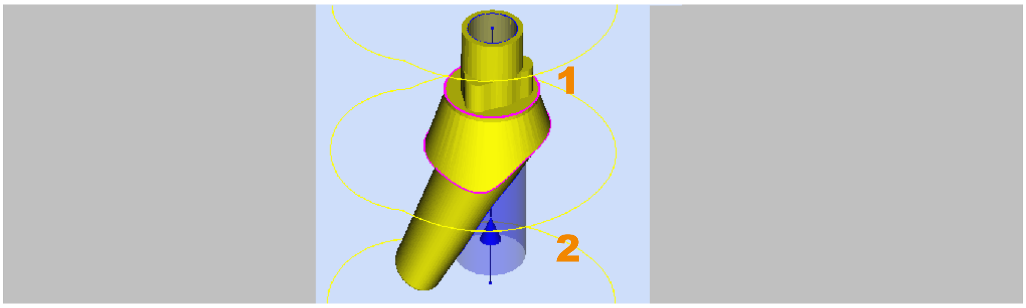


Figure 19-21

2. If not already done, determine the abutment base (1) and the screw channel (2): --> “Identify part features”
   --> “Determine abutment base”,
   --> “Determine screw channel”.

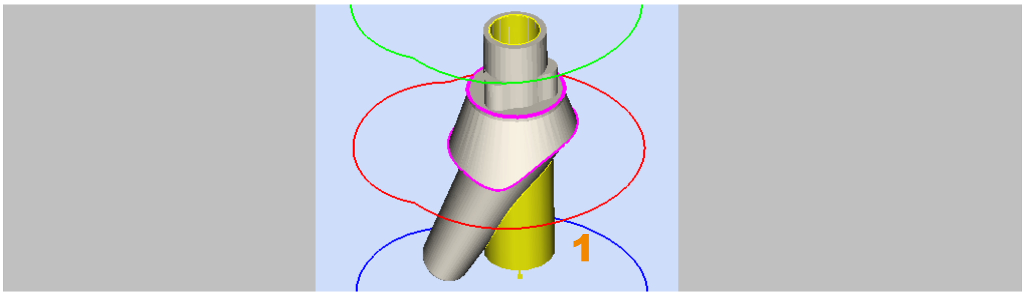
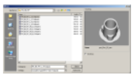


Figure 19-22

3. Select the screw channel (1):
   Click on the button for the sections of the connection geometry.
4. Select the type of geometry description:
   Click on [External].

5. Open directory with the stored connection geometries.



6. Load the .STL file with the connection geometry:
   Click on the .hdpartz file and click on [Open] or double-click on the .hdpartz file:
   The new geometry (1) is inserted in the same place as the original geometry and aligned to the screw channel.

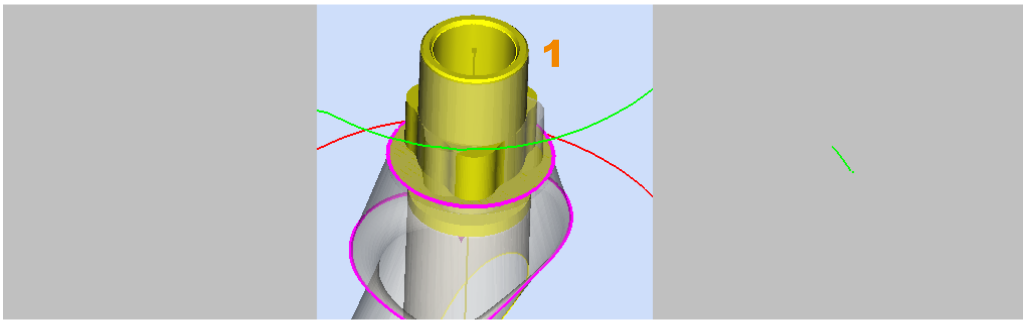


Figure 19-23

Aligning connection geometry

7. Align the new geometry precisely in terms of height and rotation:
   Click on the arrow keys:
   The new geometry is rotated around the axis of the screw channel or moved along the axis according to the set value.

It is also possible to use the automatic functions for alignment purposes. (See 8.3.3 - 8.3.5)

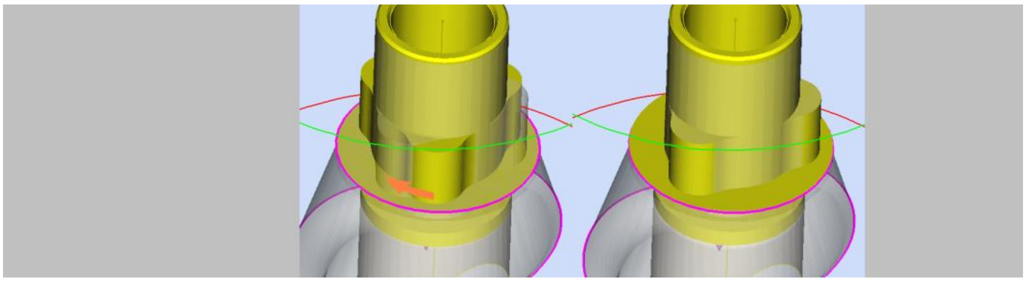


Figure 19-24

8. Save project.

For high accuracy of fit of the dental restoration, the connection geometry must be aligned very precisely in terms of height and angle. Use the automatic functions, if possible.

19.5.4 Creating connection geometry in the CAD system

You can create connection geometries with any CAD system that can export the data into STL format.

When doing so, it is important to use the same coordinate system as for the dental restoration to ensure exact placement of the new geometry on the existing CAD restoration.

Perfect match with the relevant platform of the original implant system.

The geometry can be produced on the milling unit using the present process.

The geometry data is available in STL format or can be exported into STL format.

The geometries contain the complete areas within the abutment base.

The geometries contain the screw channel and the screw seat areas.

The STL model must be closed, except in the areas of the emergence profile and screw channel.

The screw channel toward occlusal must be short enough to be contained within the dental restoration. Since these areas are not modified, the screw channel portions would otherwise protrude from the restoration in an occlusal direction.

The coordinate system used to save the geometry must be at the center of the screw channel on the plane of the abutment base.

The Z-axis must correspond to the axis of the screw channel.

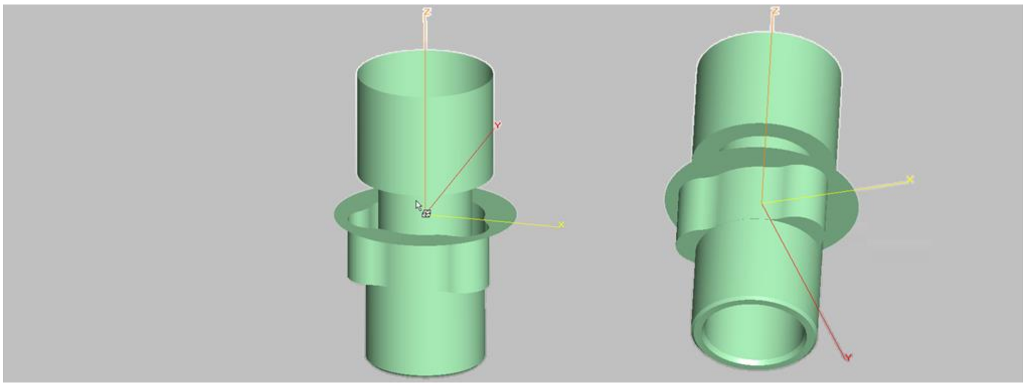


Figure 19-25

19.5.5 Preparing and saving connection geometry in hyperDENT®

The connection geometries created in the CAD system are always loaded in hyperDENT® with the part type “Abutment” and saved individually in the database (directory for connection geometries) once the process steps have been completed.

1. Load the .STL file with the connection geometry as the part type “Abutment”: --> “Load part” > Type “Abutment”.

- Ensure that the part type “Abutment” is selected.

2. Complete the subsequent process steps:

   --> Set milling direction

   --> Identify part features >

   --> Determine screw channel.

   --> Split planes into work steps and allocate categories.

   --> Set corner machining of inner geometries.

The abutment base line does not need to be defined, as it is automatically calculated. In individual cases, it can be necessary to define a different abutment base line.

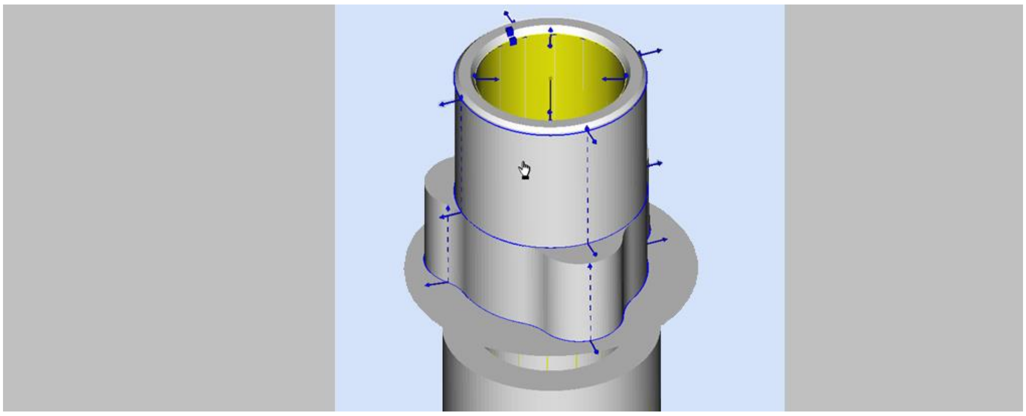


Figure 19-26

3. Delete part transformations so that the alignment of the coordinate system is adjusted to hyperDENT®:
   Select part.
   Select menu item [Extras] > [Execute command].
   Call up and execute the [Delete part transformations] function:
   Click on [Execute].
   The part is then transferred to the hyperDENT coordinate system.

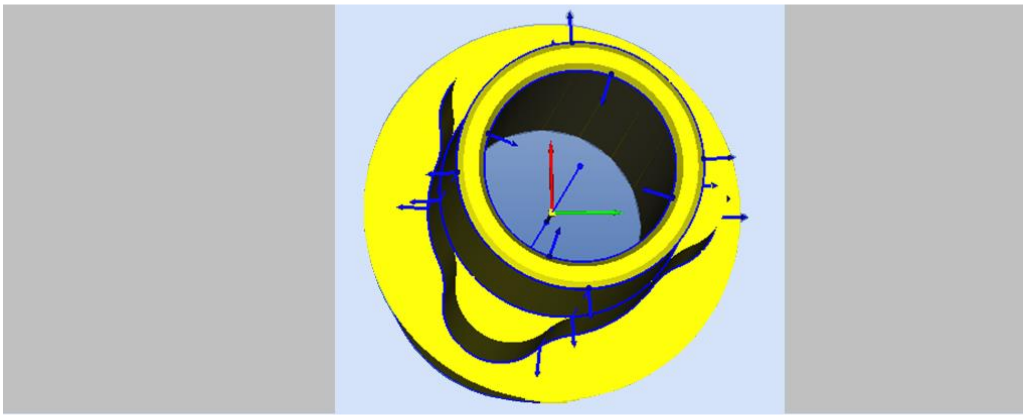


Figure 19-27

4. Save connection geometry in hyperDENT®:
   Select menu item [Extras] > [Execute command].
   Call up and execute the [Save part] function:
   Click on [Execute].
5. Select the directory.
   For automatic geometry replacement, the file must be located at the path for the connection geometries.
6. Enter the file name.

7. Select the file type ".hdpartz".
   The file type ".hdpartz" is required for automatic geometry replacement.
   The file type ".hdpartz" is necessary for automatic geometry replacement.
8. Save file:
   Click on [Save].

- For automatic geometry replacement, the connection geometries must be saved in the directory or in a subdirectory for the connection geometries: --> "Settings" > "General".
  This is the only way for the automatic geometry replacement to find the relevant replacement geometry using the information of the defined interface between CAD and hyperDENT®.
- To ensure that the system identifies the correct geometry, the file name must be as specified in the defined interface transferred from the CAD system, or allocation via the mapping file is required: --> "Mapping file for geometry replacement" > "Location of the geometry type".

19.5.6 Supported CAD systems

The automated replacement of ruled geometry portions of the CAD output data with the stored data is possible using the following CAD systems, provided the interface data is available:

3Shape DentalDesigner

Exocad

Dental Wings

19.5.7 Supported implant platforms

These predefined connection geometries are compatible with the implant platforms of the following manufacturers:

Nobel Biocare Replace®

Nobel Biocare Brånemark®

Nobel Active®

Nobel Multi unit®

Biomet 3i Certain®

Biomet 3i Osseotite®

Straumann Institute Bone Level®

Straumann Institute SynOcta®

Zimmer Tapered Screw Vent®

Astratech OsseoSpeed®

Dentsply-Friadent® Frialit Xive

19.5.8 Predefined geometry data

The file names of the predefined geometry data are constructed using a standard system according to the geometry types.

There is a mapping file (allocation file) for each geometry type. Using this allocation file, assign the stored geometry data to the data in the CAD output.

File name construction

FM_[Company]_[Type].alias
FM_[Company]_[Type]_L/UN_[Size].hdpartz

FM_	FollowMe
[Company]	Company name
[Type]	Type code
_L_	Locked
_UN_	Unlocked
.hdpartz	hyperDENT® project file
.alias	hyperDENT® mapping file, allocation file for automatic geometry replacement.

File names of geometry types

Manufacturer File name Locked	Label	 File name Unlocked
Nobel Biocare Replace®	NB_RP_	
FM_NB_RP_L_3.5		FM_NB_RP_UN_3.5
FM_NB_RP_L_4.3		FM_NB_RP_UN_4.3
FM_NB_RP_L_5.0		FM_NB_RP_UN_5.0
FM_NB_RP_L_6.0		FM_NB_RP_UN_6.0
Nobel Biocare Brånemark®	NB_BR_	
FM_NB_BR_L_3.5		FM_NB_BR_UN_3.5
FM_NB_BR_L_4.1		FM_NB_BR_UN_4.1
FM_NB_BR_L_5.1		FM_NB_BR_UN_5.1
Nobel Active®	NB_ATC_	
FM_NB_ACT_L_3.5		FM_NB_ACT_UN_3.5
FM_NB_ACT_L_4.3_5.0		FM_NB_ACT_UN_4.3_5.0
Nobel Biocare Multi unit®	NB_MU_	
–		NB_MU_UN_4.8
–		NB_MU_UN_6.0

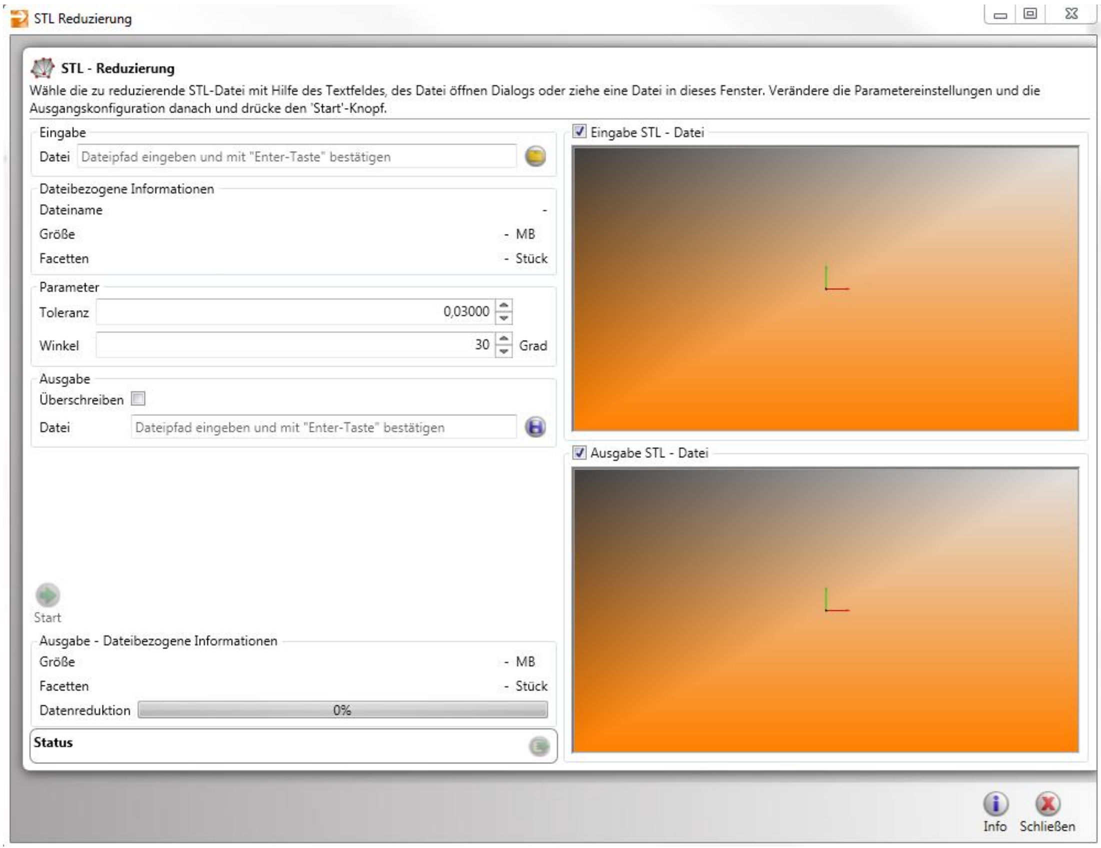
Biomet 3i Certain®	3i_CER_	
FM_3i_CER_L_3.4		FM_3i_CER_UN_3.4
FM_3i_CER_L_4.1		FM_3i_CER_UN_4.1
FM_3i_CER_L_5.0		FM_3i_CER_UN_5.0
Osseotite®di Biomet 3i	3i_OSS_	
FM_3i_OSS_L_3.4		FM_3i_OSS_UN_3.4
FM_3i_OSS_L_4.1		FM_3i_OSS_UN_4.1
FM_3i_OSS_L_5.0		FM_3i_OSS_UN_5.0
Bone Level® Straumann	STR_BNL_	
FM_STR_BNL_L_3.5		FM_STR_BNL_UN_3.5
FM_STR_BNL_L_4.1_4.8		FM_STR_BNL_UN_4.1_4.8
Manufacturer	Label	
File name		File name
Locked		Unlocked
SynOcta® Straumann	STR_SYN_	
FM_STR_SYN_L_3.5		FM_STR_SYN_UN_3.5
FM_STR_SYN_L_4.8		FM_STR_SYN_UN_4.8
FM_STR_SYN_L_6.5		FM_STR_SYN_UN_6.5
Tapered Screw Vent® Zimmer	ZI_TPV_	
FM_ZI_TPV_L_3.5		FM_ZI_TPV_UN_3.5
FM_ZI_TPV_L_4.5		FM_ZI_TPV_UN_4.5
FM_ZI_TPV_L_5.7		FM_ZI_TPV_UN_5.7
OsseoSpeed® Astratech	AS_OS_	
FM_AS_OS_L_3.5_4.0		FM_AS_OS_UN_3.5_4.0
FM_AS_OS_L_4.5_5.0		FM_AS_OS_UN_4.5_5.0
Frialit-Xive Dentsply-Friadent®	DE_XI_	
FM_DE_XI_L_3.4		FM_DE_XI_UN_3.4
FM_DE_XI_L_3.8		FM_DE_XI_UN_3.8
FM_DE_XI_L_4.5		FM_DE_XI_UN_4.5
FM_DE_XI_L_5.5		FM_DE_XI_UN_5.5

20 STL reducer

This application was developed to shorten the calculation time. Large STL models can be optimized and compressed.

Extras > Reduce mesh

Input file		
	Enter file paths and confirm	
File related information		
File name		
Size		MB
Patches		Pieces
Parameters		
Tolerance	The higher the tolerance, the greater the reduction	
Angle	Triangle angles that are smaller than the set angle are optimized.	
Output file	Enter file path and confirm	
Overwrite	The original file is overwritten	
Output related information		
Size		MB
Patches		Pieces
Data reduction		%


STL Reduzierung
STL - Reduzierung
Wähle die zu reduzierende STL-Datei mit Hilfe des Textfeldes, des Datei öffnen Dialogs oder ziehe eine Datei in dieses Fenster. Verändere die Parametereinstellungen und die Ausgangskonfiguration danach und drücke den 'Start'-Knopf.
Eingabe
Datei
Dateipfad eingeben und mit "Enter-Taste" bestätigen
Dateibezogene Informationen
Dateiname
-
Größe
- MB
Facetten
- Stück
Parameter
Toleranz
0,03000
Winkel
30
Grad
Ausgabe
Überschreiben
Datei
Dateipfad eingeben und mit "Enter-Taste" bestätigen
Start
Ausgabe - Dateibezogene Informationen
Größe
- MB
Facetten
- Stück
Datenreduktion
0%
Status
Eingabe STL - Datei
Ausgabe STL - Datei
Info
Schließen

21 Glossary

Postprocessor	Computer program that converts the results from a different program into a new format. The hyperVIEW® postprocessor converts the machine-independent program from hyperDENT® which describes the manufacturing steps of the part, into a machine-dependent format that can then be transferred to the relevant machine and processed. For different machines, special postprocessors or settings are required that adapt the data to the machine-specific conditions.
Abutment	Dental restoration that is placed directly onto the implant.
Abutment base	Part of the interface geometry, starting from the screw channel which can be defined as a separate machining area to determine an individual machining direction.
Emergence	Part of the geometry joined to the abutment base and located in the gums, which can be defined as a separate machining area to determine an individual machining direction.

Bounding box

Boundary area that encompasses the part in a cube or box shape and delimits the machining area. The boundary lines are not displayed in hyperDENT®.

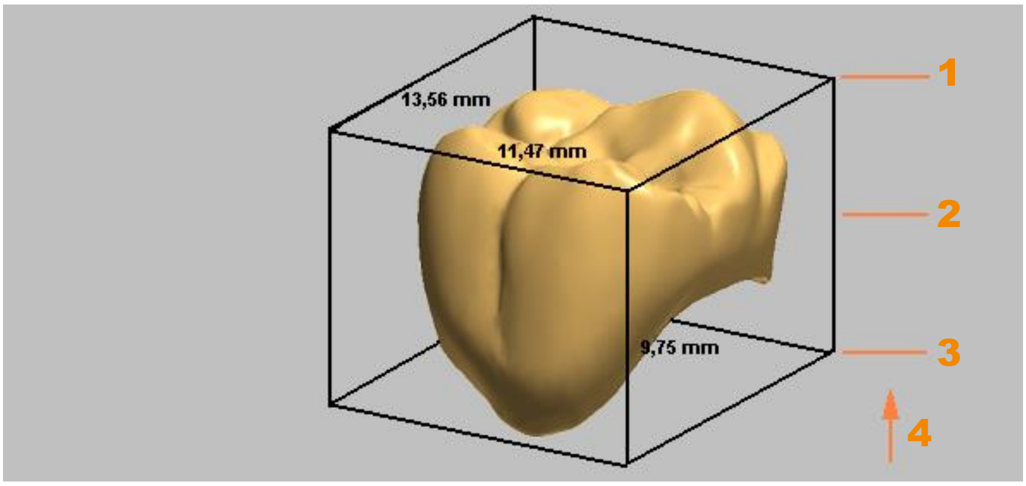


Figure 21-1

1. Max. of bounding box
2. Center of bounding box
3. Min. of bounding box
4. View direction

Prefabs – prefabricated blanks

Apart from the standard blanks, you can also use prefabricated blanks (prefabs) with a finished screw channel, screw fit, and interface geometry. These prefabricated blanks are usually provided by the system supplier to match the corresponding holder. You can request further information from our support.

22 Index

hyperDENT

Copyright and Contact

Copyright

Address	FOLLOW-ME! Technology GmbH Regerstr. 27 81541 Munich, Germany
Contact	Phone + 49 (89) 45 21 70 60 Fax + 49 (89) 45 21 70 69 E-mail info@follow-me-tech.com www.follow-me-tech.com

Version	2016-08	160801en