Version 7.0

hyperDENT®

CLASSIC · OPTIONS

Instruction manual

FOLLOW-ME!

DENTAL ENGINEERING

FOLLOW-ME!
DENTAL ENGINEERING

Copyright

Address FOLLOW-ME! Technology Systems GmbH
Regerstr. 27
81541 Munich, Germany

Contact Tel. + 49 (89) 45 21 70 60
Fax: + 49 (89) 45 21 70 69
E-mail info@follow-me-tech.com

Version 2014-01 140318en

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
- Atomical crown
- Atomical bridge, Maryland bridge
- Abutment
- Abutment crown
- Abutment bridge
- Anatomical abutment bridge
- Inlay/onlay
- Inlay/onlay bridge
- Anatomical inlay/onlay
- Inlay/onlay crown bridge
- Telescope
- Overpress
- Occlusal splint
- Model
- Model stub
- Bridge (stub and implant-supported)
- Abutment crown bridge, using adhesive and screwed
- Abutment/abutment bridge, using adhesive and screwed
- User-defined parts

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

Comparison of hyperDENT® program versions

Function		Classic	Compact
All materials		+	+
Multi Start	Starting hyperDENT® multiple times	+	–
Multi Machine	All machine types and any number of machines permitted	O	O
Project management		+	+
Load part wizard		+	+
Autonesting		+	–

Autonesting Lite	+	+	
Calculation Merge	+	+	
Free machining areas	+	–	
Prefabs	+	+	
Splints	+	+	
Sinter frame	+	+	
Traverses	+	+	
Models	O	O	(Where available)
Template editor	O	–	
Abutment Creator	O	–	
Geometries	O	–	
Polishing	O	O	
Multipart Roughing	+	–	(Where available)
CAD Connect	Call hyperDENT® from the CAD program		
Exocad	+	+	
3Shape	+	+	
DentalWings	+	+	(Where available)

\+ Present
– Not present
O Option

1.4 Version number

The current version number of hyperDENT® is additionally provided in the:

- 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

Prerequisite

hyperDENT® has the following software requirements:

- Microsoft® .NET Framework 4.0
- CAM Utilities, version 2013

Microsoft .NET Framework, version 4.0

- .NET Framework 4.0 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.

CAM Utilities

- hyperVIEW® is also needed for the installation.

Installing the program via Setup

1. Insert the installation CD into the CD drive.

The installation program starts up automatically after a few seconds.

Tip: Automatic start of CD
If the computer is configured so that CDs do not automatically start, then you must start the installation program manually:
Click on <Start> and select <Execute>.
Enter the letter of the CD 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 path/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.

3 Operating functions

3.1 Overview of program interface

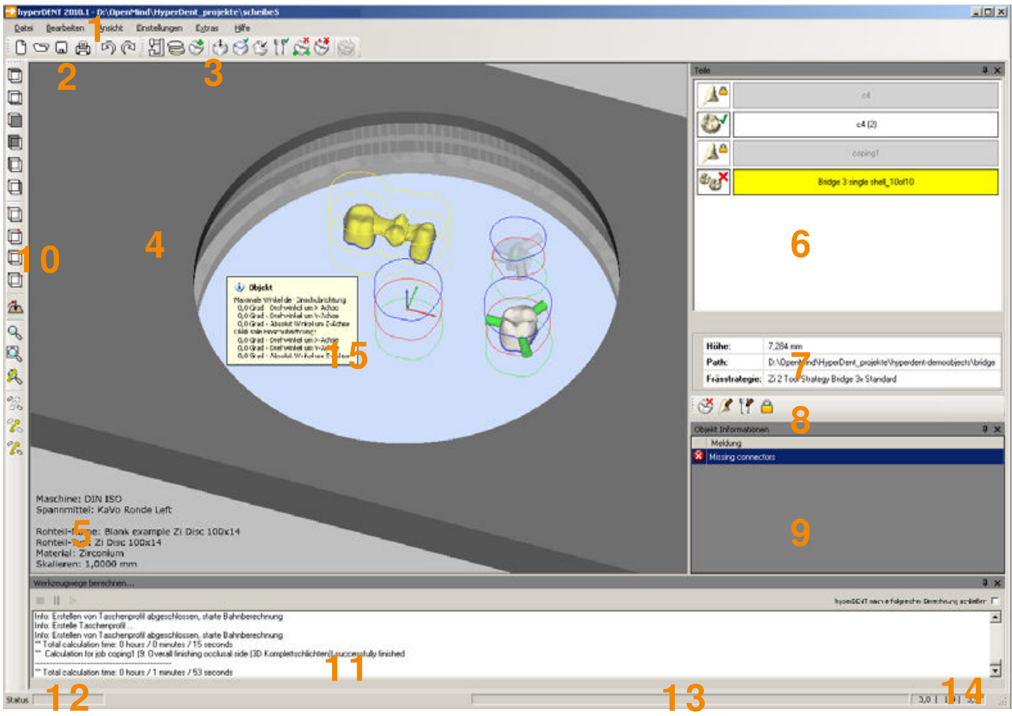


Figure 3-1

1. Menu bar
2. Toolbar – Project management
3. Toolbar – Process steps
4. Workspace
5. Project information
6. Part browser
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 line with progress bar for the calculation and information display (text)
13. Information line with details of required actions for the currently selected function.
14. Angle information for the viewing position of the blank.
15. Information window, contents depend on the position of the cursor. Dynamic display of angle information for the part, preparation line, or 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 preparation line and pontics, screw channel, and interface geometry, select, set 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 copy of the 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 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.

> Change part name (Objektnamen bearbeiten)
Change the part name, if the system suggestion is to be changed.

> Nest part in blank – local (Objekt im Rohteil schachteln – Lokal)
Automatically and optimally place the part near the current position in the blank.

> Nest part in blank – global (Objekt im Rohteil schachteln – Global)
Automatically and optimally place the part in the blank.

> Align part with screw channel axis (Objekt an Schraubenkanalachse ausrichten)
Align part as appropriate in the prefabricated blank.

> Set part sintering pin top plane
Create the top plane for sintering pins.

> Set occlusal insertion direction
Set current view direction to the part as occlusal insertion direction (= stepover direction), rotate if necessary.

> Open output directory (Ausgabeverzeichnis öffnen)
Open 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 (Öffnen in hyperDENT® Calculation Merge)
Show toolpaths in additional module hyperDENT® Calculation Merge after successful calculation.

> Lock part
Lock selected part.

> Save part
Save selected part.

> Export part
Export selected part.

Preparation line

> Edit preparation line

> Change type of preparation line

e.g. from coping to inlay/onlay, if the existing setting is to be changed.

> Change undercut property of coping

e.g. from “Coping has no undercuts” to “Coping has 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 (= stepover direction), rotate 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 sinter 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 and align with surface (Ansicht drehen und zur Oberfläche ausrichten)
Rotate view and set the view direction of the part perpendicular to the surface at the selected point.

- Blank

Rotate blank — Rotate blank in the fixture, e.g. for better use of the space left, adapt the tilt part to the swiveling axis of the fixture for 3+1-axis machining.

- Part

Edit template parameters
Change template for part. Only available in template generator 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.

Align part with screw channel axis (Objekt an Schraubenkanalachse ausrichten)
Align part as appropriate 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 direction insertion direction (= stepover 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.

- Preparation line

Edit preparation line

Change type of preparation line
e.g. from coping to inlay/onlay, if the existing setting is to be changed.

Change undercut property of coping
e.g. from “Coping has no undercuts” to “Coping has 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 (= stepover direction), rotate 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 (Objektinfos anzeigen)
Show part messages.

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 and align with surface (Ansicht drehen und zur Oberfläche ausrichten)
Rotate view and set the view direction of the part perpendicular to the surface at the selected point; e.g. for primary parts attachment, optimally align the stepover direction of the user-defined areas.

Adjust view to insertion direction (Ansicht auf Einschubrichtung einstellen)
Adjust the view of the part in line with the 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 selected part, hide the rest.

Show all — Show all parts.

Menu [Settings]

Machining — Call up submenu, 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, define defaults for connectors and sintering pins.
> Connectors	Define connectors, set default.
> Screw-channel connectors (Schraubenkanal-Konnektoren)	Define screw-channel connectors, set default.
> Sintering pins	Define sintering pins, set default.
Tools	Call up submenu, create and manage tools and tool holder.
> Tools	Define tools (template generator option).
> Tool holder	Define tool holder (template generator option).
Milling strategies	Call up submenu, create milling strategy: Name, Parameters, Allocation (template generator option).
Parts	Call up submenu, create and manage parts and part information.
> Part types	Manage part types, create and manage user-defined part types.
> Part information (Objektinformation)	Enter designations for additional part information to be displayed in the part browser.
Import 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 details for directories, messages, information, accuracy.
> Load wizard	Wizard and wizard functions for loading the part.
> Project management	Settings for project management.
> Part tracking	Settings for part management.
> Consistency checks	Type and scope of the consistency check.
> Calculation	Output directory, settings for calculation and behavior in the event of an error.
> Postprocessing	Output directory, file name for NC file, settings for creation of the NC file.
> Navigation	Assignment of mouse buttons for zoom, rotate, pan.
> Display	Edge smoothing, color assignments for holders, part, preparation lines, workspace building elements.
> Connector behavior	Updating and behavior in the event of overlapping.

Menu [Extras]

Refresh licensing	Update license.
Clean up part administration collection	Clean up part group (optional).
hyperVIEW®	Call up simulation program (optional).
License Center	Function for license management and activation.
hyperDENT® Calculation Merge	Call up additional module, show NC file after successful calculation, and merge individual calculations.
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
> Save external geometry (Speichere externe Geometrie)

3.4 Part browser

Parts

Displays parts saved in the project.

Context menu

Delete part	Delete selected part.
Edit template parameters	Edit/change template for part (template generator option).
Change part type	e.g. from bridge to crown, if the system suggestion is to be changed.
Change part name (Objektnamen bearbeiten)	Change the part name, if the system suggestion is to be changed.
Calculate toolpaths	Calculate construction data with the toolpaths.
Print selected parts	Print parts with current print setting selection.
Lock part	Lock selected part A part can be locked as follows: - Manual locking -Locked by the system during the calculation of the toolpaths -Locked after the toolpaths have been calculated, as identification for used blank material, for simultaneous display of the part as “Outline”

3.5 Part data

Additional part information
: Display and entry of additional, customerspecific part information. The information (designation) must first be entered under the menu item [Settings] > [Part information...].

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 part (template generator option).


Lock part
: Lock selected part
A part can be locked as follows:
-Manual locking
-Locked by the system during the calculation of the toolpaths
-Locked after the toolpaths have been calculated, as identification for used blank material, for simultaneous display of the milled area of the part as "Outline"

3.7 Information window

The data displayed in the information window for parts in the blank varies depending on the cursor position. It shows data for the elements at which the cursor is currently pointing: part, preparation line, screw channel, etc.

Part

Name/Label	Name of part

Maximum angle of insertion direction

Rotation angle around X, Y axis

Absolute angle around Z axis

Maximum deviation of the stepover direction within the relevant axis.

Occlusal insertion direction

Rotation angle around X, Y, Z axis

Maximum deviation of the occlusal stepover direction within the relevant axis.

Preparation line

Insertion direction

Rotation angle around X, Y axis

Absolute angle around Z axis

Maximum deviation of the insertion direction of the coping for the hyperDENT coordinate system. The deviation depends on the set insertion direction and on the inclination of the part in the blank.

Type	Type of preparation line
Preparation line	
Abutment base	
Emergence profile	

Screw channel

Insertion direction

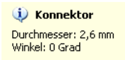
Rotation angle around X, Y axis

Absolute angle around Z axis

Deviation of the tilt of the screw channel.

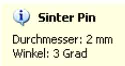
Min./max. diameter	Min. and max. diameter of the screw channel

Connector



Diameter	Diameter at the part.
Angle 0 deg	 Cylindrical

Sintering pin



Diameter	Diameter at the part.
Angle 0 deg	 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, drop down 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	Drop down list, List box: Open menu: click on the arrow icon, 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...]
- Selected part
- 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

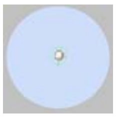
Once the program is running, the toolbar displays the icons for project management and the process steps that are required for machining.

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

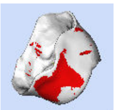
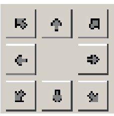
1. Either create a new project and save.
   Or open an existing project.
   Or start with the next step to construct an individual part and discard the details or save at a later time in a project.
2. Select milling unit.
   Set the milling machine and the associated holder for the machining process. The details are incorporated in the program calculation.
3. Load blank.
   Select the blank from the part administration and load into the project. All parts are placed on the blank.
4. 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.

The other icons are only active if a part is selected

5. Set milling direction, adjust direction.
   - Set the general alignment (= stepover 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 check undercut 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 preparation line, other part characteristics and depending on the part type, the pontics, the screw channel, or the interface geometry.

- Select the [Preparation line] tab and select Type.
  - Copings, inlay
    Click into the part, for bridges click into the cavity or inside the preparation line:
    The crown edge – preparation line – is determined and highlighted in blue.
  - Abutment base, Emergence profile
    Click onto the relevant boundary line:
    The boundary line is identified and highlighted in color.

The type of preparation 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 marked with 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.

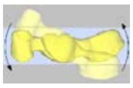
Set template correctly.

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

- If necessary, set the user-defined machining area:
  Select the [Preparation line] tab and select [User-defined area] (Benutzerdefinierter Bereich) as the type. Call up the [Plot outline] (Kontur zeichnen) function and plot the user-defined area. Further details --> “Identify part features” (Objekteigenschaften bestimmen) > “User-defined area” (Benutzerdefinierter Bereich).

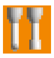
Set template correctly.
Set details correctly under “Tilt part in blank”.

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.



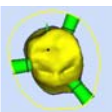
   - Tilt part.
     - Minimize height.
       If the part is askew in the blank and is therefore too high.
     - Minimize inclination (angle optimization).
       If the part is at a steep angle and if this produces large undercuts with unfavorable tilt angles.
     - Enter degree value for maximum tilt 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 and configure the new settings for the milling unit and material for your parts, and the dental restorations.

5.2 Open 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.
 Delete filter	Delete 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 Open 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 Adjust 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 order of columns 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.

Show/hide 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

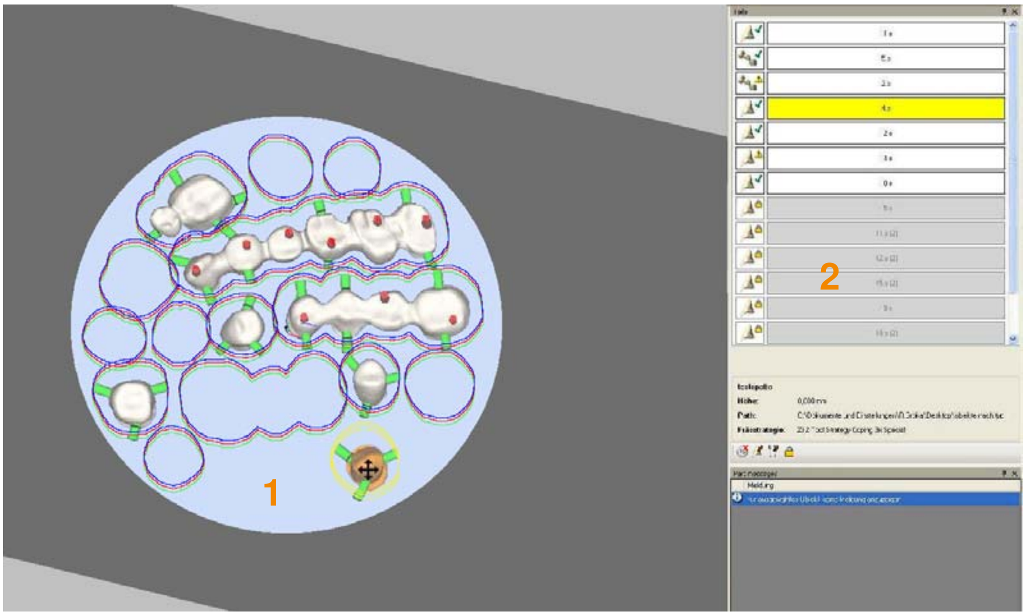


Figure 5-1

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 Print project

5.4.1 Page layout, Print project

Print Page Setup..., Page layout

The selection window shows the functions to use to arrange the print page: Size, Alignment, Margins.

Paper	
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.

Print project

The selection window displays details about the selected printer, the selection menu for the printer, and the printer properties, along with the input options for the pages to be printed (print area) 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 following icons: Print, Page layout, 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 with 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 for the print preview (Ansicht für die Druckvorschau)	View direction of the part for the printout:
Occlusal side	Display of the occlusal side.
Cavity side	Display of the cavity side.
Display project information	Activate/deactivate printout of project information.
Display machine	Select project information for the printout.
Display fixture	Select project information for the printout.
Display blanks (Rohteile anzeigen)	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	
Display part path	

5.4.3 Automatically print project after calculation

The data set under "Print project Page Setup" is automatically sent as a printout to the default printer as soon as the calculation is complete and the NC file has been created. You therefore obtain, for example, the appropriate setup plan for the NC file with blank information, part information, and tool information.

- The settings for the data to be printed are controlled under the [File] menu, [Print project Page Setup] submenu. Further details --> "Print project", > "Print project Page Setup".
- The settings for the printer are controlled under the [File] menu, [Print project] submenu. Further details --> "Print project" > "Page layout, Print project".

- Automatic printout is activated and deactivated under the [Settings] menu, under [General settings] > [Postprocessing] >[Automatically print out project after calculation].

6 Select 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.
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”.

6.1 Fixture

The fixture is the receptacle 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 provided by the system supplier to match the geometry of the fixture or are provided as a service by the software provider.
- For the optional function of automatically placing the parts in the blank, the relevant settings must be configured for the fixture: --> “Settings” > “Machining” > “Fixture”, > “Additional settings”.

7 Load 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.

As well as the usual blanks, you can also use prefabricated blanks (prefabs) with a prefabricated screw channel, screw fit, and interface geometry. These prefabricated blanks are usually provided by the system supplier to match the corresponding holder.

7.1 Load 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 administration with the list of blanks, preview, and data for the selected blank, 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 selected blank: Displays the outline of the placed and calculated parts 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 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.
Parts in blank	Displays the parts that are placed and saved in the selected blank.
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 blank in order to quickly find a suitable blank. 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.
Delete filter	Delete filter setting, show all.

FileSelection window for loading blanks from the file system.	
New	Call up selection window for loading new blanks.
Open	Load selected blank.
Cancel	Cancel process, do not load blank.

7.1.1 Load blank

Either

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

7.1.2 Adjust 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 order of columns 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.

Show/hide 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, only suitable blank types for the project and fixture are shown.

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. 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.

7.3 Create, edit blank type

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

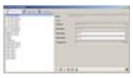
- Blanks for machining can only be loaded from the previously defined blank types.
- Blank types can only be created using the previously defined geometries and materials.

7.3.1 Create, edit 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] under the menu item [Blanks], call up the menu item [Blank types...]:
   The window [Blank type settings] 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).

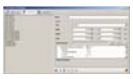
7.3.2 Create, edit geometry

The geometry describes the shape of the blank:

- Disc — Disc shape
- Cylinder — Cylinder shape
- Box — Box shape
- Extrusion — Blanks of any shape that are described by a closed outline and defined by the input of a height.
- Freeform — Blanks of any shape that are described by a model file (closed STL model).
  - Prefabricated blanks (prefabs)
    Prefabricated blanks of any shapes with prefabricated screw channel, screw fit, and interface geometry, which 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. The attachment models 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.
 Pre-defined abutment geometry (Vordefinierte Abutment Geometrie)	Call up submenu, details of the nesting point for the part if the interface geometry is already fully processed, --> “Prefabricated blanks – Prefab”. The freeform model with the interface geometry must be available as an *.STL file.
Zero point (Nullpunkt)	Call up the submenu for details of the position of the zero point for nesting the part.

Screw channel axis (Schraubenkanalachse)	Call up the submenu for details of the position of the screw channel axis for nesting the part.
X axis (X-Achse)	Call up the submenu for details of the position of the X axis for nesting the part.
Hole diameter	Diameter of the screw channel.
 Alias settings	Call up submenu for further settings for the mapping function for the defined interface for nesting the part. Display, add, delete setting.
Name	Name of the Alias setting
Rotation	Angle for the rotation around the screw channel axis.
Offset	Default value for the offset 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, fixture, and part must have the same alignment.

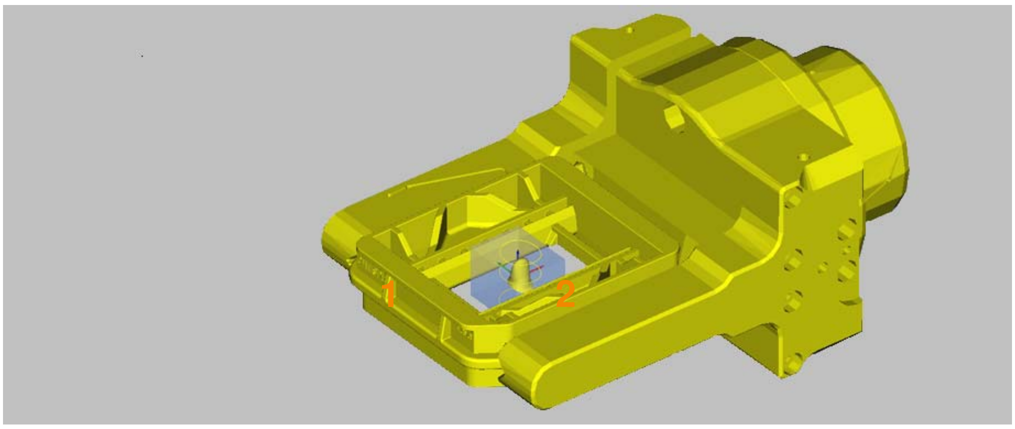


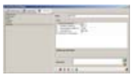
Figure 7-1

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 Create new material, edit it

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 values for uniform scaling.
No Default X, Y, Z-scaling	Default values for variable scaling in the X, Y, and Z-direction
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

Prefabricated blanks have a pre-machined screw channel, screw fit, and interface geometry.

The blanks, blank data, and holder data are provided by the system manufacturer accordingly. For further information, please contact hyperDENT® Support: --> “Contact”.

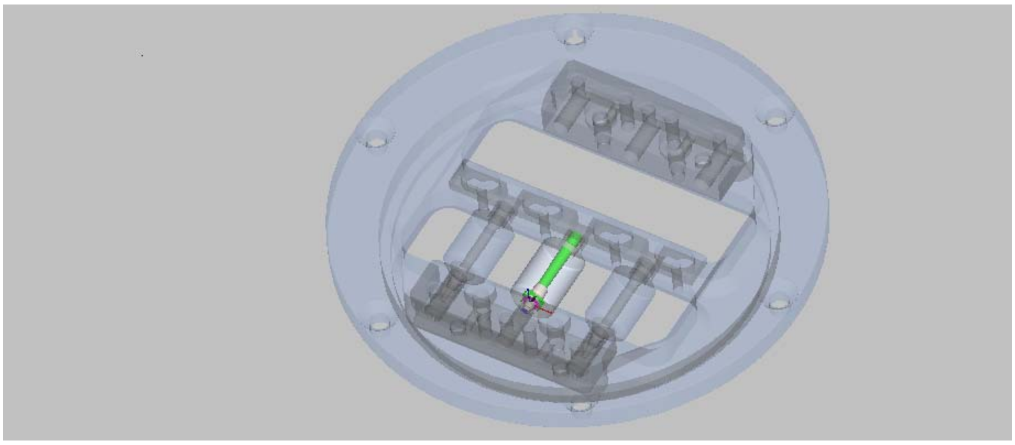


Figure 7-2

- The nesting point for the blank and part must match the present holder definition.
- The coordinate systems for the holder, blank, and part must match.
- The STL models for the holder and blank must be closed.

Nesting point for the part

The details of the nesting point for the part arise from the definition of the geometry of the blank: --> “Load blank” > “Create, edit blank type” > “Create, edit geometry”.

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

8 Load 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, manually move the part to a suitable place in the blank or use automatic placing.

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

8.1 Load 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
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

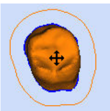
Display nested in blank	Name of blank on which the part is already nested.
Measure	Select axis (X, Y, Z) for the measurement display.
Size	Max. dimension for the selected axis.
In blank	Max. blank dimension for the selected axis, if a blank with scaling factor is already loaded.
Type	Selection for the type of dental restoration. Automatic transfer via the defined interface.

- Coping
- Coping bridge
- Atomical crown
- Atomical bridge, Maryland bridge
- Abutment
- Abutment crown
- Abutment bridge
- Anatomical abutment bridge
- Inlay/onlay

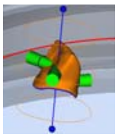
- Inlay/onlay bridge
- Anatomical inlay/onlay
- Inlay/onlay crown bridge
- Telescope
- Overpress
- Occlusal splint
- Model
- Model stub
- User-defined part

Material	Selection field for the raw material.

8.2 Place 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.
   This enables you to move the parts to be fully machined into the blank. If the blank is prefabricated, the part must be placed with the utmost accuracy: --> “Load part” > “Place part precisely in prefabricated blank – Align with screw channel”



   The axis line of the part is additionally shown in the page 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

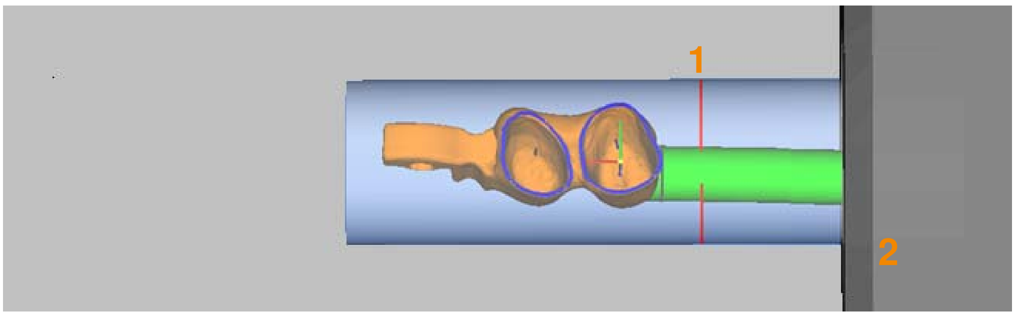
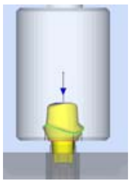


Figure 8-1

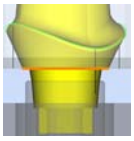
If the “Fixture” boundary is selected, the boundary line (1) (the prescribed distance to the holder (2)) is displayed when placing the parts in moving mode. --> “General job parameters” > “Bounding strategy”.

8.3 Place part precisely in prefabricated blank

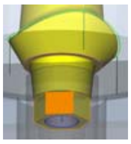


Prefabricated blanks (prefabs) have a pre-machined screw channel, screw fit, and interface geometry. To ensure that the intended construction is correct, the part must be placed with the utmost accuracy and rotated to the correct position.

- Alignment of the screw channel axis
  Align the screw channel axes of the part and the blank, i.e. so that the axes lie directly over one another and are pointing in the same direction.



- Height of the part – offset
  The reference surfaces lie directly over one another 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 object is at a certain angle to the interface geometry.



  - The interface geometries of the part and blank lie directly over one another, i.e. they are precisely aligned with one another in the angle position around the screw channel axis, e.g.: the prefabricated hexagon on the blank and the prefabricated hexagon on the part.

The part data is usually transferred from the CAD system such that it matches the blank data. Manual adjustment is possible where required: --> “Align part with screw channel”, “Offset...”, and “Rotate...”.

8.3.1 Align 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] (Teil an Schraubenkanal ausrichten).

The window displays the buttons for precise alignment and rotation of the part.

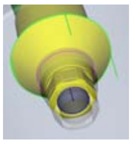
Move up/down arrow buttons (Pfeiltasten verschieben auf/ab)	Move the part along the screw channel axis in single increments (offset) or multiple increments.
Rotate left/right arrow buttons (Pfeiltasten drehen links/rechts)	Rotate the part around the screw channel axis in single increments (angle) or multiple increments.
Undo	Cancel process, undo offset by increments.
Offset increment (Offset Inkrement) 0.01 to 10 mm	Increment for the offset along the screw channel axis
Angle increment (Winkel Inkrement) 0.01 to 10°	Degree of rotation around the screw channel axis.
Move part to reference surface (Objekt auf Bezugsfläche verschieben)	Move part along the screw channel axis until the reference surfaces match.
Rotate part to selected point (Objekt auf markierten Punkt drehen)	Rotate part around the screw channel axis until the selected points match.
Rotate part to reference surface (Objekt auf Bezugsfläche drehen)	Rotate part around the screw channel axis until the selected reference surfaces match.
Close	Exit process, close window.

8.3.2 Manual offset and rotation

1. Select the part, call up the [Edit] menu, and select the following menu item: [Part] > [Align part with screw channel] (Teil an Schraubenkanal ausrichten).
2. Enter the offset increment.
3. Press the arrow buttons and move the part along the screw channel to the required position.
4. Enter the angle increment for rotation.

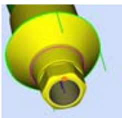
5. Press the arrow buttons and rotate the part around the screw channel axis to the correct position.

8.3.3 Move to reference surface semi-automatically



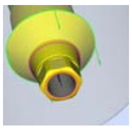
Move the part along the screw channel axis until the selected reference surfaces match.

1. Select the part, call up the [Edit] menu, and select the following menu item: [Part] > [Align part with screw channel] (Teil an Schraubenkanal ausrichten).
2. Call up the [Move part to reference surface] (Objekt auf Bezugsfläche verschieben) function.
   Click on the function button.
   The holder and blank are hidden, the part is displayed for selecting the reference surface.



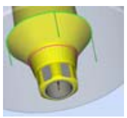
3. Select the reference surface on the part:
   Click on the reference surface on the part that matches the corresponding reference surface on the blank.
   The part is hidden and the blank is displayed.

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



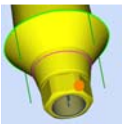
5. The part is automatically moved along the screw channel axis until the selected reference surfaces match.
   The new position is displayed on the screen.

8.3.4 Rotate to selected point semi-automatically



Rotate the part around the screw channel axis and align in any position with 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] (Teil an Schraubenkanal ausrichten).
2. Call up the [Rotate part to selected point] (Objekt auf markierten Punkt drehen) function.

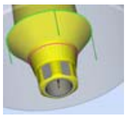


3. Select point on the part:
   Click on a point on the part that is to match the corresponding point on the blank.

4. Select point on the blank:
   Click on a point on the blank that is to match the corresponding point on the part.

5. The part is automatically rotated around the screw channel axis until the selected points match.
   The new position is displayed on the screen. The angle information is displayed in the status line.

8.3.5 Rotate to reference surface semi-automatically

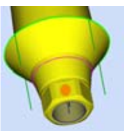


Interface geometries with rotation safeguarding require precise orientation of the part to ensure that the alignment of the part matches the prefabricated interface surfaces of the blank.

Rotate the part around the screw channel axis and align with the prefabricated interface geometry such that, for example, the surfaces of the hexagon on the part match 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] (Teil an Schraubenkanal ausrichten).

2. Call up the [Rotate part to reference surface] (Objekt auf Bezugsfläche drehen) function.



3. Select the reference surface on the part:
   Click on a reference surface on the part that is to match the corresponding reference surface on the blank.

4. Select the reference surface on the blank:
   Click on a reference surface on the blank that is to match the corresponding reference surface on the part.

If the corresponding reference surfaces on the blank are now shown in this view, it is sufficient to click near the presumed reference surface, e.g. click on the same screen position as before when selecting the part.

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

8.4 Place part automatically – autonesting (option)

hyperDENT® can automatically and optimally place the part (1) in the blank when loading.

The settings are separate for each holder (fixture): Menu [Settings] > menu item [Machining] > tab [Fixture].

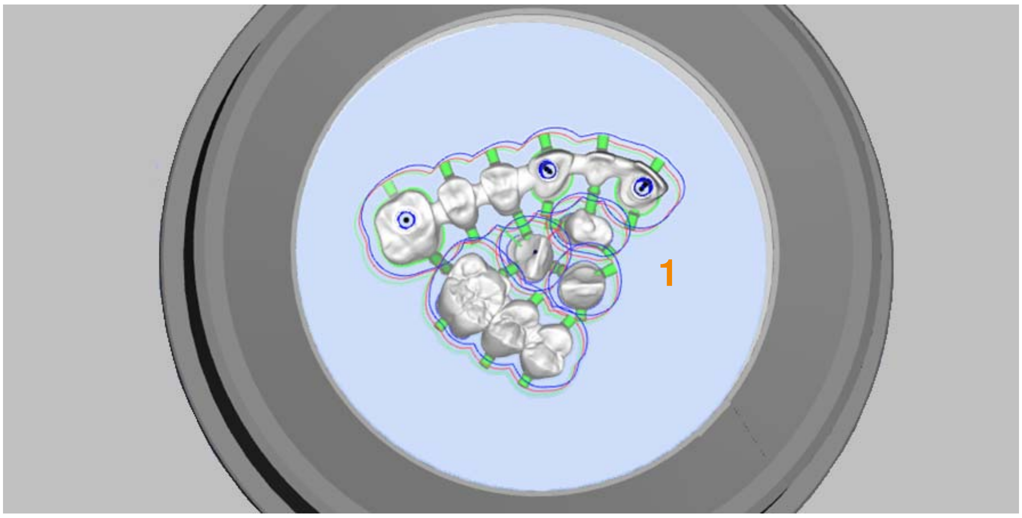


Figure 8-2

8.4.1 Configure automatic placing

1. Call up the machining settings:
   Select the menu [Settings] and call up the menu item [Machining].

2. Call up the [Fixture] tab.
3. Select the fixture (holder):
   Click in the row that contains the required holder.
4. Select the function [Edit]:
   Click on the button.

Activate

5. Configure the parameter for placing:
   Click on parameter [Automatic] and enter “Yes”.
6. Configure the other parameters as required:
   --> “Settings” > “Machining” > “Fixture”, > “Additional settings” > “Placing parameters”.
7. Accept setting and close window.
   Automatic placing is activated for this holder.

Deactivate

8. Configure the parameter for placing:
   Click on parameter [Automatic] and enter “No”.
9. Accept setting and close window.
   Automatic placing is deactivated for this holder.

8.4.2 Automatically place part when loading (nesting)

The function for automatic placing must be activated for the selected holder. The part is then automatically placed in the blank during loading in a way that saves space.

The part can be moved manually afterward.

8.4.3 Place part automatically afterward (nesting)

If automatic placing is activated for the selected holder, you can automatically place the preloaded part (1) in the blank in a way that saves space via the menu or the context menu.

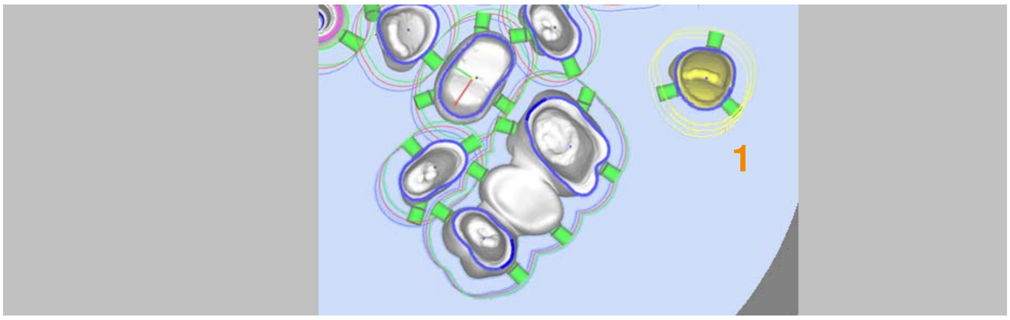


Figure 8-3

1. Select the part (1).

2. Call up the menu [Edit], select the menu item [Part], or call up the context menu.

Either

3. Select the menu item [Nest part in blank – local] (Objekt im Rohteil schachteln – Lokal):
   The part is optimally placed near the current position in the blank (2).

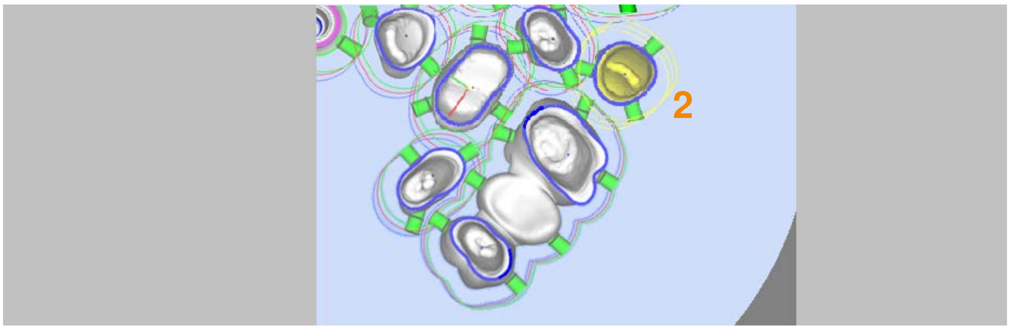


Figure 8-4

Or

4. Select the menu item [Nest part in blank – global] (Objekt im Rohteil schachteln – Global):
   The part is optimally placed in the blank (3).

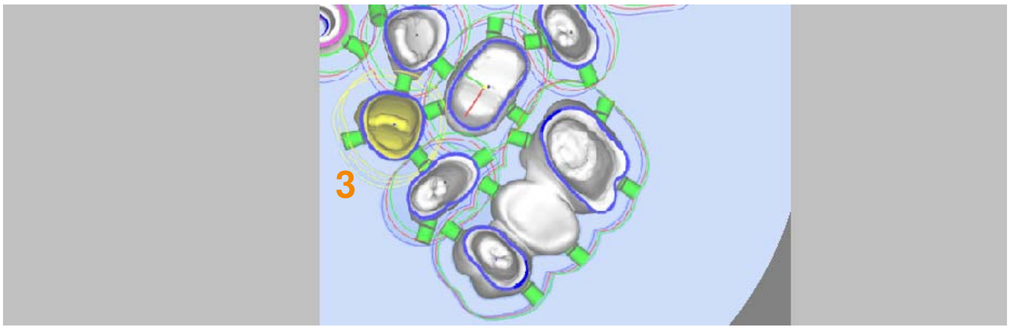


Figure 8-5

8.5 Consistency check

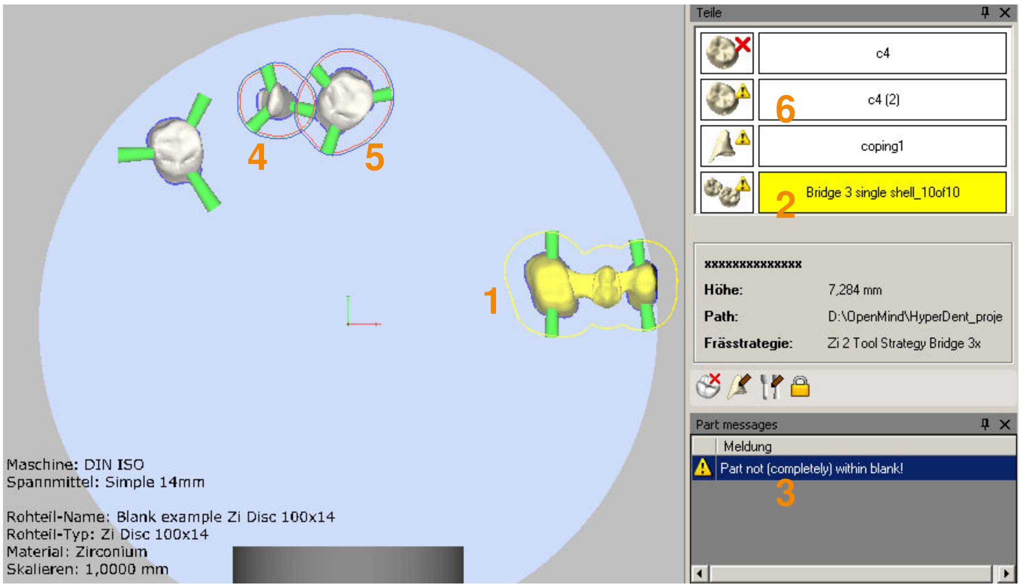


Figure 8-6

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).

8.6 Defined interface – Using the wizards

The [Load wizard] function can be used to enter the process parameters much more quickly since the dialog boxes for the next process steps are automatically opened in the sequence of the process chain.

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

Yes	Activate function
No	Deactivate function
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	Enable wizard: 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 preparation 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.
Place part in blank (Objekt im Rohteil platzieren)	
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 defaults.

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

8.7 Direct interface – working with original CAD data

The direct interface with CAD programs makes it easy to transfer CAD data for processing in hyperDENT®.

3Shape Dental Designer

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

Settings for 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 filename with OrderID	No
Milling	Yes

8.8 Direct interface – Connection to the CAD system

The direct connection (CAD Connect) with CAD programs makes it easy to transfer CAD data for processing in hyperDENT®:

- The CAD data is automatically transferred to hyperDENT®.
- The new parts appear in the list of parts yet to be machined and no longer need to be loaded via the file system.
- hyperDENT® can also be started from the CAD system with the transferred parts.

Available for CAD system

- Exocad
- 3Shape

Activate function

Activate the function via menu [Settings], submenu [General settings], menu item [Part tracking] under parameter [Activate list of new parts] (Liste neuer Teile aktivieren); --> “Settings” > “General” > “Part tracking” > “Activate list of new parts” (Liste neuer Teile aktivieren).

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

Then load the parts from the list of parts yet to be machined.

Load parts: --> “Load part”.

9 Set 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.

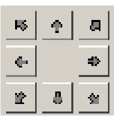
The icon is only active if a part is selected.

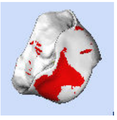
Depending on the data that is loaded, you must set the milling direction:

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

9.1 Set milling direction

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

	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	Run undercut check.



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 stepover direction

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

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

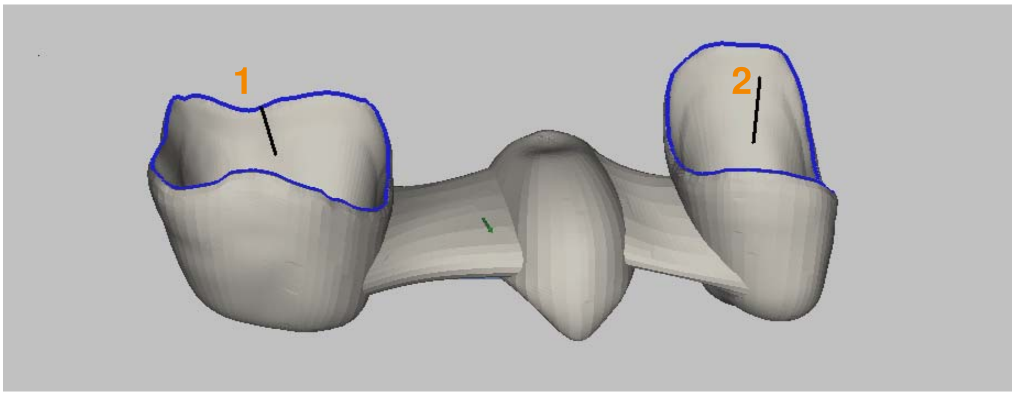


Figure 9-1

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

Setting the specific alignment

1. Select the preparation line of the desired coping (1, 2):
   Click on the preparation line.
   The selected preparation line is displayed in a different color.
2. Set the new stepover direction for the coping (1, 2):
   Using the right mouse button, rotate the part to the new position so that if possible there are no undercuts for this coping.

3. Call up the context menu and select the menu item [Insertion direction from view direction].
   The new stepover direction is shown by a line in 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 preparation line and delete it.

9.3 Occlusal stepover direction (insertion direction)

An occlusal stepover direction can be defined for individual copings.

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

Setting occlusal stepover direction

1. Set the new stepover direction for the occlusal side of the coping:
   Holding the right mouse button down, rotate the part to the new position so that if possible there are no undercuts on this coping.
2. Call up the context menu and select the menu item [Set occlusal insertion direction].
   The new stepover direction is shown by a line on the coping.

Deleting occlusal stepover 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 preparation line and in the context menu, select the menu item [Change undercut property of coping] > [Coping has undercuts].

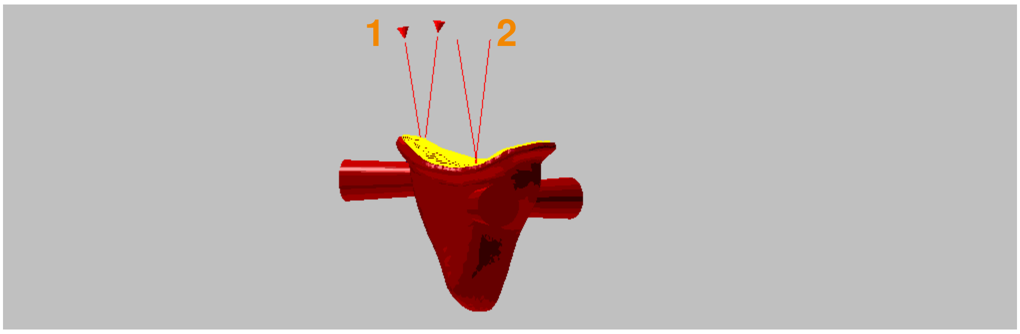


Figure 9-2

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

Selecting undercut machining

1. Select the preparation line of the coping:
   Click on the preparation line.
   The selected preparation line is displayed 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] > [Coping has undercuts]:
   Undercut machining is displayed by a checkmark in the menu item and an icon 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 preparation line of the coping.
2. Call up the context menu and select the menu item [Change undercut property of coping] > [Coping has no undercuts]:
   The undercut machining is deselected and the icon on the coping is deleted.

10 Identifying part features

The part features include the edge of the crown (= preparation 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 only active 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, at least one preparation line per coping.

10.1 Preparation line

1. Select the [Preparation lines] tab:
   The selection window shows the settings for labeling the preparation line.

Parameters	Set type of preparation line and type of creation.
Type	Select the preparation line. The preparation line type must match the part type, otherwise the calculation is not possible.

- Coping
- Inlay/onlay
- Abutment base
- Emergence
- User-defined area

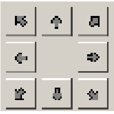
Mode

Manual	Manually label the area.
Automatic	Automatically determine the preparation line.

Plot outline (Kontur zeichnen)
Plot the outline for the user-defined area.

Curves (Kurven)	Connect contour points with curves
Straight lines (Gerade Linien)	Connect contour points with straight lines
Freehand line (Freihandlinie)	Create contour with a freehand line.
Angle	Angle range of coping edge.
Preparation line	Preparation line details
Selection	Display and selection for preparation line count and number.
Offset	Default value (width of coping edge) to move the preparation line to the other edge of the coping edge.
Positive value	Move inwards.
Negative value	Move outwards.
Undercuts present (Hinterschnitte vorhanden)	Select undercuts for multiple machining with fixed tilt. The correct settings must be configured in the template.
Category	Number of the corresponding machining template.
Move	Move preparation line by the offset value.
Back	Undo last move.
Insertion direction	Display, determine insertion direction.
Alignment	Set coping-specific insertion direction (stepover direction): --> “Set milling direction” > “Coping-specific stepover direction”.
From view (Aus Ansicht)	Set current view direction to the part as occlusal insertion direction (= stepover direction).

Fine-adjust direction	Adjust the part with precision.
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 for height minimization is already factored in here. Other angle deviations from the machining (template) still need to be added.

Calculate undercuts in check direction.

Update	Run undercut check.

Adjust view to insertion direction (Ansicht auf Einschubrichtung einstellen)

Rotate view and adjust the view direction to the insertion direction.

Angle

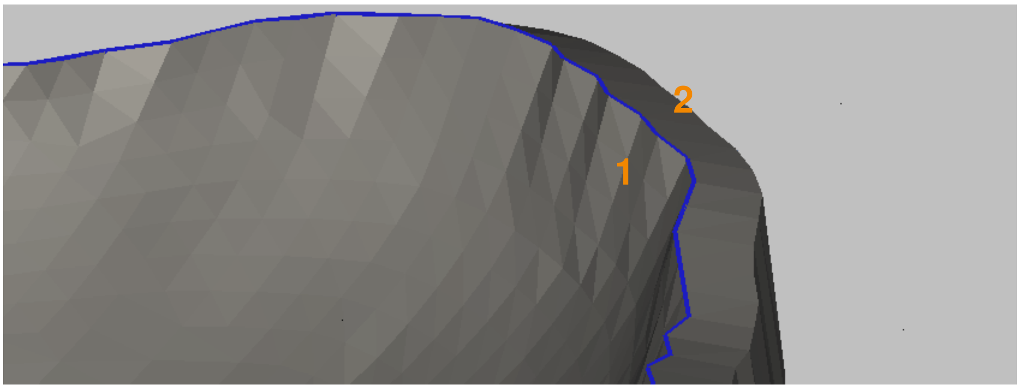


Figure 10-1

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

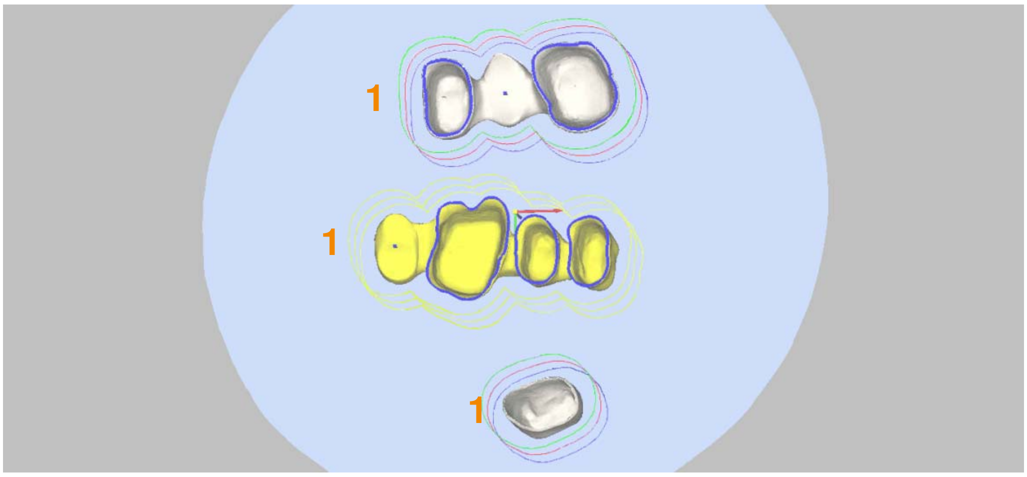


Figure 10-2

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

10.1.2 Manually determining preparation line

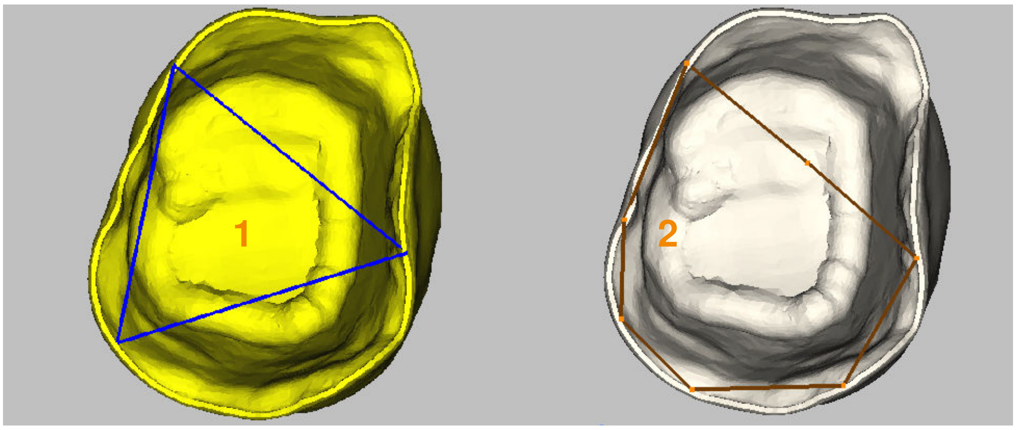


Figure 10-3

1. Select the [Preparation lines] tab.

2. Select “Manual” mode.
3. Select type.
4. Click 3 times on the coping edge to create a triangle (1).
5. Click on the sides of the triangle to create new anchor points and keeping your finger on the mouse button, drag them to the coping edge in place of the preparation line (2).
6. Manually and accurately recreate the preparation line using other anchor points.

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

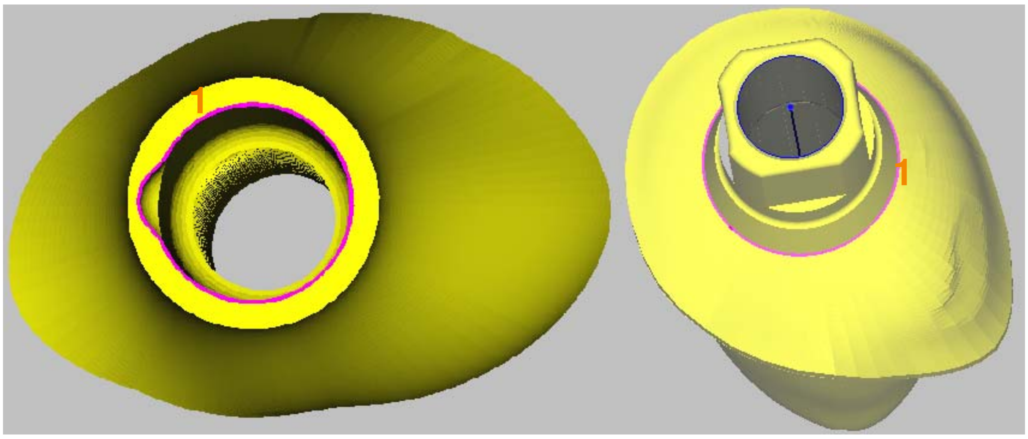


Figure 10-4

1. Select the [Preparation 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 preparation line” and place the anchor points onto the boundary line of the abutment base.

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

10.1.4 Determining emergence profile

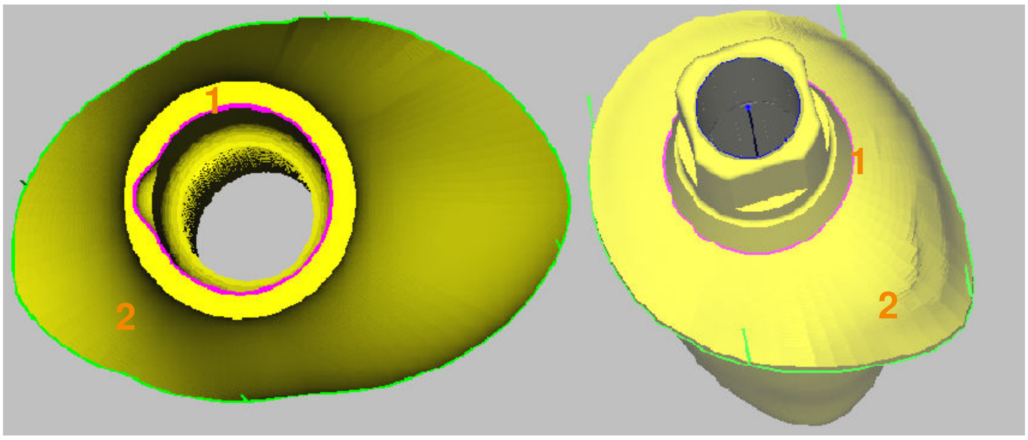


Figure 10-5

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

You can determine a separate stepover direction for the machining area emergence: --> “Coping-specific machining”.
If, for example the abutment is steeply angled, you can also set an occlusal stepover direction: --> “Set occlusal insertion direction”.

10.2 User-defined area

To enable separate machining of specific part areas, e.g. specific contour geometries (1, 2) or undercut areas (3), you can set user-defined areas and assign to these areas your own categories with their own machining parameters.

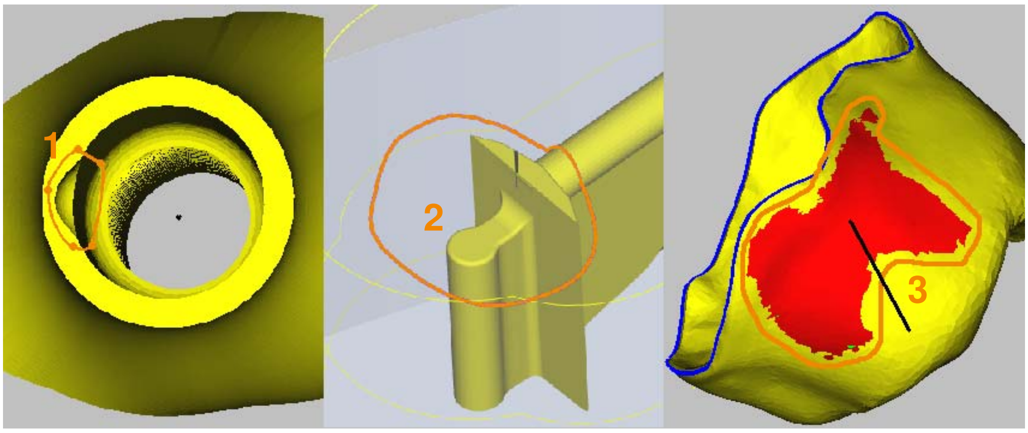


Figure 10-6

Plot area

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

5. Call up the [Plot outline] (Kontur Zeichnen) function.

Either

6. Select the line type.

Curves (Kurven) Connect contour points with curves

Straight lines (Gerade Linien)
Connect contour points with straight lines

7. Plot the contour line.
Click 3 times on the edge of the area to create a triangle (1).

8. Click on the sides of the triangle to create new anchor points and keeping your finger on the mouse button, drag them to the point in the desired area.
9. Use additional anchor points or a freehand line to manually and accurately recreate the area.#

Or

10. Select the line type.

Freehand line (Freihandlinie)

Create contour with a freehand line.

11. Plot freehand line:
    Hold down the left mouse button and use a freehand line to manually and accurately recreate the area (2, 3).
12. Accept line:
    Click on [Accept].
    The contour line is saved.
13. Plot another contour line, change the contour line, or close the window.
14. Assign a category for the machining template:
    Enter the category number of the machining template that is to be used for this area.
15. Create categories of the machining template: --> “Milling strategies” > “Machine user-defined areas”.

Delete area

1. Select and delete area:
   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 to label the pontics.

Count	Number of markings for the pontics.

Marking the pontics

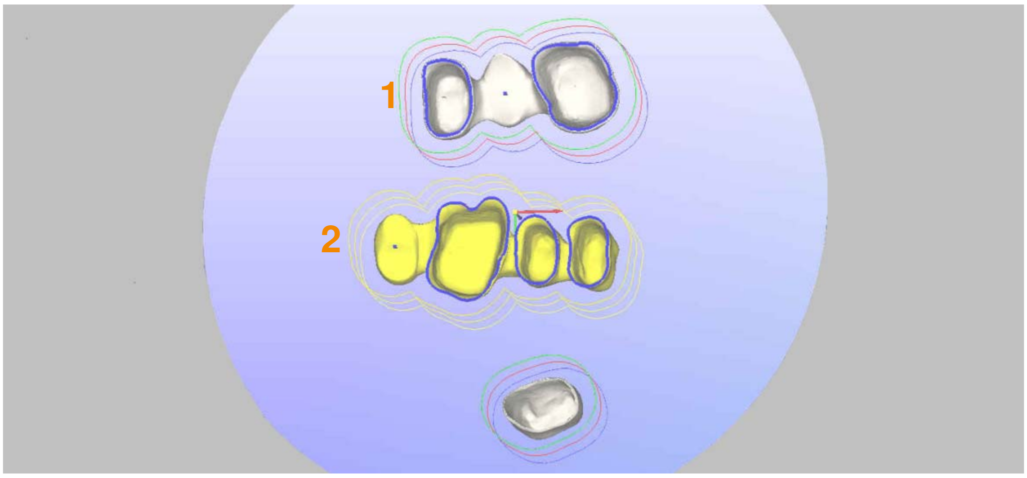


Figure 10-7

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

10.4 Edit 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 preparation line. This is necessary if for instance there is a coping preparation (1) and an inlay preparation (2).

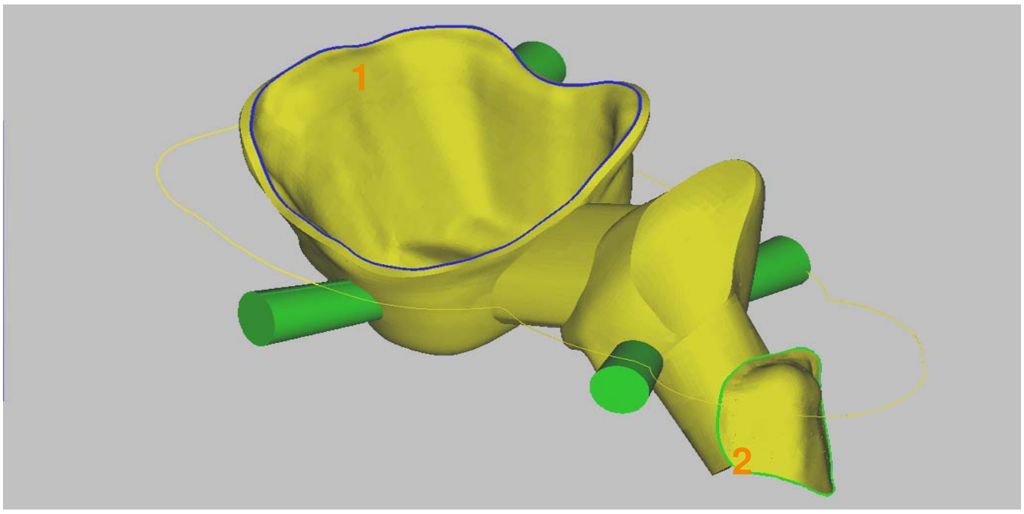


Figure 10-8

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

Changing preparation line type

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

Deleting a type that has been changed

1. Select the preparation 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 machining template to each of these areas for the individual machining.

10.5.1 Enter implant interface

1. Select the [Implant interface] tab:
   The selection window shows the settings for highlighting the interface geometry and the screw channel of implants, as well as 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	Set interface geometry in hyperDENT®.
External	Load interface geometry from file.

Explicit

Determining areas of the interface geometry in hyperDENT®.

Arrow	Move the tab display.

	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 (XY-Aufmaß)	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 machining 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 info line.

	Delete	Delete external interface geometry.
	Preview	Preview window for external interface geometry.
	Open	Open file with external geometry.
	Global XY allowance (Globales XY-Aufmaß)	Global allowance for all machining template categories allocated to the interface. 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 (Pfeiltasten verschieben auf/ab)	Move the part along the screw channel axis in increments (offset).
	Rotate left/right arrow buttons (Pfeiltasten drehen links/rechts)	Rotate the part around the screw channel axis in increments (angle).
	Offset increment	0.01 to 10 mm Increment for the offset along the screw channel axis
	Angle increment	0.01 to 10° Degree of rotation around the screw channel axis.

10.5.2 Global XY allowance – save on a part-specific basis

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

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

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

Example data input for global allowance

```
<?xml version="1.0" encoding="UTF-8"?>
<Part>
<partdata>
<xyoffset>0.001</xyoffset> [1]
</partdata>
```

1. Input for global allowance

10.5.3 Save external interface geometry, save categories

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

You can also read the categories for the integrated interface geometries and save these in a file.

Save data

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

2. Call up the menu [Extras] and menu item [Execute command].
3. Enter the command [Save external geometry] (Speichere externe Geometrie) and accept:
   Click on [Execute].
4. Select file path, select file type “.hdpartz” and accept:
   Click on [OK].
   The file is saved in the specified directory.

Read category

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

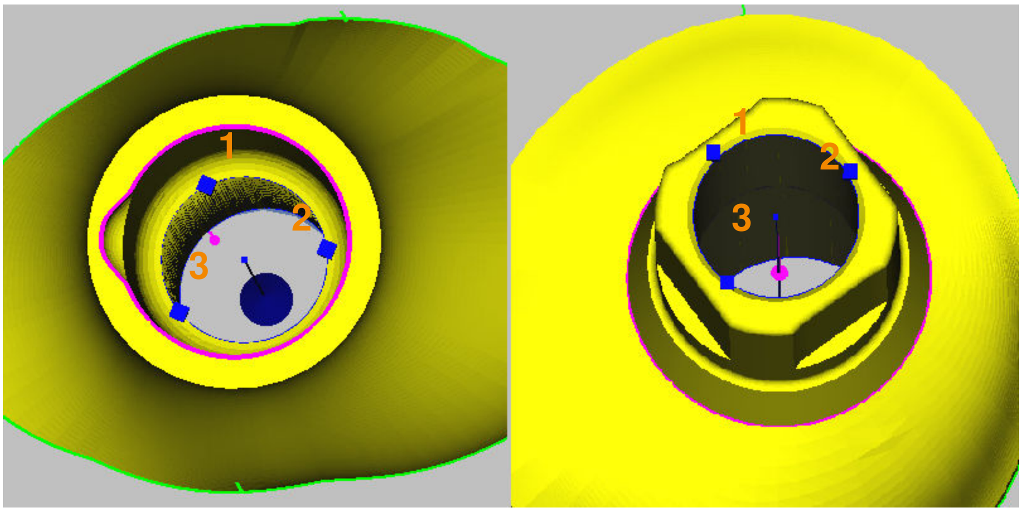


Figure 10-9

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 (rotational axis) is also highlighted in color.

The screw channel forms its own machining area.

10.7 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 machining template to each of these sections for the individual machining.

A template can therefore produce very different interface geometries.

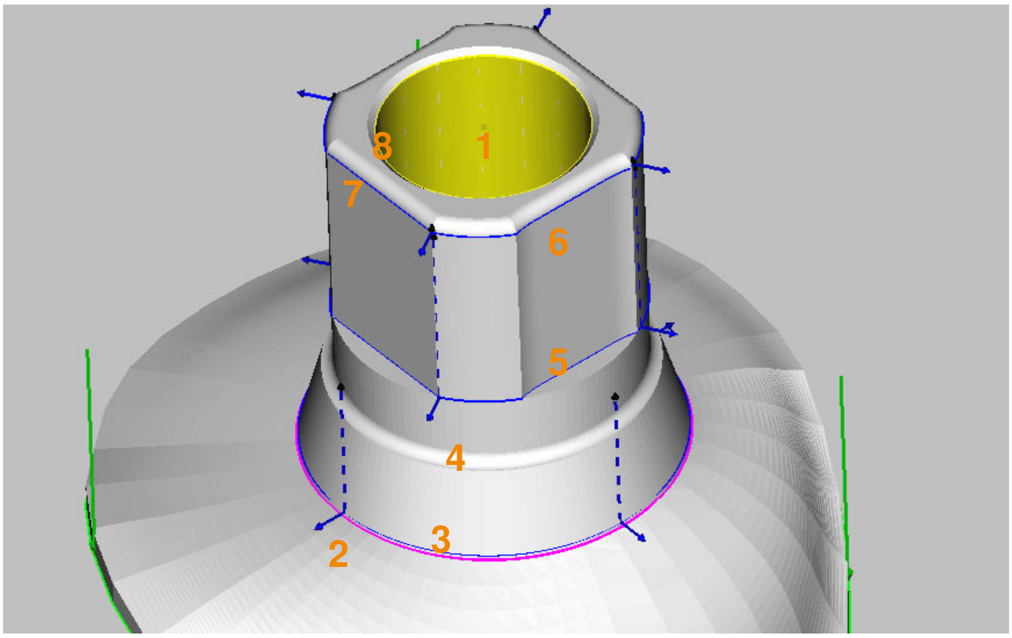


Figure 10-10 Interface geometry with machining areas

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 machining 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 machining template.
14. Click on [Close]:
    The machining sections are saved.
15. Create categories of the machining template: --> “Abutment” > “Machining method for finishing implant interface geometry”.

10.8 Corners – setting the machining of inner geometries

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

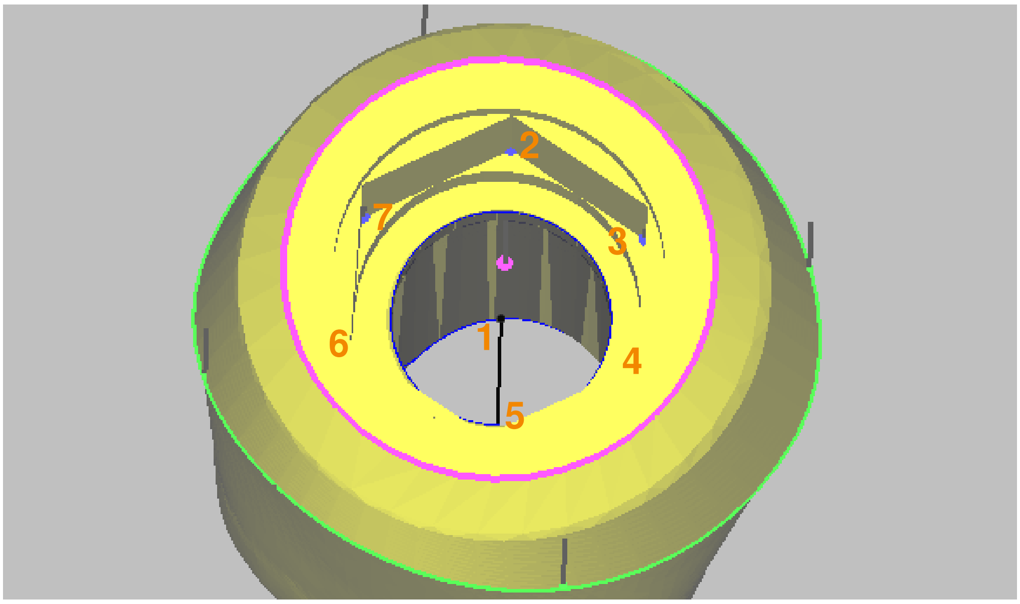


Figure 10-11 Corner machining of inner geometries

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.

10.9 Thread cutting

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

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

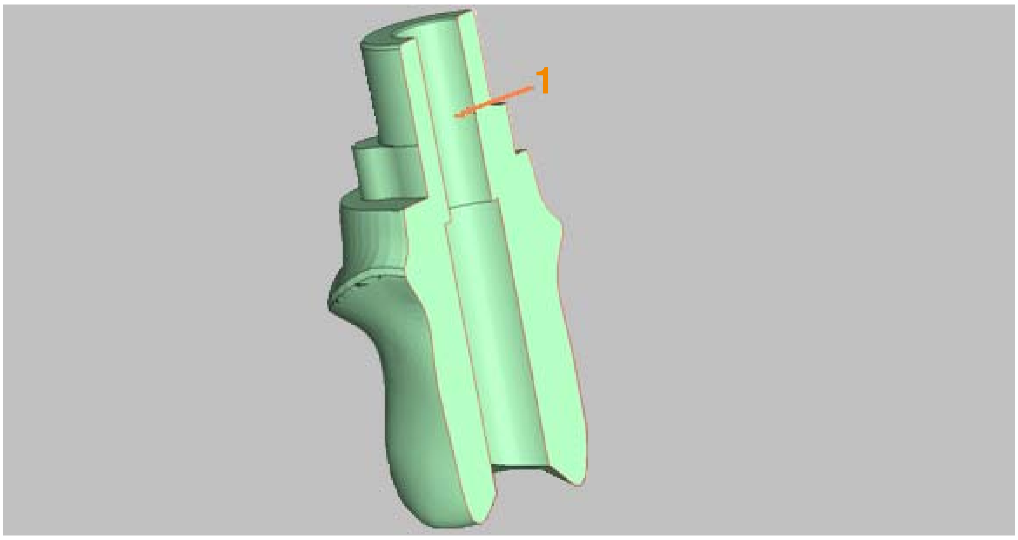
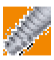


Figure 10-12 Thread milling

Pre-drill 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 drop down menu.
6. If necessary, enter the offset: Start of thread and end of thread.

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

10.9.1 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.

10.10 Individual stepover directions for abutments

You can set individual stepover 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 stepover direction

1. Select the preparation line:
   Click on the preparation line.
   The selected preparation line/boundary line is displayed in a different color.
2. Set the new stepover direction for the machining area:
   Holding the right mouse button down, rotate the part to the new position so that if possible there are no undercuts in this machining area.
3. Call up the context menu and select the menu item [Insertion direction from view direction] or [Set occlusal insertion direction]:
   The new stepover direction is shown by a line on the machining area.

Deleting individual stepover direction

Select the preparation line and delete it.

11 Tilt part in blank

The insertion direction (= stepover 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 only active if a part is selected.

Minimize 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, fixture, and 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.

Minimize inclination , angle optimization

In the case of steeply angled parts, e.g. abutments, the tilt of the stepover 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, 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 rotation	Lock the alignment to the rotation axis.
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 preparation 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.
Tilt	Create an 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

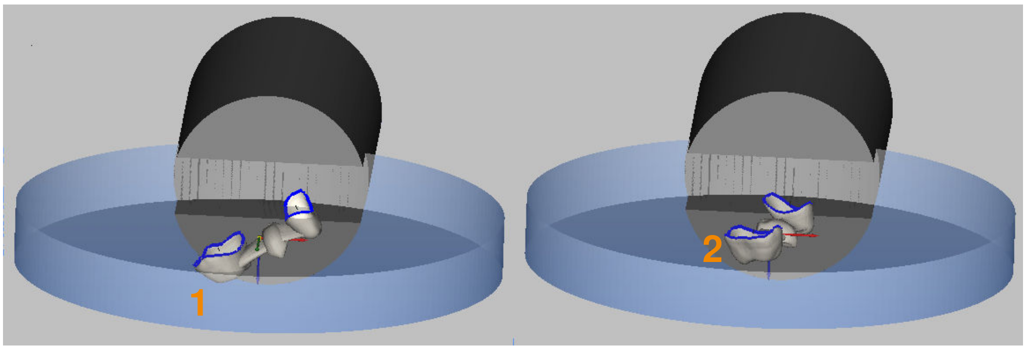


Figure 11-1

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 accordance with the rotation axis (2) of the machine, so that the tilt can be set for the machining.

and

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

Example of angle optimization

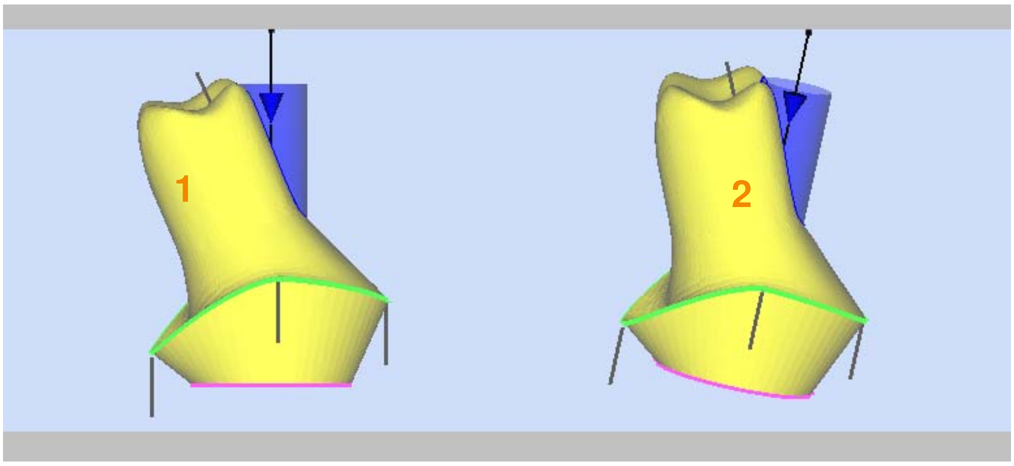


Figure 11-2

Part (1) has been manually placed and the occlusal insertion direction has been set --> “Occlusal stepover 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 Select 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.

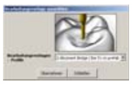
Select the template that you would consider from experience to be most suitable for the selected part.

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.

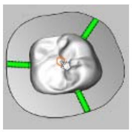


The selection window shows the buttons used to select the templates.

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

13 Set 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.

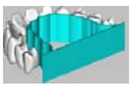
You can change each set connector individually, assign a cut to it, or delete it.

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 with prefabricated blanks (prefabs). It runs along the prefabricated screw channel of the occlusal side of the part to the opposite holder and surrounds the screw channel depending on the angle input in cylinder shape (0°) or cone shape.

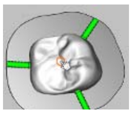
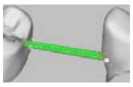
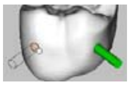
Sintering frames



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

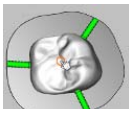
13.1 Set connectors

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 (Schraubenkanal Konnektor)	Click part: Set screw channel connector in accordance with the settings in the track of the prefabricated screw channel. Optional with cut. Apply modified settings.
	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 preparation line	Distance to preparation 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 connectors at the pontics.

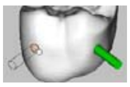
Cut	Connectors that are automatically cut at the end of the machining. Manually set individual connectors with cut or assign cut at a later stage. The job must be stored in the template.
Cut depth	Percentage value for the cut:
0%, None	No cut.
100%, Complete	Complete cut, the connector is completely 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 types.
 Save	Call up selection window [Save profile]. Save settings, save under a new name, set as default.

13.1.1 Setting connectors automatically



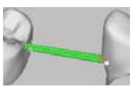
1. Click on the part:
   All connectors are set automatically according to the settings.

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 (optional)



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.

13.1.4 Set 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 automatically set in accordance with the settings.

13.2 Deleting connectors

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

13.3 Edit 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.

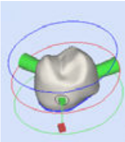
Edit	Activate settings for the machining.
Angle	Taper angle of the connector: Cylinder = 0
Diameter	Diameter at the part.

- Caution!
  With screw channel connectors, the diameter must be greater than: Screw channel diameter + 2x 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.

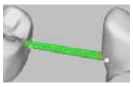
13.4 Moving connectors

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 Autoconnect the connectors



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

Reload parts

1. Place parts:
   Ensure that the milling boundaries of the parts overlap, but do not protrude into other, unfinished parts.
2. Set connectors in automatic mode, --> “Set connectors”.
   The connectors are also set between the parts.

Move 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.

Remove connection

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

13.6 Edit connector settings

13.6.1 Connector settings, screw channel connectors

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

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 preparation line	Distance of the connector to preparation 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).

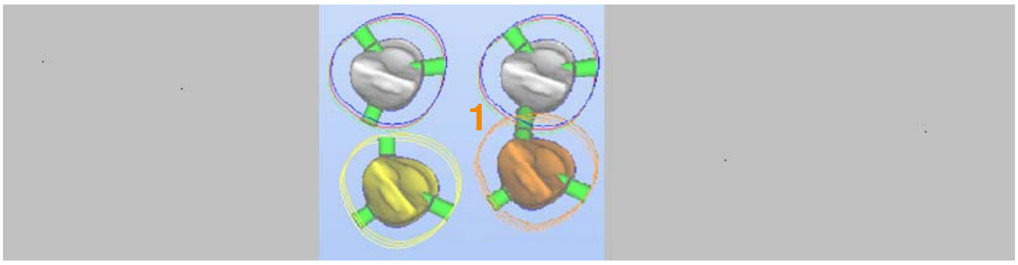


Figure 13-1

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 the 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.

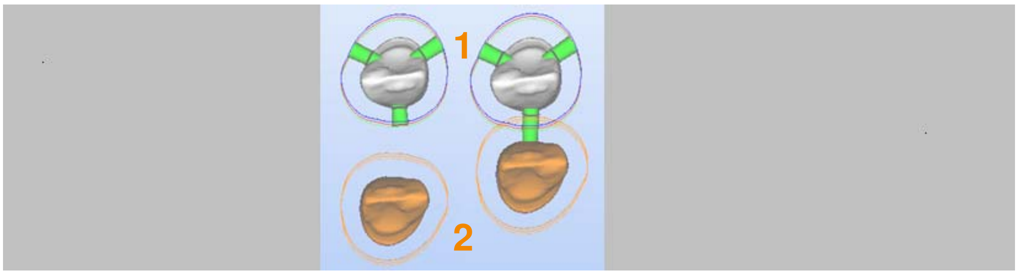


Figure 13-2

Connect connectors of the 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.

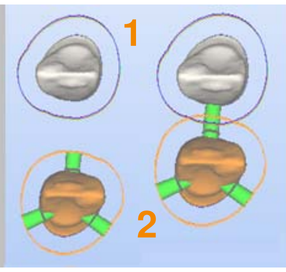


Figure 13-3

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

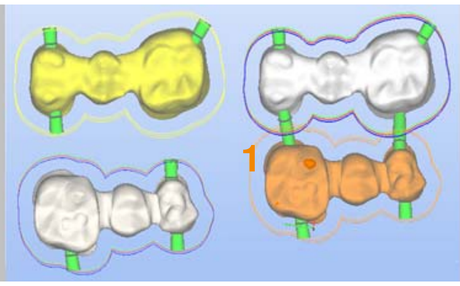


Figure 13-4

13.7 Creating sintering frames

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

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

2. Click on the part:
   The sintering frame is created.

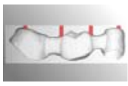
3. Connect sintering frame with the dental restoration using connectors.

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.

Detaching the inner area

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

14 Set 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 profile	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]

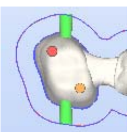
1. Select sintering pin.
2. Call up context menu or menu [Edit] > [Sintering pin].
3. Select the menu item [Edit sintering pins].

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

Edit	Activate settings for the 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 Edit 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 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 top plane for one or more parts.

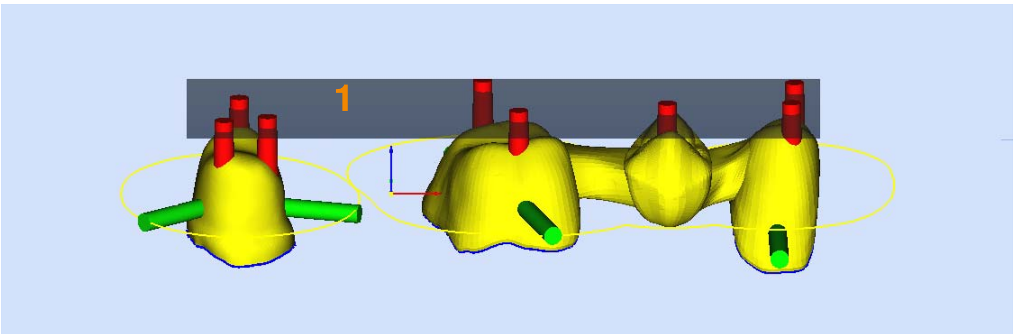


Figure 14-1

1. Select one or more parts.
2. Either call up the context menu or from the menu [Edit], 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 Calculate 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.

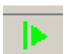
Click on the icon to start the calculation.

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 preparation 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 stepover is incorrect to a significant degree, a warning message is displayed prior to the start of calculation.

15.2 Merge calculations

There are part-specific toolpaths in individual calculations in hyperDENT® for fixtures with several blanks, e.g. for prefabricated blanks (prefabs).

hyperDENT® Calculation Merge allows you to merge the individual files containing the individual calculations into a joint file containing all calculations.

This joint file can then be converted in a postprocessor cycle in hyperVIEW® into the machine-specific NC file with optimized tool changing, --> "Create NC file, simulation (option)".

You can load this NC file on your machine and start joint machining of all blanks with optimized tool changing.

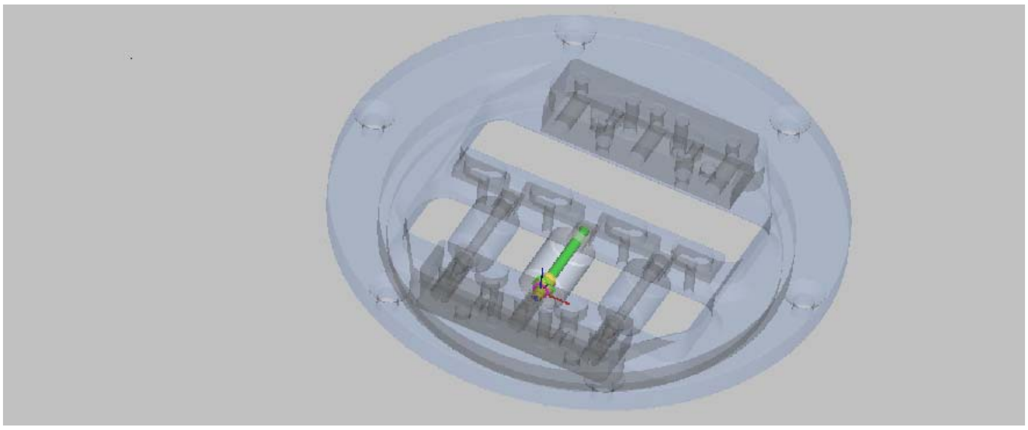


Figure 15-1 Holder for several blanks

Start hyperDENT® Calculation Merge

Either

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

Or

1. After calculation, call up the context menu and select the menu item [Part] > [Open in hyperDENT® Calculation Merge] (Öffnen in hyperDENT® Calculation Merge).

The selection window shows the administration for the calculations with the list of calculations, the parts and tools, the merged parts and tools, the blank name, and the preview for the fixture, as well as the function for updating the display and inserting new calculations.

Menu bar, menu items, icons

File	
Exit hyperDENT® Calculation Merge (hyperDENT® Calculation Merge beenden).	
Edit	
Update	
Load calculation (Berechnung laden)	
Show work directory (Berechnungsverzeichnis anzeigen)	
Show toolpaths	
Settings	
Machining	Call up submenu, --> “Settings”.
Tools	Call up submenu, --> “Settings”.

General	Call up submenu.	
Miscellaneous	Call up area.	
Filter calculation (Filter Berechnung)	Activate/deactivate filter for calculation time.	
Filter after X hours (Filter nach X Stunden)	Filter hours specification.	
Extras		
Refresh licensing		
Help	Call up Help and information.	

Display for parts and calculations

List containing the calculations and information on the part, administrative	data, blank type, fixture (holder), and information on the selected part and its calculation.
Checkmarks	Selection of the part for calculation merging.

- Caution!
  Only select parts for merging if they are suitable for the joint holder and do not overlap each other.
- Check the selection in the merge preview.

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

Display for merging

Merge preview (Vorschau Zusammenführung)
Preview of all parts selected in the list for merging (checkmarks).

- Caution!
  The selected parts (2, 3, 4) must be suitable for the joint holder (1) and must not overlap (5, 7).

Example — Permissible selection
Merging is possible

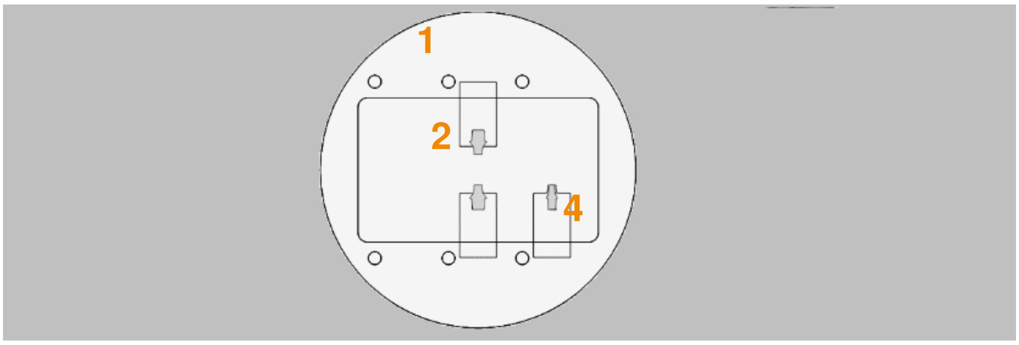


Figure 15-2 Merge preview

Example — Impermissible selection
Merging is not possible.
The parts are overlapping (6) or are calculated for different holders (5, 6, 7, 8).

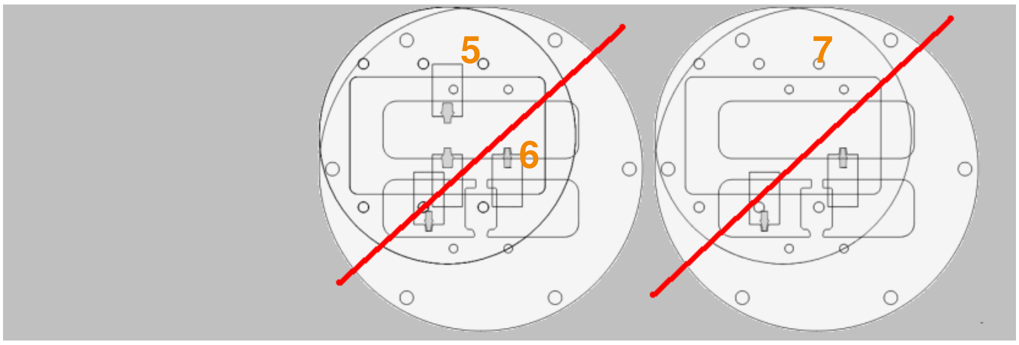


Figure 15-3 Merge preview, impermissible selection

Merged parts (Zusammengeführte Objekte)	
	List of merged parts.
Merged tools (Zusammengeführte Werkzeuge)	
	List of tools for the merged parts. These tools must be present on the machine for machining to occur.
Blank name	Name of the blank.
Start merge (Zusammenführung starten)	
Output file (Ausgabedatei)	
	Output directory with the new file for the merged parts.

16 Create 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 tool paths].

A detailed description of hyperVIEW® can be found via the menu item hyperVIEW® Help (F1), see: Select hyperVIEW®, important operations, post processor cycle.

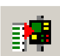
The key steps are detailed in the following description.

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. Write 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.

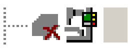
16.2 Simulation (optional)

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

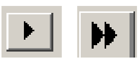
The simulation is run using the toolpaths that have been calculated in hyperDENT®.

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.

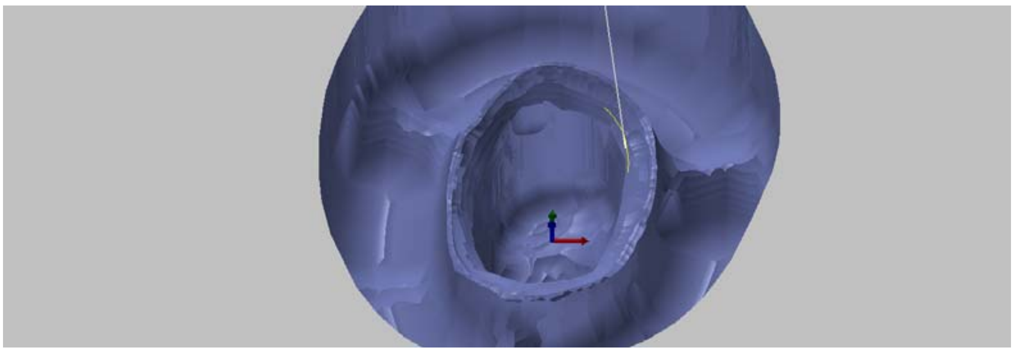


Figure 16-1

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.

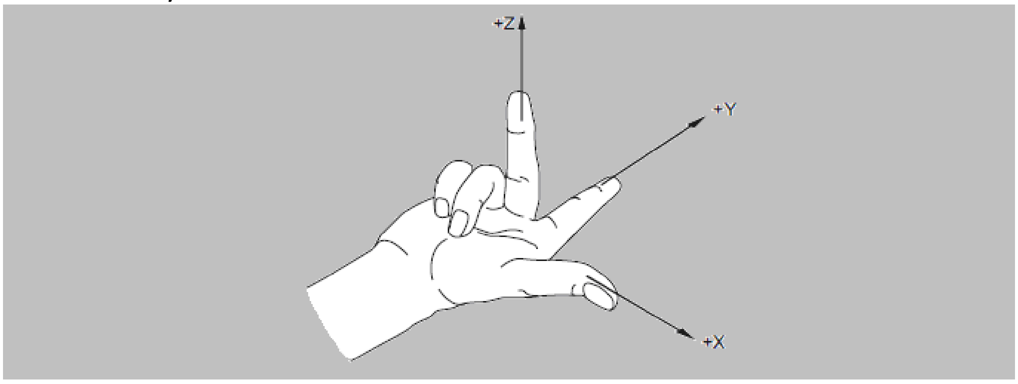


Figure 17-1

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.

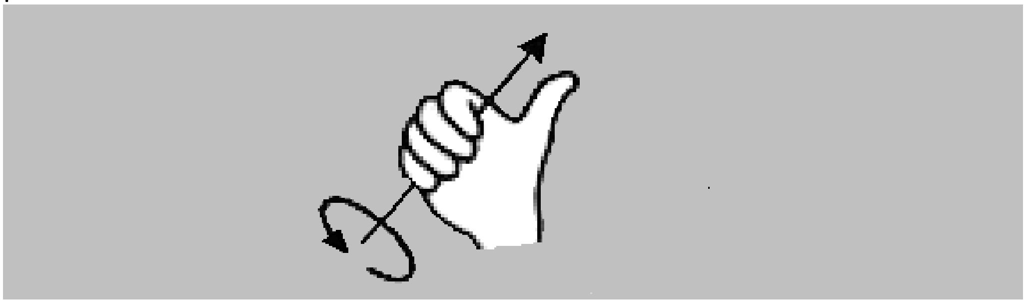


Figure 17-2

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

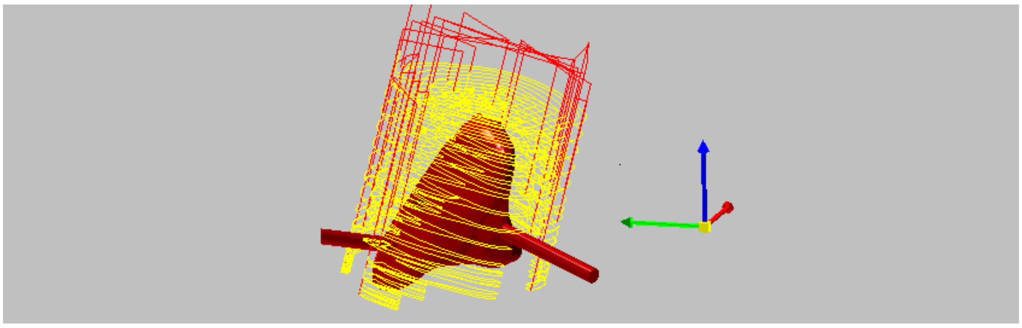


Figure 17-3

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

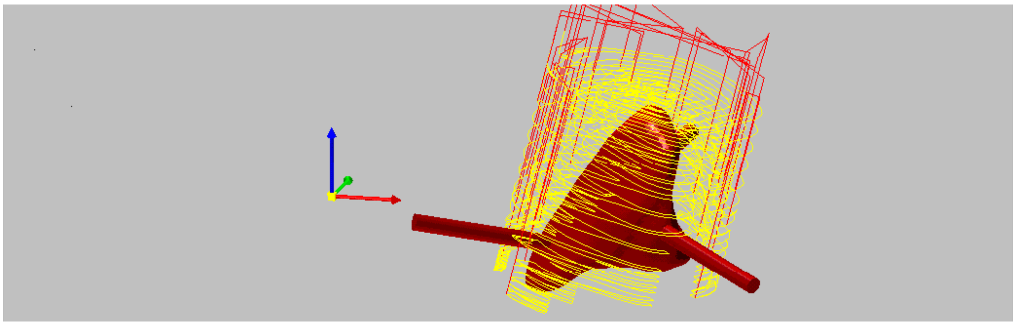


Figure 17-4

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 machinespecific settings.
Postprocessing strategy	Standard Postprocessor type: Standard or user-specific (optional).
Postprocessing mode	Settings for calling 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® work 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: Stock name, model name, timestamp (date, time).
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.

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 to 2
	Time factor for the milling time with feedrate.	
1	Milling time for G1 calculated according to defined feedrate.	
>1 to 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
Time for tool change	Time for the tool change.	s
Constant additional time	Additional time that is added to the calculated machining time, e.g. for tool change.	s
Default machine	Use selected machine as default.	

Import machine-specific settings from an XML file

You can also import the machine-specific settings from an XML file.

- The file must be in the same postprocessor directory as the .oma file, e.g.: "...CAM-Utilities\hyperVIEW\12.0.hyperDENT\oma".
- If the settings were imported from the .xml file and then changed manually in menu [Settings] > [Machining] > [Machines], the XML cannot be reimported.
- Further changes are only possible in hyperDENT® via menu [Settings] > [Machining] > [Machines].
- It is only possible to reimport the .xml file once the existing machine in hyperVIEW® has been deleted and a new one has been created following a hyperDENT® restart.

17.1.3 Fixture

Fixture, select 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 settings

Calculation in NC coordinates (Berechnung in NC-Koordinaten)	
Yes	Perform calculation with identical NC coordinate system. Required for processing NC data with hyperDENT® Calculation Merge.
No	Use existing coordinate system. Merging of NC data with hyperDENT® Calculation Merge is not possible.
Rotation axis	Preferred rotation axis for the display on the screen.
Placing parameters (Platzierungs Parameter)	
	Settings for automatic placing of parts in the blank (autonesting). Further details --> “Load part” > “Place part automatically – autonesting”.
Automatic / Manual	
Yes	Load part and place automatically.
No	Load part. The part must be placed manually.
Professional (Profi)	No
Placing direction (Platzierungs Richtung)	
	Arrangement of the parts in line with placing sequence.
Inside out	
Outside in	
Left right (von links nach rechts)	
Right left (von rechts nach links)	
Bottom up	
Top down	
Center left and right (von der Mitte nach links und rechts)	
Center up and down (von der Mitte nach oben und unten)	

Start position 1st part (Startposition 1. Objekt)	
	Display of the position of the first part. The position depends on the selected placing direction.
Offset angle (Offset Winkel)	1 to 90°
	Angle by which the part is rotated for placing in order to achieve the best possible arrangement in the blank:
Small angle	High placing accuracy, long calculation time.
Large angle	Short calculation time, low placing accuracy.
Starting from the original alignment, the part is rotated by the offset angle and each new alignment is checked to ensure optimum placing within the blank.	
Performance	Selection of the placing mode:
Very fast – less precise ...	
Less fast – very precise	
Overlap (Überlappung)	Information on milling boundary.
Yes	Milling boundaries overlap, material saving.
No	No overlap of milling boundaries.
Add connectors (Konnektoren hinzufügen)	
Yes	Add connectors, place parts with connectors.
No	Placing without connectors.

17.2 Blanks

Create and manage blank types, blank administration settings.

17.2.1 Blank types

Here, 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], [Load project].

You can show/hide the columns in the table and the options for filtering the blanks.

1. Call up the blank administration settings:
   In the menu [Settings], call up menu item [Blanks] > [Blank administration…].
   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 filter function for blank type Enable/disable filter options.
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 (preset).
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	Show/hide columns in the table for blanks already used.
Filter blank type selection	Enable/disable filter function for blank type Enable/disable filter options.

17.3 Building elements

Define defaults for 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”.

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	Selection filter for the tools:
Tool type	

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.

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.
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.
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 purpose, 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).

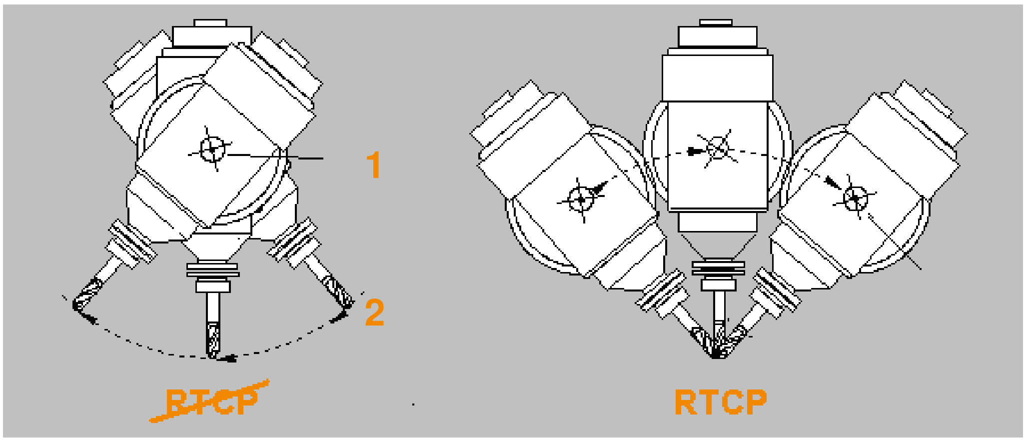


Figure 17-5

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 holder] 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	Description of the tool holder.

Data	Details about the shape of the tool holder.

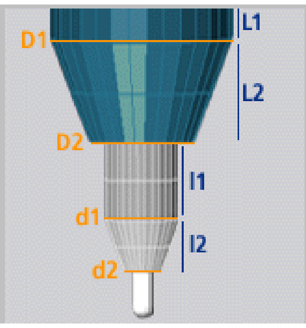


Figure 17-6

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 option: Create and edit template, further details --> "Milling strategy" > "Edit template".

17.6 Parts

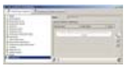
Create and manage part types and part information.

The parts, the dental restorations, are divided into part types in accordance with their specific features, which are in turn assigned to the relevant machining templates.

17.6.1 Create and edit user-defined part type

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

The new part type has the specific properties of the original part type, but can also have, for example, specially adapted machining templates (template generator option).



1. Click tab [Part type setting] (Einstellung Objekttyp):
   The selection window shows the available part types, the associated user-defined part types, and the machining functions: Edit, Add, Delete.

List (Liste)	List of the original part types available.
Name	Name of the selected part type.
User-defined part types (Benutzer definierte Objekttypen)	List of the user-defined part types created from the selected part type.
Internal name (Interner Name)	
Local name (Lokaler Name)	
Icon (Ikone)	Icon for the user-defined part type.

Add user-defined part type

2. Select part type and call up the [Edit] function:
   Click on the part type and click on [Edit].
3. Call up the [Add] function:
   The input window is displayed.
4. Enter the name for the part type.
5. Select a unique icon if required.
   Select icon from file system; otherwise, the icon of the original part type will be used.
6. Confirm input or cancel process.

Edit user-defined part type

7. Double-click on the name or icon of the user-defined part type and either change the name or select a new icon.
8. Confirm input or cancel process.

Delete user-defined part type

9. Select user-defined part type and delete:
   Click on the part type and click on [Delete].
10. Acknowledge confirmation prompt.
11. Confirm input or cancel 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.

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 (identical internal ID).

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

Yes	If identical internal IDs are present, replace the existing parts with the imported parts.
No	Keep existing parts.

4. If necessary, select [Copy existing, modified objects] (Bestehende, geänderte Objekte kopieren):

Yes	If identical internal IDs are present, create the imported parts as copies with new internal IDs. If the part names are identical: Add an index to the name of the imported part, 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 parts and the changes to the database.

The imported parts receive a new internal ID during the import process. This ID differs from the previous ID. As such, hyperDENT® is unable to detect if a previously imported part is imported again.

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 the general settings:
   Select the menu [Settings] and call up the menu item [General].
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 quick info for parts (Zeige Quickinfo für Objekte)	
	Show 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 Load wizard

Use load wizard	
Yes/No	Use wizard.
Adjust direction	
Identify parts	
Tilt part	

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 management directory is stored.
For example, once the loaded files and management 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 a management 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 to 3

Project manager directory	
	Path name 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 tracking to determine which part has been placed and machined in which blank and project.
The data needed for the administration is saved on the hard drive in one or more directories.
The directories are created relative to where the management directory is stored.
For example, once the loaded files and management 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 a management 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 to 3

Part manager directory

Path name for the directory with the data for part management.
Initial directory to which the other directories of the part management relate.

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

Activate list of new parts (Liste neuer Objekte aktivieren)

Activate the list of new, as-yet unmachined parts for direct CAD connection (CAD Connect).

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

17.8.6 Calculation

Settings for the calculation of the project data.

Work 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 operation: 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.
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 selection and is performed for all selected parts.
Toolchange optimization	Yes Process all parts that have been calculated together, for a quicker and more economical milling process with optimized tool change, i.e. first of all process all milling paths across each part using tool “A”, then tool “B”, etc.
Preserve temporary data	No

View cycle parameters		No
Check toolpath within milling boundary (Werkzeugweg innerhalb der Fräsbereichsbegrenzung prüfen)		Yes
Yes	The toolpaths in the feed must be within the milling boundary. Movements in rapid traverse mode are not checked.	
No	No check. The toolpaths may go over the milling boundary. When placing parts in tight conditions, this may damage neighboring parts.	
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 directory
c:/tmp	Temporary directory

Postprocessing strategy	Standard Postprocessor type: Standard or user-specific (optional).
Postprocessing mode	Settings for calling 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® work directory	NC files are written in the work 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 name of the blank.
[Modelname]	Model name parameter for the 1st part.
[NCIDX:1]	Counter parameter with start value (1) for sequential numbering of the NC files, e.g.: “Crown_1.nc”, “Crown_2.nc”, “Crown_3.nc”.
[TIMESTAMP]	Parameter start time of the calculation (date, time).

Parameters [Modelname] and [NcIDX] enable hyperDENT® to be integrated into an automated process:

- The output of NC files is tool-specific.
- The name of the NC file is identical to the part name; only one part is calculated.
- The individual NC files are numbered in sequence, beginning with “_1”.

Create info file	Create XML file with essential information for the NC file, such as blank and calculated parts. This is saved 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	G0 toolpath
<G1Length>	m	G1 toolpath
<G0Time>	h:min	G0 time
<G1Time>	h:min	G1 time
<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.

17.8.8 Navigation

Zoom	Zoom behavior when you move the mouse wheel.
Rotate	Key combination to rotate the blank.
Move	Key combination to move the blank.

17.8.9 Display

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

Holder color	
Fixture transparency	0 to 0.9
Part color	
Part color (new)	
Preparation line color	

Color of other preparation line	
Color abutment base lines	
Color emergence profile	
Outline color	
Outline top color	
Outline bottom color	
Connector color	
Sintering pin	
Sintering frames	
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		
Selected part color		
Modify selected part color		
Use background image	Yes/No	
Background image transparency		0 to 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 option, you can create and edit templates, --> “Edit template”.

The template generator 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 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 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
5. Special

18.1.1 Call up the 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.

Job list	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 Call 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, use, delete, copy, export, password protect, print.

Show for	Selection filter for the templates: Material, part type, permitted machines.
Name	Template name. 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 (Lizenz fehlt)	A license is required for this template. The license is not present.
 License present (Lizenz vorhanden)	A license is required and present for this template.

2. Select template.
   Click on the name of the template in the list.
3. Select the function [Edit]:
   Click on the icon.
4. If required, enter password.
   Enter the password for password-protected templates. If the template is license protected, the license must be present on the 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.

Job list	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.

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 present on the computer.

Activate, delete password protection

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.
   If no entry is made, password protect for this template is canceled.
4. Save password by clicking on [OK].

18.1.4 Editing the 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.

Enable/disable job

The calculation only processes jobs that are enabled. Deactivated jobs are not processed.

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

On The job is activated and will be processed.

Off The job is deactivated and will not be processed. The job remains in the list with its parameters.

Add 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.

Move job

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

Delete 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.

Copy 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 jobs in the job list.

Exchange job

1. Select the job, call up the context menu, and select the menu item [Exchange job template]:
   The menu is displayed for the new jobs.
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 jobs, the job name remains the same.
3. Adjust job name:
   Adapt the displayed job name to the actual job
   (--> hyperDENT® job name) to avoid mix-ups and incorrect machining.

Change job name

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

To avoid mix-ups and incorrect machining, make sure that the job name relates to the actual job.

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 the 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.
   The changes apply to all selected jobs in the case of a multiple selection.
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 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.

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

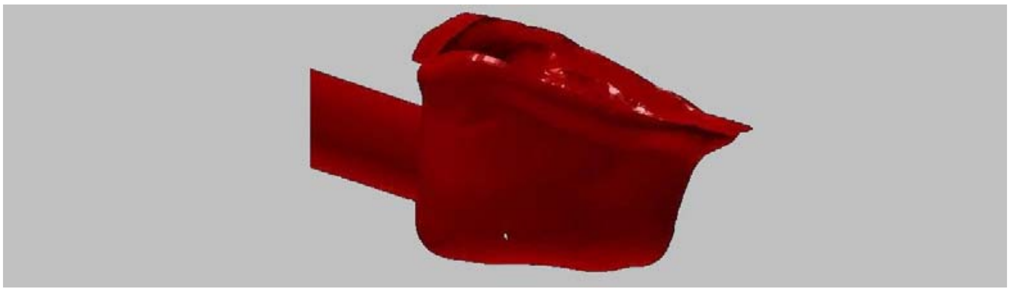


Figure 18-1

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).

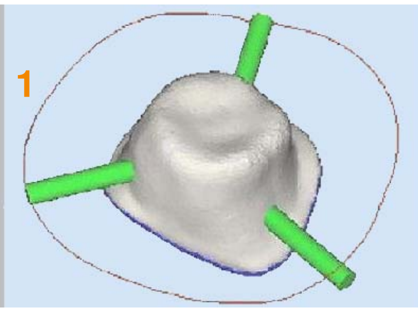


Figure 18-2

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 the section-wise machining. Connectors are correctly aligned in accordance with the specific holder.

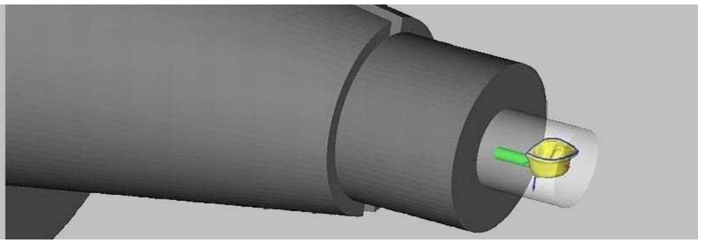


Figure 18-3

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 +0.5 mm If the preparation line is made thicker, it may also be necessary to extend the milling boundary.

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.	

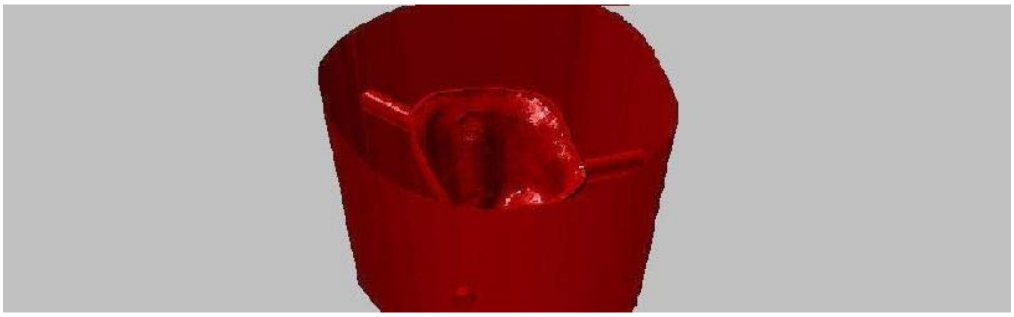


Figure 18-4

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 area 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.	

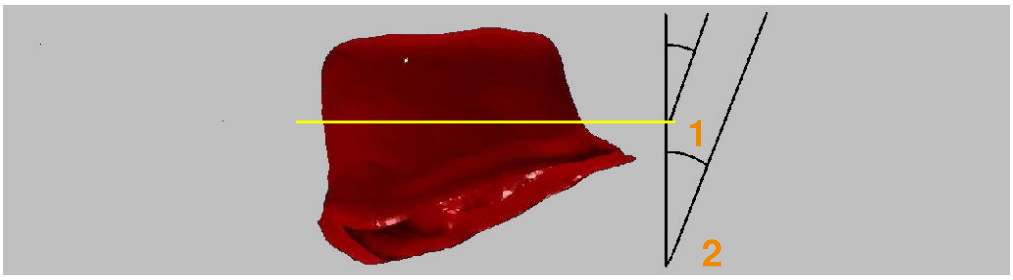


Figure 18-5

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 stepover direction is tilted (height optimization) orthogonally to the blank.	
No	Tilted machining, all milling planes are tilted toward the blank surface.	

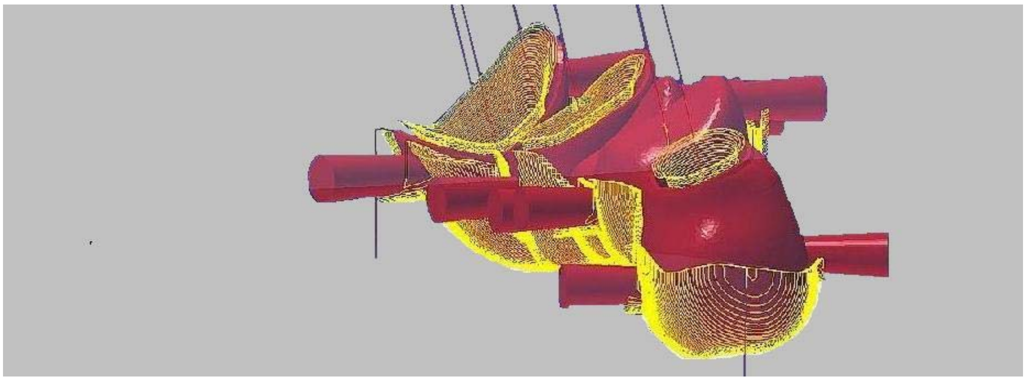


Figure 18-6

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).

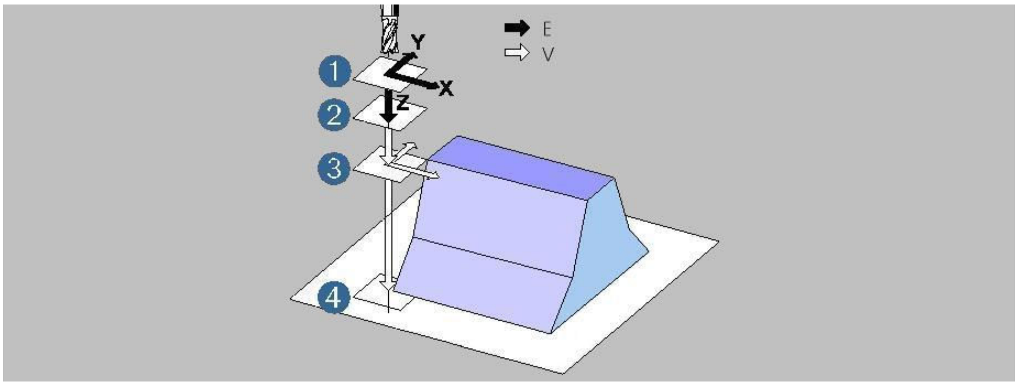


Figure 18-7

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 machining jobs		No
Yes/No	Enable/disable collision check for the fixture in 3X machining jobs. The calculation of the toolpaths with collision check results in longer calculation times.	
Check fixture for collisions in 5X machining jobs		No
Yes/No	Enable/disable collision check for the fixture in 5X machining jobs. The calculation of the toolpaths with collision check results in longer calculation times.	
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. i.e. 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.

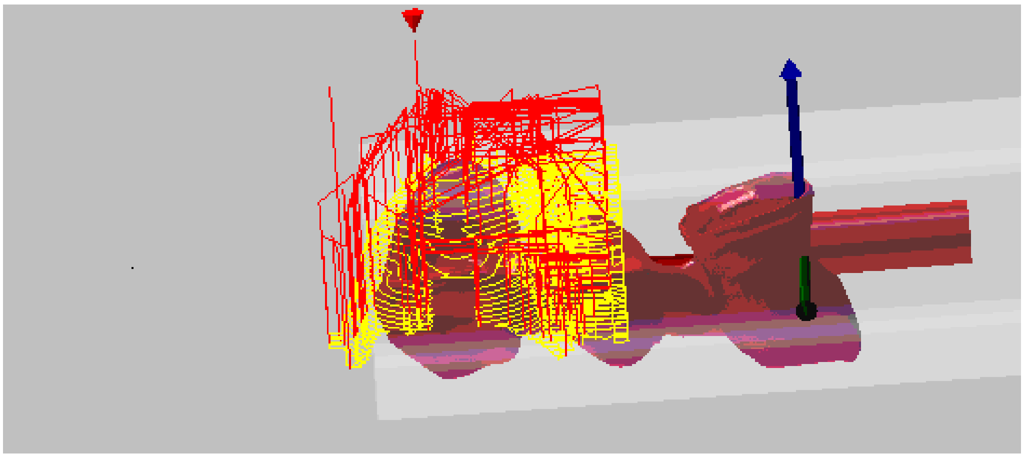


Figure 18-8

Min. segment distance to element border	mm	
	Minimum distance of the machining section to the element borders, e.g. boundary lines of preparation 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 preparation 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 preparation line and the inner areas of the copings for 3X machining.

Plane 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.

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 sintering 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 outline.
- 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 on 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, pre-drilling.

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 Pre-drilling 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.

Job template	Original job name predefined by hyperDENT®.
Jobname	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.

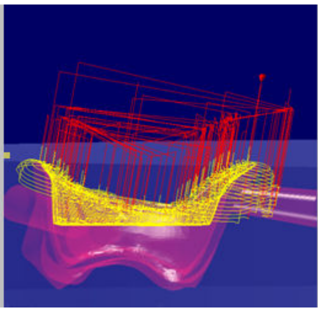


Figure 18-9

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 stepover direction.

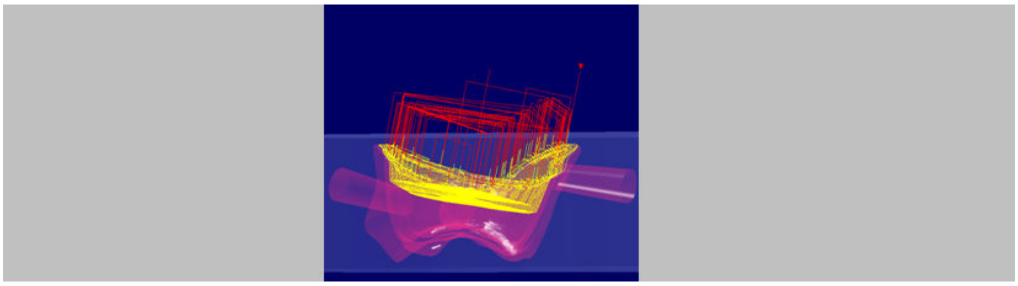


Figure 18-10

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 tool paths 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 of 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 window with a list of the 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 stepover 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 means a higher consumption of material and longer machining times. Older templates are automatically set to the tool shank for safety reasons.

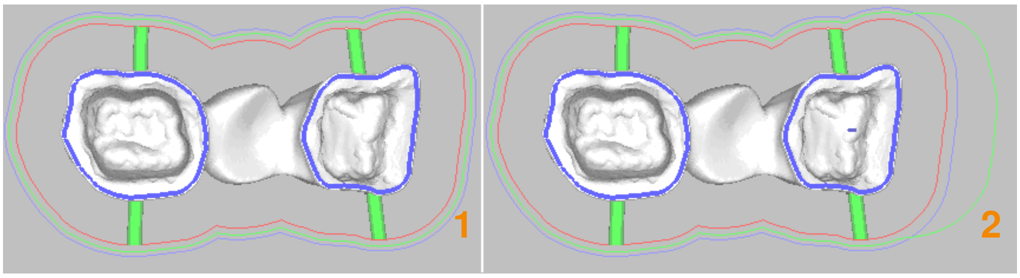


Figure 18-11

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

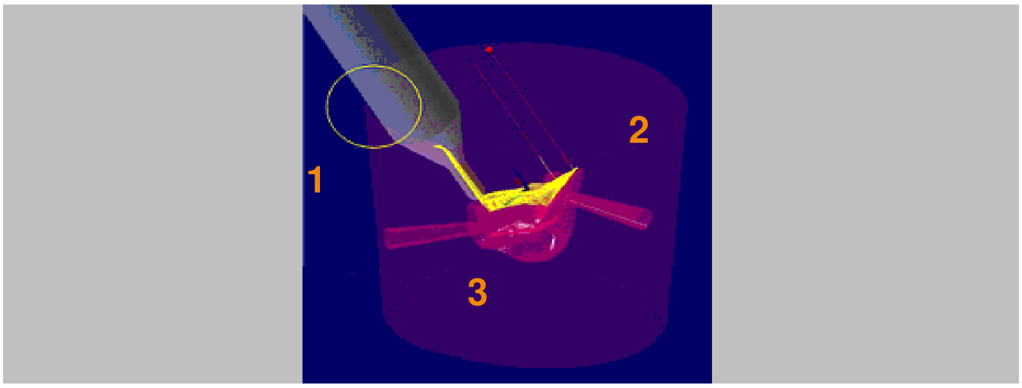


Figure 18-12

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

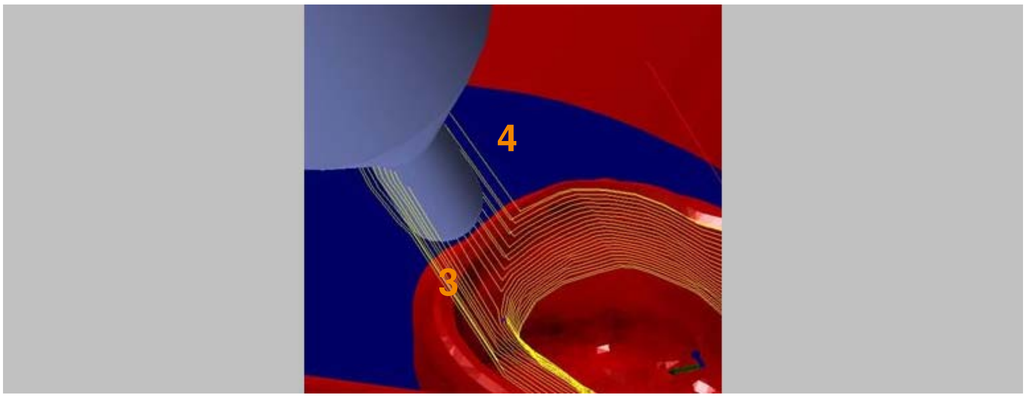


Figure 18-13

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

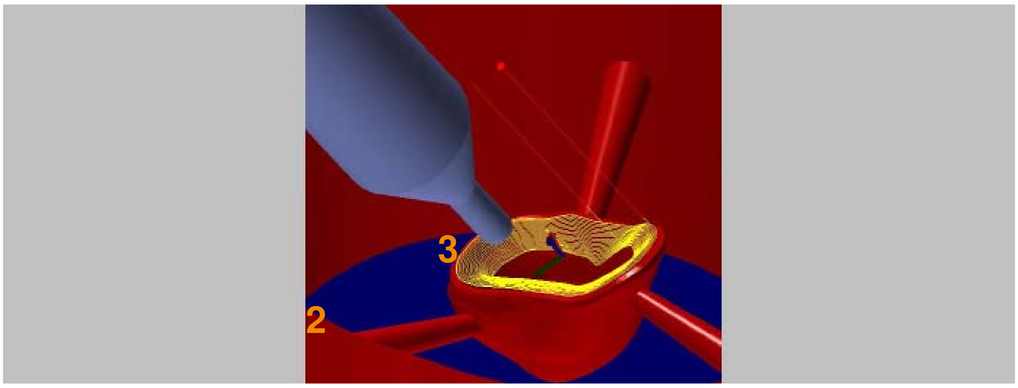


Figure 18-14

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 spindle 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 a 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.

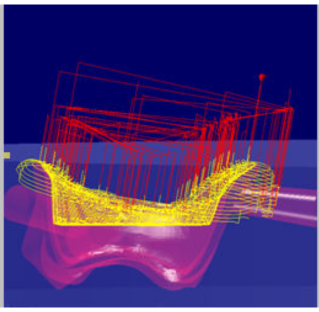


Figure 18-15

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 stepover direction.

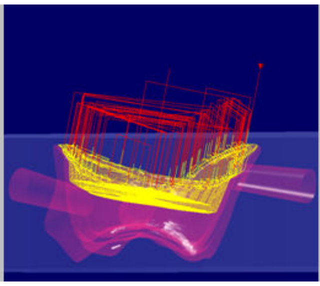


Figure 18-16

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 stepover 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:
Single plane	Step down > (surface – depth)
Several planes	Step down < (surface – depth)

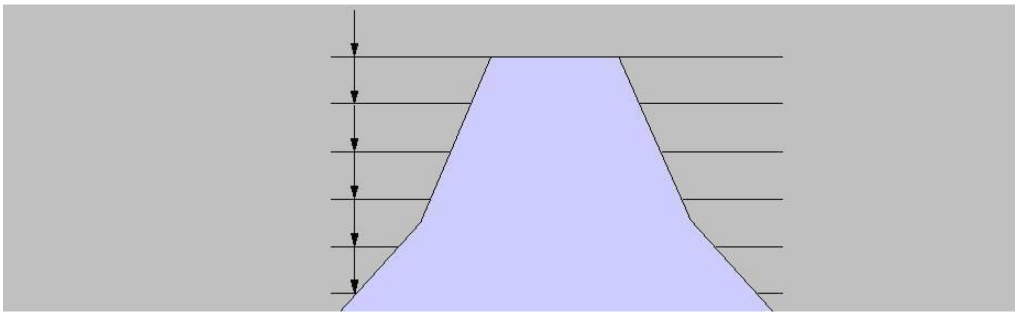


Figure 18-17

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.

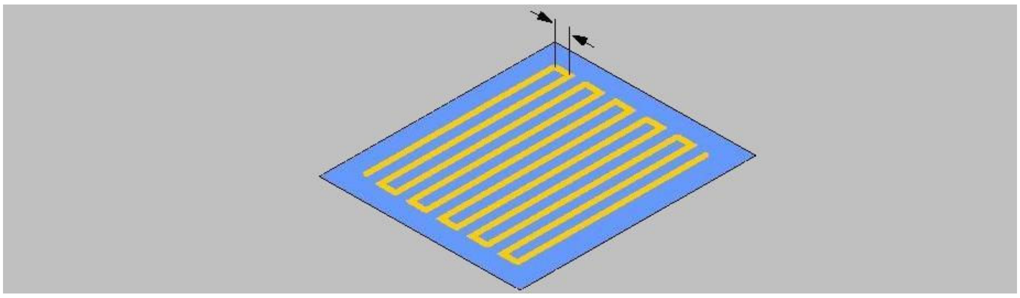


Figure 18-18

Fillet corners Yes/No	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/helix	Ramp Method by which the tool plunges into the material in a Z-direction.
Ramps/helix angle	45 Angle in degrees with which the tool plunges into the material in a Z-direction: "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: 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.

\n	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.

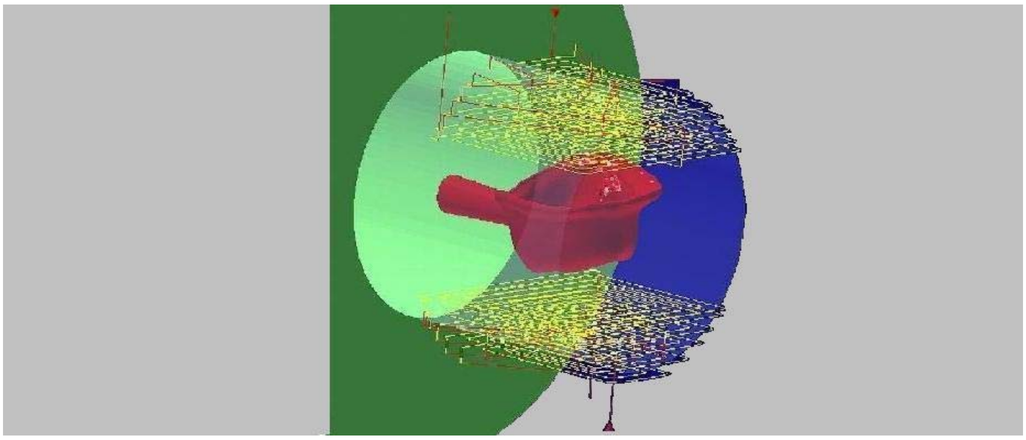


Figure 18-19

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.

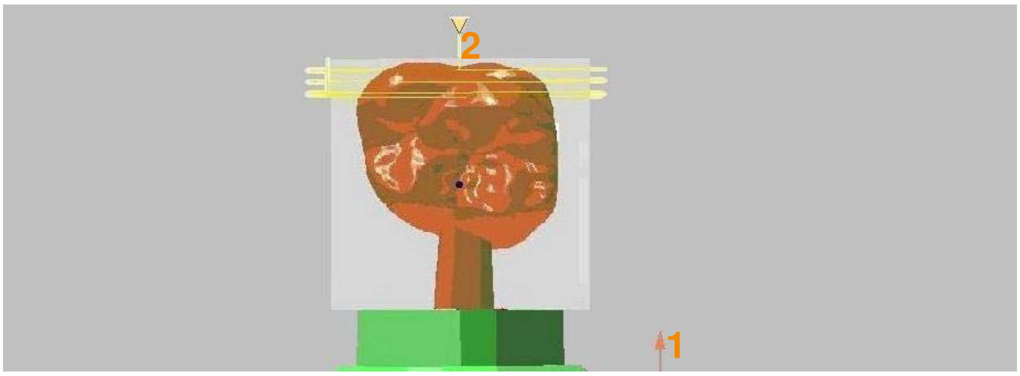


Figure 18-20

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 “rear” 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

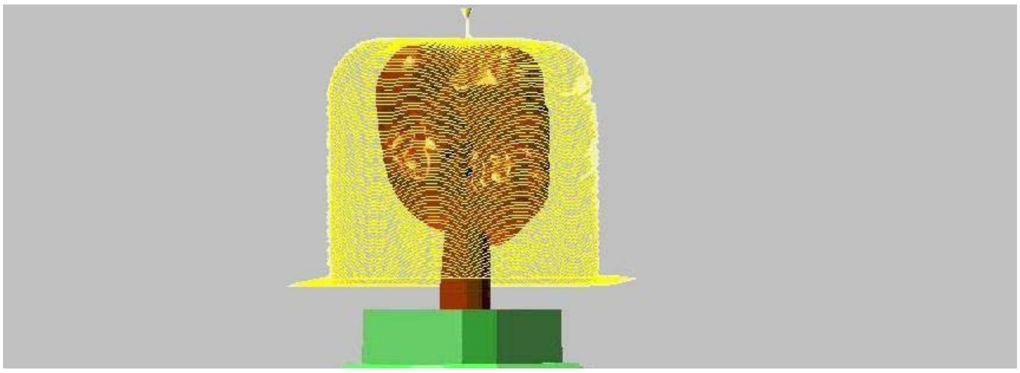


Figure 18-22

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 center 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 blank 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.

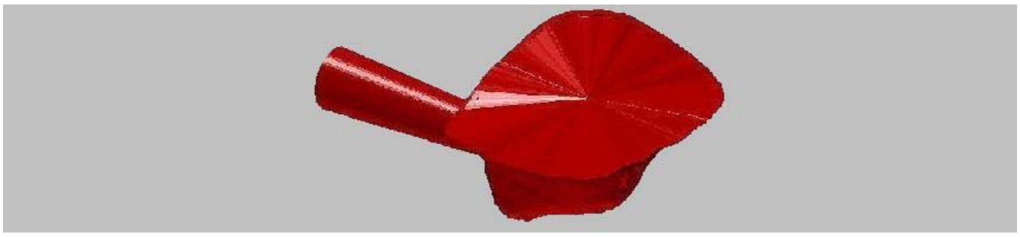


Figure 18-23

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"
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

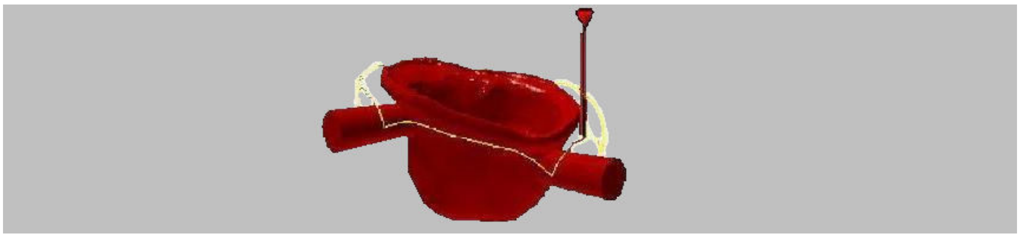


Figure 18-24

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.

- Caution!
  Tool's full cut possible. Therefore the setting [Optimized in] is unsuitable for grinding strategies.

18.8.3 Finishing/fine finishing the preparation line

Targeted pre/fine finishing in the preparation 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 preparation line, only the outer area is machined: [Inner offset] from preparation line is smaller than or equal to the value for the stepover. Machining starts at the preparation line.

- If this area is divided into two jobs – from the preparation line in and from the preparation 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.
Inner offset	Area (1) that is machined from the preparation line to inside the coping.

Outer offset

Area (2) that is machined from the preparation line to outside the coping.

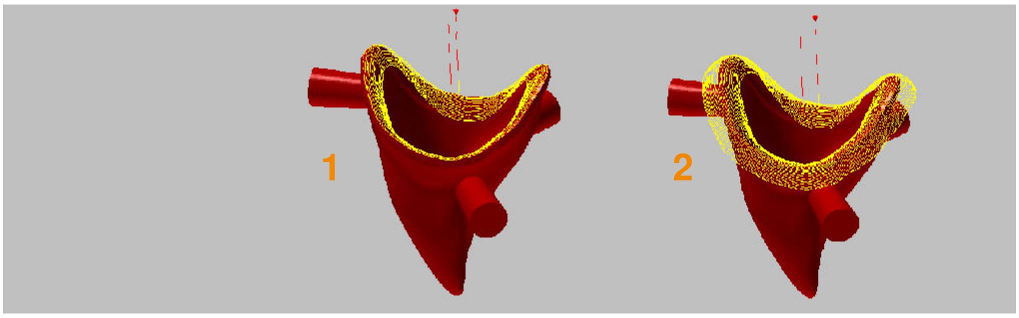


Figure 18-25

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 preparation line.

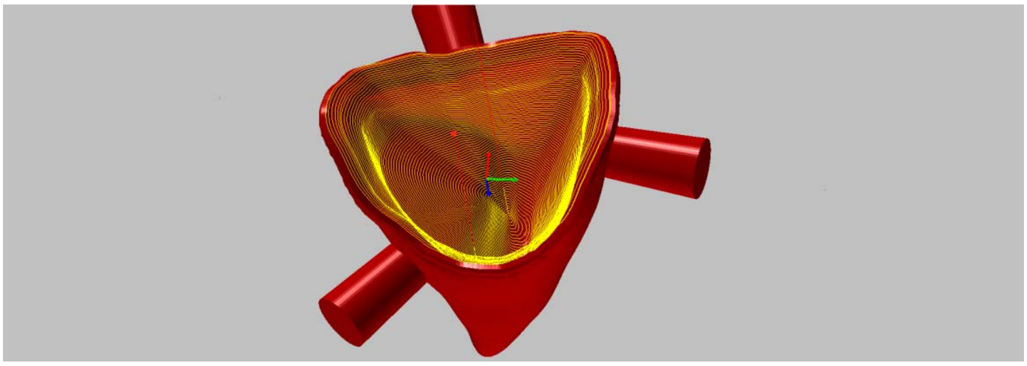


Figure 18-26

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 preparation 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 preparation line.
Values < 0	The offset is generated inside the preparation line.

- Take care with undercut areas that start directly at the preparation 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 , --> "Align parts" > "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 tool path 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 Automatic Center of coping Deepest point in coping	according to drilling job Position in the coping for the plunging of the tool, coordinated with the previous drilling job [Drilling inside copings].
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 preparation line for bridges with inlay/onlay parts or Maryland bridges

If the preparation 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.

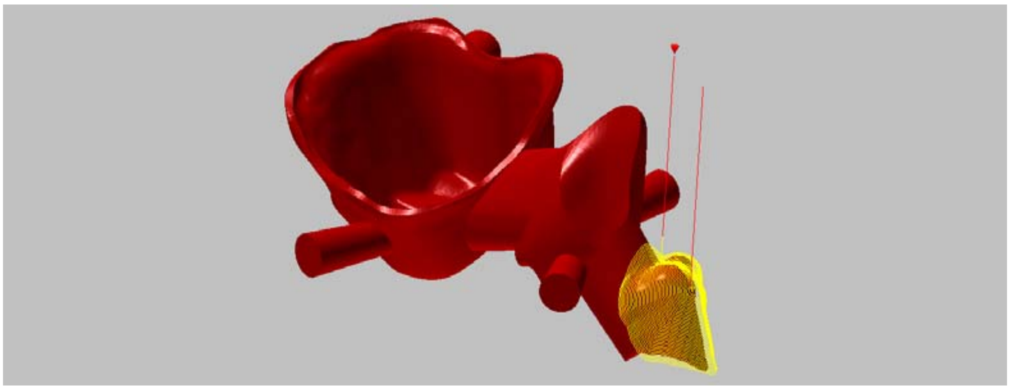


Figure 18-27

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, otherwise unmachined areas may be left. No other function of this, no protection of the preparation 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 preparation 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”.

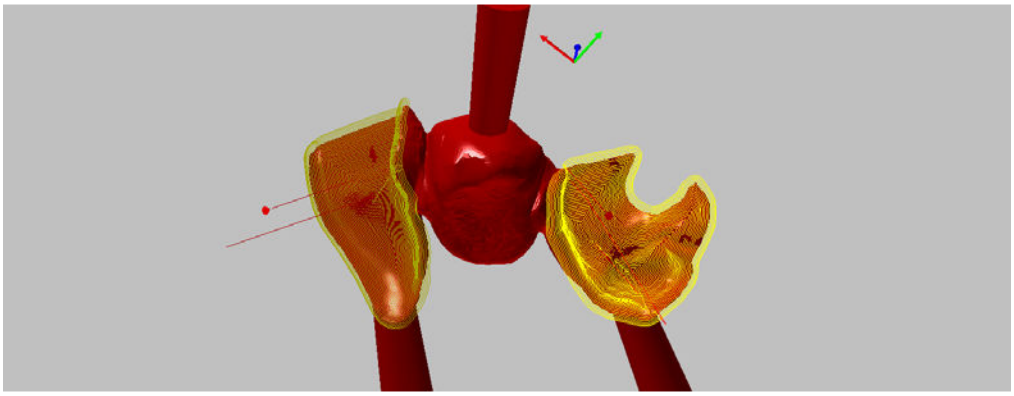


Figure 18-28

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).

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 preparation line for zirconium oxide. This prevents the cycle from identifying and machining space left at the preparation line.
Values < 0	The offset is generated inside the preparation line. If possible, keep the value as low as possible, up to approx. – 0.5. Use the offset in combination with an activated “Clip boundary”.
Z-level optimization	
Yes	Machining in narrow areas (pockets) is also done with depth step. Suitable for clearing out deep pockets that are deeper than the current tool diameter.

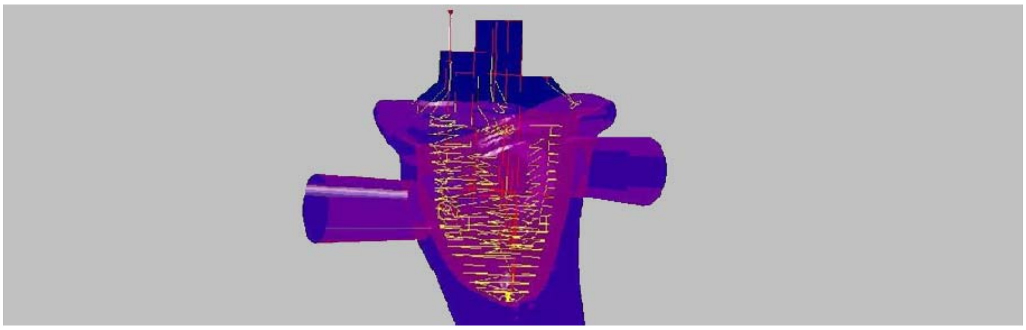


Figure 18-29

No Z-level optimization is disabled.

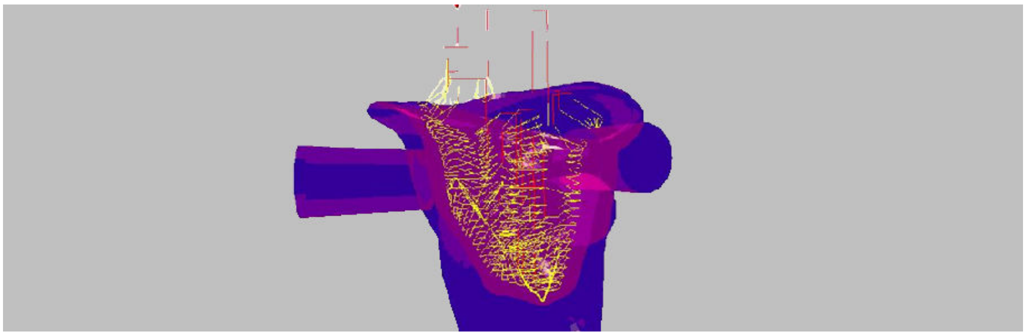


Figure 18-30

Clip boundary:

Yes Also set the boundary of the preparation line as the stop area so as to better protect the preparation line from unintentional machining.
Where possible, set to “Yes” if a “Boundary offset” is entered.

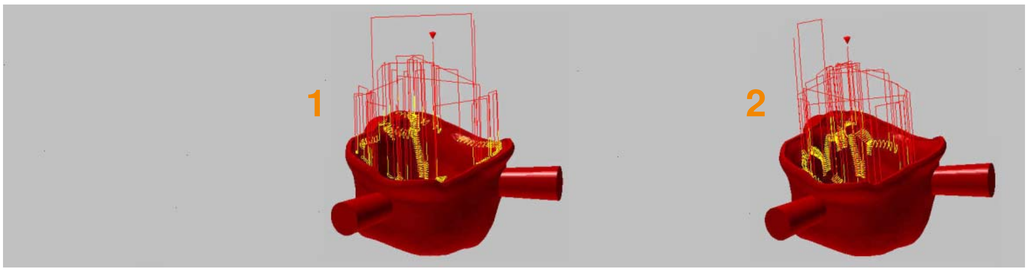


Figure 18-31

1. Tool path without set parameter “Clip boundary”
2. Tool path with set parameter “Clip boundary”

Strategy flat areas		
Parallel	The tool paths run in flat areas (1) parallel to the surface of the workpiece.	
Normal	The tool paths 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.	

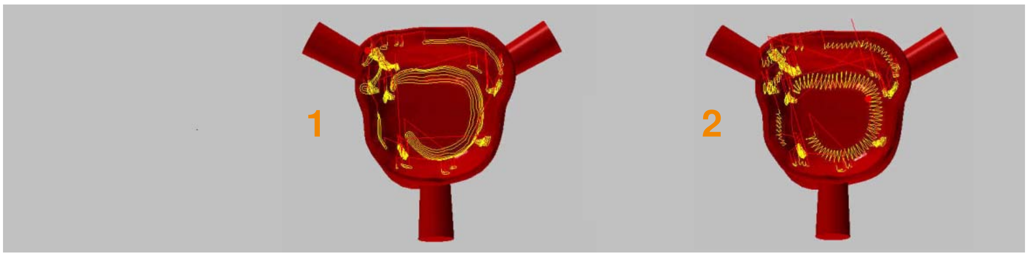


Figure 18-32

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.

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 range		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

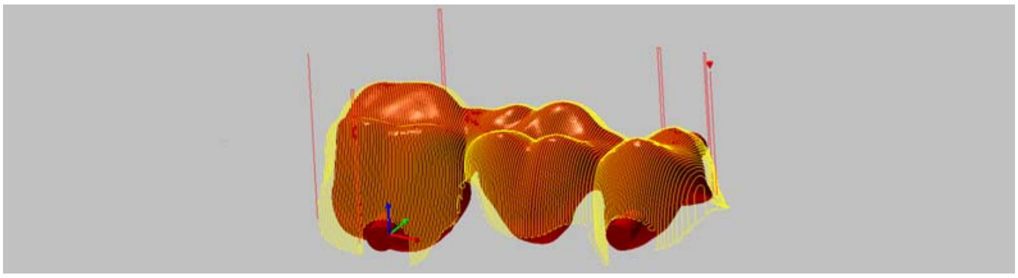


Figure 18-33

3D profile finishing strategy ruled (copings)
5X profile finishing strategy ruled (copings)

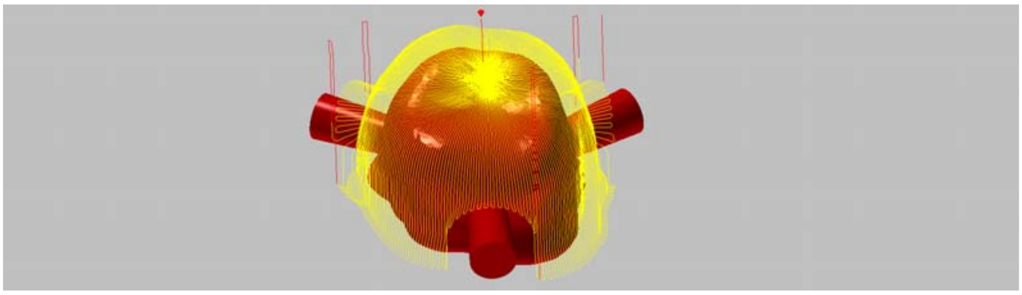


Figure 18-34

3D profile finishing X-direction (copings)
5X profile finishing X-direction (copings)

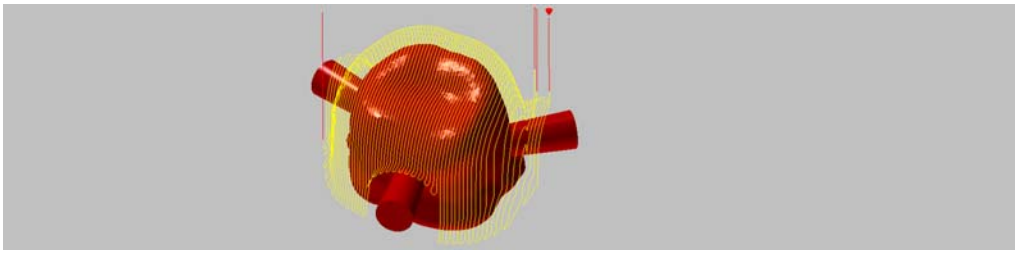


Figure 18-35

3D profile finishing XY-direction (copings)
5X profile finishing XY-direction (copings)

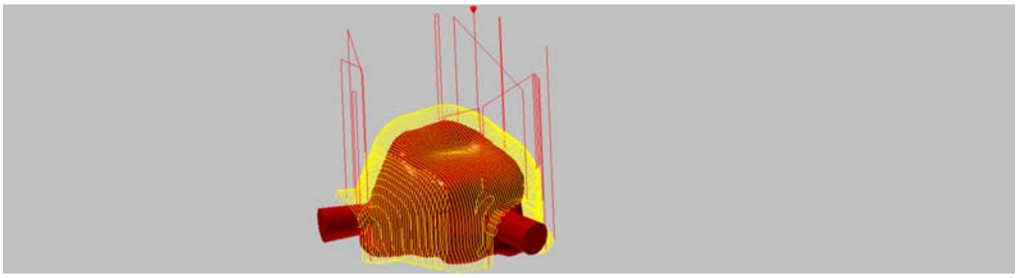


Figure 18-36

3D Z-level finishing

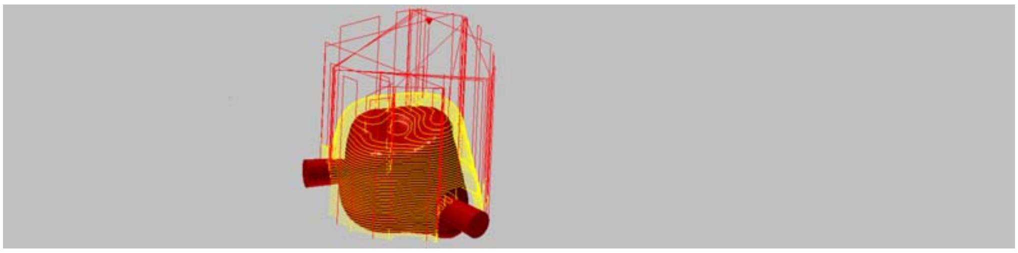


Figure 18-37

3D complete finishing

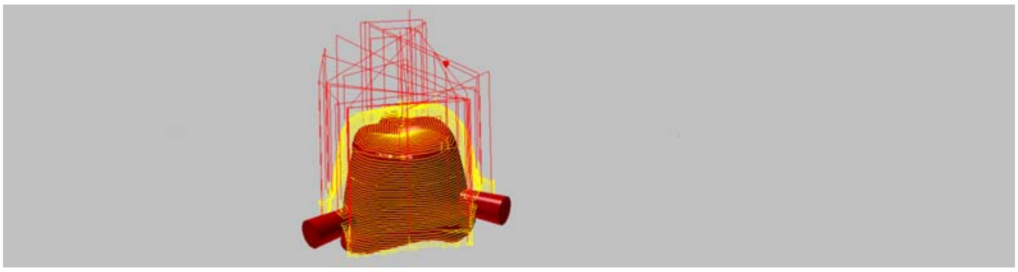


Figure 18-38

3D profile finishing pocket mode (bridges cavity side)
5X profile finishing pocket mode (bridges cavity side)

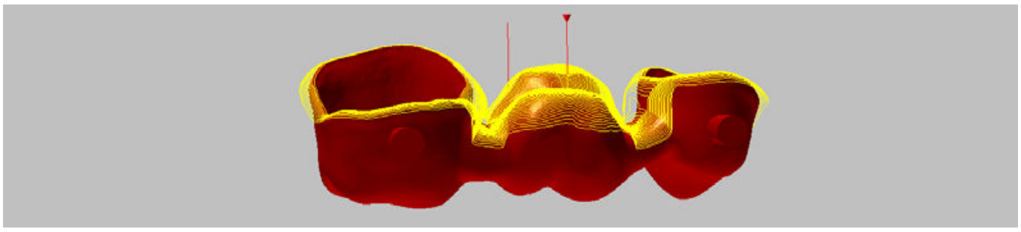


Figure 18-39

3D equidistant finishing

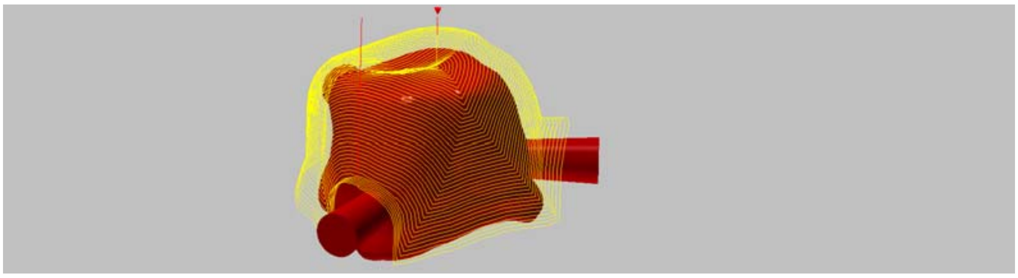


Figure 18-40

3D equidistant finishing flow (without rework)

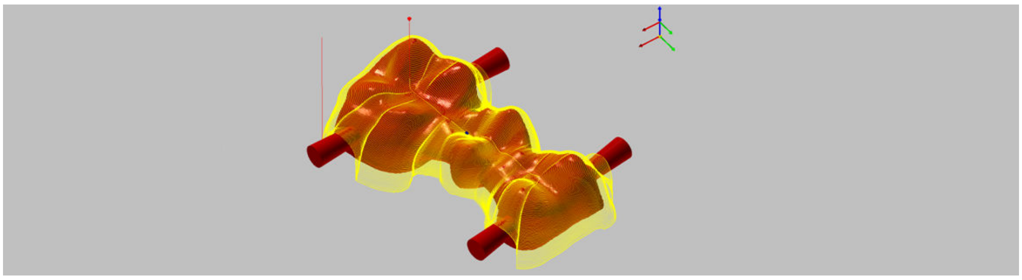


Figure 18-41

5X boss finishing (occlusal side)

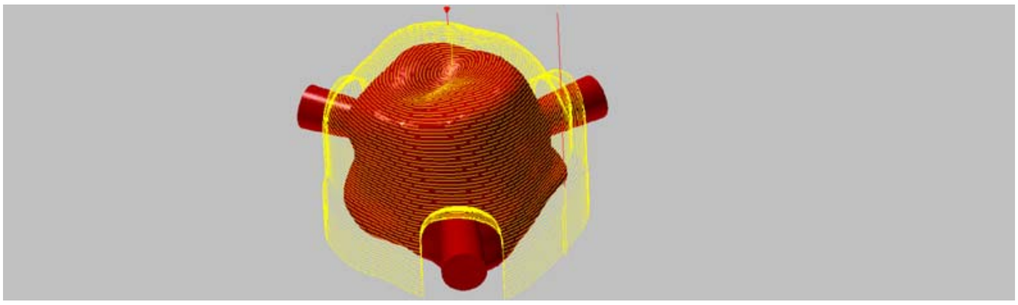


Figure 18-42

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 tool paths and increases safety in the preparation line/milling boundary area.

Add rework step	Yes
Yes	Optimum surfaces thanks to equal intervals between tracks throughout the entire milling area.

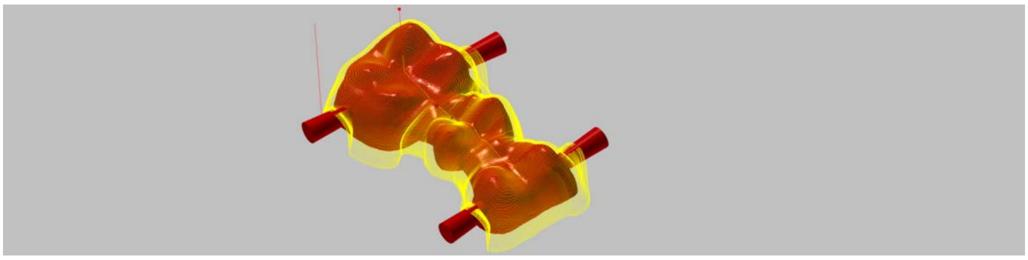


Figure 18-43

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.

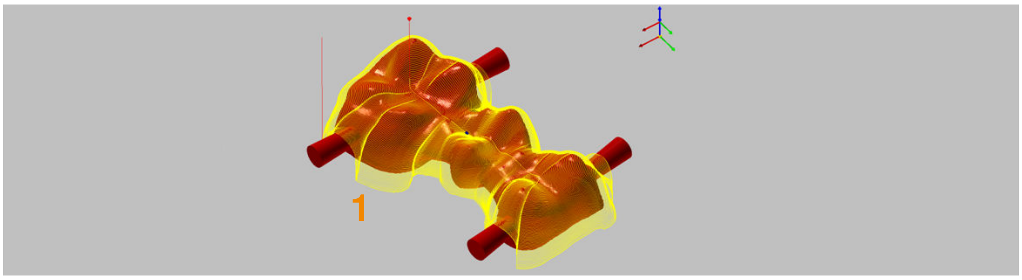


Figure 18-44

For 3D equidistant finishing

The shortest tool paths are created for equidistant finishing.

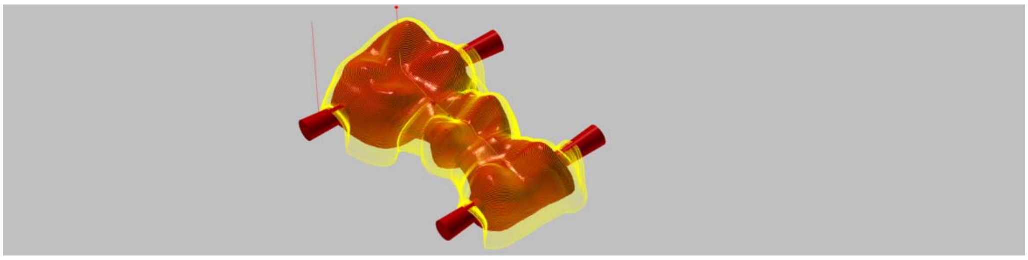


Figure 18-45

Stepover direction Inside out Outside in		
Inner Offset		mm
	Default value for machining from the outside in.	

For all 5X cycles without undercut machining option

Desired tilt angle		Max. approx. 3°
Tilt angle range		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 range		≥ 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 tool path 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.
X	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

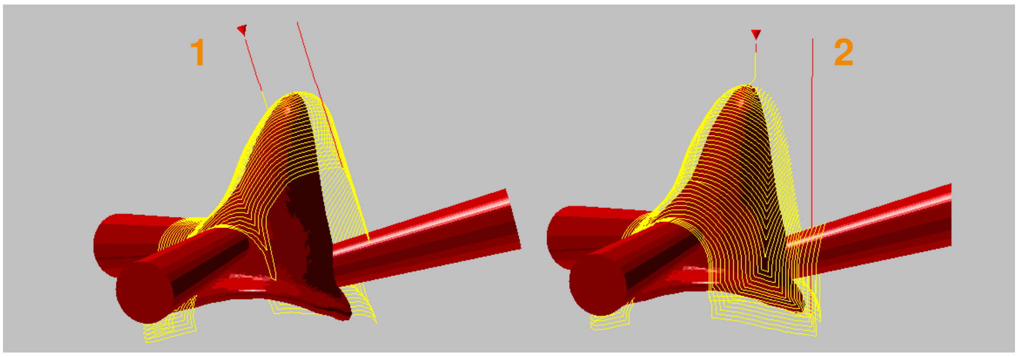


Figure 18-46

The first tool path (1) is calculated with a tilt of 10 degrees and the second tool path (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 preparation line allowance	
	Allowance to protect the preparation line that is added to the value --> “General setting” > “Preparation line thickness”. Important when finishing zirconium oxide so that the preparation line area is not machined any further as it generally has already been fully machined.

For all cycles machining cavity side

Protect preparation line	
Yes	Protection of preparation line, important when finishing zirconium oxide so that the preparation 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).

For 3D complete finishing

Slope dependent machining	
Yes	Slope dependent machining so that the tool is lifted less often from the part: Yes/No

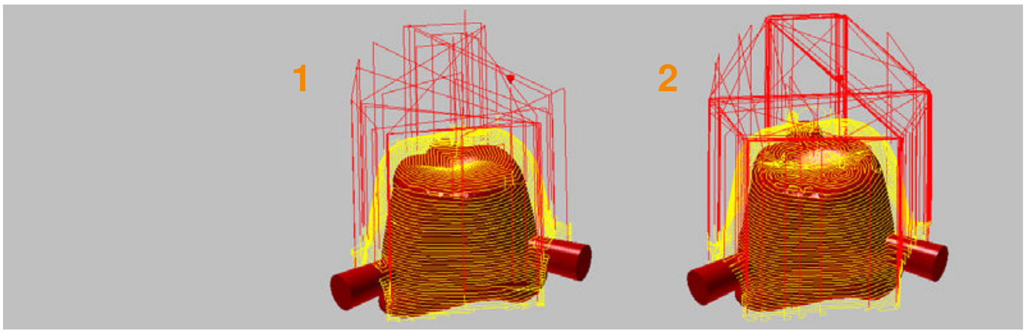


Figure 18-47

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 independently 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 Inside out 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.

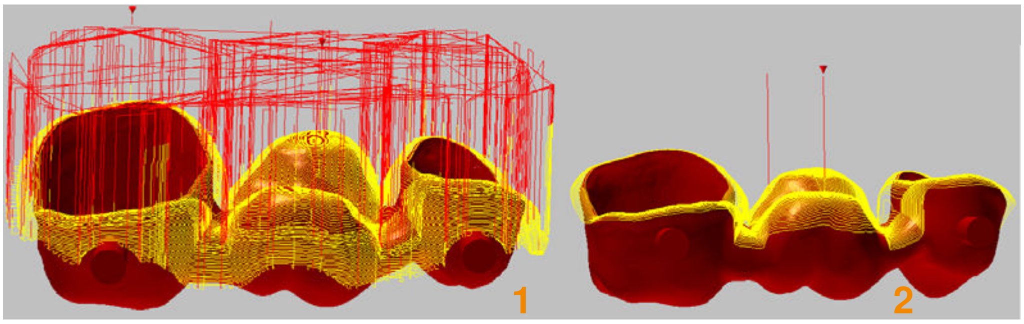


Figure 18-48

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 blank fixture).
Tilt strategy	
Automatic	The lead angle is automatically calculated.
Radial Z	Enter lead angle manually.

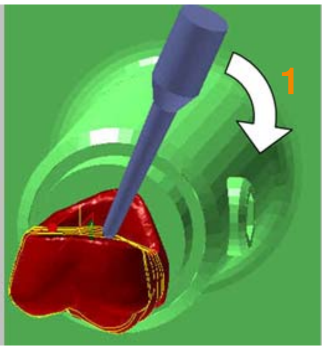


Figure 18-49

1. Lead angle

Lead angle	Value for the lead angle.

Desired tilt angle/max. tilt angle

Tilt angle in relation to the Z-axis.

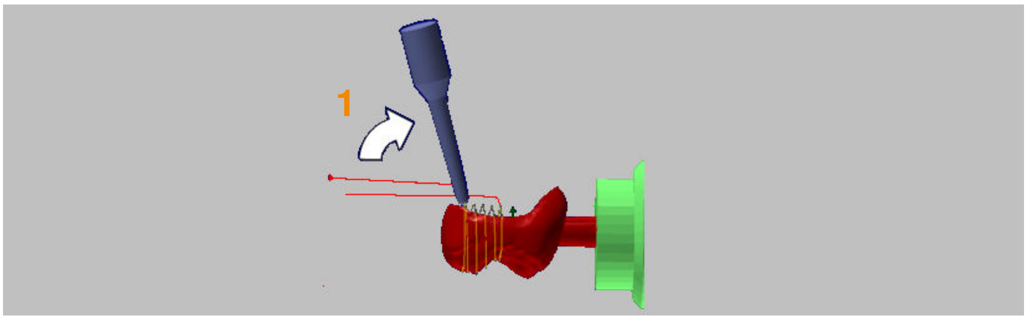


Figure 18-50

1. Tilt angle

Max. avoidance angle A/B

Maximum angle to avoid collision with regard to the A/B-axis.

Invert infeed direction — Reverse the stepover 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.

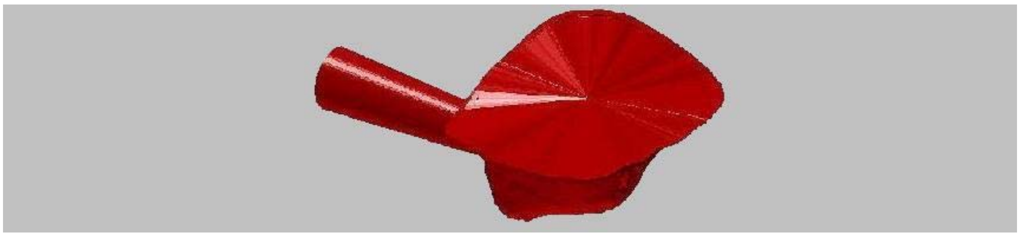


Figure 18-51

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.

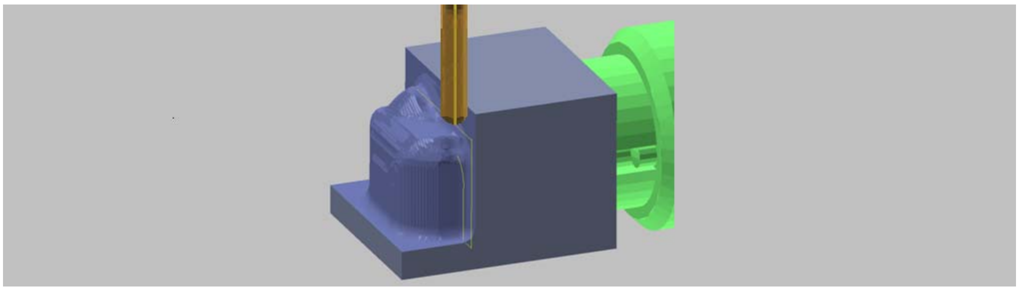


Figure 18-52

Cycles	3D axis parallel peeling Drilling inside copings (preparation for machining with 5X peeling). 5X peeling
Stepover direction	X/Y positive/negative Entry of stepover direction in X/Y positive or negative direction.
Inside out	
Outside in	
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 pre-drilled, 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 the holder side, from “front” 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 “front” view on the blank (bounding box).

For 3X Peeling, roughing each side

Convex shell (Konvexe Hülle)

Yes	Create a convex shell and place around the object for calculating the toolpaths.

The convex shell prevents an excessive axial step (2), e.g. during 5X Z-level finishing and boosts safety during grinding.

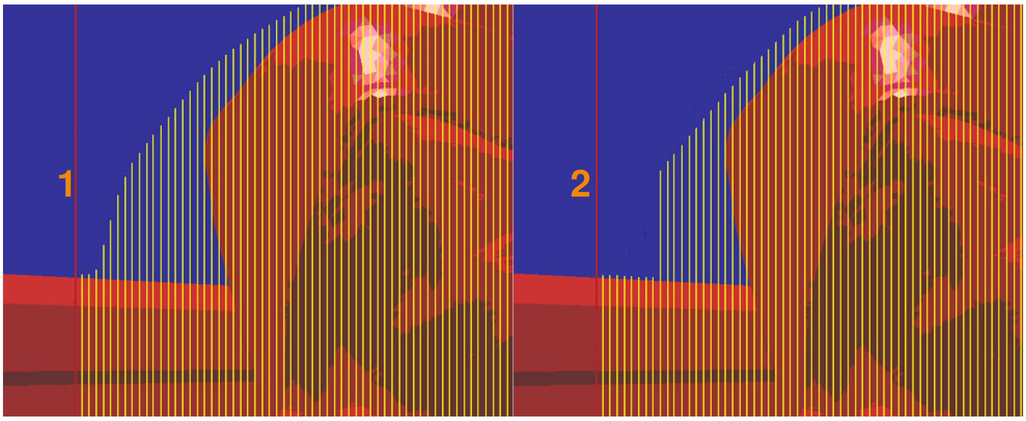


Figure 18-53

1. Toolpaths with convex shell
2. Toolpaths without convex shell

As the highest point of the preparation line (1) coincides with the convex shell and the milling path always begins at the highest point when finishing the preparation line and finishing inside copings/inlay, this also makes the grinding process safer.

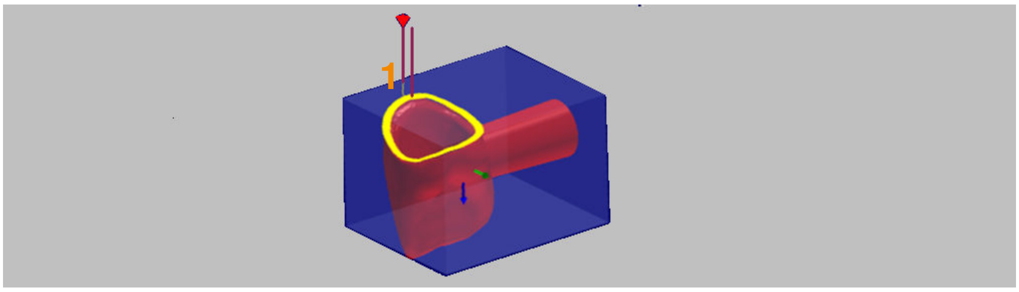


Figure 18-54

Extend machining boundary

Move out the beginning of the milling paths over the machining boundary (1) to ensure safer machining even for complex inlays/outlays with simultaneous offset for mill radius + allowance + tolerance (steep areas). Application is possible for roughing and finishing.

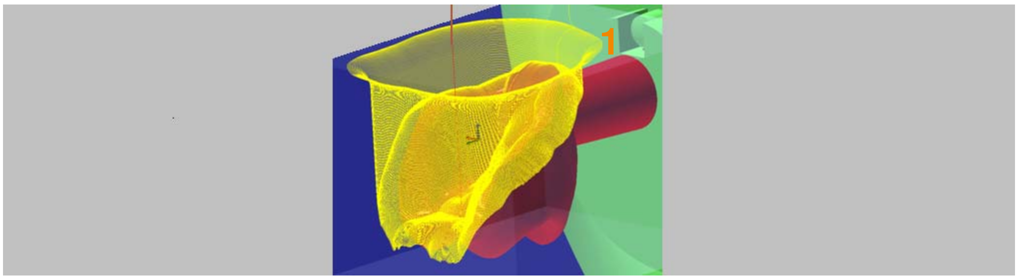


Figure 18-55

No	No extension of machining boundary.
Part maximum (Objekt Maximum)	Extension up to the maximum machining boundary for the part.
Maximum of blank (Maximum des Rohteils)	Extension up to the highest point of the end of the blank.
Blank surface (Rohteiloberfläche)	Extension up to the blank surface.
Automatic	Automatic extension by the program.

Distance to connector interface point (Abstand Konnektoranschlusspunkt) mm

The distance (1) between the connector interface point on the part (2) and the end of the convex shell in the center of the connector (3) determines the path of the convex shell (4).

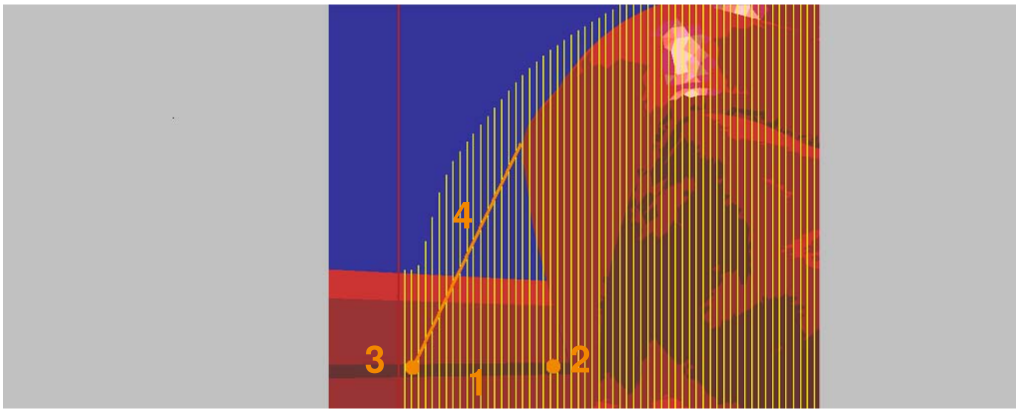


Figure 18-56

1. Distance to connector interface point
2. Connector interface point on part
3. End point of convex shell
4. Convex shell (Konvexe Hülle)

Equidistant spiral finishing (Spiralförmiges Äquidistantes schlichten)

The toolpaths run equidistant to each other in spirals from the inside out.

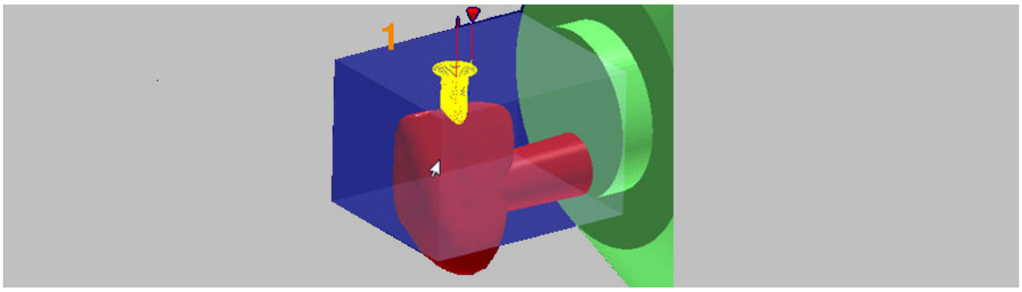


Figure 18-57

Stepover direction

Inside out

Outside in

Offset mm

Default value for machining from the outside in.

Inner boundary offset (Offset innere Boundary) (preparation line)mm

Default value for inner machining boundary (preparation line).

Outer boundary offset (Offset äußere Boundary)	1.25 mm Default value for outer machining boundary

18.8.10 Fissure machining

Only for full wax-up crowns and bridges.

Cycles	3D automatic space left 3D arbitrary blank roughing

The cycle “3D arbitrary blank 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.

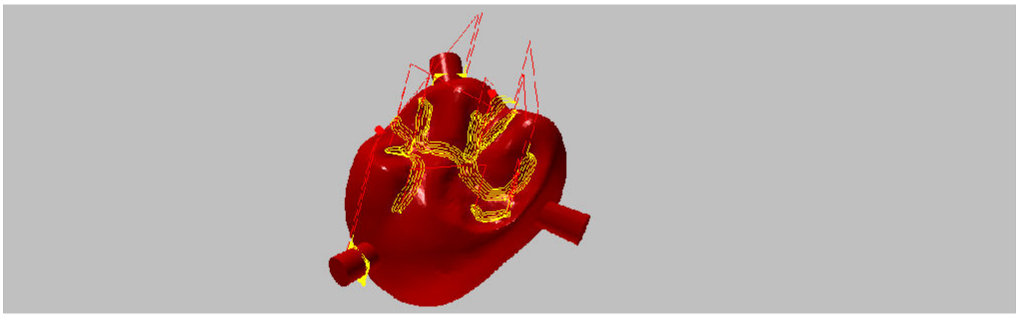


Figure 18-58

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.

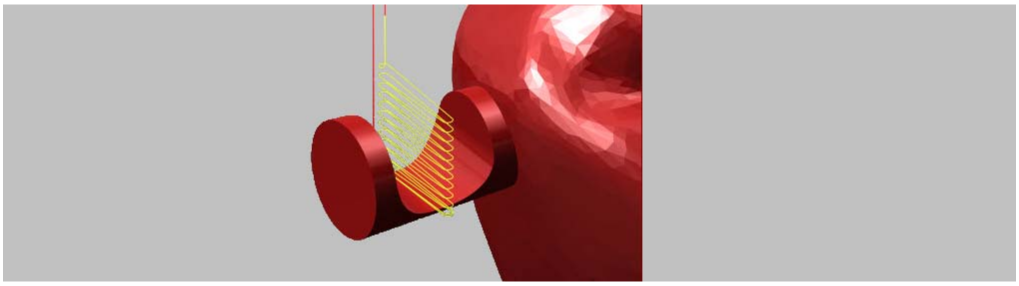


Figure 18-59

3D freepath milling
The cut follows the outline (viewed in the stepover direction).

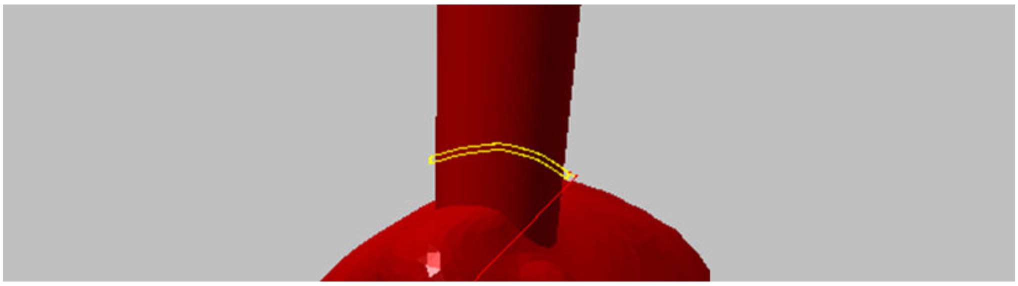


Figure 18-60

3D arbitrary blank roughing
The cuts follows the surface (viewed in the stepover direction). The connector is cut from the part without residue.

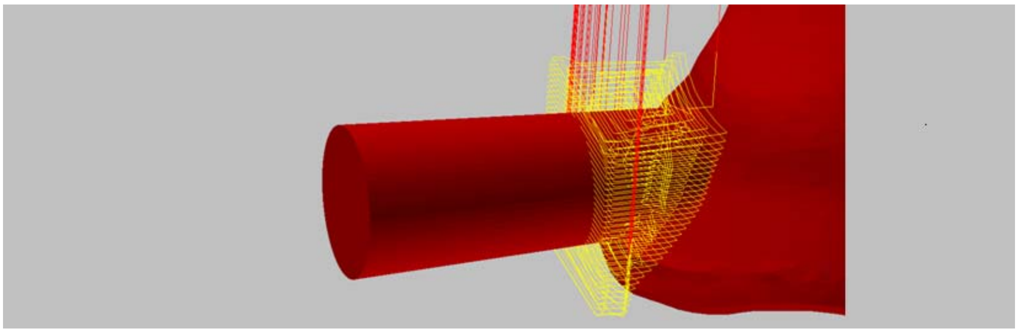


Figure 18-61

Additional cut of connectors on cav. side
Break connectors.

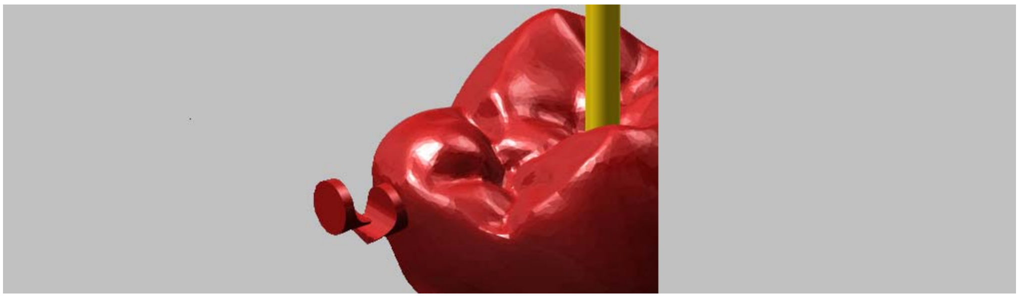


Figure 18-62

Cut/reduce 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 stepover direction) is generated automatically.

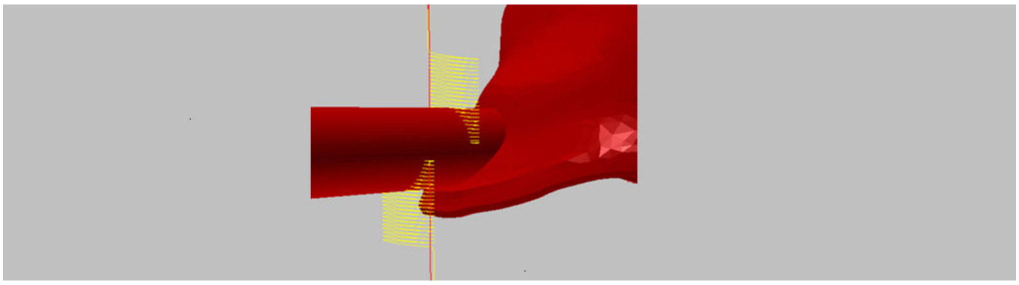


Figure 18-63

From both sides	
Yes	Creates cuts from occlusal and cavity side. Caution! Do not place cuts from the cavity side in undercut areas.
No	Only cuts from the occlusal or cavity side.
Reduce percentage	0 to 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.

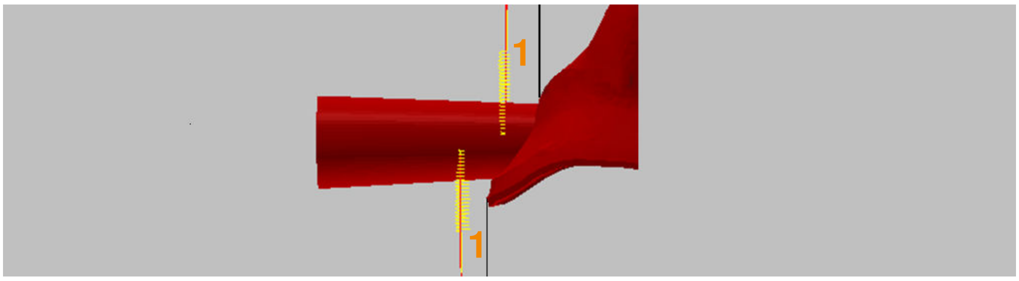


Figure 18-64

Parallel to outline	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.

Additional cut of connectors on cav. 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.

Break connectors

Spindle 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 Cut 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 stepover direction).

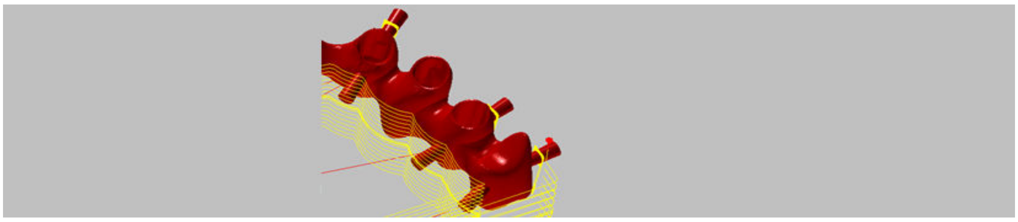


Figure 18-65

Side	Side for machining.
Occlusal side/cavity side/both sides	

Remaining material height, inner frame 0.1 to 0.05 mm

Remaining material thickness after the cut, so that the inner section (1) of the frame can be pressed out.

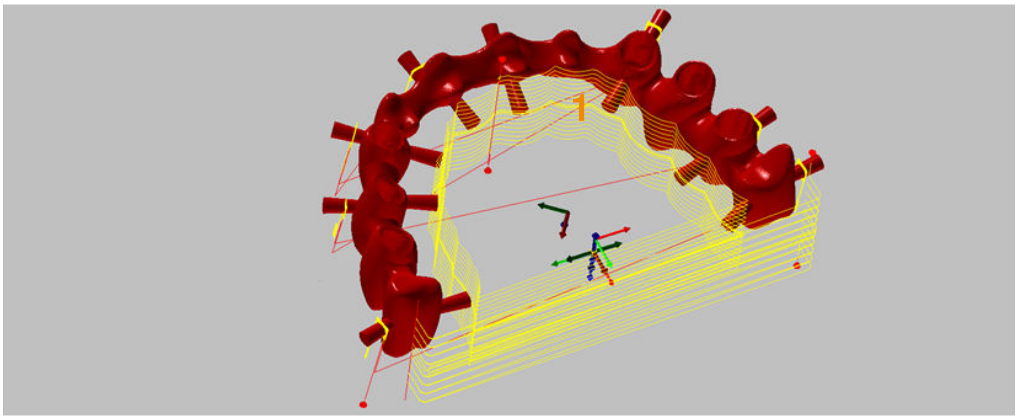


Figure 18-66

Remaining material height, base connectors 1 mm

Material thickness for an additional connector (1) in the area at the base of the sintering frame.

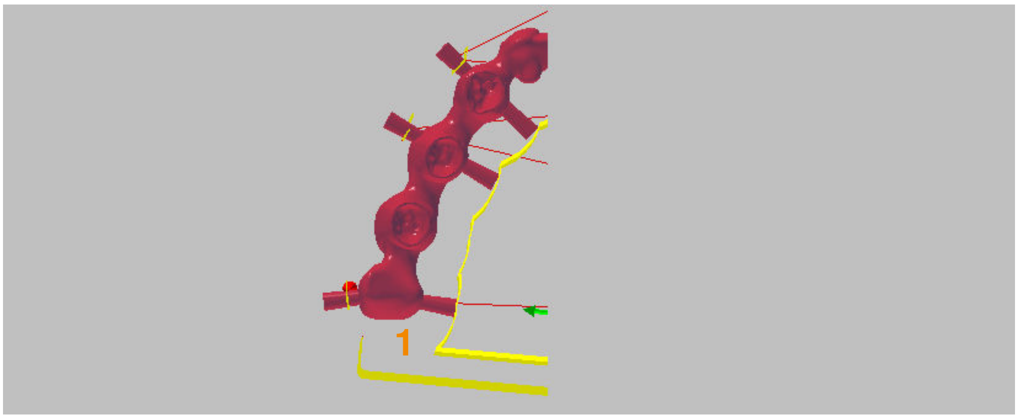


Figure 18-67

18.8.13 Machine splints

Machining of long cavities for producing bitesplints.

The splints lie over the existing teeth as a guard and must surround them below the maximum tooth thickness to ensure a secure grip. As such, the splints have undercuts. The undercuts extend over the entire cavity area and have opposing angles on both sides of the cavity for the insertion and stepover direction.

Depending on the settings in the template, hyperDENT® can detect these long splint cavities and divide them into individual segments for machining on 3+2 machines.

- Automatic division into segments for machining the long cavities.
- Separate machining for the end areas.
- Machining of splint for an entire dental arch with gaps.

Sample splint with 3 segments (1, 2, 3)

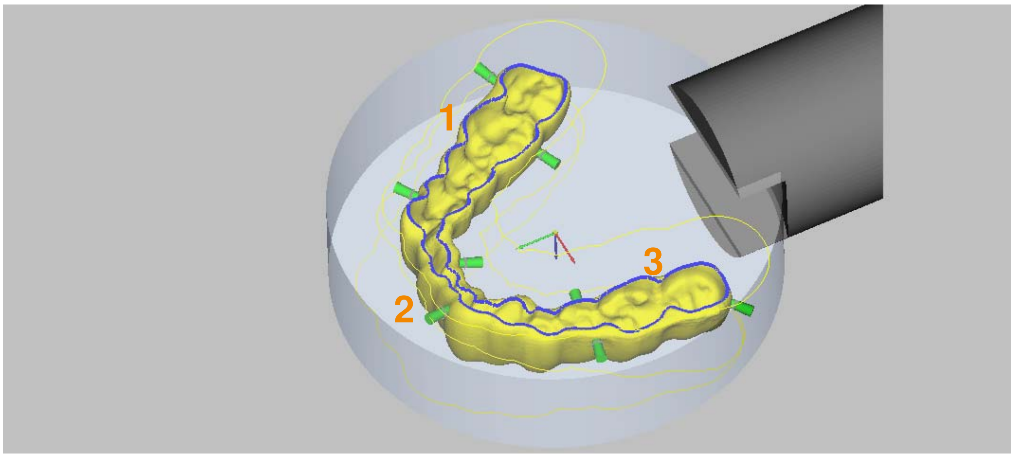


Figure 18-68

The machining takes place, for example, in 6 or more steps:
3 segments, each with 2 different angles for the tilting.

Cycles	Segmented profile finishing, Segmented profile finishing, XY-optimized, Segmented equidistant finishing.

Finishing on the inside of long cavities

Angle difference of segments (Winkeldifferenz der Segmente)	
	Angle deviation of the center line of the cavity for automatic division into part sections (segments). The smaller the value, the more segments are created.
End segment angle (Winkel Endsegment)	
	Angle for the size of the end segment.
0	No end segment
>0	End segment is created.

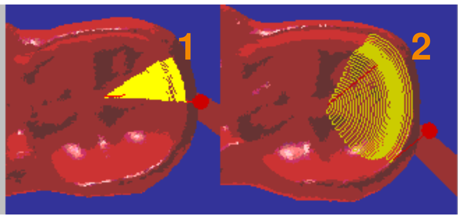


Figure 18-69

1. End segment angle of 30°
2. End segment angle of 120°

Desired tilt angle	Tilt angle for machining the segment.

Finishing inside coping

A splint for an entire dental arch with gaps can contain small cavities for individual teeth, which are machined with this job.

Calculate if	Small cavity
Small cavity	The calculation is only performed for a small cavity. This is required when machining individual teeth within a splint, e.g. for gaps in teeth.

18.8.14 Drilling inside copings

Pre-drilling 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, pre-drilling.

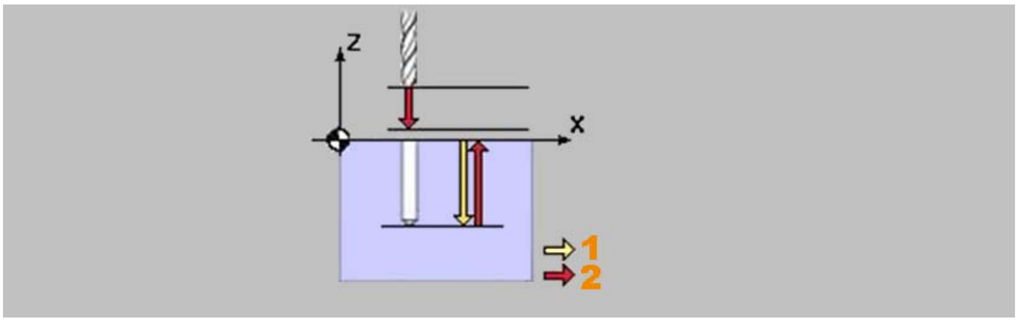


Figure 18-71

1. Feedrate
2. Rapid traverse

Drilling with chip break
The drill hole is created in several steps (Z1, Z2, Z...). 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.

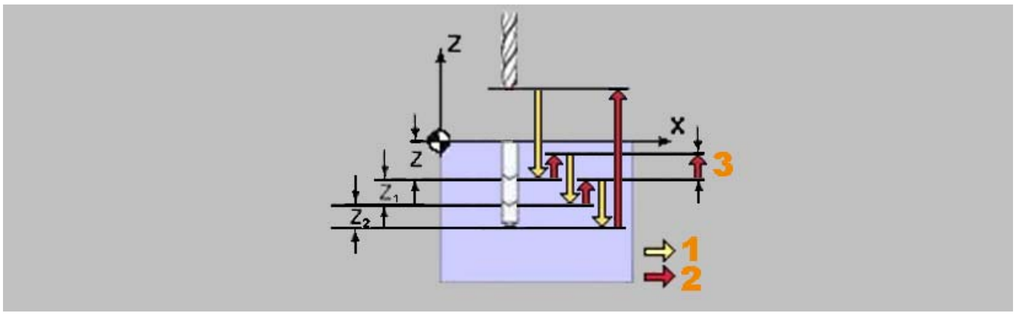


Figure 18-72

1. Feedrate
2. Rapid traverse
3. Retract value

Drilling with pecking
The drill hole is created in several steps (Z1, Z2, Z...). 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.

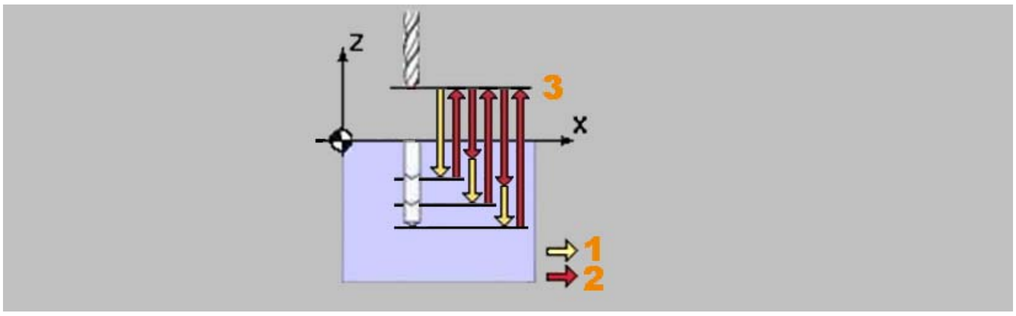


Figure 18-73

1. Feedrate
2. Rapid traverse
3. Clearance distance

Spiral milling
Produce the precise end diameter of the drill hole in a spiral milling process.

The combination of pre-drilling with a drill and spiral milling at any diameter enables fast, flexible production with just a few tools.

Hole diameter (Loch Durchmesser)	mm End diameter of the drill hole when spiral milling.
Side	Side on which machining takes place.
Front	
Back	
Left	
Right	
Top	
Bottom	
Inside copings (Innenseite von Kappen)	
Occlusal insertion direction	
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.
Top	Blank Determination of the start of the drilling.
Blank	Blank surface.
Convex shell (Konvexe Hülle)	Uppermost point of the convex shell.
Automatic	The program determines the start 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 for spiral milling.

18.8.15 Machine all user-defined areas

You can set user-defined areas for specific part areas and assign to these areas your own jobs, which you can precisely adapt to the required machining.

Cycles	Roughing any side (Restmachining) 3D automatic restmachining 5X automatic restmachining 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”. This means you can machine specific part areas with the optimum template.
	Complete machining for a category Restmachining Finishing

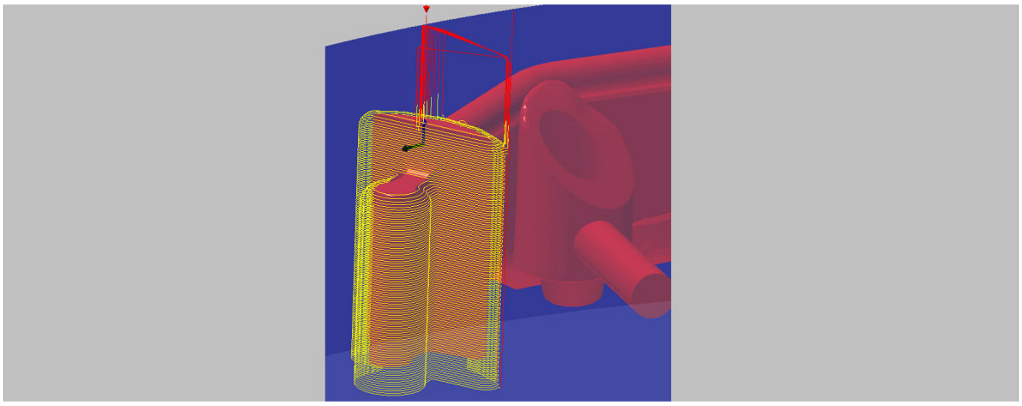


Figure 18-74

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 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 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 for 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 always performed, regardless of the screw channel diameter.
	Screw channel diameter bigger	Calculation only performed if the screw channel diameter is bigger than the default value.
	Screw channel diameter smaller	Calculation only performed if the screw channel diameter is smaller than the default value.
Screw channel diameter between		Calculation only performed if the screw channel diameter is bigger than the 1st default value and smaller than the 2nd default value.

- 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 blank roughing

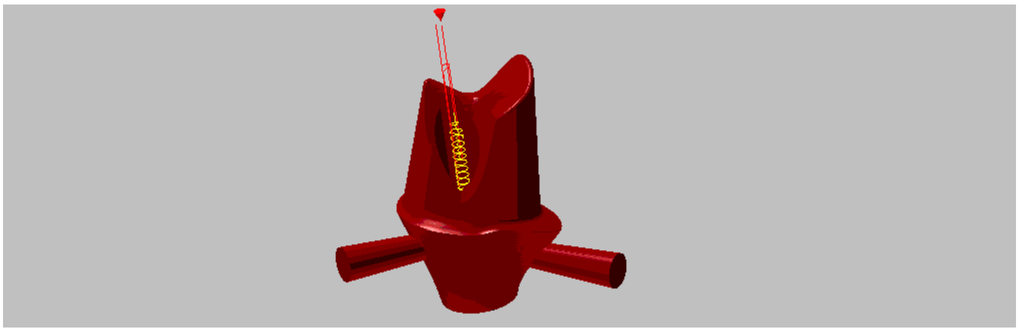


Figure 19-1

5X boss finishing

3D Z-level finishing

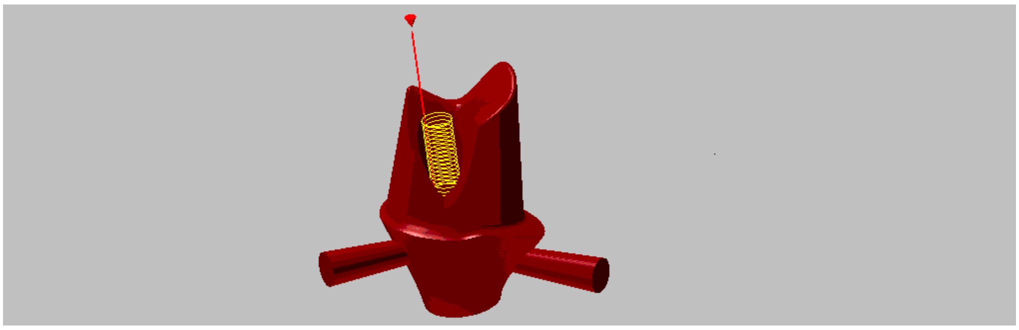


Figure 19-2

Simple drilling

Drilling with chip break

Drilling with pecking

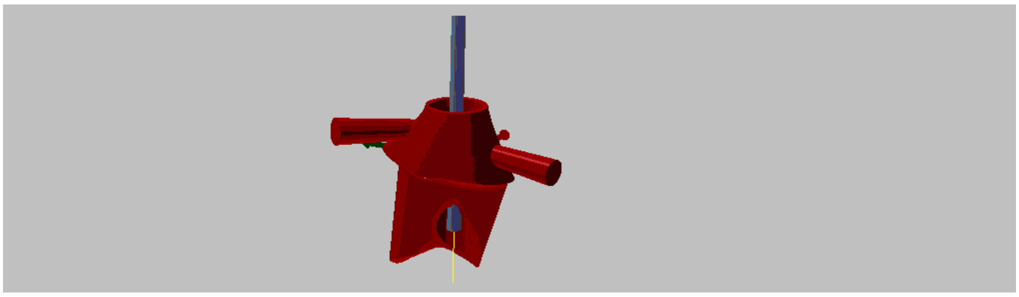


Figure 19-3

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 for the other general job parameters under:
--> “Milling strategies” > “General job parameters”.

Calculate if	Off Conditional calculation, dependent on the screw channel diameter. This means that you can select the tool to suit the screw channel diameter.
Off	Calculation always performed.
Screw channel diameter bigger	Calculation only performed if the screw channel diameter is bigger than the default value.
Screw channel diameter smaller	Calculation only performed if the screw channel diameter is smaller than the default value.
Screw channel diameter between	Calculation only performed if the screw channel diameter is bigger than the 1st default value and smaller than the 2nd 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.: mill, thread mill.
Off	Calculation always performed.
Thread exists	Calculation only performed if a thread exists, e.g. for thread mill.
Thread does not exist	Calculation only performed if no thread exists.

For 3D arbitrary blank 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		
Add. screw channel radius offset		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 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		
Through		

- Caution!
  Do not use the "Automatic" setting if the drill hole has already been pre-drilled 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
Begin of screw channel	
Mid of screw channel	
Begin of screw fit area	
End of screw fit area	
End of screw channel	
Through	
Depth offset	mm Additional offset for the end point 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.

For drilling, drilling with chip break, drilling with pecking

Simple drilling

The drill hole is created in a step, e.g. for center drilling, pre-drilling.

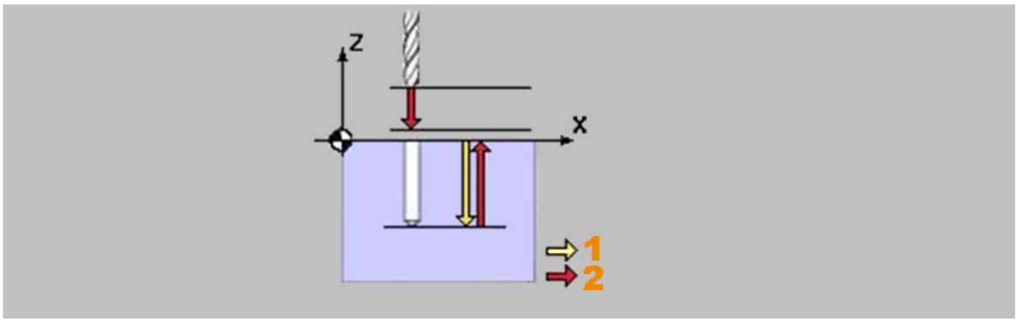


Figure 19-4

1. Feedrate
2. Rapid traverse

Drilling with chip break

The drill hole is created in several steps (Z1, Z2, Z...). 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.

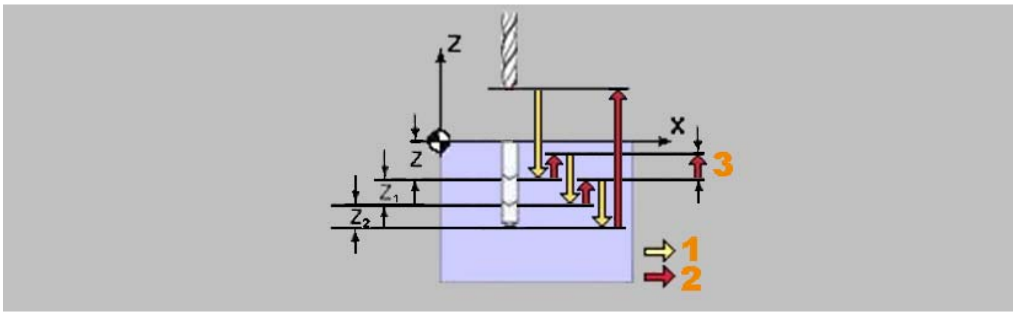


Figure 19-5

1. Feedrate
2. Rapid traverse
3. Retract value

Drilling with pecking	The drill hole is created in several steps (Z1, Z2, Z...). 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.

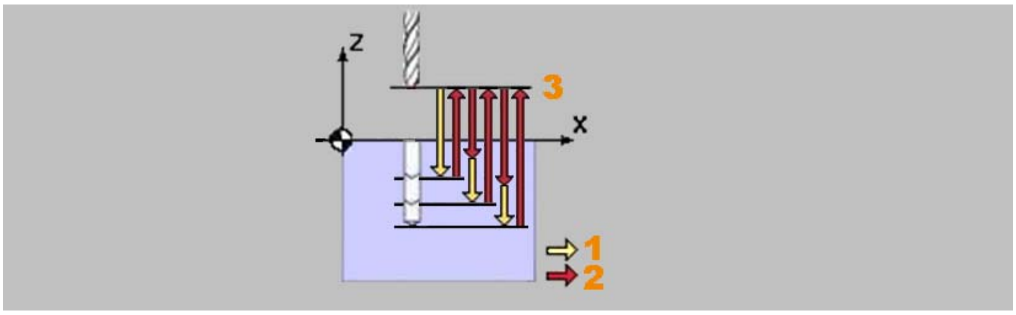


Figure 19-6

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

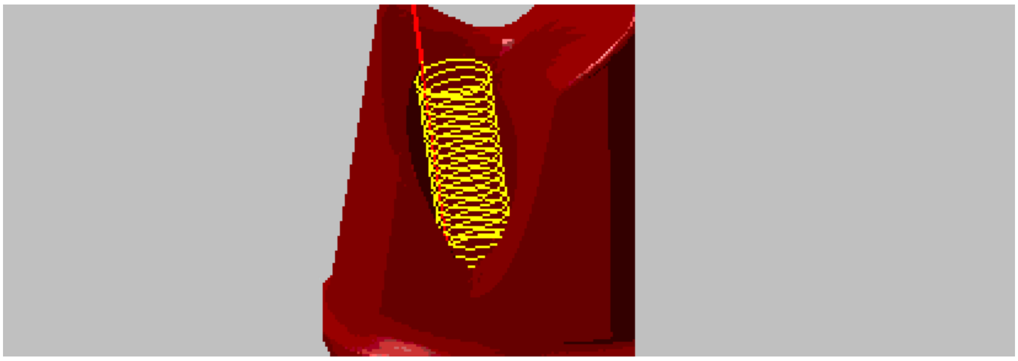


Figure 19-7

Side — Side that is to be machined by this cycle.

Occlusal side/cavity side

Add. screw channel radius offset — 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 the machining.
- Maximum inside abutment area
- Blank — Blank surface
- Begin 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 pre-drilled 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 point 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.
Stepover direction	Upward
Downward/upward	Direction in which machining is to take place.

Nr. of steps

Number of steps (stepovers) for the production of a thread.

Nr. of spring cuts per step

Number of repeat (or empty) steps used to create the thread pitch.

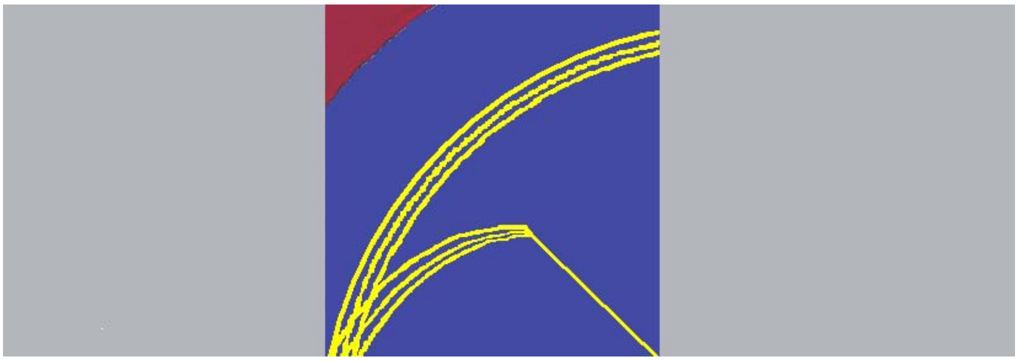


Figure 19-8

19.4.3 Drilling implant interface geometry

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).

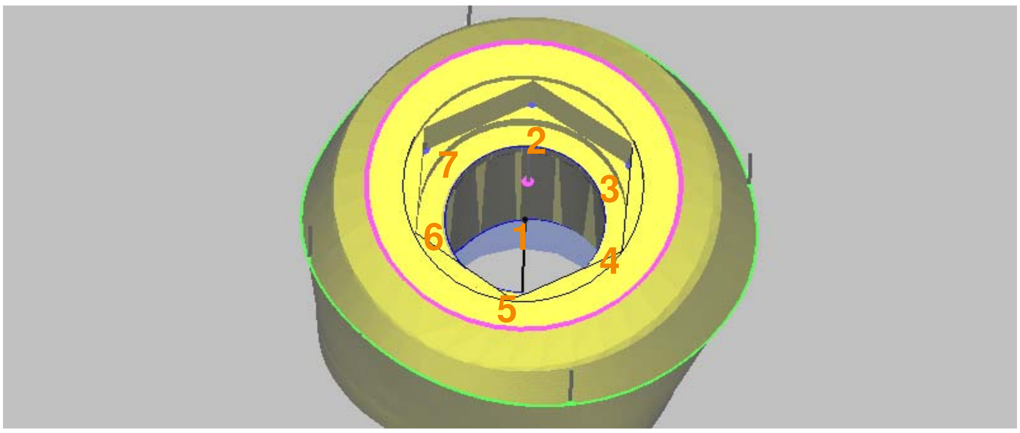


Figure 19-9

19.4.4 Plane finishing inside abutment bases

Machining to create an absolute plane surface at the abutment base.

Cycles	3D equidistant finishing

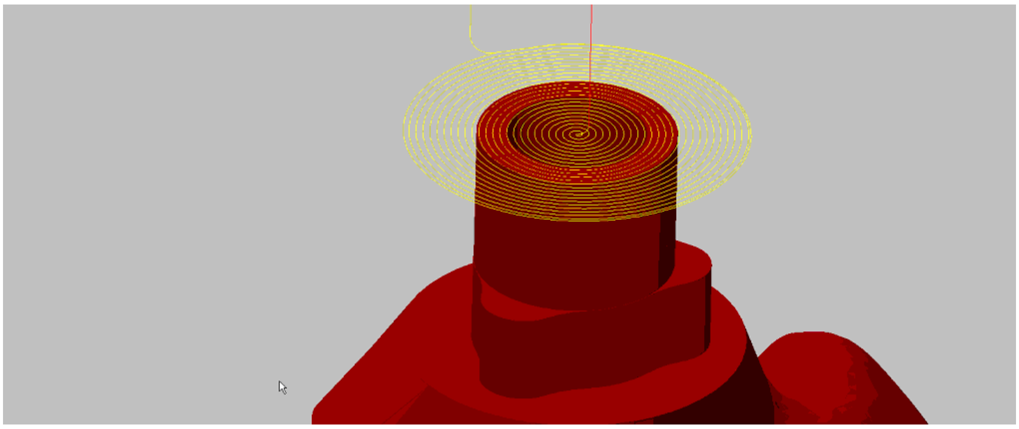


Figure 19-10

Offset	mm
0	Milling boundary as for abutment base.
>0	Milling boundary is extended.
<0	Milling boundary is reduced.
Stepover direction	
Inside out	
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, tool path 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 stepover direction.

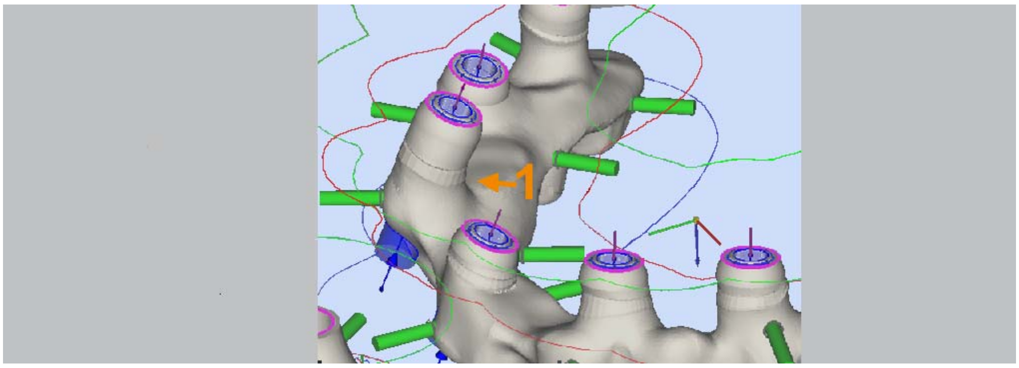


Figure 19-11

Rest machining outside abutment bases

Cycles 3D arbitrary blank roughing (with blank tracking)

Finishing outside abutment bases

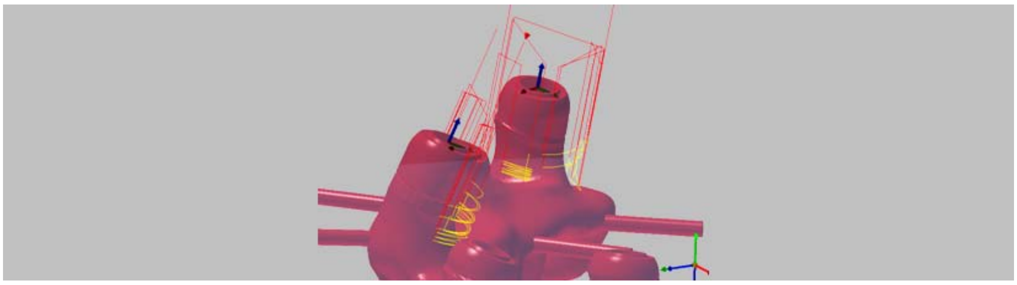


Figure 19-12

Cycles 3D Z-level finishing

Side	Side that is to be machined by this cycle.
Occlusal side/cavity side	
Use coping-specific tilting No	Details of the rotation axis for the coping-specific tilting. If the stepover 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 to 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

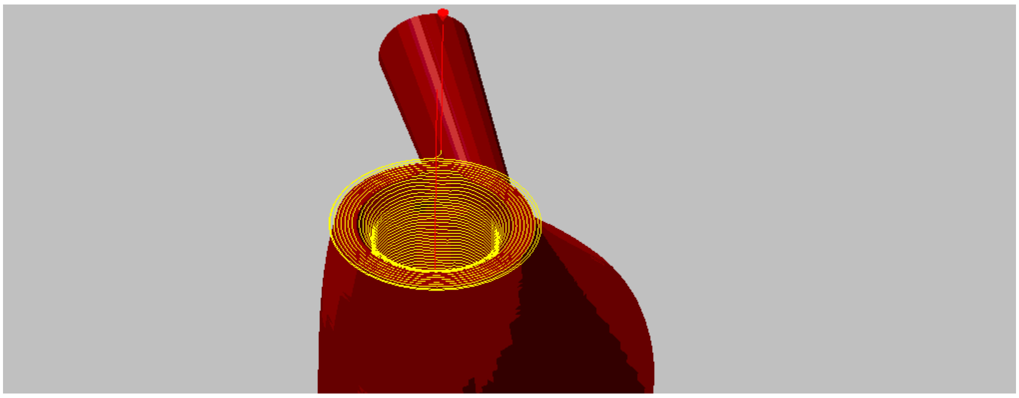


Figure 19-13

Exclude geometry inside	
Yes/No	Exclude defined abutment base areas, e.g. 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, tool path follows profile (in offset area) up to “outer depth”.
Stepover direction	
Outside in	
Inside out	
	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

Equidistant finishing is usually the best cycle for the emergence profile, as this area requires a high surface quality.

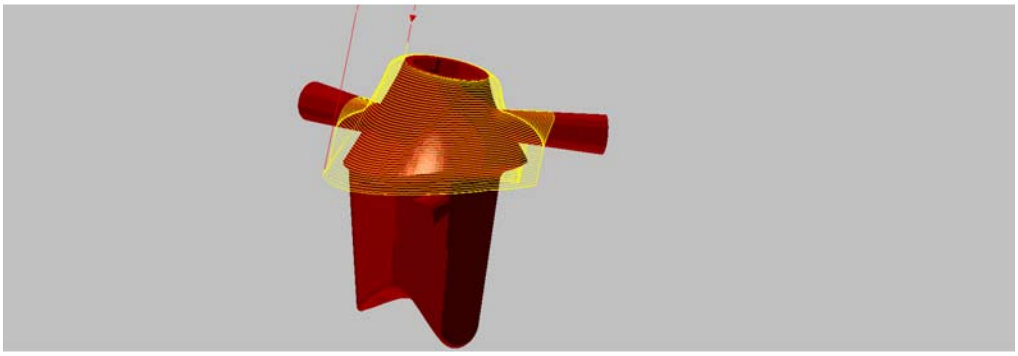


Figure 19-14

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.

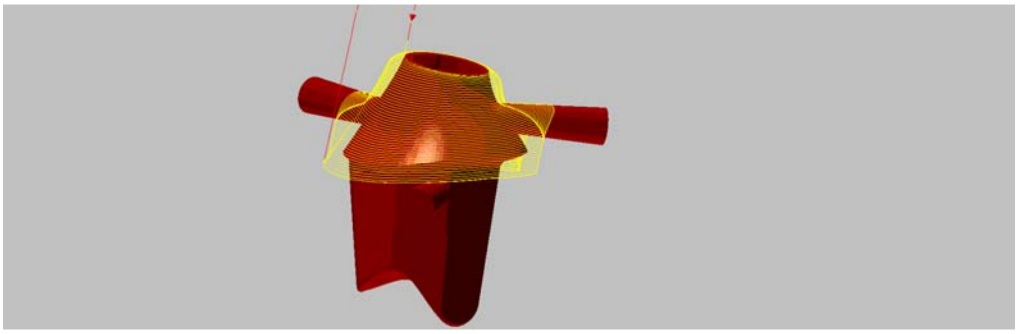


Figure 19-15

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...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.	
Stepover direction		
Top down		

Bottom up	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.

Partial machining for a category

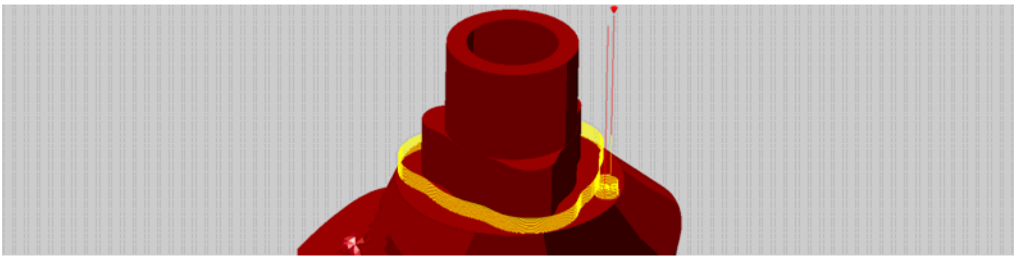


Figure 19-16

Complete machining for a category

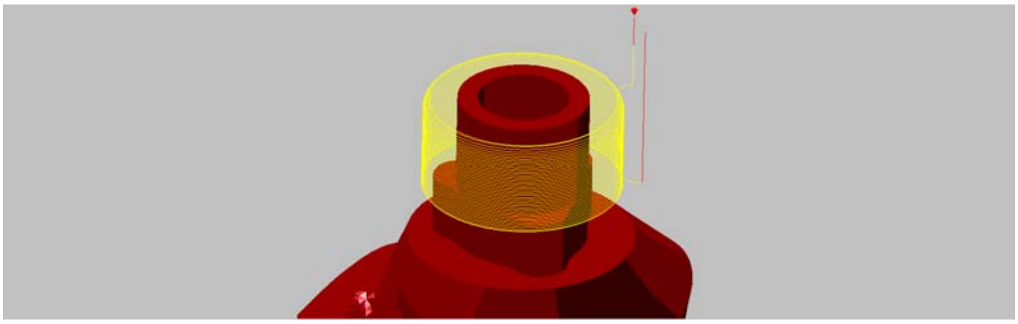


Figure 19-17

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.

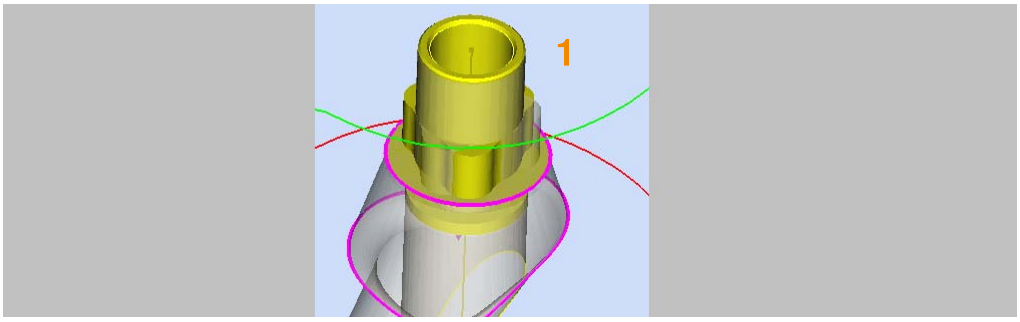


Figure 19-18

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

Create, amend 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 line:
   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 geometries.
6. Repeat the procedure for further 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 line.
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

Line	Designation in CAD	=	Designation in 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

Line	Designation in CAD = Designation in 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

Line	Designation in CAD = Designation in 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 1 October 2010 or later. In earlier versions, errors will be encountered during placement.

3Shape File: ImplantDirectionPosition.xml

Example

ImplantDirectionPosition_0.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>
```

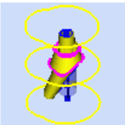

Figure 19-20

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 restoration 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:

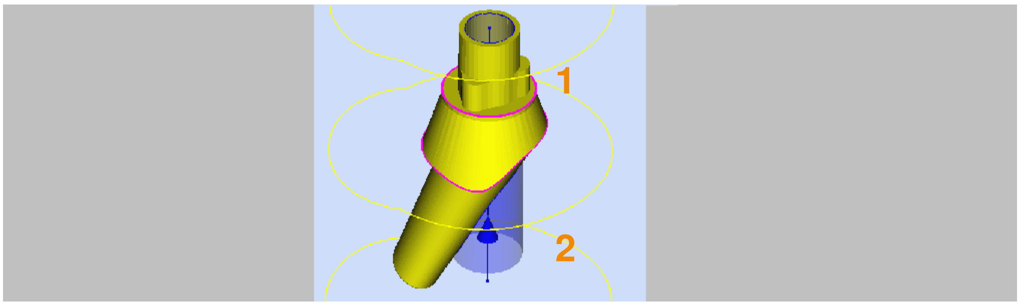


Figure 19-21

2. If not already done, determine the abutment base (1) and the screw channel (2): --> "Identify part features"
   --> "Determine abutment base",
   --> "Determine screw channel".

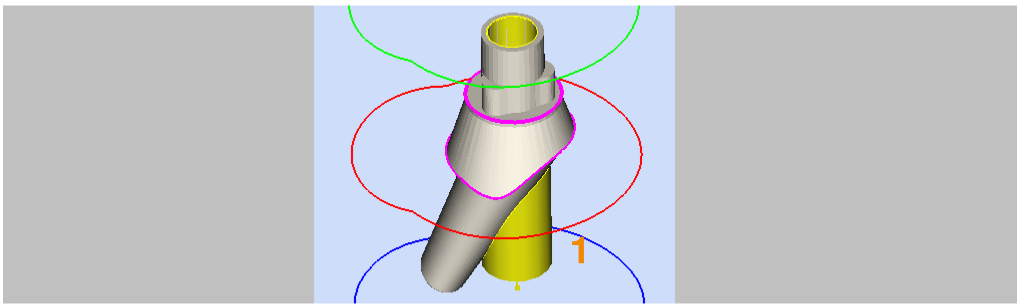
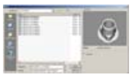


Figure 19-22

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.

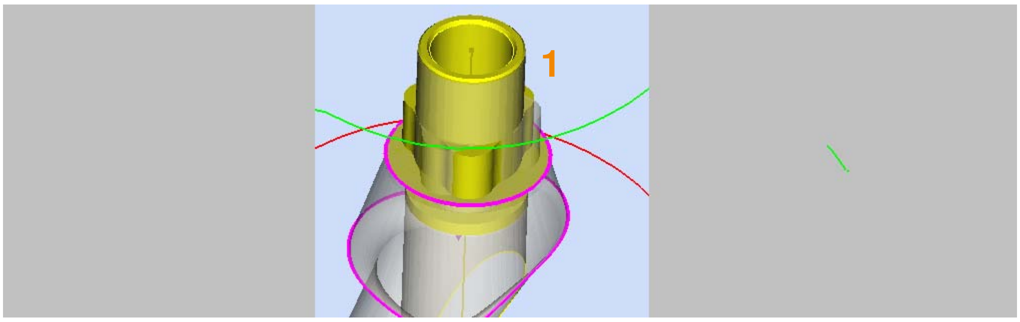
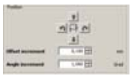


Figure 19-23

Align 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.

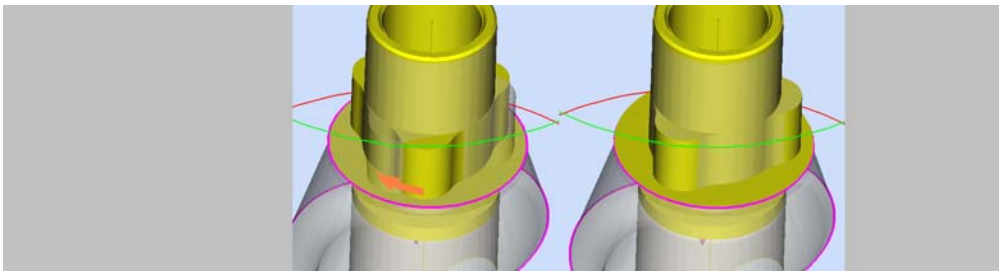


Figure 19-24

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.
For this, set a smaller increment for the angle and the movement parallel to the axis.

19.5.4 Create connection 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 fit 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.

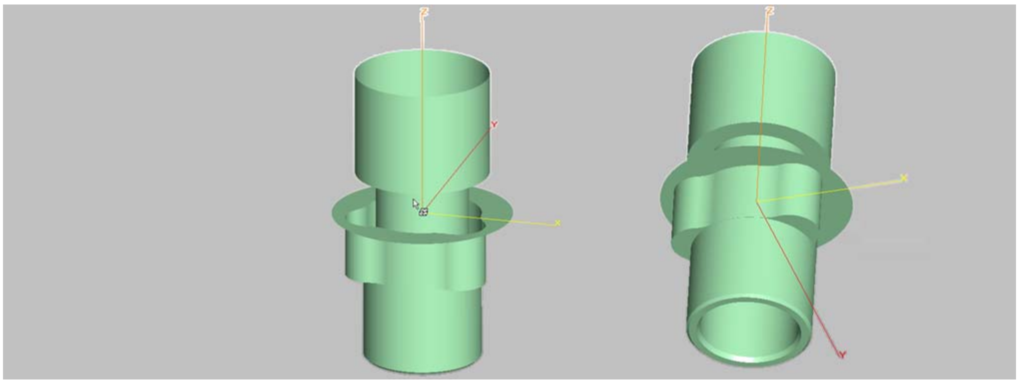


Figure 19-25

19.5.5 Prepare and save 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 geometries 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.

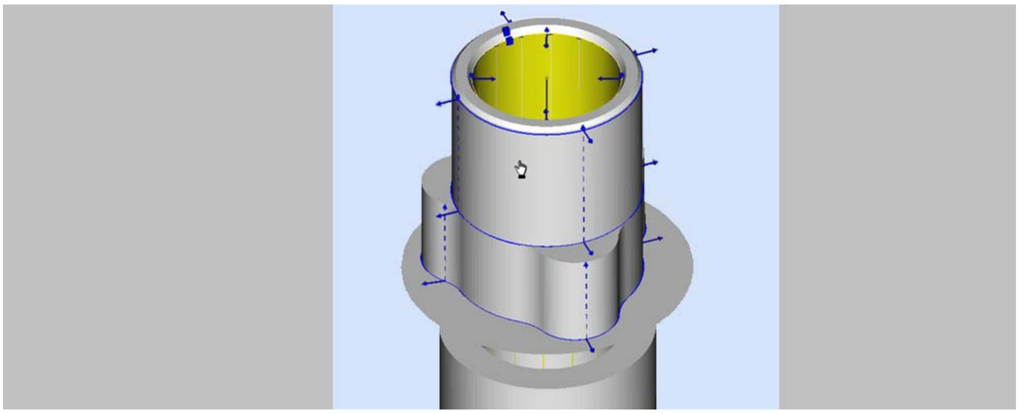


Figure 19-26

3. Delete all 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.

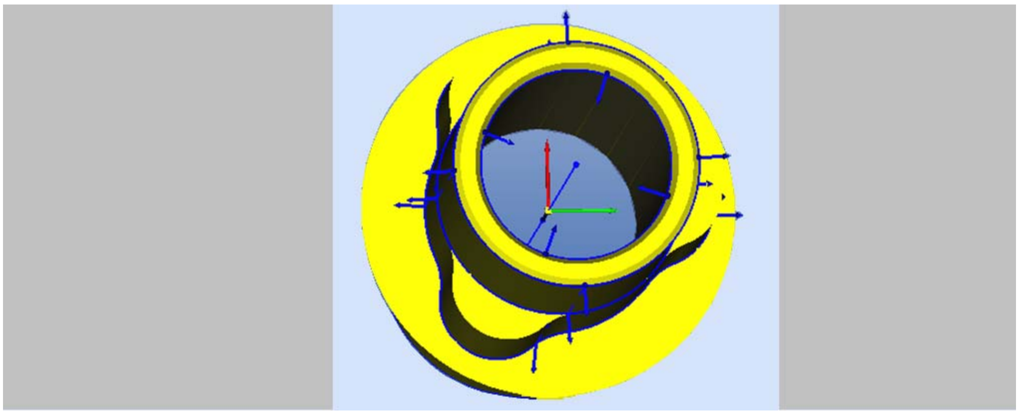


Figure 19-27

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 file type ".hdpartz".
   File type ".hdpartz" is required 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

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 Institut Bone Level®
- Straumann Institut 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 identifier
[Type]	Type identifier
_L_	Locked
_UN_	Unlocked
.hdpartz	hyperDENT® project file
.alias	hyperDENT® mapping file, allocation file for automatic geometry replacement.

File names of geometry types

Manufacturer	Identifier	
File name Locked		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	Identifier	
File name Locked		File name 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 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 stepover 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 stepover 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®.

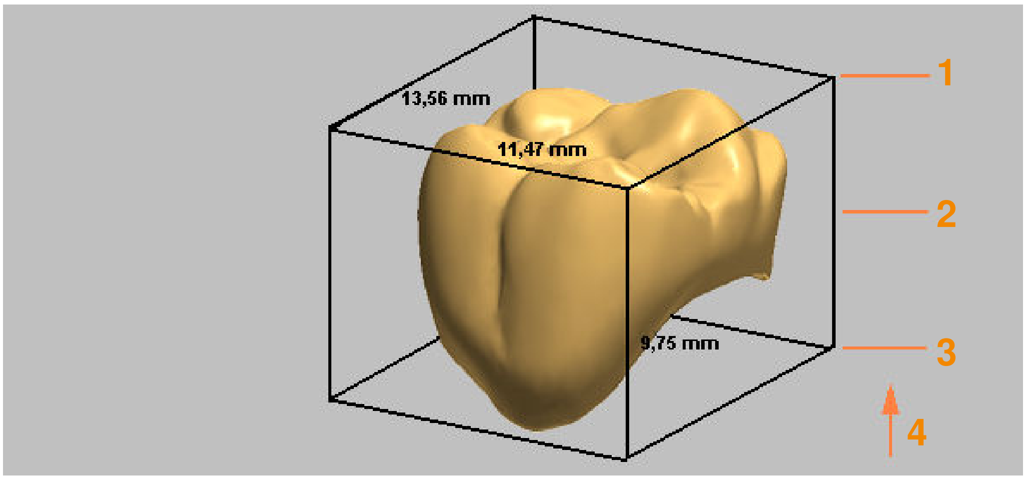


Figure 20-1

1. Max. of bounding box
2. Center of bounding box
3. Min. of bounding box
4. View direction

Prefabs – prefabricated blanks

As well as the usual blanks, you can also use prefabricated blanks (prefabs) with a prefabricated screw channel, screw fit, and interface geometry. These prefabricated blanks are usually provided by the system supplier to match the corresponding holder.
For further information, please contact our support team.

21 Index

hyperDENT®
Instruction manual

Software for dental CAM applications

Copyright

Address	FOLLOW-ME! Technology Systems GmbH Regerstr. 27 81541 Munich, Germany
Contact	Tel.: + 49 (89) 45 21 70 60 Fax: + 49 (89) 45 21 70 69 E-mail info@follow-me-tech.com

Version	2014-01	140318en