

hyperMILL®



© The helmet was programmed and produced by OPEN MIND

hyperMILL®

Release notes 2024 | Update 1

 **OPEN MIND**
THE CAM FORCE

This document is intended for users and administrators. It applies to *hyperMILL*®, *hyperMILL*® SHOP Viewer, *hyperMILL*®CAD, CAD Viewer, *hyperMILL*® for SOLIDWORKS, and *hyperMILL*® for Autodesk® Inventor®.

The document is installed in the directory `...\OPEN MIND\doc\[Version number]\Readme...`

Useful information about hardware and software requirements, graphic cards for *hyperMILL*®CAD, installation requirements as well as an installation guide can be found on our website at: [Useful information](#)

If you work with third-party software that uses *hyperMILL*® data (such as postprocessors and simulation tools), you should consider the following: The format of all data generated by *hyperMILL*® can be changed by OPEN MIND **without prior notice** and at any time for the purposes of further development. This primarily affects the output of machine- and control-neutral programs (POF format). OPEN MIND does not offer any guarantee for problems that are due to compatibility issues involving third-party software.

OPEN MIND Technologies AG

Argelsrieder Feld 5
82234 Wessling
Germany
Tel.: (+49-8153) 933-500
Fax: (+49-8153) 933-501
E-mail: <sales.europe@openmind-tech.com>
Web: www.openmind-tech.com

Compliance Intelligence

The Software may include a compliance intelligence mechanism for security and reporting purposes ("Security Mechanism") that automatically collects and transmits data regarding the installation and use of the Software to OPEN MIND Technologies AG, its licensor and the manufacturer of the Security Mechanism for purposes of verifying End Customer's compliance with the terms of the applicable EULA, identifying unauthorized use and users, and otherwise protecting and enforcing intellectual property rights. Data processed through the security mechanism may include, but is not limited to, user, device and network identification information, location and organizational domain information and software usage information. For more information about the processing of personal data through the Security Mechanism, please see our privacy policy at <https://www.openmind-tech.com/en/privacy/>.

(produced on Mon, Jun 10, 2024)



Table of Contents

1. Supported Versions	2
Operating systems und CAD platforms	2
CAD interfaces	2
Interfaces Tool database	5
Supported EDM formats	5
Supported OPTICAM Software Versions	6
Interfaces NC simulations	6
2. New commands and additions	7
CAM	7
User interface	7
<i>hyperMILL</i> and CAD program	7
The <i>hyperMILL</i> browser	7
Basics of CAM editing	7
Specifying the basic settings	8
NC system and frames	8
Tool management	8
Structure CAM project	9
Simulate machining	12
<i>hyperMILL</i> additional functions	14
Turning	15
Roughing	15
Grooving	16
Drilling	17
Optimized deep Hole Drilling (new)	17
2D Machining	21
Contour Milling on 3D Model	21
Chamfer Milling on 3D Model	22
3D Machining	22
Profile Finishing	22
Z Level Shape Finishing	25
Plane Machining	25
Cutting edge	26
5X Machining	27
Cavity Milling	27
Additive Manufacturing	31
Tube Milling	32
Multiblade Milling	33
Tool database	33
Define turning tool	33
Create indexable cutter	33
Optimal barrel cutter	35
Tool series and technology templates	37
Define coupling	58
VIRTUAL Machining Center	58
Program run (simulation)	58
Controlling visibility in the graphics area	58
Simulating the VNC or NC program file	59
Collision check and material removal	60
Configuration	60
TOOL Builder	60
Introduction	60
Steps for creating	61
Turning tool: Creating a static holder for machining with a turret	61
CAD	63
Introduction	63
Documentation and help	63
User interface	64
Tabs	64



Default settings	64
Options / properties	64
Sketcher options	64
Data interfaces	64
OPEN MIND Software	64
Import PMI with MBD	65
Analysis	66
Attributes	66
Edit and modify	66
Edit	66
Modify	69
Points, curves and faces	69
Drafting	69
Shapes	72
Parametric modeling	75
Automatic rebuild on / off	75
Solids, features and meshes	75
Features	75
Solid	77
Meshes	77
Design electrodes	77
Electrode options	77
Create partial	78
Create user defined	78
Modify eroding path	78
Using user-defined tags for electrode documentation	79
Export to EDM	79
Preparing generator programs	80
CAM programming	80
Overview	80
Adaptive construction	83
Analyzing probing results	83
Probing tab	84
Import probing data	87
Probing options	87
3. Release notes	89
Release 2024	89
CAM	89
CAD	93
Release 2024 Update 1	98
CAM	98
CAD	102



OPEN MIND has always offered an innovative CAD/CAM solution whose CAD functionalities are seamlessly linked to CAM programming.

We Push Machining to the Limit

The resulting time savings in workpiece preparation, among other things, make it clear that CAM without CAD is not an option. From version 2024, *hyperMILL*® now combines CAD and CAM under one name (instead of *hyperCAD-S*), thus consolidating “CAD for CAM” for the future. The familiar CAD functionalities remain the same, only their names change.



1. Supported Versions

Operating systems und CAD platforms

64-bit operating systems	Windows 10, Windows 11
64-bit CAD platforms	hyperMILL® Inventor 2022, 2023, 2024 SolidWorks 2022, 2023, 2024
Server operating systems (License server only)	Windows Server 2012 R2, Windows Server 2016, Windows Server 2019
hyperMILL exclusively supports 64-bit operating systems.	



Attention: hyperMILL® 2023 is not compatible with any thinkdesign version!

CAD interfaces

The following CAD models can currently be imported and/or exported (depending on the license purchased):

Product	File type	Up to version		Import	Export
		Technology up to 31 December 2019	Current technology		
CATIA V4	*.model	4.2.4	4.2.5	x	
	*.exp	4.2.4	4.2.5	x	
CATIA V5	*.CAT-part	6R2018 (R28)	V5-6 R2023	x	
	*.CAT-product	6R2018 (R28)	V5-6 R2023	x	
	*.CGR	Not supported	V5-6 R2023	x	
CATIA V6	*.3dxml	Not supported	2024	x	
PTC Creo Parametric	*.prt *.prt. *	6.0 F000	10	x	



Product	File type	Up to version		Import	Export
		Technology up to 31 December 2019	Current technology		
	*.asm *.asm. *	6.0 F000	10	x	
	*.neu	Not supported	10	x	
PTC Creo	*.xpr	6.0	10	x	
	*.xas	6.0	10	x	
Siemens NX	*.prt	NXCR	NX2312	x	
SOLIDWORKS	*.sldprt	2019	2024	x	
	*.sldasm	2019	2024	x	
Autodesk® Inventor®	*.ipt *.iam	2019	2025	x	
Rhinoceros®	*.3dm	Not supported	8	x	
Solid Edge	*.par	Not supported	2024	x	
	*.asm				
	*.pwd				
	*.psm				
PRC (Product Representation Compact)	*.prc	Not supported	All versions	x	
Parasolid	*.x_t	31	36.1	x	
	*.x_b	31	36.1	x	
JT-Open	*.jt	10.2	10.9	x	
IGES	*.igs	5.2, 5.3	5.1, 5.2, 5.3	x	
	*.iges	5.3	5.3		x
STEP	*.stp, *.step	AP 203	AP 203 E1/E2 ^a .	x	
		AP 214	AP 214 ^b .		
		AP 242	AP 242 ^c .		
		AP 214	AP 214		x
AutoCAD	*.dwg	2018 (AC1032)	2019	x	



Product	File type	Up to version		Import	Export
		Technology up to 31 December 2019	Current technology		
	*.dxf	2013-2017 (AC1027)	2019		x
Point clouds	*.pt, *.asc *.xyz *.txt	No version	No version	x	
	*.pt				
Polygon mesh	*.stl *.stla *.stlb	No version	No version	x	x
3MF Reader (3D Manufacturing Format)	*.3mf	Not supported	1.2.3	x	
ACIS	*.sat *.sab	Not supported	2023	x	
Wavefront OBJ	*.obj	Not supported	All versions	x	

^a(ISO 10303-203) "Configuration controlled 3D design of mechanical parts and assemblies"

^b(ISO 10303-214) "Core data for automotive mechanical design processes"

^c(ISO 10303-242) „Managed model-based 3D engineering“



Interfaces Tool database

Tool Management System	Required licenses	Required software
tdm systems	TDM Base modul (TDM / TDMGL) TDM Class / group structure V (CLGR) CAM Interface TDM - <i>hyperMILL</i> (AME) (iM-HYP) Optional for 3D Tool data transfer: 3D-Solid Converter for <i>hyperMILL</i> (iCHYP)	TDM Systems - Base Installer TDM Systems - Data Installer TDM Application Server Installer TDM GlobalLine Interfaces Installer (for the Smart Interface Client <i>hyperMILL</i>)
Zoller TMS	<i>hyperMILL</i> v2 Interface first license TMS Tool Management Solutions	TMS Tool Management Solutions BRONZE package TMS Tool Management Solutions from version 1.17.0
WinTool AG	WinTool <i>hyperMILL</i> Interface	WinTool 2020 (WT2020.2.1) Microsoft Server 2012 or higher Microsoft SQL Server 2012 or higher <i>hyperMILL</i> Interface (2.13.5)
Hexagon Manufacturing Intelligence	NCSIMUL Tool NCT-CAM-HY (<i>hyperMILL</i> interface)	NCSIMUL Tool NCSIMUL Tool Client NCSIMUL Tool Server NCSIMUL Tool Interface FlexLM

Supported EDM formats

Reports for the following eroding machine types can currently be converted.

Manufacturer	Software	Version	Output 3-point path	Virtual electrode	Rotating electrode	Output 3D path
Zimmer & Keim	Alphamoduli		x	x	x	x
OPS Ingersoll	Multiprog		x	x	x	
ONA			x	x	x	
+GF+HMI	AC FORM HMI		x	x	x	
Exeron	Exoprog		x	x	x	



Supported OPTICAM Software Versions

The following OPTICAM software version can be used for the respective hyperMILL® version:

hyperMILL®	OPTICAM
2024 Update 1	2024.2
2024	
2023.2 Update 9	2024.1 2023.2
2023.2 Update 8	2023.2 2023.1
2023.2 Update 7	
2023.2 Update 6	
2023.2 Update 5	
2023.2 Update 4	
2023.2 Update 3	

Interfaces NC simulations

VERICUT from version 7.0
NCSimul from version 2020.0


2. New commands and additions

Information on new commands and additions, as an extract from the software documentation:

CAM

User interface

hyperMILL and CAD program








Icon	Function	Explanatory note
	Optimal barrel cutter	Optimize parameters of barrel cutters based on surface and curve information.

The hyperMILL browser

Jobs tab

Turning with turret

The following information is available for turning jobs with a turret:

	A machine with turret is used in the job list and the definition of the turret setup is free of errors and warnings.
	A machine with turret is used in the job list and the definition of the turret setup contains errors.
	A machine with turret is used in the job list and the definition of the turret setup is free of errors and warnings. No NC file has been created yet.
	A machine with turret is used in the job list and the definition of the turret setup contains errors. No NC file has been generated yet.
	A machine with turret is used in the job list and the tool used in the job is set up on the turret.
	A machine with turret is used in the job list and the tool used in the job is not set up on the turret.
	No machine for turning jobs with a turret has yet been set up in the job list

Basics of CAM editing



Specifying the basic settings

Dialog page CAM Plan

Activate the **Project** → **Activate CAM Plan** function to display the CAM plan dialog page in the hyperMILL® browser. Select a suitable project template depending on the type of part to be machined.

NC system and frames

Functions on the shortcut menu: Coordinate systems


 **Preselect**: Preselect the marked origin or frame. This can then be selected directly from the list in the job definition (see **ToolFrame**) by clicking the icon . A preselected frame or origin is displayed in **bold** in the browser.

 **Remove preselection**: Cancel the preselection for the selected origin or frame.

Tool management

Define tool

Edit icons in the Tool type dialog

	<p>Optimize barrel cutter parameters.</p> <p>Optimize parameters of barrel cutters based on surface and curve information so that the surface to be machined is completely machined as required.</p>
---	--

Define tool in a job

Cutting profile

Technology data defined for the tool used in the job is displayed in the cutting profile.

Tools included in a tool series




Profile mode: The cutting profile parameters are defined in the technology template of the tool series. For information on creating a tool series, see section **OPEN MIND tool database** → **Tool series and Technology templates**.

Select the **Static (compatible)** profile mode if a type of cut suitable for machining is available for the tool. Then select this under **Type of cut**.

Type of cut: The type of cut is taken from the series definition of the tool that is defined in the tool database for the material used.

Select the **Manual** profile mode if no suitable type of cut is available for the tool. The parameters of the cutting profile correspond to the standard parameters of a tool that is not included in any tool series. When switching from profile mode **Static (compatible)** to profile mode **Manual**, the parameters of the previously set type of cut are transferred to the cutting profile.

Icons in the Tools dialog

	<p>Available if a machine with turret for turning jobs is defined in the job list. Only the mounted tools are displayed in the tool selection list.</p>
	<p>Available if a machine with turret for turning jobs is defined in the job list. All defined tools are displayed in the tool selection list.</p>
	<p>Get preselection: Preselect a frame or origin from the list of frames/origins for job definition.</p>



Structure CAM project

CAM Plan

Der CAM-Plan beinhaltet Funktionen, die den Ablauf des Programmierens vereinfachen. Vom Analysieren des Bauteils, Anzeigen und Reparieren möglicher Flächendefekte bis hin zum automatischen Erzeugen von Features und für das Programmieren notwendiger Geometrien. Zusätzlich stellt der CAM-Plan eine Plattform für weiterführende Technologien und mehr Interaktivität beim Programmieren dar. Der CAM-Plan ist eine Dialogseite im *hyperMILL*®-Browser, unterteilt in einen Strukturbaum (im oberen Teil) und die Bereiche **Auswählen** und **Aufgaben** (im unteren Teil).

After activating the CAM plan in the *hyperMILL*® settings (CAM menu **Setup** → **Settings** → **hyperMILL settings** → **CAM Plan** → **Activate CAM Plan**), select a predefined project template. Depending on which template was selected, the CAM plan contains different functions and tasks.



Use the  /  icons in the structure tree to control the visibility of elements.



To avoid possible issues during programming, all the tasks offered should be carried out.

General (complementary): Control the visibility of all geometries not selected as a part using the  /  icons.

Part: Select the surfaces of the part to be programmed without unnecessary additional surfaces or hole capping. Then go through the tasks step by step to prepare the part completely.

Remove duplicate surfaces: Check the part for double surfaces and remove them if present.

Check surfaces: Check the selected surfaces for possible defects that could cause problems in the subsequent programming process. If a damaged surface is found, the first step is to attempt to repair the surface automatically. In the next step, a new layer is created, the corresponding surface is moved there and removed from the selection made in **Part**. Ensure that this or any newly constructed surface is included in the part selection again.

hyperMILL® converter check: Check whether the selected part can be converted by the converter.

Check for gaps: Check whether there are gaps between the selected surfaces. If gaps are found, a layer is automatically created upon which the gaps are displayed using curves and points. Decide whether the gaps should be corrected or not. Skip this task by deactivating the corresponding option at **Computable task**.

Orientate surfaces: Check whether the selected surfaces are correctly orientated. After starting the task, the corresponding CAD function opens automatically and aligns the surfaces. Manually align any surfaces that are not aligned correctly. The corresponding surfaces are displayed in a dark color.

Hole recognition / add capping: This task starts the Feature mapping based on the selection made for the **Part**. Capping surfaces are automatically created for the recognized holes, a new layer is created and the surfaces are moved there. Skip this task by deactivating the corresponding option at **Computable task**.

Create milling area(s): The milling areas required for programming are created automatically. Depending on the selected project template one or two milling areas are created. If a template "with holes" has been selected, two milling areas are created. One milling area that contains the part and the holes and one milling area that contains no holes but the capping surfaces. If the "without holes" template has been selected, only one milling area is created.

HPM Milling: Select all surfaces that are relevant as product surfaces for 3D finishing and create topology information from them. The curves generated represent the exact topology information of the product surfaces. They are saved internally and used for the point distribution of 3D finishing.

Create topology information

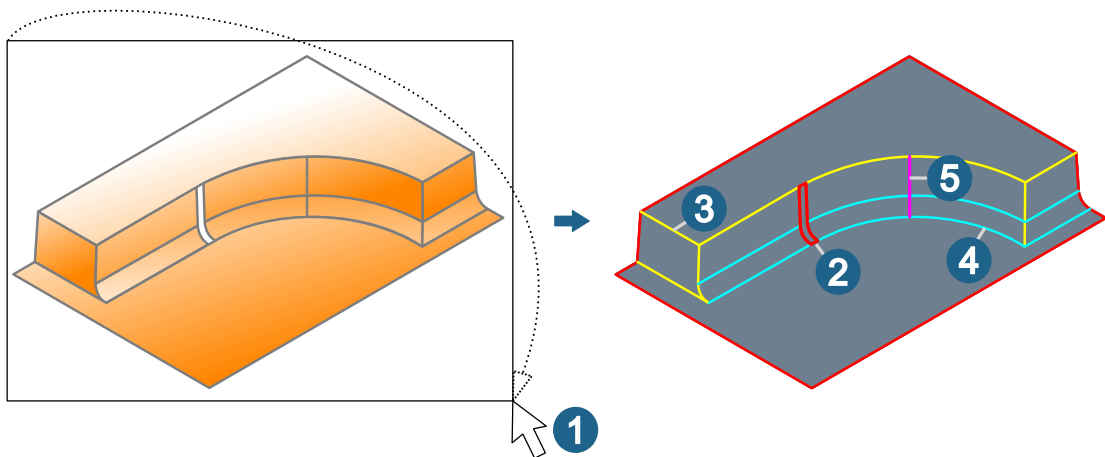


Generate topology curves for 3D finishing.

hyperMILL® browser → CAM Plan → HPM Milling → Create topology information

Select

Shapes: Select entities ①.



Invert orientation: Invert the orientation of the faces of the shape.

Mapping

Select the continuities to be analyzed: **Gap** ②, **Sharp** ③, **Curvature** ④, **Tangential** ⑤. The number of curves found is displayed. The results are marked with the colors displayed in the dialog. The colors cannot be changed.

Sharp edges

Search for all, only convex or only concave edges.

Reference values

Tolerance: Enter a value for the permissible deviation in the position for the size of the gap.

Angular tolerance (degrees): Enter a value for the maximum permissible deviation of the tangency of the transition.

Curvature difference (%): Enter a value for the maximum permissible deviation of the curvature of the transition.

Deburring: Select all surfaces as edge surfaces that are relevant for deburring the part and generate the edge geometries for deburring. These curves are saved internally and used for the toolpath calculation of the cycle **5X Deburring**.

Kantengeometrie erstellen

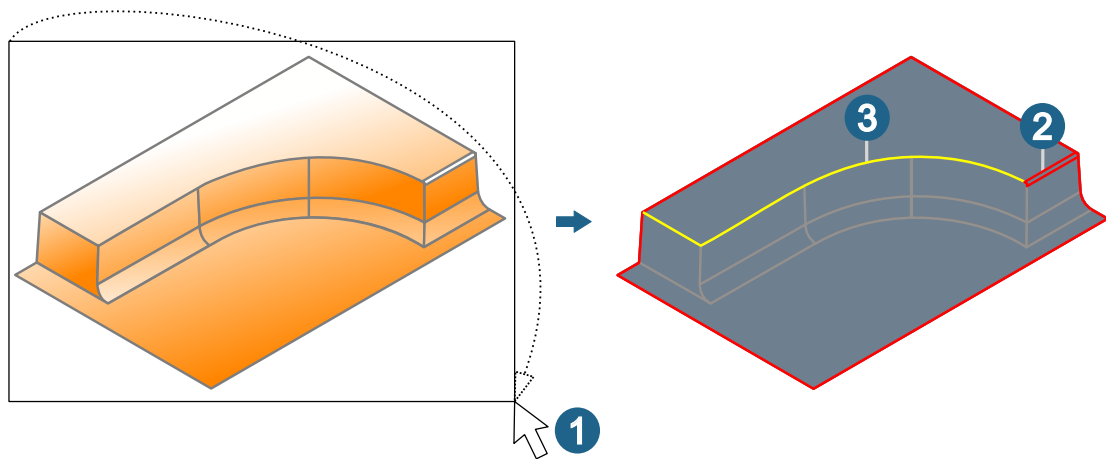


Konturen für das Entgraten ermitteln.

hyperMILL®-Browser → CAM-Plan → Entgraten → Kantengeometrie erstellen

Auswählen

Formen: Elemente auswählen ①.



Zuordnung

Die zu untersuchenden Stetigkeiten auswählen: **Lücke** ②, **Scharf** ③

Die Anzahl der gefundenen Kurven wird angezeigt. Die Ergebnisse werden mit den im Dialog angezeigten Farben kennzeichnen. Die Farben können nicht geändert werden.

Referenzwerte

Toleranz: Ein Wert für die zulässige Abweichung in der Position für die Größe der Lücke eingeben.

Max. Entgratwinkel (Grad): Den maximalen Winkel zwischen zwei Flächen eingeben, für die eine zu entgratende Kante erzeugt werden kann. Um alle möglichen scharfen Kanten zu erzeugen, 180 Grad eingeben.

Werkzeugdurchmesser: Einen Durchmesser eingeben, der gleich oder kleiner ist als der kleinste Werkzeugdurchmesser, der zum Entgraten verwendet werden soll.

The Job list dialog

Job list settings

Compound job IDs/Job IDs: Use **Start** to define the start ID and **Increment** to define the incremental increase of the start ID for compound jobs/jobs.

Probing

Create logs for CAD import: Visualize the measured values generated by the **3D Point Probing** cycle in the CAD system (via the **Probing** tab). Various filter functions are available. For further information, see the product documentation for the CAD system.



The **Create logs for CAD import** function is executed via the VIRTUAL Machining Center. The prerequisite for this is a corresponding Virtual Machine that is activated in the **Job list** → **Postprocessor** → **Machine** → **Machine administration** dialog and is available for selected machine controls. Please contact your OPEN MIND dealer to arrange this.

Postprocessor



* **Turret setup:** If a Virtual Machine with support for turning jobs with a turret is defined as a machine (see **Machine administration** → **Edit** → **Machine properties** → **Virtual Machine Directory**), this is symbolized by an icon. Click the icon to open the **Turret setups administration** dialog to assign the required **Names** and, if required, a **Comment** for this machine configuration. Click the **New** button to define a new turret setup. The following functions are also available: **Edit...**, **Delete...**, **Copy...**, **Export...**, and **Default...**



Simulate machining


Machining planner





Start the VIRTUAL Machining Center in **Machining planner** mode and equip a turret for turning jobs with graphical support. No toolpaths are loaded. The function is available if, in the job list (**Postprocessor** dialog page) a corresponding Virtual Machine has been loaded.

	In the VIRTUAL Machining Center, switch to the Configuration operating mode and
	open the Turret setup dialog.

Turret setup

Both turret holders and NC tools from the tool database or from *hyperMILL*® can be imported for turret setup (1). Double-click in a line and then click on the button (2) to rotate a turret holder and an NC tool around the Z-axis.

Station	Name	NC-Tool mounts	Shift X	Shift Y	Tool reach	Rotation Z
1	Turret holder_01	1	0	0		0
1.1	Drilling tool_01	1	0	0	120	
2	Turret holder_02	1	0	0		90 
2.1	Roughing	1	0	0	45	
3	Turret_holder_03	1	0	0		0
3.1	Drilling tool_02	1	0	0	50	
4	Turret holder_04	1	0	0		0
4.1	Finishing	1	0	0	45	
5						
6						

-  Add turret holder from Tool Database
-  Add turret holder from hyperMILL
-  Add NC-Tool from Tool Database
-  Add NC-Tool from hyperMILL

Station: Position of the tool mount on the turret.

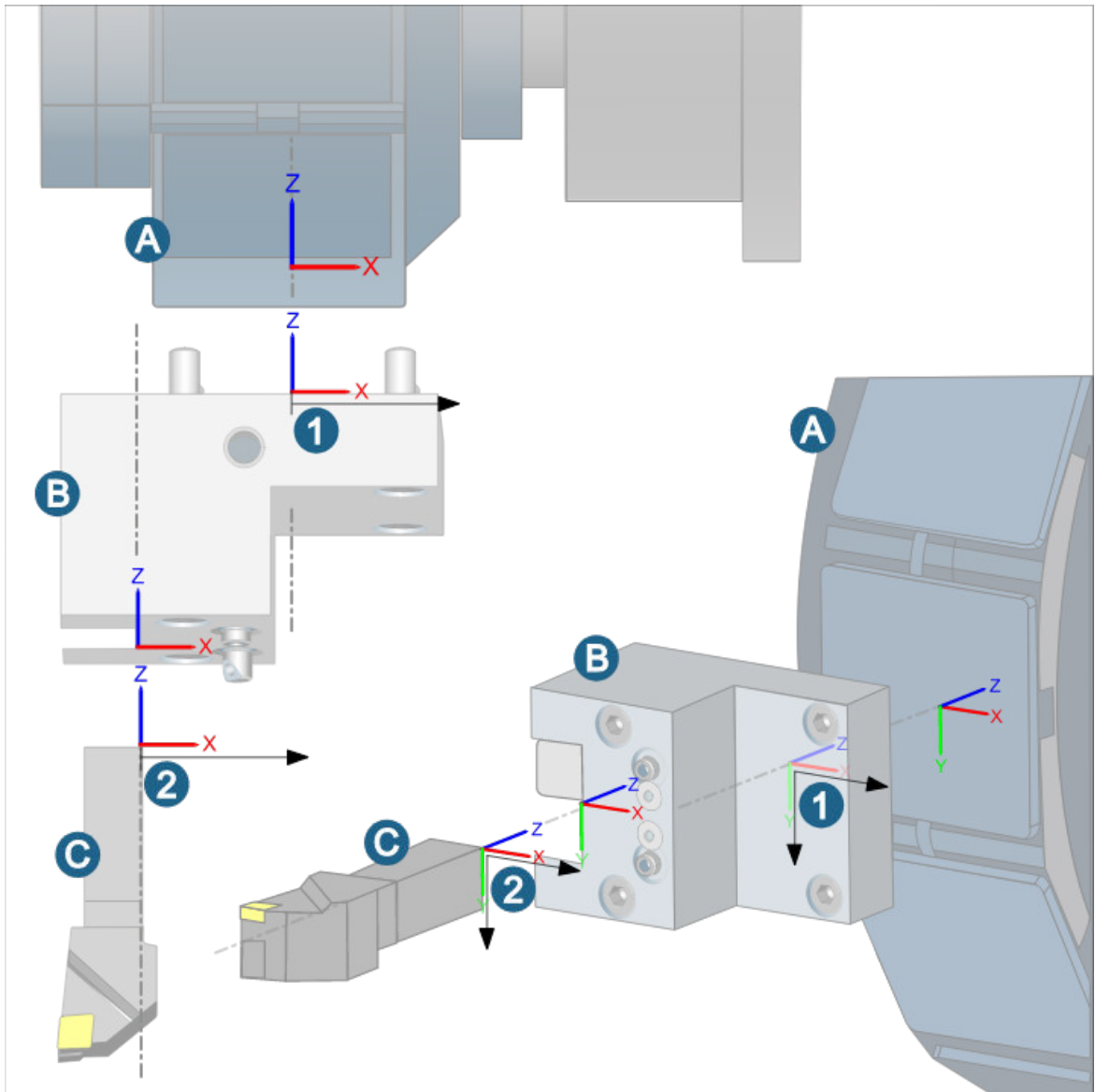
Name: Name of the static holder/NC tool in the location.

NC-Tool mounts: Number of tool mounts of the static holder.



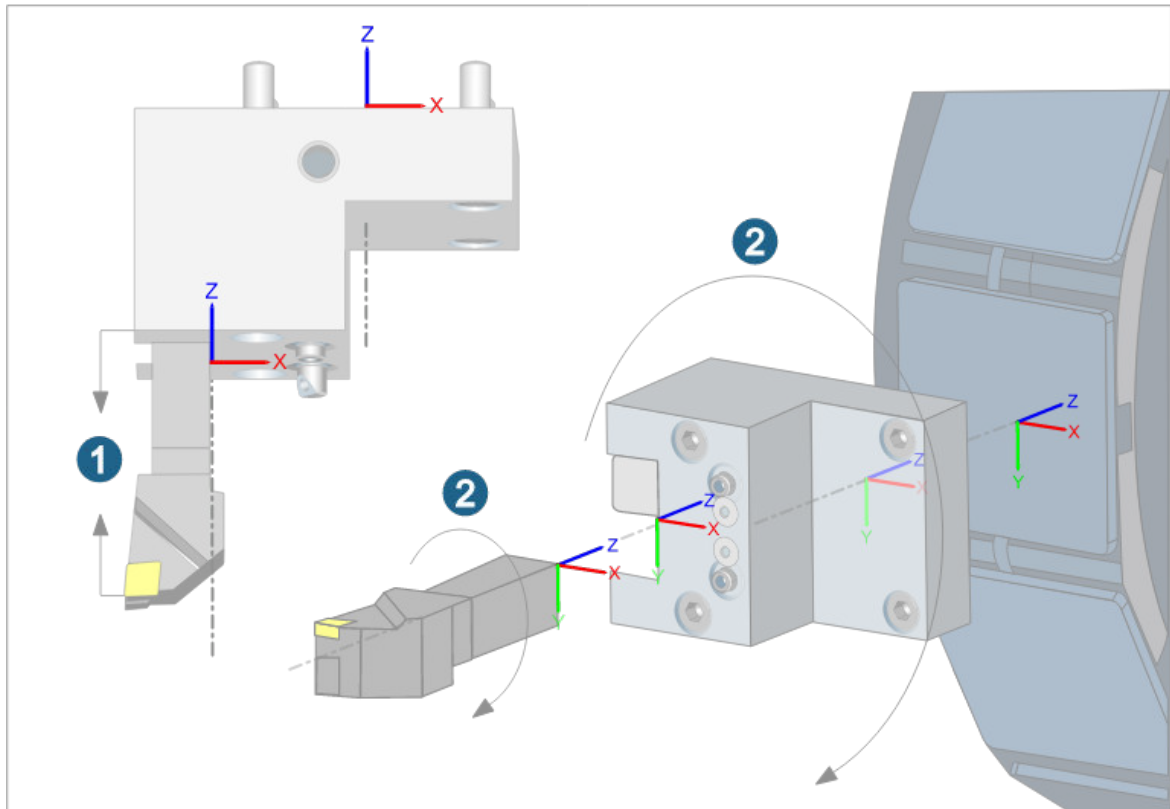
Shift X / Shift Y: Offset between the coordinate system of the **Machine side coupling (1)** and the **Work piece side coupling (2)** of the static holder in the X or Y direction.

(A) Turret, (B) Turret holder, (C) NC tool.



Tool reach: (1) Section of the NC tool (measured up to the tool tip) that sticks out of the turret holder.

Rotation Z: (2) Rotation of the NC tool or the turret holder around the Z-axis. The turret holder is rotated in 90° steps and the NC tool in 180° steps.



hyperMILL additional functions

Rest material display



Display remaining rest material for visual analysis.

CAM browser → Jobs → Shortcut menu → Utilities → Rest material display

2024

Display remaining rest material for visual analysis. A comparison of the pre-calculated current machining status with a workpiece is carried out and the differences are displayed. A current stock chain is required for this.

Color/rest material thickness

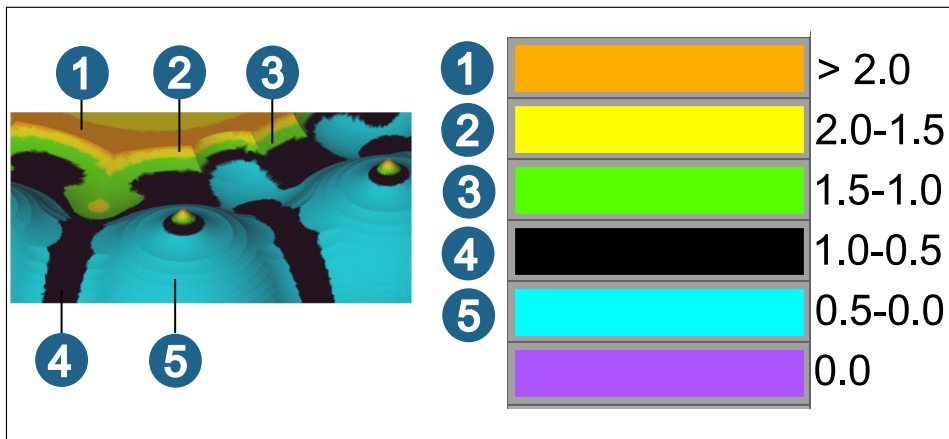
Color: Double-click with the left mouse button on the color field in the table to open the dialog for changing the color of the interval.

Rest material thickness: Double-click with the left mouse button on a table row to change a value of the interval limit.

Limits

Minimum rest material that is taken into account in the rest material display. Maximum rest material that is taken into account in the rest material display. Enter **Min.** and **Max.**

Example: Default values for color and rest material thickness



Tolerance

Model geometry: Value for the accuracy of the rest material display.

Update: Update the display of the rest material.

More options

Number of intervals: The rest material is displayed in five intervals. A higher value increases the fineness of the display.

Exact computation: Refine the calculation of the rest material display. The calculation is performed automatically in relation to the size of the workpiece.

Turning

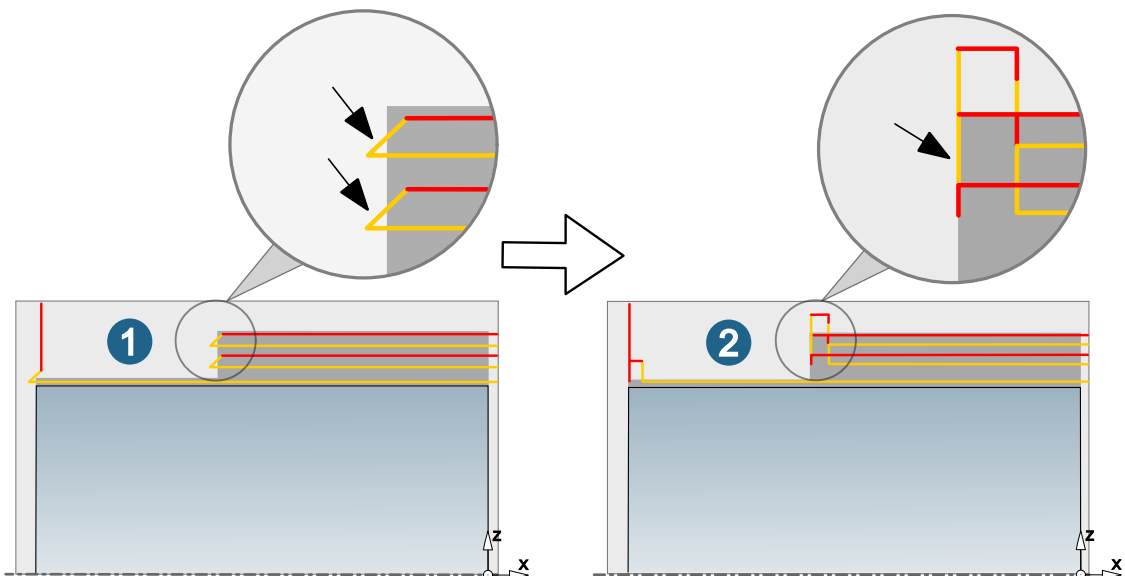
Roughing

Strategy

Options

Ring removal: Remove ring chips from material breakthroughs. Finished surfaces are protected and process reliability is improved. The function is executed instead of the defined macro, and only if there are material breakthroughs. It is supported for the **Axial roughing** strategy in combination with all cutting side options, and for the **Radial roughing** strategy only in combination with the **Plane machining** function.

(1) **Ring removal** is activated, (2) **Ring removal** is not activated.



High Performance

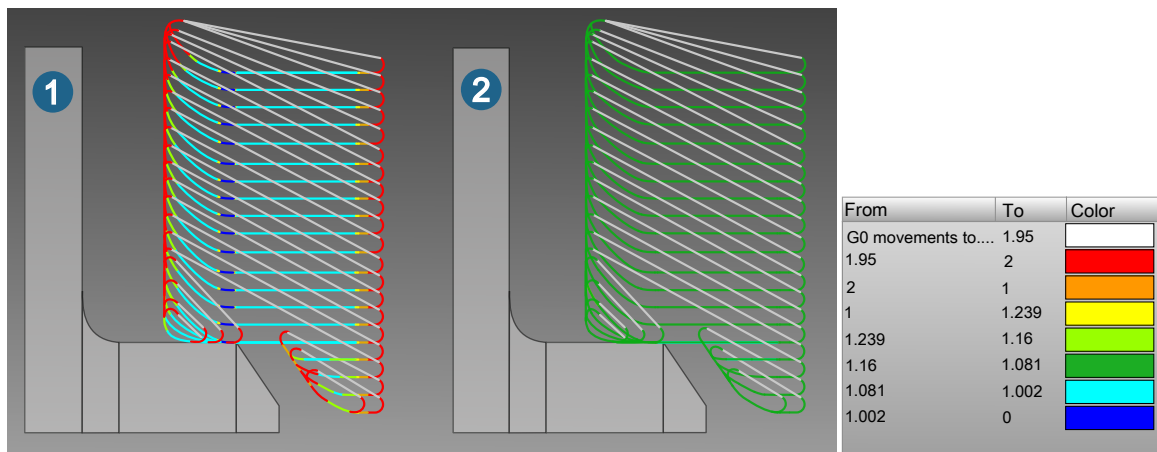
Chip thickness control

The **Chip thickness control** function ensures that the chip thickness remains as constant as possible during machining, thus guaranteeing reliable chip breaking during lead in and retraction and at shallow cutting depths.

Use the two parameters **Min. feedrate** and **Max. feedrate** to define the range for adjusting the feedrate. The target chip thickness is calculated from the feedrate of the tool (see cutting profile of the tool).

1) **Chip thickness control** is activated, (2) **Chip thickness control** is not activated.

Feedrate analysis with the function **CAM** → **Analysis toolpath** → **Feedrates by color map**.

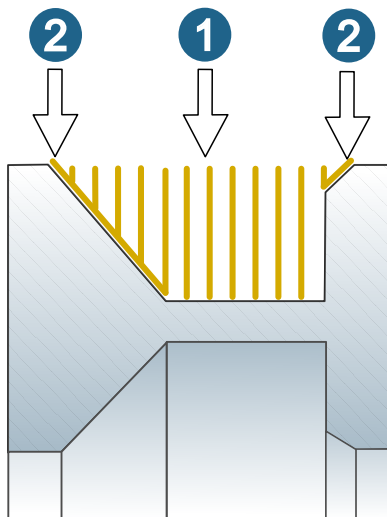


Grooving

Strategy

Options

Finish path: (2) Removal of machining marks from the previous roughing operation (1).





Drilling

Optimized deep Hole Drilling (new)

Process

The process sequence of the tool is controlled in eight phases. For each phase, the **Spindle** orientation, **Feedrate**, spindle speed (), **Coolant**, and **Dwell time** can be defined.

Phases

Spindle: Use and to define the spindle orientation or to define a spindle stop.



For threading into the pilot hole (phase 1) and when retracting to the start of the pilot hole or to the clearance distance (phase 7/phase 8), the orientation of the spindle is set to by default.

Feedrate / : The values for and **Spindle speed** are taken from the definition of the tool in the OPEN MIND tool database. They correspond to the values set in the job definition on the **Tool** tab under **Cutting profile**. To adjust both values, enter the % value in the **Feedrate** and columns.



To manually overwrite the values for **Feedrate** and that are preset in the tool definition, right-click the value in the required **Phase**, select the **Disconnect** option, and enter the required value.

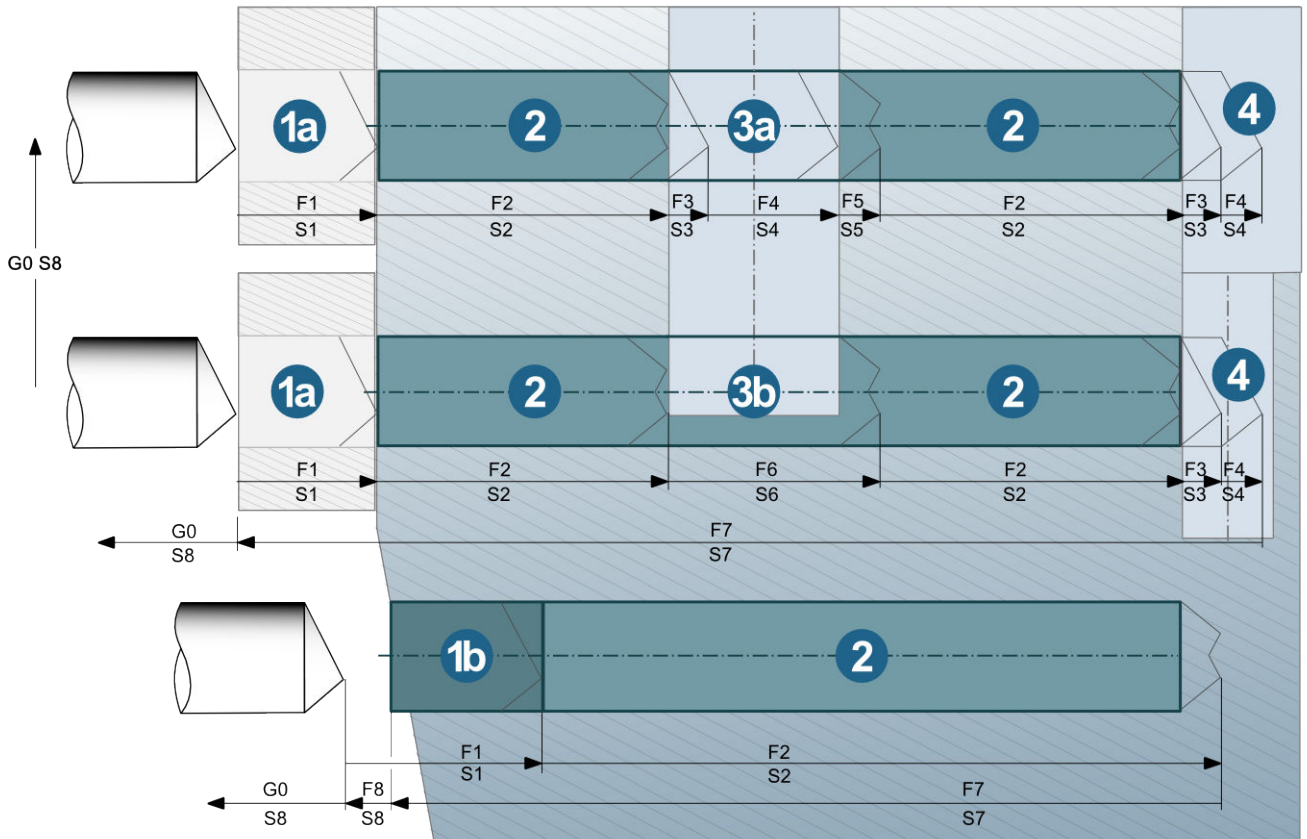
The value is now displayed **in bold**, the definition in the tool and the adjustments in the **Feedrate /** column are **no longer** taken into account.

To establish a new connection between feedrate/spindle speed and % values and to the tool definition, right-click the modified value and select the **Link** option.

Coolant: Define the coolant for each phase.

Dwell time: Define the dwell time in seconds for each phase.

Movement sequence of the tool



Feedrates and spindle speeds assigned to the tool during its movement:

(1a) Guide sleeve: rapid movement up to the start of the guide sleeve; with F1 and S1 until the end of the guide sleeve.

(1b) Pilot hole: rapid movement up to the clearance distance. With F1/S1 until the end of the pilot hole.

(2) With F2/S2 until the start of the crosshole.

(3a) Crosshole: from the start of the crosshole until the end of material removal with F3/S3. For crossholes with F4/S4. From the start of material removal until full material removal with F5/S5. (3b) Crosshole: for continuous material removal in a crosshole: until the end of the crosshole/material removal with F6/S6.

(4) Outlet for a through hole/crosshole) with F3/S3 or F4/S4.

Retract movement: to the clearance distance or the start of the guide sleeve in F7/S7.

Linking movements: between drill holes with F8/G0 and S8 (reduced feedrate/spindle speed).

Additional feedrate settings

Stepwise feedrate increase: Enable the option to gradually increase the feedrate values between F1 and F2, F6 and F2, as well as F4 and F2. The **Length** defines the total length of the stepwise feedrate increase (default = tool diameter * 1.5). The **Number of steps** determines how often the increase takes place in the defined **Length**.

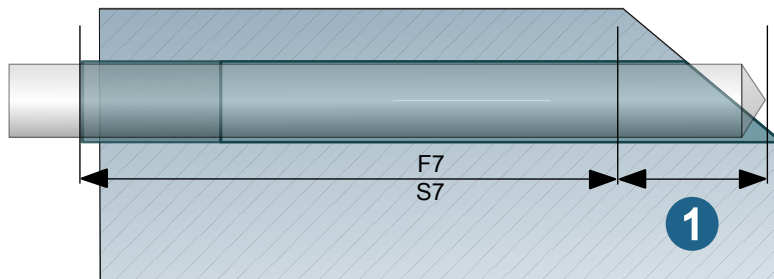
Process: end of hole

Dwell time: Time in seconds for which the drill remains on the hole bottom to cut material free.

Incremental retract: Define a retract movement to retract the tool at the end of the dwell time on the hole bottom in a defined retract length and with a defined **Retract feedrate**. This prevents possible damage to the tool during direct retraction with F7 from the hole bottom.

Retract length: (1) Specify the length of the incremental retract movement.

Retract feedrate: Specify the feedrate of the incremental retract movement.



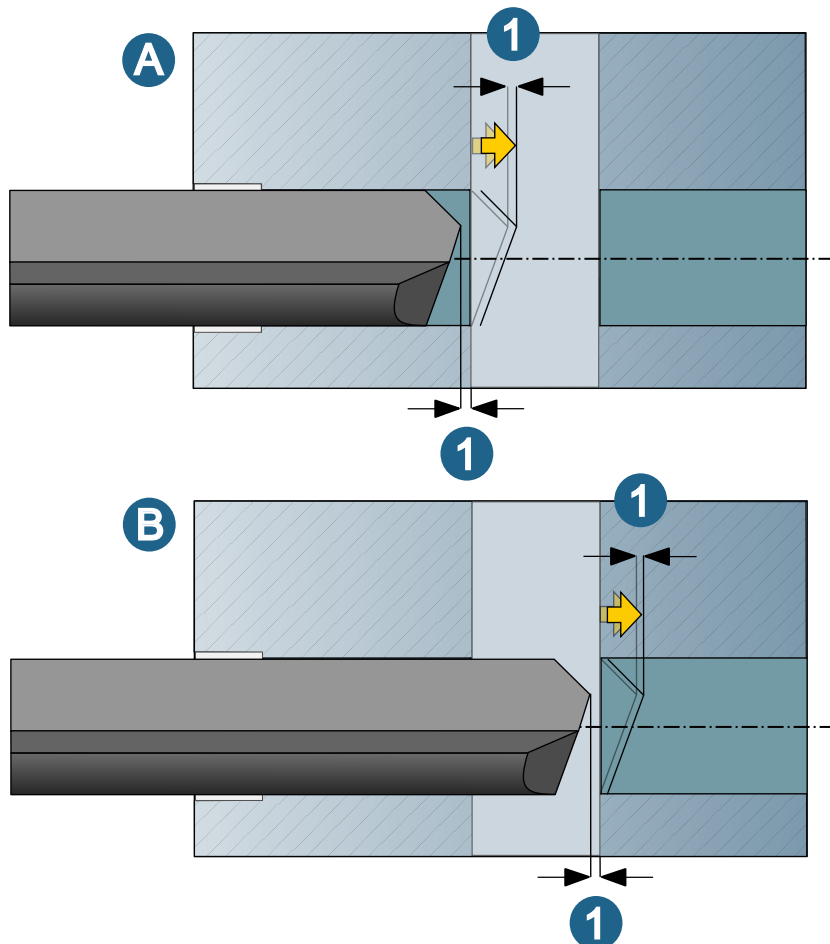
Parameters

Cross hole safety

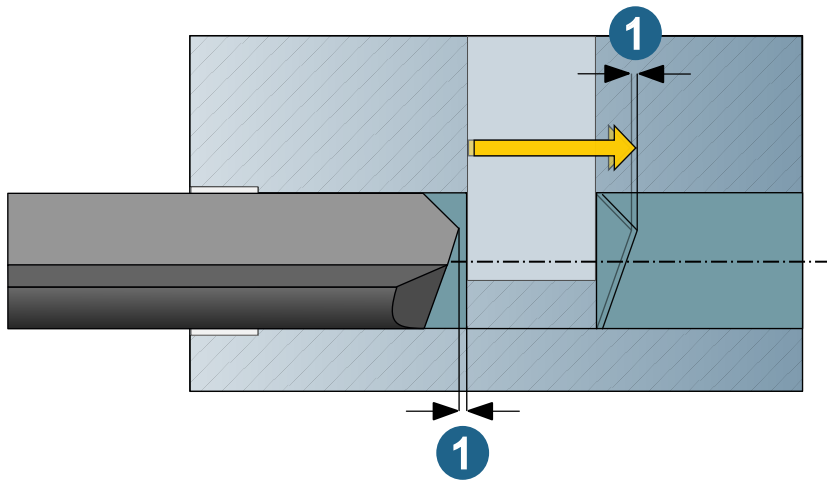
Clearance distance: To ensure a safer drilling process, the phases 3 and 5 (cross drilling without material removal) or the phase 6 (cross drilling with material removal) are each extended at the beginning and end by the amount of the clearance distance (1).

Examples:

Cross drilling without material removal: Phases 3 (A) and 5 (B) start earlier and end later. Phase 4 is shortened by the corresponding amount. (1) = Clearance distance.



Cross drilling with material removal: Phase 6 starts earlier and ends later. (1) = Clearance distance.



Machining parameters

Chip break: Activate if a chip break is to take place.

Initial pecking depth: Infeed depth in the first drilling stroke, that is, when entering the hole (without pilot hole and guide sleeve).

Pecking depth: Infeed depth of the drilling strokes performed after the initial drilling stroke.

Min. infeed depth: Minimum infeed depth in a drilling stroke if a **Reduce value** is defined.

Reduce value: Amount by which the pecking depth is reduced for each drilling stroke (after the initial infeed).
 $Z_{n+1} = Z_n - \text{Reduce value}$.

Incremental retract: (A) Incremental retract after each drilling stroke. (1) Initial pecking depth. (2) Pecking depth. (3) Retract value.

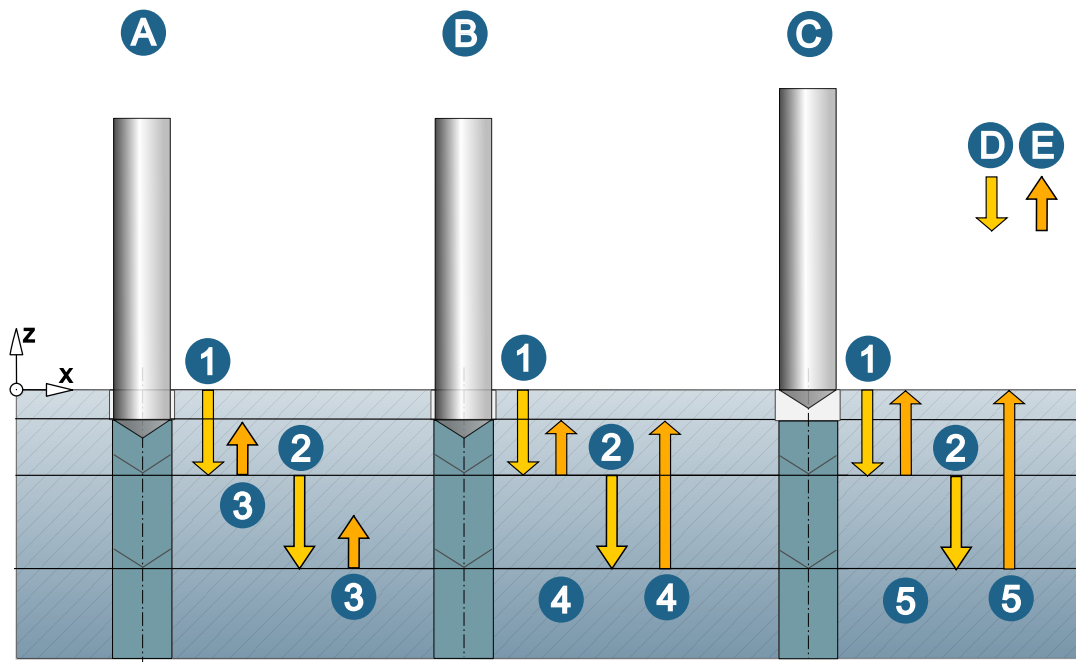
Complete retract → Retreat to pilot depth: (B) Complete retraction after each drilling stroke to pilot depth. Make sure that the tool does not leave the hole completely, as there is a risk of tool breakage. The tool remains in the hole with **Tip angle** length + additional clearance.

(1) Initial pecking depth. (2) Pecking depth. (4) Retract to pilot depth.

Complete retract → Retreat to top: (C) Complete retraction after each drilling stroke to the top edge of the hole. Make sure that the tool does not leave the hole completely, as there is a risk of tool breakage. The tool remains in the hole with **Tip angle** length + additional clearance.

(1) Initial pecking depth. (2) Pecking depth. (5) Retract to top.

(D) = Lead in feedrate. (E) Retract feedrate.



Retract feedrate: Feedrate for retraction of the tool during the chip break (without consideration of cross holes).

Lead in feedrate: Feedrate for re-entry of the tool during the chip break (without consideration of cross holes).

Setup

Model

Definition of the collision checked part of the CAD model. For further information, see section [Collision check preparations](#).

Additional surfaces: Temporary safety surfaces to avoid unnecessary rapid travel movements.

For information on the holder/spindle clearance, see section [Check tool](#).

NC parameters

Accuracy : Defines the quality of the model (mesh) against which the check takes place.

Stock

Use the **Enable stock** option to optionally define a stock that is required to detect breakthroughs and/or cross holes.

2D Machining

Contour Milling on 3D Model

Contours

Technology

Feedrate options

Edge control: Optimize the feedrate at edges. Two parameters limit the reduction/increase of the feedrate: **Min. feedrate (factor)** and **Max. feedrate (factor)**. The factor value refers to the standard feedrate at the reference point of the tool and controls the feedrate at the contact point. Permissible values for the **Min. feedrate (factor)** are between 0 and 1, for the **Max. feedrate (factor)** between 1 and 10.

Chamfer Milling on 3D Model

Contours

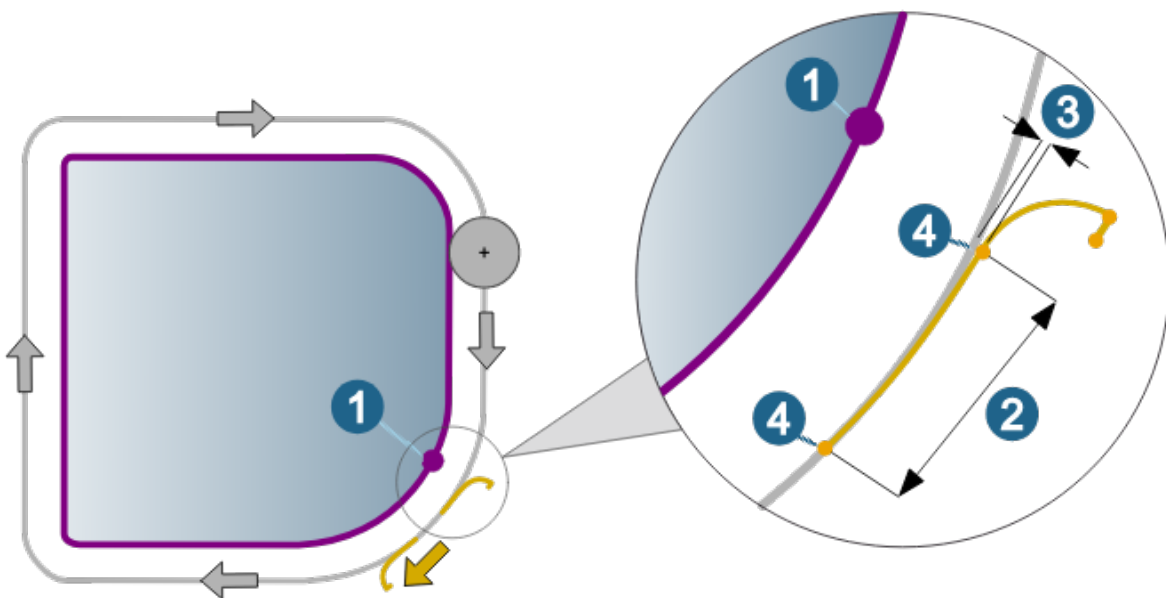
Options

Overlap off: Select this option when the tool should not perform an overlap on the closed contours.

Smooth overlap: Select if the tool is to perform a smooth overlap on the closed contour. Define the **Length** (2) of the overlap. Use **Distance** (3) to define the maximum lateral distance of the tool from the model during the overlap.

(1) Start point

(4) = Toolpath points: are shown when this is activated on the **General** dialog page (in the **Toolpath visualization** area).



Standard: Select this option when the tool should perform an overlap on the closed contours. Define the **Length** of the overlap (2).

3D Machining

Profile Finishing

Setup

Tolerance parameter

Tolerance use cases: Use the parameters of the tolerance use cases to ensure the best possible tolerances depending upon the individual machining situation. The best settings are automatically used based on the tool diameter and the selected **Use case** (**Semi finishing**, **Finishing**, **Fine finishing**, **Micro finishing**).

For **Mode** specify whether the toolpath is calculated in **Standard** mode or in **High-precision surface mode**. The available use cases represent the typical machining steps during a milling operation.

High precision surface mode: Use when maximum accuracy is required to calculate the toolpath. The calculation of the toolpath is not based on the faceted model geometry, but on the real surfaces of the model. Only available for **Ball mill**.

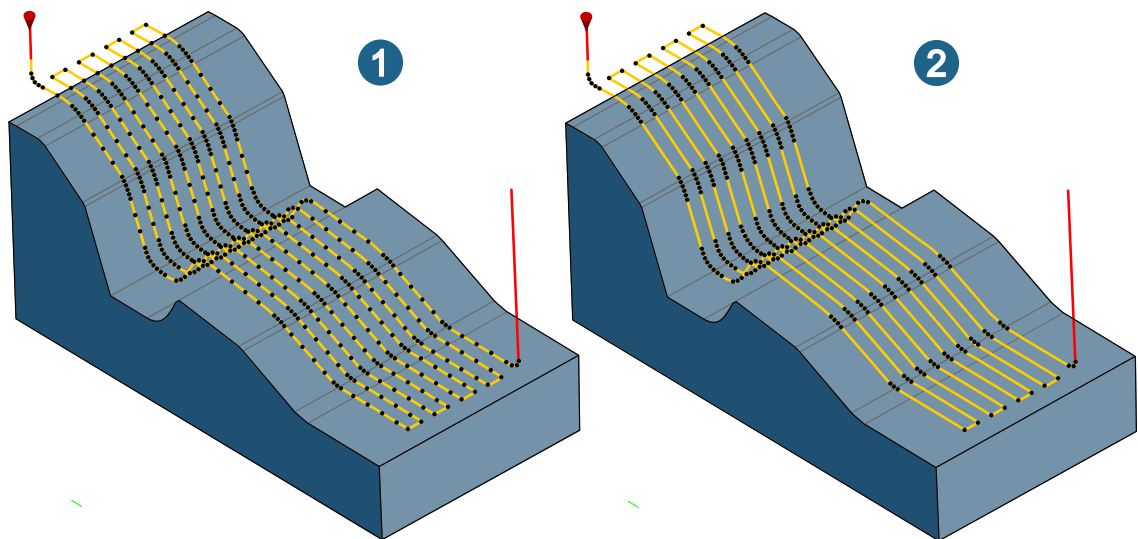
True Shape point distribution: Activate to redistribute points and guarantee an even and machine control-friendly distribution of points.



To guarantee the best possible point distribution, use the **CAM Plan** function. The machining cycle automatically takes into account the topology information that was created with the **HPM-Milling** task in the CAM plan.

Filter points: Points that are located on straight toolpath sections, for example a planar surface, and are therefore not required, are filtered out.

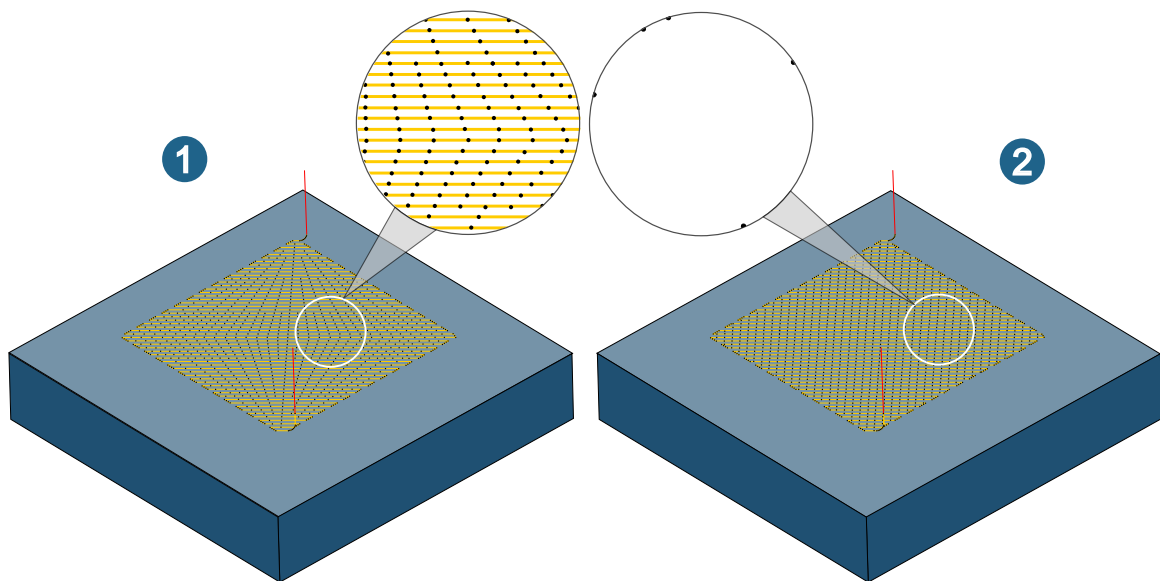
Function is not activated (1), Function is activated (2)



Please note that no or fewer points are not always the best solution for all machine controls. For more detailed information, please contact the respective machine manufacturer.

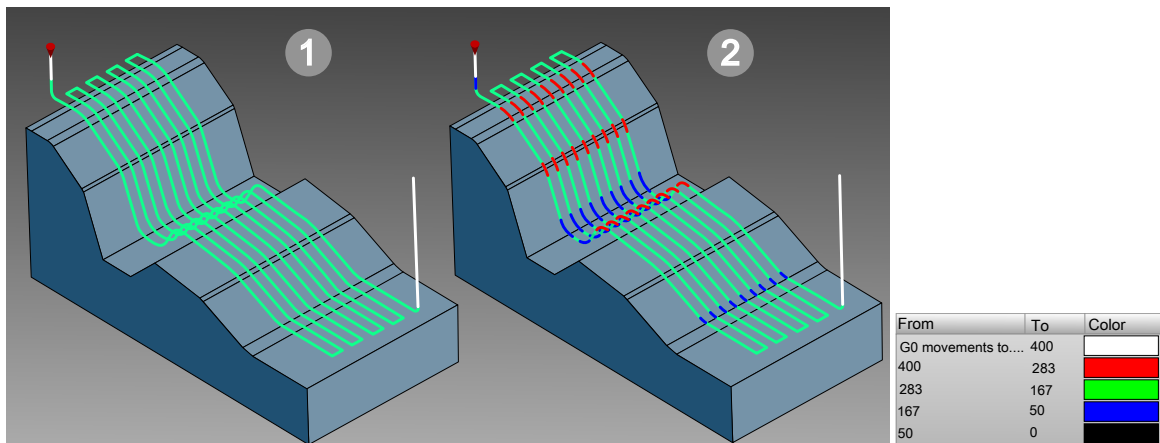
Cross align points: Align the points across the entire toolpath perpendicular to each other (i.e. at right angles to the toolpath direction).

Function is not activated (1), Function is activated (2)



Feedrate adaption: Activate to automatically adjust the feedrate depending on the curvature of the part. The feedrate is reduced for concave curvatures (1) and increased for convex curvatures (2). This means that the real contact feedrate of the tool on the part surface remains constant.

Function is not activated (1), Function is activated (2)



3D path compensation: Available for the **Ball mill** tool type. Enables various NC controls to machine the model with a smaller or larger tool than actually programmed in *hyperMILL*®. In addition to the X, Y and Z coordinates, the direction vectors I, J and K to the surface contact point must also be known.

Protect edges: When using the CAM plan, edges on the part are treated so that they are not damaged during machining. This is achieved by inserting additional points in areas where sharp edges were previously found in the component using the **HPM Milling** task.



The 3D path compensation function is available only with a specially adjusted postprocessor. Without this adjustment, the NC program cannot correct this output and, as a result, both the component and machine may be damaged if the milling geometry used differs from the milling geometry used to calculate the toolpath. To adjust your postprocessor, please contact your OPEN MIND partner.

Tolerance settings

In the tolerance settings, adjust the parameters of the **True Shape point distribution** and **Feedrate adaptation** functions.

Activate **True Shape point distribution** to guarantee an even and correct point distribution in toolpaths. The parameters **Calculation tolerance**, **Max. chordal error**, **Max. G1 length**, **Min. G0 distance**, **Reference G1 length**, control the arrangement and distances between the points.

Use the **Adjacent sync. ISO** parameter to achieve asynchronous point distribution between individual toolpath segments by entering a (factor) value less than 1.

Function is not activated (1), Function is activated (2)

The **Min. feedrate (factor)** and **Max. feedrate (factor)** define the upper and lower limit for the feed rate adjustment. The factor refers to the standard feedrate defined for the toolpath. Use **Number of steps** to control in how many intervals the increase or decrease of the feedrate adaption is performed.

Z Level Shape Finishing

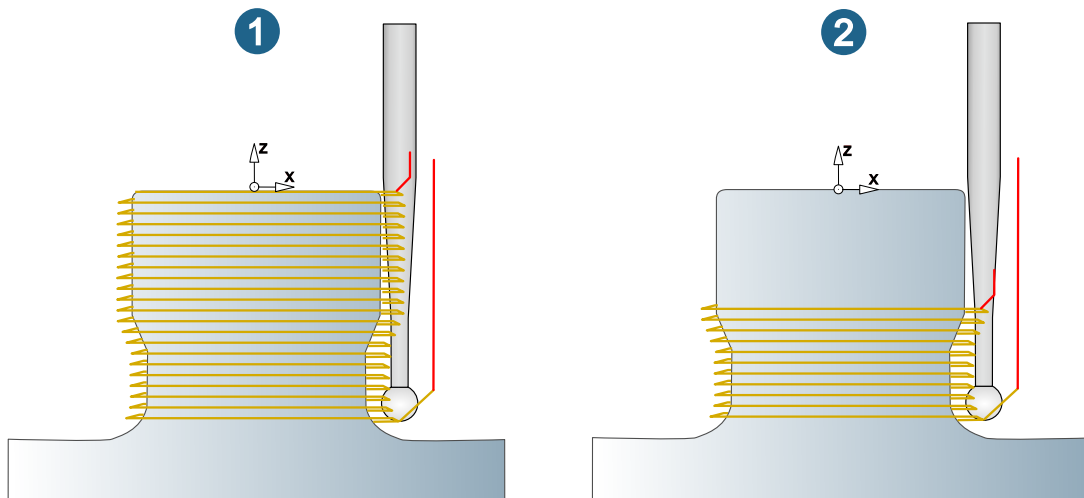
Strategy

Undercut machining

Supported tool types for undercut machining: Lollipop, T-Slot Cutter, and Woodruff Cutter.

Undercut machining: (1) Enable to machine the model completely.

Undercut machining only: (2) Enable to only machine the undercut area.

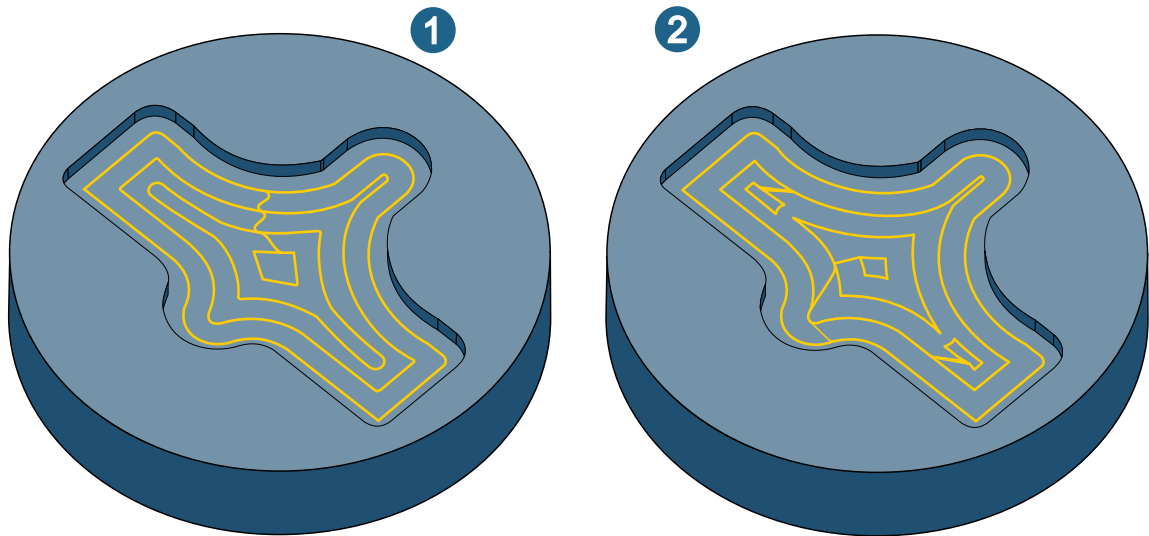


Plane Machining

Parameters

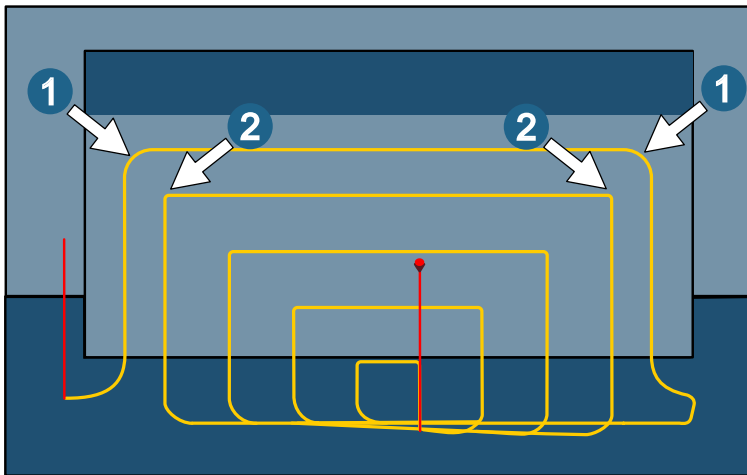
Pocket mode

Two methods are available for calculating the pocket toolpaths. Use the **High feed mode** (1) to realize large side infeeds and machine pockets with islands. Preferably use the **Standard mode** (2) with smaller side infeeds and consequently fewer repositioning movements.



Toolpath fillets

Fillet radius: Abrupt changes in direction within the cutting toolpaths are rounded. The value defined in the user interface is used to round the toolpaths that are in (lateral) contact with the model (1). The toolpaths that are not in (lateral) contact with the model are automatically rounded on the basis of the tool diameter (2).



Cutting edge

Strategy

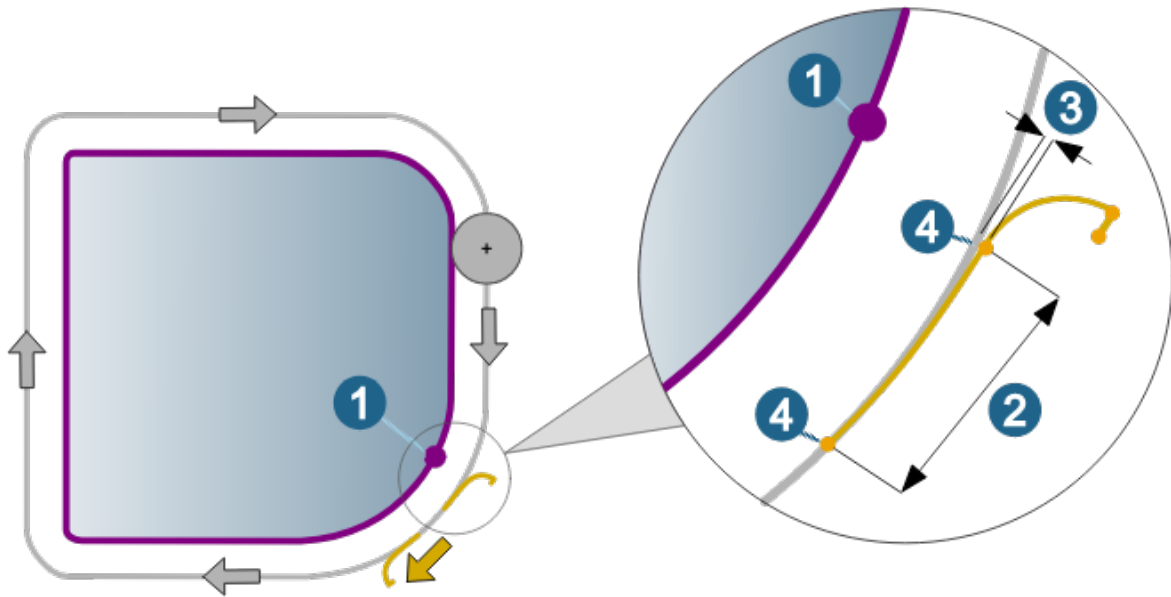
Interior corners

Fillet interior corners: Smooth filleting (rounding) of internal milling paths for pockets or islands.

Reduce corner feedrate: The interior corners are machined at a lower feedrate to protect the tool. The reduced feedrate is defined in the tool database as a technology parameter. This can also be adjusted for a specific material in a cutting profile.

Smooth overlap

Use smooth overlap: Enable if the tool is to perform a smooth overlap. Define the **Area length** (2) of the overlap. Use **Distance** (3) to define the maximum lateral distance of the tool from the model during the overlap.



5X Machining

Cavity Milling

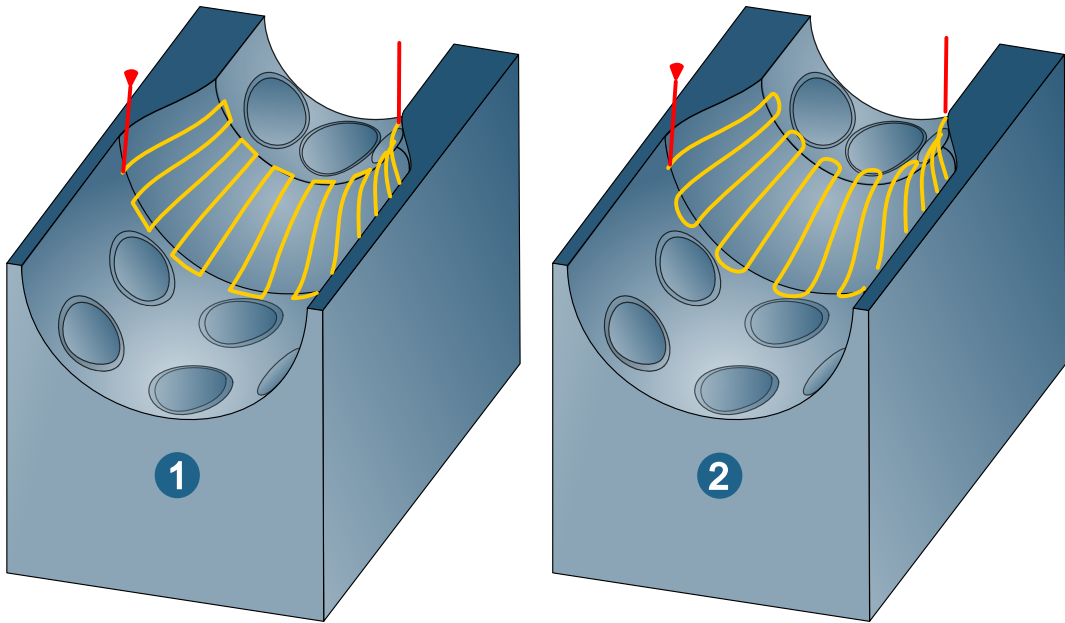
Radial machining

Strategy

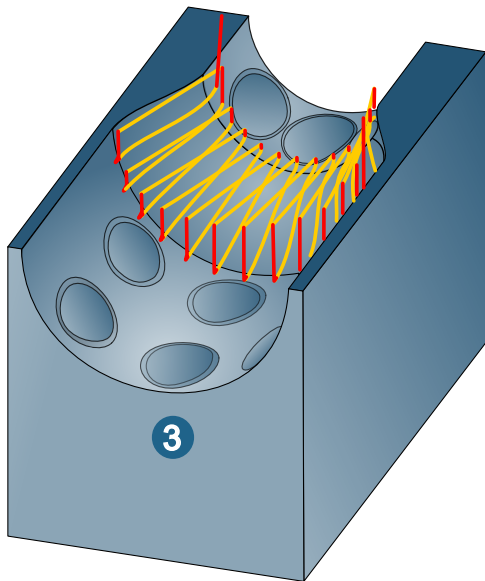
Other infeed options

Use the other infeed options to define the type of horizontal stepover between the end point of one milling path and the start point of the following milling path. The infeed options can be combined with all infeed strategies (Along axis, Offset Normal, ... and so on). In the following figures, the **Along axis** infeed strategy is used to visualize the infeed options.

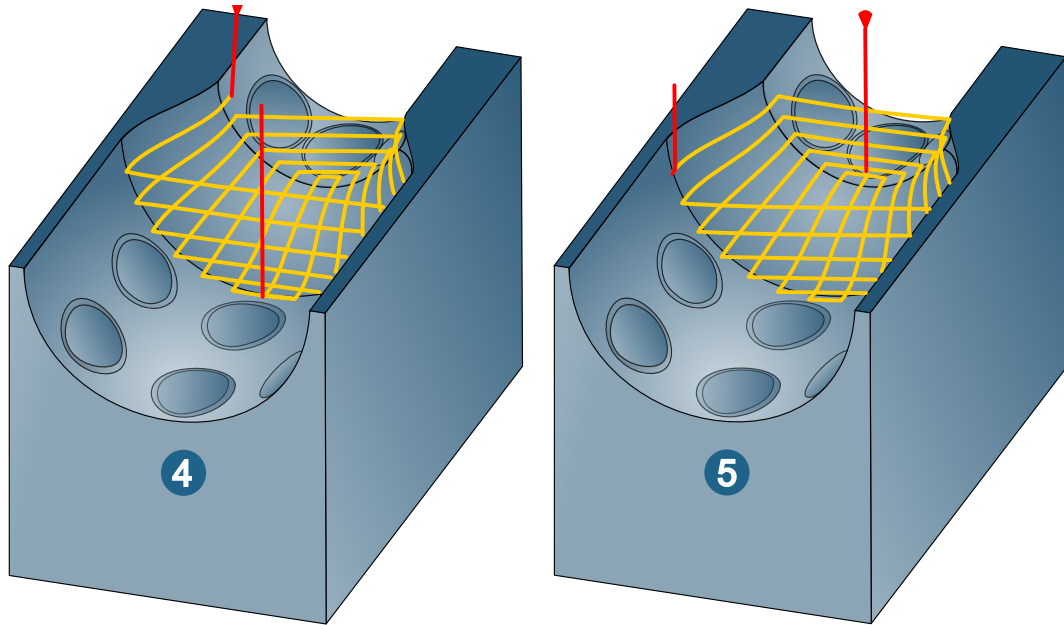
Zigzag direct (1), **Zigzag smooth** (2): The machining direction changes after each passed profile. An approach macro is used for the infeed to the first milling path and a retract macro is used for retraction after the last milling path. The **Zigzag direct** option connects the individual paths directly to each other. The **Zigzag smooth** option connects the individual paths with a fillet.



Oneway: The milling direction remains the same over the entire machining area. The infeed from the end point of one path to the start point of the next path is diagonal.



Oneway Outside-in (4) / Oneway Inside-out (5): The milling direction is the same over the entire machining area. Machining takes place using either climb milling or conventional milling.

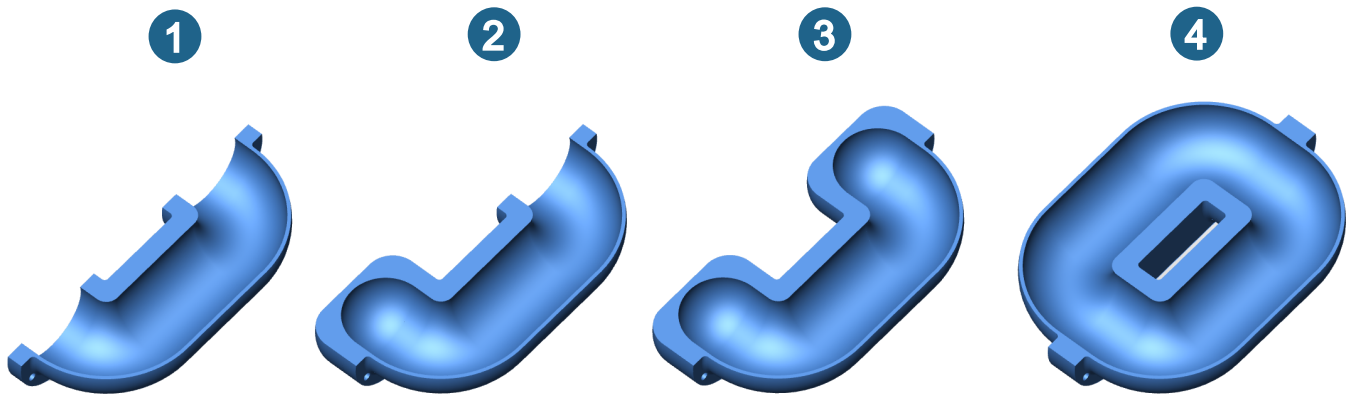


Halfpipe Machining

Strategy

Geometry type

The cycle supports the following geometry types: (1) 2 open ends, (2) 1 open end, (3) No open end, and Infinite (4).



Strategy

Infinite

Graphic	Strategy	Description/advantages
	<p>Across / Oneway outside-in</p>	<p>Plunge machining. Especially for tools with a short cutting edge.</p>



Graphic	Strategy	Description/advantages
	Across / Oneway inside-out	<p>Drawing machining.</p> <p>Small tilt angles result in longer contact on the cutting edge on the one hand and less contact with the tool tip on the other hand.</p>
	Across / Zigzag	<p>Due to the shorter distances between the paths, the total length of the toolpath is shorter.</p> <p>If pull / plunge cut does not play a role.</p>
	Along / Oneway outside-in	<p>Similar to a halfpipe with two open ends, alternating machining takes place for each infeed between both sides with linking paths at the start position.</p> <p>Advantageous if circulating machining is not possible - e.g. due to axis limitations.</p>
	Along / Oneway inside-out	<p>The strategy can be used for roughing, as both sides are machined simultaneously from the outside to the inside. Select Diameter of the tool and Allowance to achieve a sensible path distance between the left/right side. Use Start offset to move the first infeed outwards in order to achieve acceptable cutting conditions.</p>
	Along / Spiral outside-in	<p>First, continuous machining takes place on the selected start side. Then machining switches to the opposite side.</p> <p>Advantageous if machine tool and clamping allow circulating machining.</p>

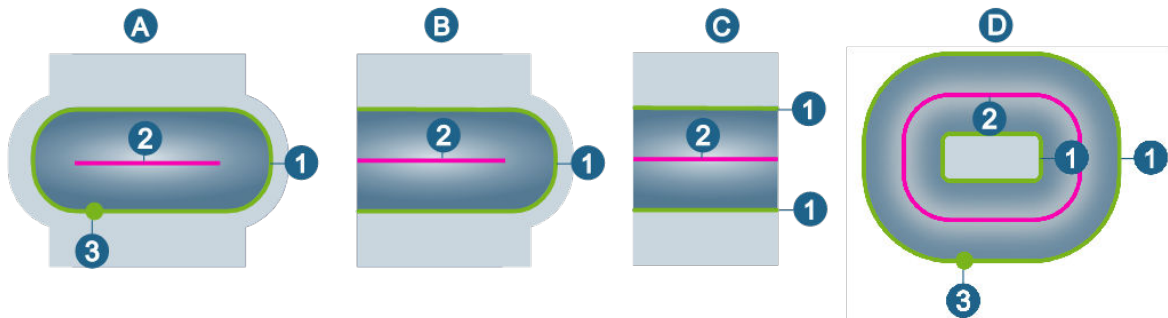
Graphic	Strategy	Description/advantages
	Along / Spiral inside-out	

Boundaries

The selection of **Boundaries** (1) depends on the geometry type of the forming groove. The following are permitted for the geometry type

- **No open end** A closed boundary around the central curve (A)
- **1 open end** An open U-shaped boundary around the central curve (B)
- **2 open ends** Two open boundaries on both sides of the central curve (C)
- **Infinite** Two closed boundaries inside and outside the central curve (D)

(2) = Central curve.



The profile selection has been technologically improved for the **5X Halfpipe Machining** cycle. The selection

- is saved in full and generates an error message if the number of profiles or profile type is incorrect.
- can be subsequently edited to close missing areas or remove elements selected by mistake.

Additive Manufacturing

Strategy

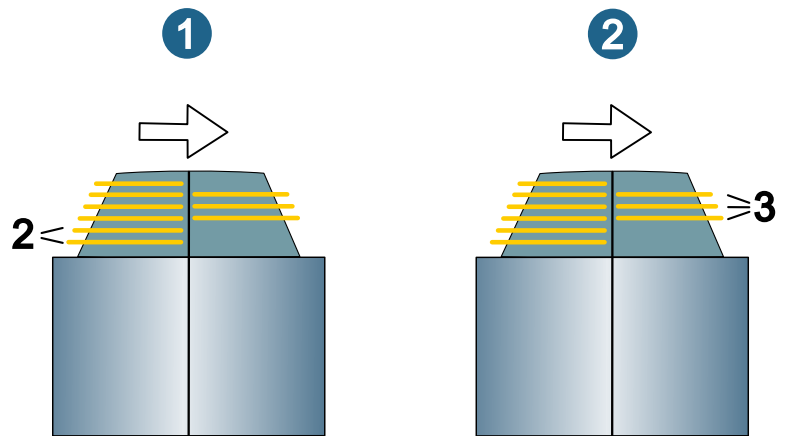
Reference path options

Layer: Group of toolpaths at the same Z depth or at the same distance from the drive surface. The number is independent of whether an infill takes place or not (see options under **Additive** → **Infill**).

Use **Number layers to skip** to define the number of layers that do not generate any additive toolpaths (1) and **Max. number of layers** to define the maximum number of layers that are created (2).

Group: If **Infill** is activated a pocket (including islands - if present) and an open contour are regarded as a toolpath/group. If **Infill** is deactivated, each contour is considered a toolpath/group.

Use **Number paths to skip** to define the number of paths that do not generate any additive paths (1) and **Max. number of paths** to define the maximum number of toolpaths that are created (2)



Tube Milling

Strategy

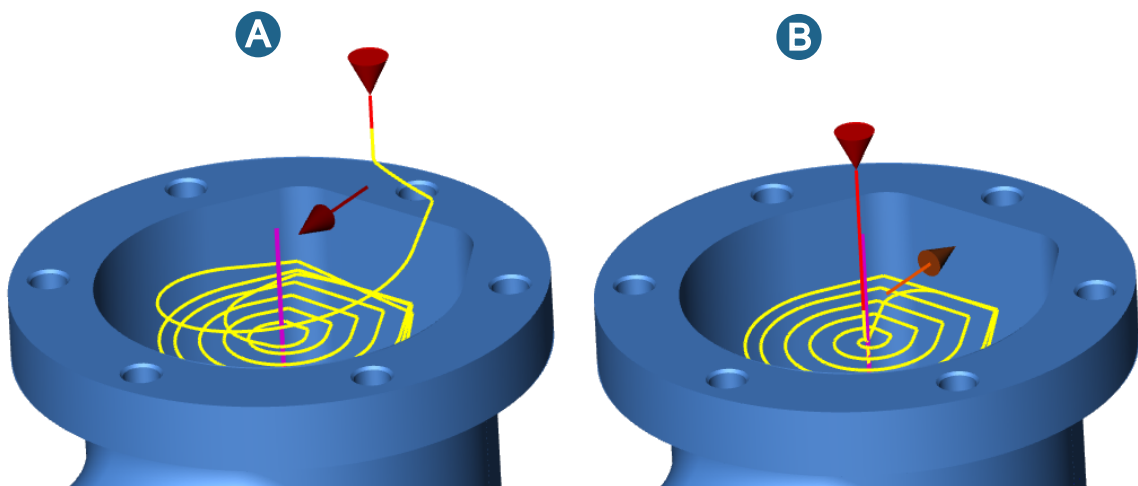
Tube Roughing

Lateral infeed strategy

There are two directions available for the lateral infeed inside each level.

Toward central curve: (A) The tool moves in a spiral along the surfaces of the tube to the next plane and then removes the material in the direction of the central curve.

Away from central curve: (B) The tool moves along the central curve to the next plane and then removes the material from the central curve in the direction of the surfaces of the tube.



If the **Away from central curve** strategy is applied, it must be ensured that the tool can enter along the central curve, e.g. by means of drilling.

Tube Finishing

The following functions are only available in the Tube Finishing cycle.



Tube Rest Machining

The following functions are only available in the Tube Rest Machining cycle.

Multiblade Milling

MB Roughing

5 axes

Lead angle up / Lead angle down

Smooth factor: The smooth factor can be used to limit the length/smoothing of the transition between the global and local tool axis settings. This makes it possible to control the area of the drill or upward cuts for both cutting directions independently of each other.

Not activated: Global smoothing between the setup at all positions that are shown in the UI. The global value is applied at positions without manually specified local value.

Enabled: Changes the lead angle from the global to the local value and back within a length of factor * tool radius. With a small factor, the transition between the specified lead angles can be relatively hard. A larger factor makes the transitions smoother, but can lead to longer drill or upward cuts.

MB Hub Finishing

5 axes

Lead angle up / Lead angle down

Smooth factor: The smooth factor can be used to limit the length/smoothing of the transition between the global and local tool axis settings. This makes it possible to control the area of the drill or upward cuts for both cutting directions independently of each other.

Not activated: Global smoothing between the setup at all positions that are shown in the UI. The global value is applied at positions without manually specified local value.


Enabled: Changes the lead angle from the global to the local value and back within a length of factor * tool radius. With a small factor, the transition between the specified lead angles can be relatively hard. A larger factor makes the transitions smoother, but can lead to longer drill or upward cuts.

Tool database

Define turning tool

Define static holder

Turret holder (static/driven) and **Guide sleeves** are supported.

1. On the **Static Holders** tab, select the **New** → **Turret holder/Guide sleeve** function via the shortcut menu.
2. Under **General** → **Type**, select an option (**static / driven**) for the turret holder, **Name**, and define a **Comment** if necessary. Select a **Manufacturer** from the list, the **Ordering code** is inserted automatically.
3. In the **Technology** area, define the factor values for the speed, feedrate, infeed width and infeed length, as well as the maximum spindle RPM and maximum feedrate.
4. In the **Couplings** area, select the **Machine side coupling** and the **Work piece side coupling**.
5. Under **Geometry** → **Geometry**, click the icon  to start the TOOL Builder to import 3D data for creating a static holder. For further information, see section **Turning tool: Create static holder** in the product documentation for the TOOL Builder.

Create indexable cutter

The indexable cutter tool type is supported as **Indexable high feed cutter** and **Indexable round insert cutter** in hyperMILL®. It is made up of the **Insert** and **Holder** components. An indexable cutter is created in two steps. In the first step, a 3D model is used to generate a rotation contour for the insert and a rotation contour for the holder; in the second step, the two rotation contours are combined to form a tool.

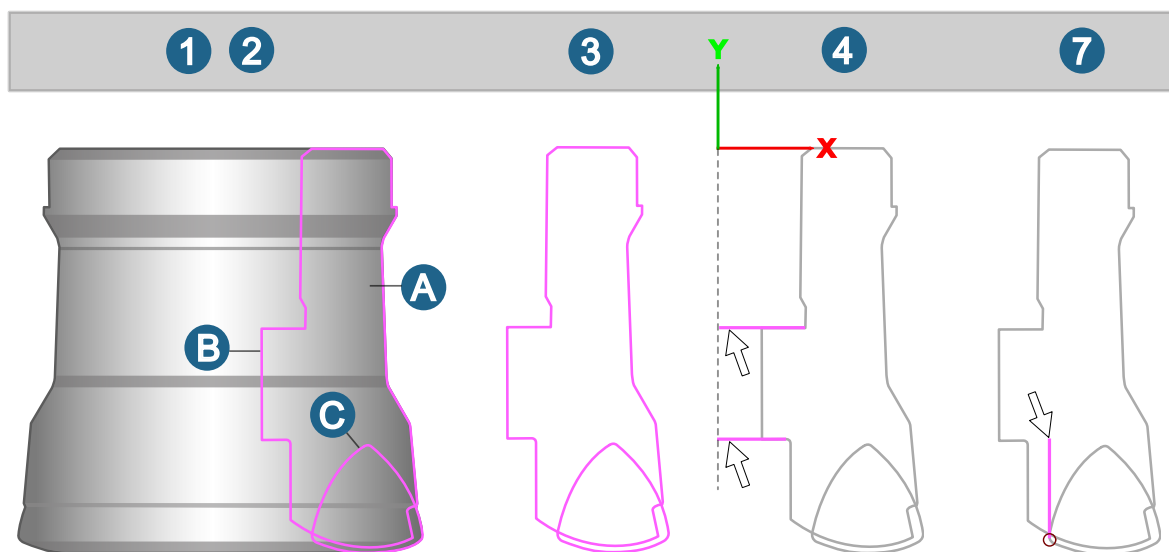
Indexable cutter with high feed inserts

To create an indexable cutter with high feed inserts, proceed as follows:

Create rotation contours in the CAD system

1. Use the CAD function **File** → **Open** to import a 3D model (*.iges, *.step) of the tool.
2. Create a rotation contour with the CAD function **Curves** → **Shape contour**. To do this, activate the **Precise** options in the **Shape contour** dialog, use the functions **Rotation** and **Direction** → **Z WP**, and select the geometry of the base body with **Select** → **Shapes**. Confirm the selection.
3. Then hide the holder and insert geometry so that only the shape contours are shown.
4. Extend the contours of the inner planar surface to the center of the tool adaptor.
5. Check the generated geometry with the CAD function **Analysis** → **Double entities**. If there are duplicate elements, delete them.
6. Check the dimensions of the tool with the CAD function **Analysis** → **Bounding box**. The tool dimensions (diameter, length) must match the tool manufacturer's data.
7. Use the CAD function **Drafting** → **Sketch** to generate a vertical line as a reference for aligning the insert later.

Example: (A) = Base body, (B) = Rotation contour holder, (C) = Rotation contour insert



Assemble tool in the CAM system

8. Use the **Tools** → **New CAM** function to create an **Indexable high feed cutter** and select the previously generated rotation contour for the holder in the tool dialog under **Body geometry** with the icon.
9. In the same dialog, select the rotation contour for the insert in **Insert geometry** with the icon, and select the previously created vertical line (see job step 6) to align the insert.
10. Finally, enter the values for **Diameter** and **Length** in the **Indexable high feed cutter** dialog (tool manufacturer's data sheet). The tool dimensions (diameter, length) must match the tool manufacturer's data, see job step 6.
11. The tool has now been created and can be used in hyperMILL®.

Indexable round insert cutter

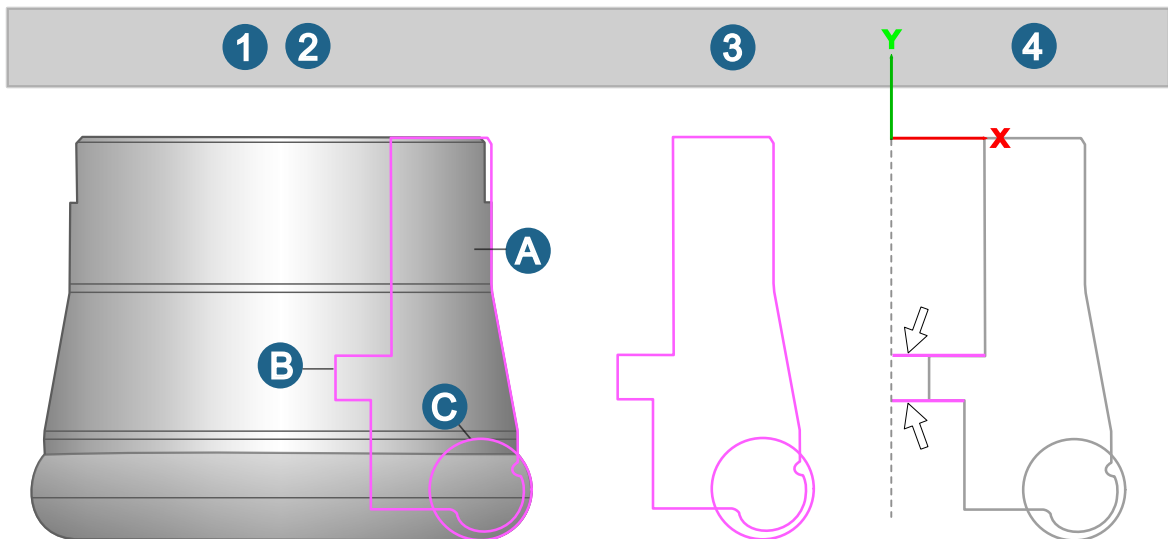
To create an indexable cutter with round inserts, proceed as follows:

Create rotation contours in the CAD system


1. Use the CAD function **File** → **Open** to import a 3D model (*.iges, *.step) of the tool.
2. Create a rotation contour with the CAD function **Curves** → **Shape contour**. To do this, activate the **Precise** options in the **Shape contour** dialog, use the functions **Rotation** and **Direction** → **Z WP**, and select the geometry of the base body with **Select** → **Shapes**. Confirm the selection.
3. Then hide the holder and insert geometry so that only the shape contours are shown.
4. Extend the contours of the inner planar surface to the center of the tool adaptor.
5. Check the generated geometry with the CAD function **Analysis** → **Double entities**. If there are duplicate elements, delete them.

- Check the dimensions of the tool with the CAD function **Analysis** → **Bounding box**. The tool dimensions (diameter, length) must match the tool manufacturer's data.

Example: (A) = Base body, (B) = Rotation contour holder, (C) = Rotation contour insert



Assemble tool in the CAM system

- Use the **Tools** → **New CAM** function to create an **Indexable round insert cutter** and select the previously generated rotation contour for the holder in the tool dialog under **Body geometry** with the  icon.
- Finally, enter the values for **Diameter**, **Length**, and **Corner radius** in the **Indexable round insert cutter** dialog (tool manufacturer's data sheet). The tool dimensions (diameter, length, corner radius) must match the tool manufacturer's data, see job step 6.
- The tool has now been created and can be used in *hyperMILL*®.

Optimal barrel cutter



Optimize parameters of barrel cutters based on surface and curve information.

CAM → **Optimal barrel cutter**

CAM browser → **Jobs** → **Job** → **Tool**: Select a barrel cutter. A function for optimizing the barrel cutter is displayed in the tool definition.

2024

For machining with barrel cutters, the optimal shape of the barrel cutter must first be found. This ensures that the surface to be machined is completely machined as required. The parameters of the required barrel cutter are optimized on the basis of surface and curve information. A surface and several curves in relation to this surface are included in the analysis. Preset values for the dimensions of the barrel cutter are taken from the tool definition in *hyperMILL*® when called.

Select

Surface: Select the surface for which an optimal barrel cutter geometry is to be found for machining.

Curve: Select one or more curves in relation to this surface that represent the planned machining path. The number of selected elements is displayed.

Barrel

When calling the dialog from the *hyperMILL*® job definition (**Tool** dialog page) and selecting a barrel cutter, the **Cutter parameters** of the selected tool are adopted. The parameters can then be changed manually.

For **Base diameter range** and **Barrel radius range**, enter the minimum and maximum values between which the optimal value is to be calculated. All other parameters are considered fixed when the calculation is running.

Both the variables and the fixed parameters are included in the calculation of the optimal barrel radius and base diameter.



If the dialog is started via the **CAM** → **Optimal barrel cutter** menu, preselect an option for a conical, tangential or general barrel cutter for **Barrel type** and an option (ball, radius, sharp) for **Tip type**.

W? If the dialog is started via the hyperMILL® job definition (**Tool** dialog page), these parameters are already preset and can no longer be changed.

Options

Contact parameter: Enter how the preview of the tool contact point of the barrel cutter should be positioned in relation to the selected curve. For example, if the value 0.5 is entered, the tool preview is displayed at the center of the curve.

Create 3D: Create a 3D model of the calculated tool geometry in the graphics area.

Verification

- **Single curve:** Select an individual curve to specify where the tool preview should be displayed. Enter a value or vary it using the slider. The determined tool geometry is moved along the curve
- Use **Face offset** to enter a value that enlarges the internal tool shape to allow a preview of the contact area between the tool and the surface. The smaller the value, the smaller the preview.

Further options

Cutting side: Change the tool position in relation to the surface normal of the selected surface (inside or outside).

Infeed direction: Reverse the infeed direction of the tool in relation to the orientation of the selected curve.

Automatic computation: The optimal tool geometry is displayed as a preview for the set values and only in the slider position that defines the contact parameter.

Global mode: If **Global mode** is selected for the calculation, every point in the selected curves is taken into account so that the optimal tool shape is calculated within all selected curves. Depending on the geometry, the calculation may take some time. Otherwise, or if the slider is moved, the calculation is performed continuously for each point currently reached. When the slider stops, the result is updated. Different tool dimensions are obtained for the various points analyzed on the curves.

Result

Optimal barrel radius: The determined maximum possible barrel radius that fits the area is output.

Optimal barrel diameter: The determined maximum base diameter (shaft side) that fits the surface is output.



The function does not carry out a collision check! This is performed in the cycle. The optimum shape is determined using an analytical calculation on the selected surface, taking into account a safe tolerance, in order to obtain the best possible shape.

Current restriction: Currently, the tool is always considered normal to the curve during the calculation. A lead angle is expected to be introduced with one of the next versions.

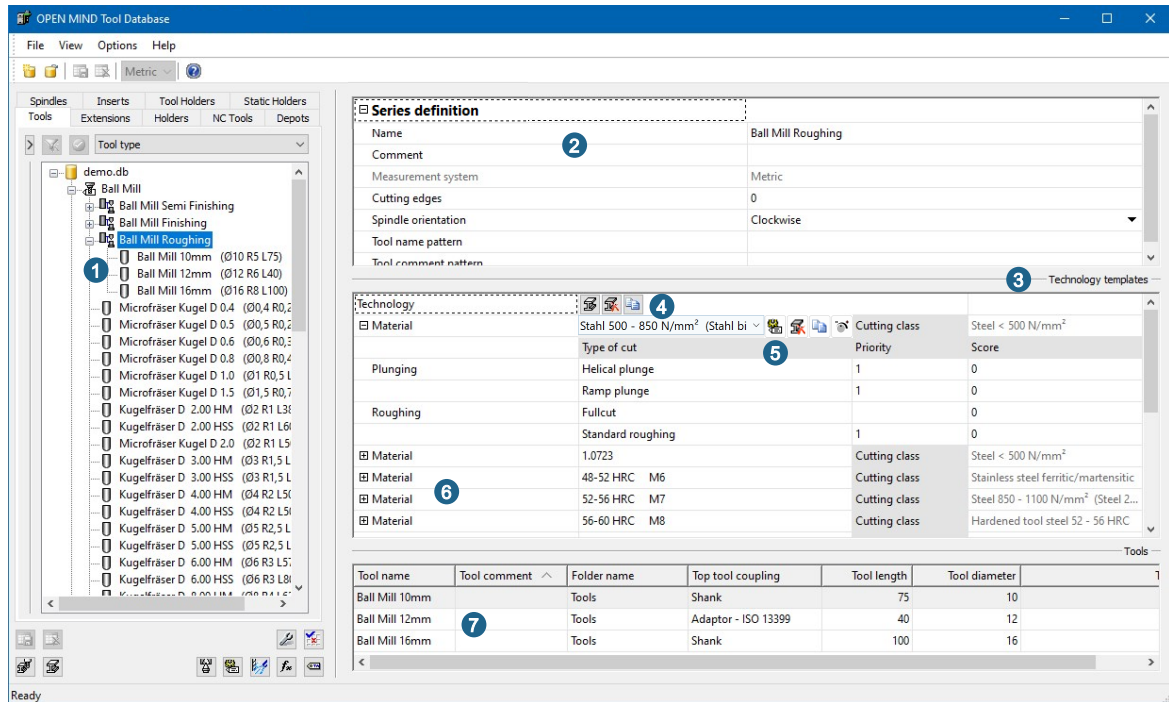


Tool series and technology templates

Definitions	
Tool series	A Tool series can only be created in the Tool type view and comprises one or more tools of the same type. As the technology in the tool series is defined for all tools, a series should only contain tools that have the same purpose. The names and comments for the individual tools can also be defined in the tool series using a pattern.
Technology template	A Technology template contains a Type of cut for a Cutting class or a Concrete material that can be machined with the tools in the tool series.
Cutting class	A Cutting class is a virtual material (that is, a type of reference material) that is superordinate to a group of concrete materials with similar machining properties (machinability, hardness, and so on).
Concrete material	A Concrete material is a single material that belongs to a specific cutting class. The assignment is based on machinability.
Type of cut	Each Type of cut contains predefined, application-related technology parameters that can be defined via constant values, calculation rules, or sample tables. For further information, see section Type of cut parameter and associated calculation rules .

User interface

- (1) List of tools in a tool series in the tool browser under the tool type view
- (2) Overview: features of the tool series
- (3) Definition and overview of the technology templates (materials and types of cut)
- (4) Material definition, (5) Type of cut definition
- (6) List of materials in a tool series
- (7) List of tools in a tool series



Define tool series

To define a tool series, proceed as follows:

1. On the **Tools** tab of the database, show the **Tool type** view, select the required tool type and choose **New** → **Tool series** from the shortcut menu.
2. Define a unique **Name** and, if necessary, a **Comment** for the tool series. Define the measurement system, the standard **Cutting edges**, and the **Spindle orientation** of the tools to be saved in this tool series.
3. Tool names and tool comments can (optionally) follow a defined pattern. Click the line **Tool name pattern** and **Tool comment pattern** and then click the icon to compile the required name pattern from the available elements.
4. If available, select the tool manufacturer from the list under **Manufacturer**. This must be created in advance in the **Edit manufacturers** dialog.

Pattern format for tool name and comment

Specific number formatting (for example, for diameter, length, and so on) is permitted to define the pattern for the tool name and the tool comment. The following characters are permitted for formatting numbers:

“#”: Serves as a placeholder for digits of the number that can be discarded. Leading and trailing zeros can be discarded.

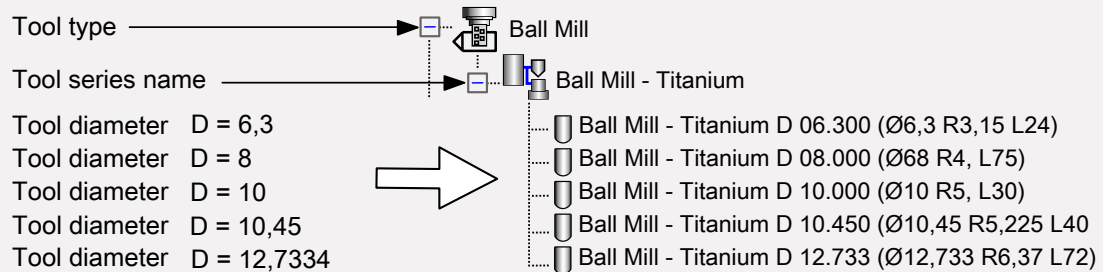
“0”: Serves as a placeholder for digits of the number that are to be filled with “0” if they are not present. If they are present, the digit from the number is used.

“.”: Used as a decimal separator if a “.” is required as the display style.

“,”: Used as a decimal separator if a “,” is required as the display style.

Example: Tool name pattern

```
<TOOL_SERIES> D <TOOL_DIAMETER format = "00.000" decimal-separator = ".">
```


MEASUREMENT SYSTEM CONVERSION IN PATTERN

It is possible to display the geometry values of the tool in a measurement system other than the one selected in the tool series. To do so:

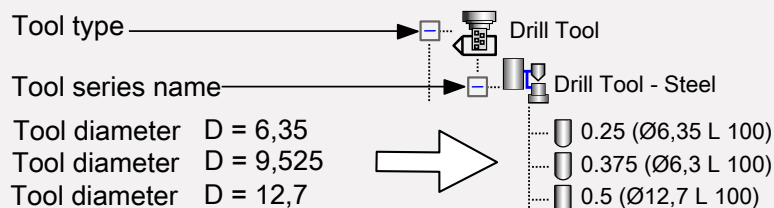
```
unit = "<measurement units>"
```

Add to the pattern (measurement units are "in" or "mm"). For example, to display the tool diameter 9.525 mm in inches (0.375) in the metric measurement system, use the following pattern for the tool name:

```
<TOOL_DIAMETER unit = "in">
```

Example: Tool name pattern

```
<TOOL_DIAMETER unit = "in">
```


Add a new tool to a tool series

To add a new tool to a tool series, proceed as follows:

1. On the **Tools** tab of the database, show the **Tool type** view and select the required tool type.
2. Select the required tool series and choose **New** → **New tool** on the shortcut menu.
3. **Name** and **Comment** are assigned automatically if a corresponding name scheme has been defined. The tool type, tool series, number of cutting edges, measurement system, and manufacturer are taken from the tool series.
4. Define the required geometry parameters.

The technology of the tool is defined in the **Technology templates**. The parameters cannot be changed on the tool.

Add an existing tool to a tool series

To add an existing tool (for example from a previous version of the database) to a tool series, proceed as follows:

1. On the **Tools** tab of the database, show the **Tool type** view and select the required tool.
2. While holding down the left mouse button, drag the tool to the entry for the required tool series and release the mouse button (**drag and drop**).



Please note that all technology data of an existing tool is discarded when the tool is added to a tool series.

Define technology template

A tool series can contain several technology templates. Each technology template comprises a material and an assigned cutting type.

Proceed as follows to define a technology template:

1. In the **Technology** area, click the icon (1) to add the required **Material**.
2. Click the name of the material and then click the icon (6) to add a **Type of cut** for the material. The types of cut available depend on the tool type in the tool series.

Edit type of cut

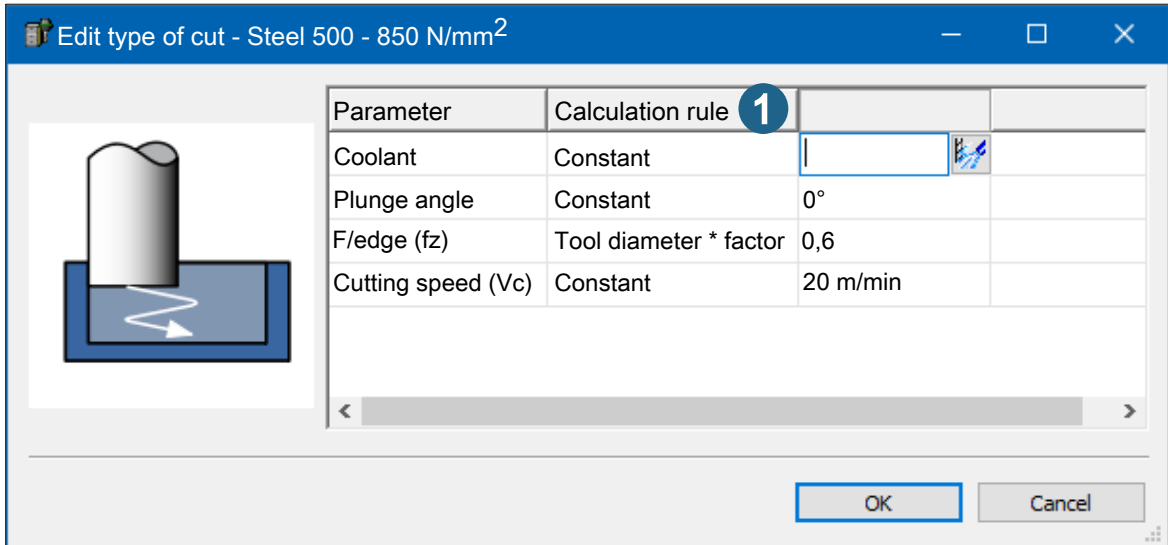
Each type of cut consists of a **Parameter** set and a **Calculation rule** for each parameter. Click the name of the type of cut and then click the icon (8) to open the **Edit type of cut** dialog.

Other functions

(2) Delete a material and all assigned types of cut. (3) Copy the data of all technology templates to the clipboard (3) or paste them from the clipboard (4). (5) Define a scaling for rule parameters. (7) Delete a type of cut. (9) Lock a technology template to prevent unintended changes or to exclude it from scaling.

Technology		1	2	3	4		
Material	Steel						Steel < 500N/mm ²
Type of cut							Priority
Plunging	Ramp plunge						Score
Roughing	Standard roughing						

The parameters of the type of cut cannot be changed, only the calculation rule assigned to a parameter, and the associated values. To do this, select the required rule in the **Calculation rule** (1) column and specify using the available methods.



SCALE RULE PARAMETERS



Use the **Scale rule parameters** to scale the value of the selected parameters. The factor value applies to all types of cut of the selected material if the **Type of cut** is not locked.



To derive a new material from an existing material, copy the existing material with all types of cut and then scale all parameters using the **Scale rule parameters** function.

Type of cut parameter and associated calculation rules




Type of cut parameters are tool technology parameters that are required in a *hyperMILL*® job or are necessary to calculate the technology parameters. Only the type of cut parameters that are actually required are available. The following table contains all the type of cut parameters that can be defined in *hyperMILL*® and their calculation rule(s).

Type of cut parameter	Calculation rule
Axial feedrate: Feedrate for axial G1 movements, usually at the start of machining (clearance distance to the first toolpath point).	Feedrate * factor
Coolant: Call the coolant (multiple selection possible). The call of the coolant must be created globally in advance using the Edit coolants function.	Constant
Cutting speed (Vc): Calculation of the spindle speed, depending on the tool diameter.	Constant Tool diameter sample table Tool diameter and corner radius sample table



Type of cut parameter	Calculation rule
<p>F/edge (fz): Feedrate for each tool cutting edge for calculating the machining feedrate.</p>	<p>Tool diameter * factor</p> <p>Tool diameter sample table</p> <p>Tool diameter and corner radius sample table</p> <p>Standard chip thinning formula</p> <p>Feedrate per edge formula</p>
<p>F/edge max.: The parameter F/edge (fz) always refers to a specific Infeed depth (ap). If a value lower than the defined value for the infeed depth (ap) is used, the feedrate can be increased. This feedrate adjustment can be limited using the F/edge max. parameter.</p>	<p>Tool diameter * factor</p> <p>Tool diameter sample table</p> <p>Tool diameter and corner radius sample table</p>
<p> If there is no value for the F/edge max. parameter, use the same value as for the F/edge (fz) parameter.</p>	
<p> The parameter is not currently used by <i>hyperMILL</i>®. The definition with meaningful values is necessary for future functions and is therefore made available in advance.</p>	
<p>Plunge angle: Plunge angle for helical or ramp-shaped movements.</p>	<p>Constant</p> <p>Tool diameter sample table</p> <p>Tool diameter and corner radius sample table</p>
<p>Infeed length max: Maximum length with which the cutting edge of the tool may be engaged when plunging.</p>	<p>Cutting length * factor</p> <p>Tool diameter * factor</p>
<p>F/edge full cut: Feedrate for each tool cutting edge in full cut.</p>	<p>Standard material removal rate * factor</p> <p>Tool diameter * factor</p> <p>Tool diameter sample table</p> <p>Tool diameter and corner radius sample table</p>
<p>F/edge plunge: Feedrate for each tool cutting edge for helical or ramp-shaped movements when plunging.</p>	<p>Tool diameter * factor</p> <p>Tool diameter sample table</p> <p>Tool diameter and corner radius sample table</p>
<p>Infeed width (ae): Lateral infeed.</p>	<p>Tool diameter * factor</p> <p>Tool diameter sample table</p> <p>Tool diameter and corner radius sample table</p>



Type of cut parameter	Calculation rule
<p>Infeed depth (ap): Vertical stepdown.</p>	<p>Cutting length * factor</p> <p>Tool diameter * factor</p> <p>Tool diameter sample table</p> <p>Tool diameter and corner radius sample table</p>
<p>Cutting speed plunge: Cutting speed (Vc) for calculating the spindle speed during plunging movements, depending on the tool diameter.</p>	<p>Constant</p> <p>Tool diameter sample table</p> <p>Tool diameter and corner radius sample table</p>
<p>Infeed width min/Infeed width max: Only available for the Trochoidal roughing type of cut.</p> <p>Define the range for the minimum and maximum infeed width in which the infeed width (ae) may be located.</p>	<p>Tool diameter * factor</p>
<p>Infeed length min: Minimum vertical length with which the tool must be engaged.</p>	<p>Cutting length * factor</p> <p>Tool diameter * factor</p> <p>Tool diameter sample table</p> <p>Tool diameter and corner radius sample table</p>
<p> The parameter is not currently used by hyperMILL®. The definition with meaningful values is necessary for future functions and is therefore made available in advance.</p>	
<p>Cutting speed reference: Cutting speed reference value for calculating the Cutting speed (Vc) using the Standard adjusted speed formula or Cutting speed formula calculation rules.</p>	<p>Constant</p> <p>Tool diameter sample table</p> <p>Tool diameter and corner radius sample table</p>
<p>F/edge reference: Reference value for the feedrate for each cutting edge for calculating the chip thickness.</p>	<p>Tool diameter * factor</p> <p>Tool diameter sample table</p>
<p> As technology data is generally available for conventional roughing strategies with large lateral infeed, the feedrate is adjusted based on this reference technology and in relation to the Infeed width (ae) parameter. The chip thickness is retained.</p>	
<p>Infeed width reference: Reference value for the infeed width for calculating the chip thickness.</p>	<p>Tool diameter * factor</p>
<p> As technology data is generally available for conventional roughing strategies with large lateral infeed, the feedrate is adjusted based on this reference technology and in relation to the Infeed width (ae) parameter. The chip thickness is retained.</p>	




Type of cut parameter	Calculation rule
Expected tool engagement (ae): Side allowance for which the defined technology data is designed.	Cutting length * factor Tool diameter * factor Tool diameter sample table
Expected tool engagement (ap): Hub allowance for which the defined technology data is designed.	Tool diameter * factor Tool diameter sample table Tool diameter and corner radius sample table
Steep slope feedrate factor: Reduce the machining feedrate for plunging movements using the Steep slope angle parameter.	Constant
Steep slope angle: Define the angle from which the Steep slope feedrate factor parameter is applied to the machining feedrate.	Constant
Scallop height: Theoretical scallop height that defines the infeed using the Scallop height infeed mode.	Constant
Infeed: Infeed using the Constant stepdown , Constant stepover , or 3D stepover infeed mode.	Stepover formula Tool diameter sample table Tool diameter and corner radius sample table
Expected tool engagement (3D): 3D allowance (allowance on bottom and side) for which the defined technology data is designed.	Tool diameter * factor Tool diameter sample table Tool diameter and corner radius sample table
Feedrate/revolution: Machining feedrate per revolution.	Tool diameter * factor Tool diameter sample table Tool diameter and corner radius sample table
Retract feedrate/rev: Machining feedrate per revolution for the retract movement.	Tool diameter * factor Tool diameter sample table Tool diameter and corner radius sample table
Lateral infeed:	Tool diameter * factor Tool diameter sample table Tool diameter and corner radius sample table
Edge control: Only available in the Thread milling type of cut. Define whether the edge control function is switched on or off.	

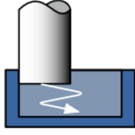

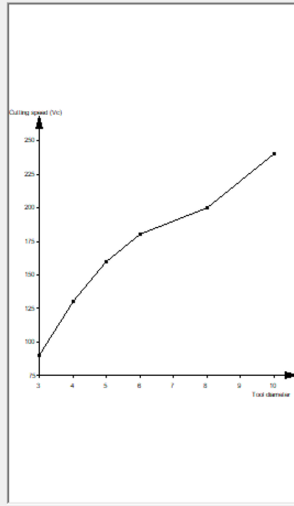


Type of cut parameter	Calculation rule
Max. drilling depth: Define the depth to which the tool can machine with the respective strategy.	Tool diameter * factor Tool diameter sample table Tool diameter and corner radius sample table
Drill size: Define the diameter with which the hole will be pre-machined.	Tool diameter * factor Tool diameter sample table
<p>The following types of cut can only be used in the jobs of the same name. The parameters of the individual types of cut reflect the parameters of the respective job.</p> <ul style="list-style-type: none"> • Simple drilling • Drilling with chip break • Drilling with pecking • Centering • Gun drilling • Optimized deep hole drilling (with/without chip break) • Boring • Tapping 	

Calculation rules and their meaning


Calculation rule	Meaning
Feedrate * factor	Define the axial feedrate, for example, based on the regular feedrate using a factor. The feedrate of the individual tool is always taken into account.
Constant	Use Constant to set a specific parameter to a constant value for all tools in a tool series. The value applies to all tools.
Tool diameter sample table	Use the Tool diameter sample table to define specific values for individual diameters. The value range between two examples is interpolated linearly.
 Only the range between the smallest and largest diameter is defined!	



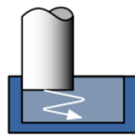

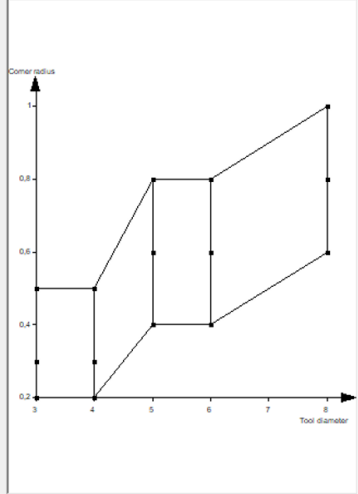
Calculation rule		Meaning																																							
<div style="display: flex; justify-content: space-between;"> <div style="width: 25%;"> <p>Helical plunge</p>  </div> <div style="width: 50%;"> <table border="1"> <thead> <tr> <th>Parameter</th> <th>Calculation rule</th> <th></th> <th></th> </tr> </thead> <tbody> <tr> <td>Feedrate axial</td> <td>Feedrate * factor</td> <td>1</td> <td></td> </tr> <tr> <td>Coolant</td> <td>Constant</td> <td>1</td> <td></td> </tr> <tr> <td rowspan="6">Cutting speed (Vc)</td> <td rowspan="6">Tool diameter sample table</td> <td>Tool diameter</td> <td>Cutting speed (...)</td> </tr> <tr> <td>3</td> <td>90</td> </tr> <tr> <td>4</td> <td>130</td> </tr> <tr> <td>5</td> <td>160</td> </tr> <tr> <td>6</td> <td>180</td> </tr> <tr> <td>8</td> <td>200</td> </tr> <tr> <td></td> <td></td> <td>10</td> <td>240</td> </tr> <tr> <td>F/edge (fz)</td> <td>Tool diameter * factor</td> <td>0,025</td> <td></td> </tr> <tr> <td>Plunge angle</td> <td>Constant</td> <td>6°</td> <td></td> </tr> </tbody> </table> </div> <div style="width: 20%; text-align: right;">  </div> </div>				Parameter	Calculation rule			Feedrate axial	Feedrate * factor	1		Coolant	Constant	1		Cutting speed (Vc)	Tool diameter sample table	Tool diameter	Cutting speed (...)	3	90	4	130	5	160	6	180	8	200			10	240	F/edge (fz)	Tool diameter * factor	0,025		Plunge angle	Constant	6°	
Parameter	Calculation rule																																								
Feedrate axial	Feedrate * factor	1																																							
Coolant	Constant	1																																							
Cutting speed (Vc)	Tool diameter sample table	Tool diameter	Cutting speed (...)																																						
		3	90																																						
		4	130																																						
		5	160																																						
		6	180																																						
		8	200																																						
		10	240																																						
F/edge (fz)	Tool diameter * factor	0,025																																							
Plunge angle	Constant	6°																																							
																																									



Tool diameter and corner radius sample table

Use the **Tool diameter and corner radius sample table** to define specific values for individual diameter/corner radius combinations. The value range between the minimum and maximum values is interpolated.





Only the range between the smallest and largest values is defined!

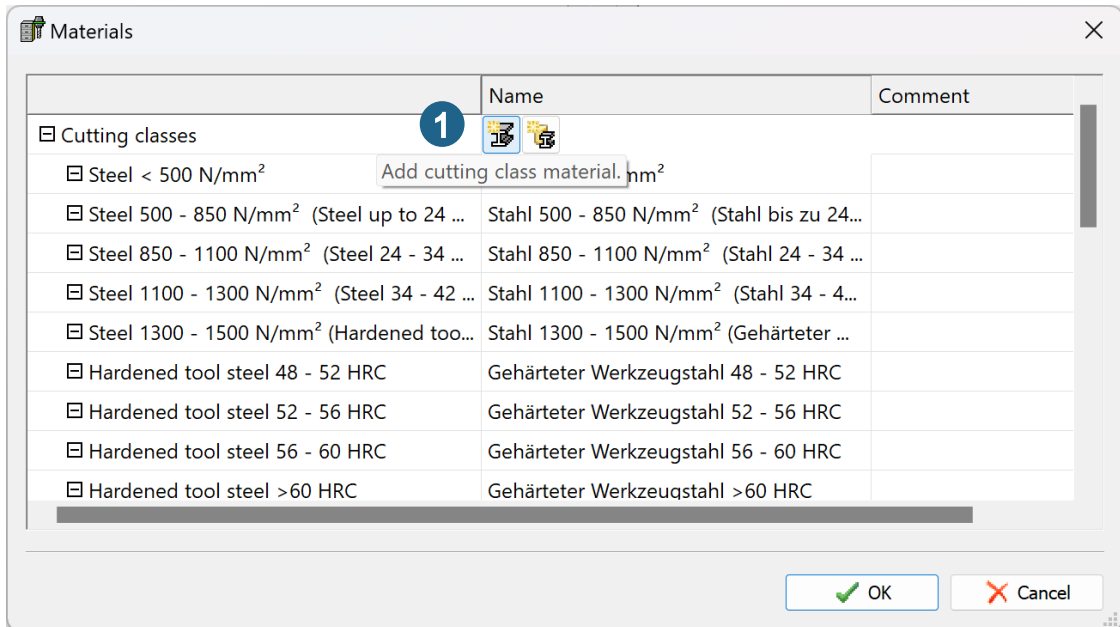
Calculation rule		Meaning																																											
<div style="display: flex; justify-content: space-between;"> <div style="width: 25%;"> <p>Helical plunge</p>  </div> <div style="width: 50%;"> <table border="1"> <thead> <tr> <th>Parameter</th> <th>Calculation rule</th> <th></th> <th></th> </tr> </thead> <tbody> <tr> <td>Feedrate axial</td> <td>Feedrate * factor</td> <td>1</td> <td></td> </tr> <tr> <td>Coolant</td> <td>Constant</td> <td>1</td> <td></td> </tr> <tr> <td rowspan="6">Cutting speed (Vc)</td> <td rowspan="6">Tool diameter and corner r...</td> <td>Tool diameter</td> <td>Cutting speed (Vc)</td> </tr> <tr> <td>3</td> <td></td> </tr> <tr> <td>5</td> <td></td> </tr> <tr> <td>6</td> <td></td> </tr> <tr> <td>8</td> <td></td> </tr> <tr> <td>Corner radius</td> <td>Cutting speed (Vc)</td> </tr> <tr> <td>0,2</td> <td>120</td> </tr> <tr> <td>0,3</td> <td>130</td> </tr> <tr> <td></td> <td></td> <td>0,5</td> <td>150</td> </tr> <tr> <td>F/edge (fz)</td> <td>Tool diameter * factor</td> <td>0,025</td> <td></td> </tr> <tr> <td>Plunge angle</td> <td>Constant</td> <td>0°</td> <td></td> </tr> </tbody> </table> </div> <div style="width: 20%; text-align: right;">  </div> </div>				Parameter	Calculation rule			Feedrate axial	Feedrate * factor	1		Coolant	Constant	1		Cutting speed (Vc)	Tool diameter and corner r...	Tool diameter	Cutting speed (Vc)	3		5		6		8		Corner radius	Cutting speed (Vc)	0,2	120	0,3	130			0,5	150	F/edge (fz)	Tool diameter * factor	0,025		Plunge angle	Constant	0°	
Parameter	Calculation rule																																												
Feedrate axial	Feedrate * factor	1																																											
Coolant	Constant	1																																											
Cutting speed (Vc)	Tool diameter and corner r...	Tool diameter	Cutting speed (Vc)																																										
		3																																											
		5																																											
		6																																											
		8																																											
		Corner radius	Cutting speed (Vc)																																										
0,2	120																																												
0,3	130																																												
		0,5	150																																										
F/edge (fz)	Tool diameter * factor	0,025																																											
Plunge angle	Constant	0°																																											
																																													

Calculation rule	Meaning
Tool diameter and pitch sample table	Only available in the Tapping type of cut. Use the Tool diameter and pitch sample table to define specific values for individual diameter/pitch combinations. The value range between the minimum and maximum values is interpolated.
Tool diameter * factor	Based on the tool diameter, use a factor to define the value for the F/edge parameter, for example. The diameter of the individual tool is taken into account.
Cutting length * factor	Based on the cutting length of the tool, use a factor to define the value for the Infeed length max parameter, for example. The cutting length of the individual tool is taken into account.
Standard material removal rate * factor	The calculation rule is only available for the Standard roughing type of cut. The F/edge for full cuts is derived on the basis of the material removal rate during the standard roughing cut. The feedrate can also be reduced using a factor.
Standard adjusted speed formula	The calculation rule is only available for the Trochoidal roughing type of cut. Due to the reduced lateral infeed during trochoidal roughing, the tool life increases. The cutting speed is increased accordingly to such an extent that the tool life is maintained in relation to the tool's reference technology.
 The Cutting speed formula calculation rule allows tool manufacturers to use their own formulas to adjust the cutting speed.	
Standard chip thinning formula	Use to keep the chip section constant depending on the lateral infeed (Ae).
 The Feedrate per edge formula calculation rule allows tool manufacturers to use their own formulas to adjust the mean chip section.	
Stepover formula	Use to calculate the Constant stepdown , the Constant stepover , or the 3D stepover based on the Scallop height parameter.

Cutting classes and materials

To define a new cutting class or a new concrete material, proceed as follows:

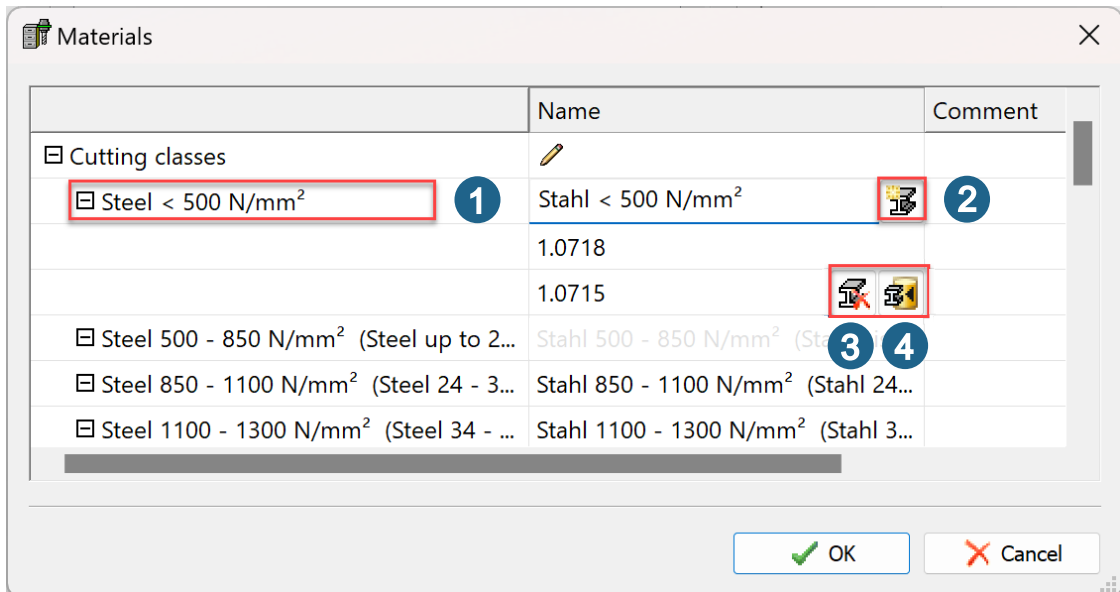
1. Click **Yes** to open the **Materials** dialog.
2. In the **Materials** dialog, click the icon (1) and then select the  **Add cutting class material** or  **Add cutting class material from material database** function.



- As soon as a cutting class exists (1), click the button (2) (**Add concrete material**) to define a material belonging to the cutting class. To change the name of the cutting class, click the required row in the **Name** column. Add a comment if required.

Cutting classes can be deleted or reassigned as long as no specific materials are assigned.

The following functions are also available in the dialog. (3) **Delete concrete material**. (4) **Reassign concrete material**.



Use the **Reassign concrete material** function to convert a legacy material from an older database to the new material structure or to reassign a missing material (as a user-defined material).

- Then confirm the selection with **OK**.

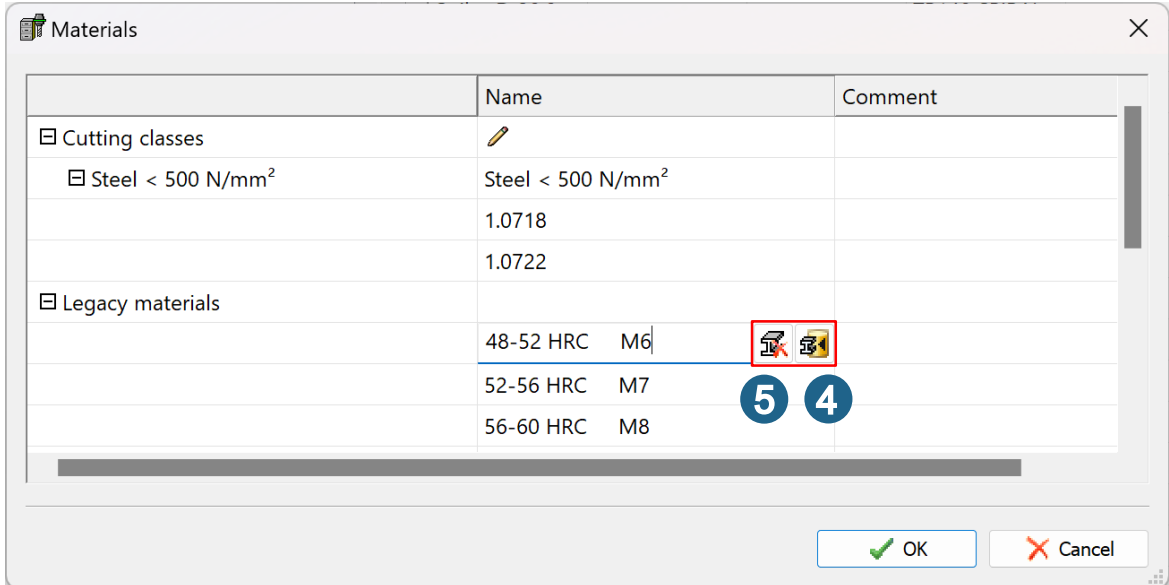


Cutting classes and specific materials cannot be changed, which means that only **one** instance of the cutting class or the specific material is ever available in the database.



Convert legacy material to standard material / delete legacy material

1. Click the icon (4) to convert a legacy material into a standard material by selecting a required material from the material database.
2. Click the icon (5) to delete a legacy material.





Overview: jobs and types of cut

List of types of cut that can be used in a job. A type of cut is defined in the tool database as part of the series definition for a tool type and depending on the material used.

Symbol	Type of cut
Drilling	
Centering	Centering
Simple Drilling	Simple drilling
Drilling with chip break	Drilling with chip break
Drilling with pecking	Drilling with pecking
Reaming	Reaming
Tapping	Tapping
Helical Drilling	Helical plunge Ramp plunge Side semi-finishing Side finishing
Circular Pocket	Standard roughing High-feed roughing Side semi-finishing Side finishing
Optimized deep Hole Drilling (new)	Optimized deep hole drilling Optimized deep hole drilling with chip break
Boring	Boring
Gun Drilling	Gun drilling
Back Boring	Back boring
Thread Milling	Thread milling
Chamfer Milling on 3D Model	Chamfer milling
2D Milling	
Pocket Milling	Standard roughing Trochoidal roughing High-feed roughing Side semi-finishing Side finishing



Symbol	Type of cut
Contour Milling	Full cut Standard roughing Trochoidal roughing High-feed roughing Side semi-finishing Side finishing Face semi-finishing Face finishing
Contour Milling on 3D Model	Full cut Standard roughing Trochoidal roughing High-feed roughing Side semi-finishing Side finishing Face semi-finishing Face finishing
Inclined Contouring	Standard roughing High-feed roughing Side semi-finishing Side finishing Face semi-finishing Face finishing
Inclined Pocketing Rectangular Pocket	Standard roughing High-feed roughing Face semi-finishing Face finishing
Rest Machining	Full cut Standard roughing High-feed roughing Side semi-finishing Side finishing Face semi-finishing



Symbol	Type of cut
	Face finishing
Face Milling	Standard roughing High-feed roughing Face semi-finishing Face finishing
Playback Milling	Full cut Standard roughing High-feed roughing Side semi-finishing Side finishing Face semi-finishing Face finishing
Plunge Milling	Plunge Milling
3D Milling	
3D Optimized Roughing	Standard roughing
3D Arbitrary Stock Roughing	Trochoidal roughing High-feed roughing Face semi-finishing Face finishing
3D Profile Finishing	3D semi-finishing (fine – medium – rough) 3D finishing (fine – medium – rough)
3D Z Level Shape Finishing	Standard roughing High-feed roughing Side semi-finishing Side finishing 3D semi-finishing (fine – medium – rough) 3D finishing (fine – medium – rough)
3D Iso Machining	3D semi-finishing (fine – medium – rough) 3D finishing (fine – medium – rough)
3D Free Path Milling	Ramp plunge Full cut Standard roughing



Symbol	Type of cut
	Trochoidal roughing High-feed roughing Side semi-finishing Side finishing Face semi-finishing Face finishing 3D semi-finishing (fine – medium – rough) 3D finishing (fine – medium – rough)
3D Plane Machining	Standard roughing High-feed roughing Face semi-finishing Face finishing
3D Z Level Finishing	Standard roughing High-feed roughing Side semi-finishing Side finishing 3D semi-finishing (fine – medium – rough) 3D finishing (fine – medium – rough)
3D Advanced Milling	
3D Plunge Roughing	Plunge Milling
3D Complete Finishing	3D semi-finishing (fine – medium – rough) 3D finishing (fine – medium – rough)
3D Equidistant Finishing	3D semi-finishing (fine – medium – rough) 3D finishing (fine – medium – rough)
3D Automatic Rest Machining	Standard roughing High-feed roughing 3D semi-finishing (fine – medium – rough) 3D finishing (fine – medium – rough)
3D Corner Rest Machining	Standard roughing High-feed roughing Face semi-finishing Face finishing



Symbol	Type of cut
	3D semi-finishing (fine – medium – rough) 3D finishing (fine – medium – rough)
3D Cutting Edge Machining	Standard roughing High-feed roughing Side semi-finishing Side finishing
3D Rework Machining	Ramp plunge Full cut Standard roughing Trochoidal roughing High-feed roughing Side semi-finishing Side finishing Face semi-finishing Face finishing 3D semi-finishing (fine – medium – rough) 3D finishing (fine – medium – rough)
3D Curve Flow Machining	Standard roughing High-feed roughing Side semi-finishing Side finishing
5X Cavity Milling	
5X Optimized Rest Roughing	Standard roughing High-feed roughing
5X Profile Finishing	3D semi-finishing
5X Radial Machining	3D finishing
5X Z Level Finishing	Standard roughing High-feed roughing Side semi-finishing Side finishing 3D semi-finishing (fine – medium – rough) 3D finishing (fine – medium – rough)



Symbol	Type of cut
5X Equidistant Finishing	3D semi-finishing (fine – medium – rough) 3D finishing (fine – medium – rough)
5X Rest Machining	Standard roughing High-feed roughing 3D semi-finishing (fine – medium – rough) 3D finishing (fine – medium – rough)
5X Corner Rest Machining	Standard roughing High-feed roughing Side semi-finishing Side finishing 3D semi-finishing (fine – medium – rough) 3D finishing (fine – medium – rough)
5X Free Path Machining	Ramp plunge Full cut Standard roughing Trochoidal roughing High-feed roughing Side semi-finishing Side finishing Face semi-finishing Face finishing 3D semi-finishing (fine – medium – rough) 3D finishing (fine – medium – rough)
5X Rework Machining	Ramp plunge Full cut Standard roughing Trochoidal roughing High-feed roughing Face semi-finishing Face finishing 3D semi-finishing (fine – medium – rough) 3D finishing (fine – medium – rough)



Symbol	Type of cut
5X Z Level Roughing	Standard roughing High-feed roughing
5X Cutting Edge Plunge	Plunge Milling
5X Cutting Edge Swarf	Standard roughing Trochoidal roughing Face semi-finishing Face finishing
5X Halfpipe Machining	3D semi-finishing (fine – medium – rough) 3D finishing (fine – medium – rough)
5X Surface Milling	
5X Top Milling 5X Iso Top Milling	Standard roughing High-feed roughing Face semi-finishing Face finishing 3D semi-finishing (fine – medium – rough) 3D finishing (fine – medium – rough)
5X Contouring	Full cut Standard roughing Trochoidal roughing High-feed roughing Side semi-finishing Side finishing Face semi-finishing Face finishing 3D semi-finishing (fine – medium – rough) 3D finishing (fine – medium – rough)
5X Swarf Cutting one Curve 5X Swarf Cutting two Curves 5X Swarf Cutting Planes	Full cut Standard roughing Trochoidal roughing Side semi-finishing Side finishing Face semi-finishing



Symbol	Type of cut
	Face finishing
5X Shape Offset Roughing	Standard roughing Trochoidal roughing High-feed roughing Face semi-finishing Face finishing
5X Shape Offset Finishing	Standard roughing High-feed roughing Side semi-finishing Side finishing Face semi-finishing Face finishing
5X Tangent Machining	Side semi-finishing
5X Tangent Plane Machining	Side finishing
5X Prismatic Fillet Finishing	3D semi-finishing (fine – medium – rough) 3D finishing (fine – medium – rough)
5X Tube Milling	
5X Tube Roughing	Standard roughing High-feed roughing
5X Tube Finishing	3D semi-finishing (fine – medium – rough)
5X Tube Rest Machining	3D finishing (fine – medium – rough)
5X Blade Milling	
3D Blade Roughing	Standard roughing High-feed roughing
5X Blade Point Milling	3D semi-finishing (fine – medium – rough) 3D finishing (fine – medium – rough)
5X Blade Platform Machining	Side semi-finishing Side finishing 3D semi-finishing (fine – medium – rough) 3D finishing (fine – medium – rough)
5X Blade Top Milling	3D semi-finishing (fine – medium – rough) 3D finishing (fine – medium – rough)

Symbol	Type of cut
5X Blade Tangent Milling	Side semi-finishing
	Side finishing

Define coupling



CHANGED USER GUIDANCE

Since version 2024, the coupling types **Weldon**, **Weldon shank** and **Shrink fit** became standard types and can be directly used in the coupling area of the component (Tool, Extension, Holder, Tool holder). For tools the **Diameter** of the top coupling is automatically adapted to the tool diameter or shank when present. Tools using former coupling classes are automatically converted.

VIRTUAL Machining Center

Program run (simulation)



SIMULATION REQUIREMENTS

The following requirements must be met to be able to perform a program run (simulation):

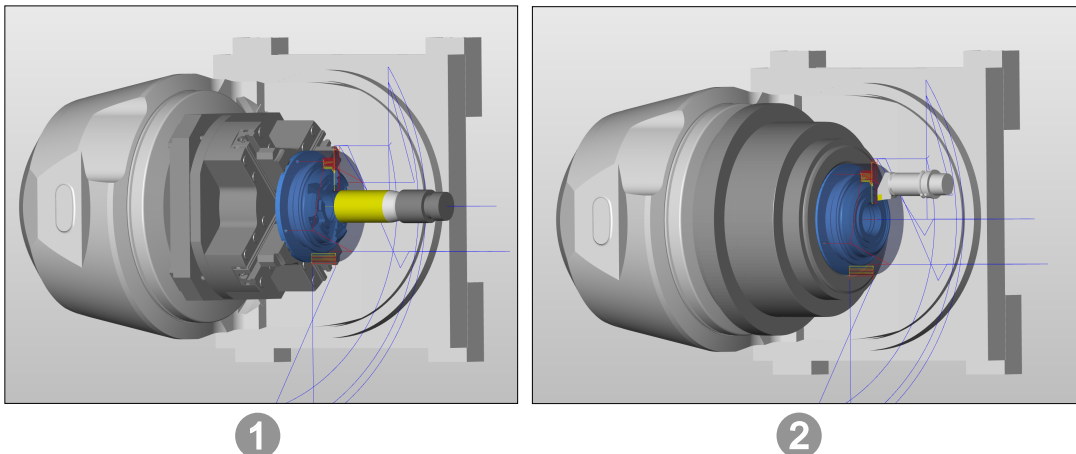
- A component (**Model**) must be defined for milling and drilling operations, see *hyperMILL®* → **Job list** → **Model** → **Model**.
- A component (**Model**) and a 2D turning contour must be defined for turning operations, see *hyperMILL®* → **Job list** → **Model** → **Turning model**.

If this is not the case, the VIRTUAL Machining Center cannot be started. A corresponding message is displayed.

Controlling visibility in the graphics area

Turning operations

For turning operations, the stock, component, and fixture are shown rotated. Prerequisite for the rotated display of the stock: The material removal simulation is activated.



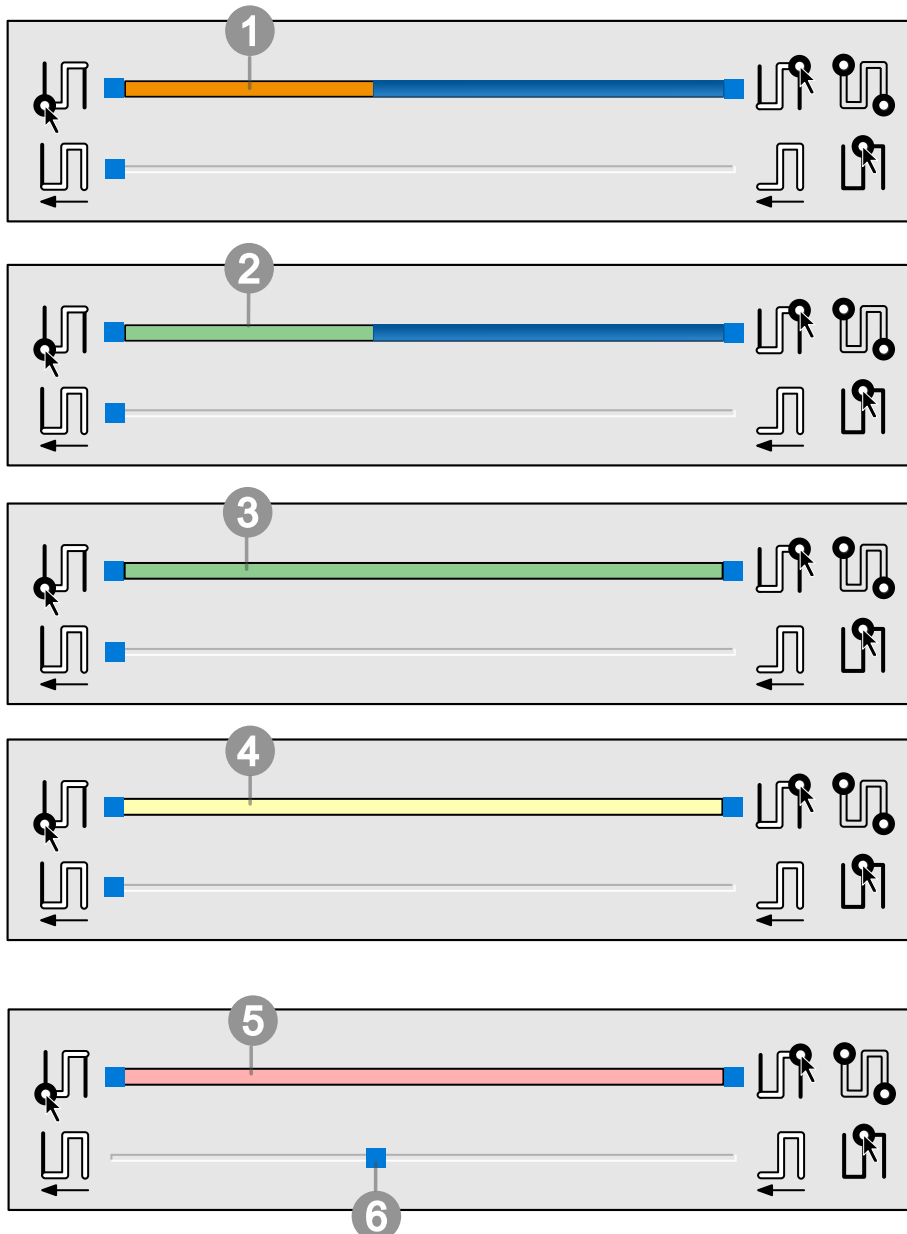
(1) Display of fixture, component, and stock for milling and drilling operations. (2) Display of fixture, component, and stock as rotational geometry for turning operations.

Simulating the VNC or NC program file

Run a collision check



After starting the collision check, the progress indicator in the simulation control panel changes as follows:







① Progress of the collision check in the loaded program file. Either a collision, component violation, or a contact was detected.

② Progress of the collision check in the loaded program file. No collision, component violation, or contact has yet been detected.


③ The collision check ended without a collision, component violation, or contact.

- ④ The collision check ended with a contact. Detailed information on this can be found in the output message and in the upper window of the display area.
- ⑤ The collision check ended with either a collision or component violation. Detailed information on this can be found in the output message and in the upper window of the display area.
- ⑥ Machine position in the loaded program file.

Collision check and material removal

	<p>Move the machine to the next warning. The button turns yellow if a warning (for example, violation of the axis limit) has occurred.</p>
	<p>Move the machine to the next warning or next error. The button turns red if an error or an error and a warning have occurred.</p>
	<p>Move the machine to the next collision area. The button turns yellow after the collision check has been carried out if a contact has been detected.</p>
	<p>Move the machine to the next collision area. The button turns red after the collision check has been carried out on detecting the following:</p> <ul style="list-style-type: none"> • A collision or a component violation • A collision and a component violation • A collision and a contact • A component violation and a contact • A collision, a component violation, and a contact

Configuration

	<p>Turret setup: Only available if the VIRTUAL Machining Center has been started in Machining planner mode and in <i>hyperMILL</i>® (Job list → Postprocessor), a corresponding Virtual Machine has been loaded.</p> <p>Equip a turret with static holders and NC tools for turning operations.</p>
---	---

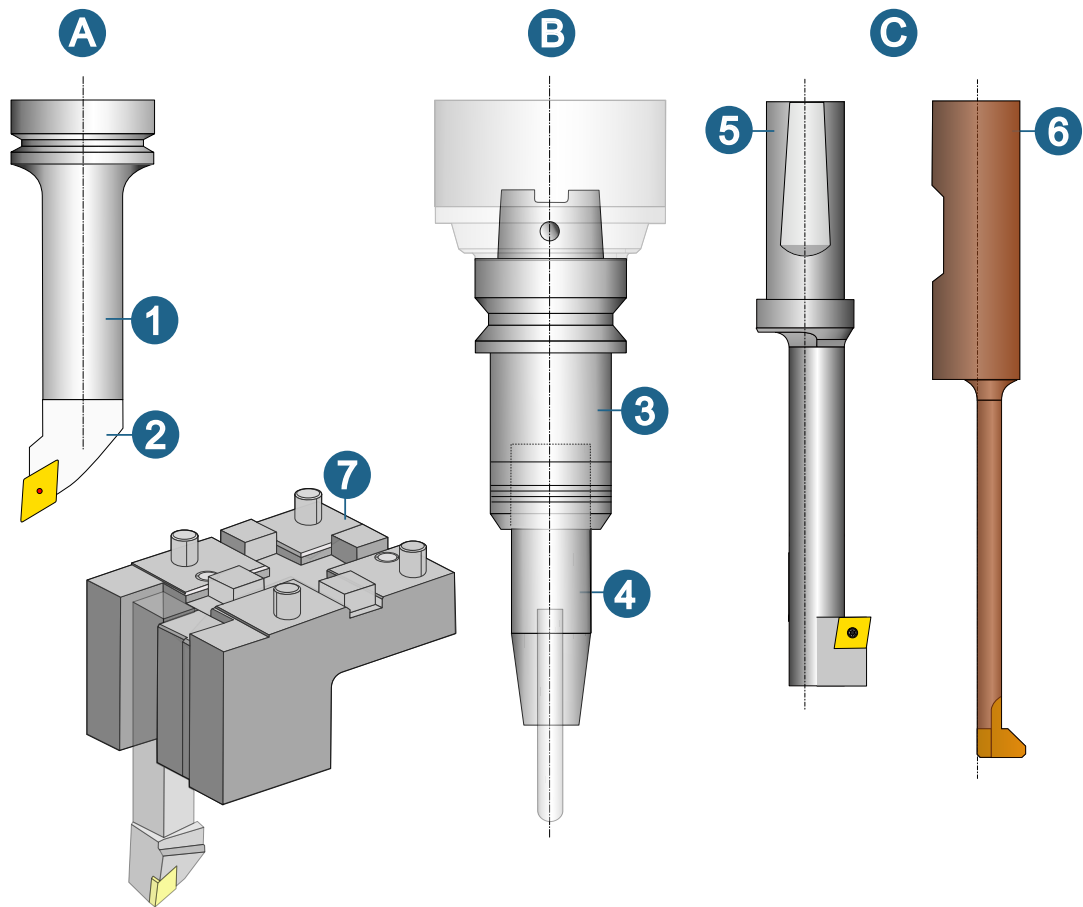
TOOL Builder

Introduction

Many tool suppliers offer 3D models of their tools. You can use the TOOL Builder to interactively prepare these models so they can be used for calculation, collision checking and simulation.

The following tools and tool elements are supported:

- ⑦ **Static holder** (turning with turret)



By default, tools are assigned the name of the document with which they were created.



CAD entities, which are used to define a tool holder, holder, static holder or extension, are locked against further editing.

Steps for creating

Turning tool: Creating a static holder for machining with a turret

Import the 3D data provided by tool manufacturers and create a static holder for turning operations with a turret.



To create a static holder for machining with a turret with the TOOL Builder, proceed as follows:

In the tool database, on the **Static holders** → **New** → **Turret holder** → **Geometry** dialog page, click the icon to start the TOOL Builder.



Then use the **File** → **Open** dialog to open a 3D geometry file in *.step, *.stp, *.iges, or *.hmc format.

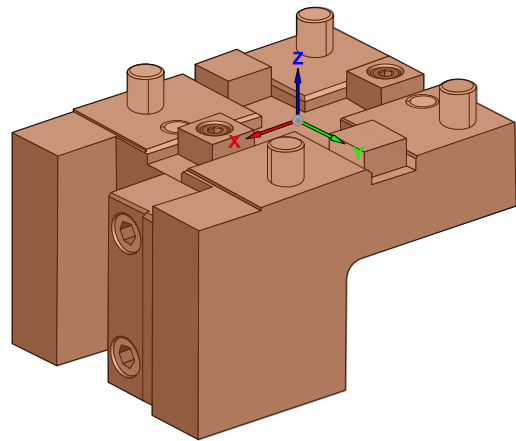
Define the body



Define the geometry of the static holder.

The geometry of the holder is automatically detected, so that after opening the file, the **Elements** of the body are already selected and highlighted in color.

The number of elements is displayed.



Define the holder axis

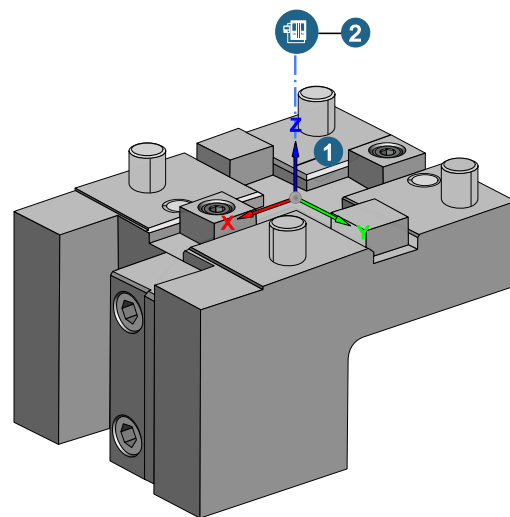


To do this, **Select** the required elements (for example, the reference coordinate system ①).

Alternatively, select **2 Points**. The order of selection defines the direction of the axis.

If necessary, invert the direction by double-clicking the arrow of the axis or activate the **Invert** function.

The program attempts to automatically detect the **Machine side** ②. Correct the machine side if necessary.



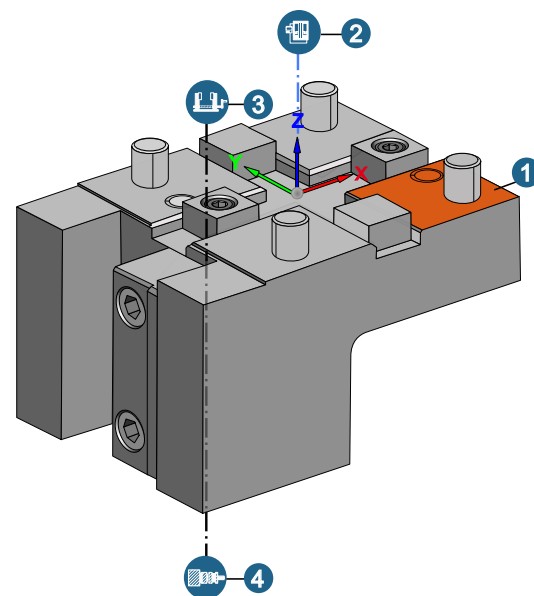
Define the holder coupling



To do this, select a **Face** or a **Point**.

The selected element ① is highlighted in color. Specify the **X direction** of the reference coordinate system. To do this, select the X axis of the reference coordinate system, **2 Points**, or one **Point**.

Machine side ②, **Clamping side** ③, and **Workpiece side** ④ are displayed.



Click the button to rotate the X axis direction by 90°.

Define the tool axis



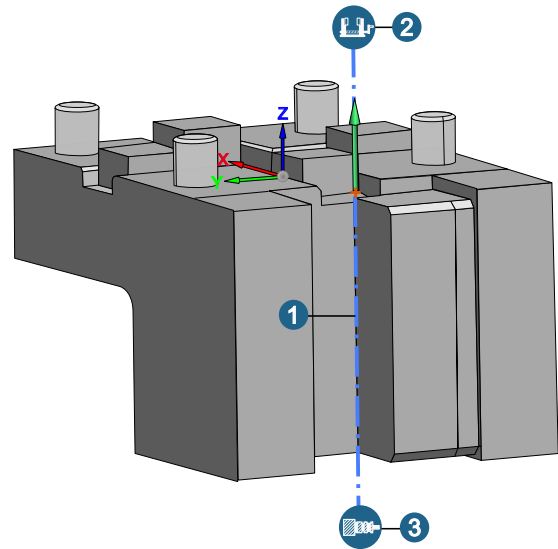
To do this, **Select** the required elements ①.

Alternatively, select **2 Points**.

The order of selection defines the direction of the axis.

Invert the **Direction** by double-clicking the axis arrow or activate the **Invert** function.

Clamping side ② and **Workpiece side** ③ are displayed.



Define the tool coupling



Select a **Face** ① or a **Point** to define the area of the tool coupling.

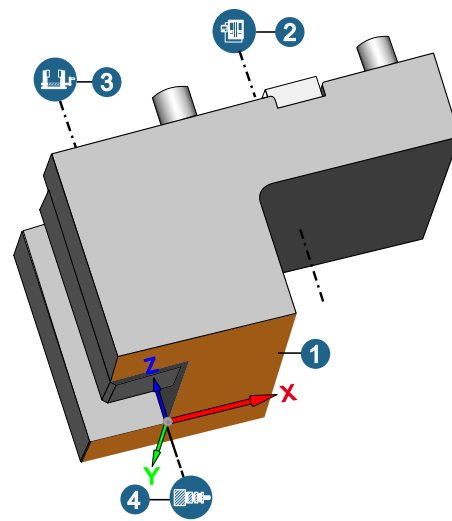
If necessary, manually adjust the **X direction** of the axis. To do this, **Select** the required elements. Alternatively, select **2 Points** or one **Point**.

The order in which the points are selected defines the direction of the axis.

Machine side ②, **Clamping side** ③, and **Workpiece side** ④ are displayed.



Click the button to rotate the X axis direction by 90°.



Export the data



Transfer the imported 3D file with the previously defined elements to the tool database as a static holder.

CAD

Introduction

Documentation and help

Send to support



Send the current document to support.

Help → **Send to support**

2024

Save the currently open document as a *hyperMILL*® SHOP Viewer document and send it as an e-mail attachment to your OPEN MIND support contact. The computer's e-mail client is started. Explain the issue. Enter the e-mail address of your OPEN MIND support contact under **To** as the recipient.

User interface

Tabs

Model

Assign entities to current layer

Assign selected entities to the current layer. To do this, select the **Assign entities to current layer** command on the context menu.

Default settings

Options / properties



Load and locally modify the defaults for the model, the model structure and the graphical attributes of the document and the software.

File → **Options** → **Options / properties**

Graphic > System > Navigation

3D input device: In order to remain compatible with older products from the manufacturer 3Dconnexion, we recommend selecting the previous driver (**Legacy** option). Select the **3DxWare SDK4** option to support the current functionality.

Sketcher options



Options for the Sketch and V-sketch commands

File → **Options** → **Sketcher options**

V-sketch options

Strict chirality: If the option is not selected, the point-arc constraint (or the point-line constraint) can slide beyond the imaginary extension of the entity (line and circle). If the option is selected, it does not slide, but remains limited to the existing part of the arc or line.

Data interfaces

OPEN MIND Software

Import and save different OPEN MIND data.

Own file formats

*.bnd **Table 1. Options for opening a *.bnd file**

Property	Description
General	Entity precision: Enter a value that is used for the same-named Entity precision property of the CAD entities. Unit of measure: Read in data in the Millimeter unit of measure or Inch unit of measure. Select the unit of measure.

*.omx **Table 2. Options for opening an *.omx file**

Property	Description
General	Entity precision: Enter a value that is used for the same-named Entity precision property of the CAD entities. Unit of measure: Read in data in the Millimeter unit of measure or Inch unit of measure. Select the unit of measure.

Import PMI with MBD

For MBD ¹ from STEP AP242, SOLIDWORKS, Siemens NX, CATIA V5, and PTC Creo, dimensions, surface finishes, and tolerances are imported, automatically evaluated by the software, and linked to faces and their measurements. Tags are assigned for this purpose.

Example 1. MBD tags (selection)

T_PMI_DIM_TOL_FORMAT

T_PMI_DIM_TOL_NUM_LOWER

T_PMI_DIM_TOL_NUM_UPPER

T_PMI_DIM_TYPE

T_PMI_DIM_VALUE

These can be viewed using the **Tags** → **Edit info** command, for example.

In the **T_PMI_DIM_REFERENCES** tag, the name of the referenced dimension, such as a length dimension, is used to create a link to the dimensioned faces. The names of the entities in the **T_NAME** tag must be unique.

In the settings for importing CAD data, enable the **Convert hidden entities** option under **Entity conversion modes**.

When creating the dimensions in the upstream CAD software, make sure that the dimensions are generated with reference to faces and not to face edges. When referencing edges, the faces cannot be clearly assigned. No subsequent manual adjustment of dimensions may be made. Tolerance values and tolerance classes must match.



Shape and position tolerances are imported as dimensions and can be visually evaluated by the user.

¹Model Based Definition



Analysis

Attributes

Local curvature



Get information about local curvature attributes of a face or a curve.

Analysis → **Local curvature**

Configuration

- For face and curve on face:
 - A point on an entity, the direction of the normal, and the UV directions can be generated individually as required.
- For individual curve:
 - A point on an entity, the direction of the normal, and the UV directions can be generated individually as required.

Edit and modify

Edit

Copy



Copy to clipboard.

Edit → **Copy**

Control the behavior for copying

The settings from the `copy_defaults.xml` file are used when copying entities from document to document via the clipboard.

The file with the general preferences is in `C:\Program Files\OPEN MIND\hyperCAD-S\[version]\files\factorysettings`. For individual presets copy this file in the folder `C:\Users\[USER]\AppData\Roaming\OPEN MIND\hyperCAD-S\[version]\commands\defaults`. If the folder does not exist, create the folder. Customize the settings in the copied file.

The file `copy_defaults.xml` has the following structure:

```
<?xml version="1.0"?>
<PBag name="filecmd.Copy"
  version="1">
  <property name="SaveAllMaterials"
    type="omsLong"
    uuid="40dff002-f0e9-59f9-b8a8-04776d96aea9"
    dimension="Dimensionless">1</property>
  <property name="SaveMating"
    type="omsLong"
    uuid="40dff002-f0e9-59f9-b8a8-04776d96aea9"
    dimension="Dimensionless">0</property>
  <property name="SaveSpreadsheet"
    type="omsLong"
    uuid="40dff002-f0e9-59f9-b8a8-04776d96aea9"
    dimension="Dimensionless">0</property>
  <property name="UseCurrentWP"
    type="omsLong"
    uuid="40dff002-f0e9-59f9-b8a8-04776d96aea9"
    dimension="Dimensionless">1</property>
</PBag>
```

Attribute	Elements or property	Configuration possibilities
SaveAllMaterials	Copy material.	0: Do not copy
SaveMating	Copy mating	1: Copy
SaveSpreadsheet	Copy variables.	
UseCurrentWP	Copy current workplane	

The attributes `uuid` contained in the XML structure are the same everywhere. Every user can use the file.

If, for example, mating properties or spreadsheet variables are required, the corresponding options must be enabled in the files `copy_defaults.xml` and `paste_defaults.xml`.

Align dimensions

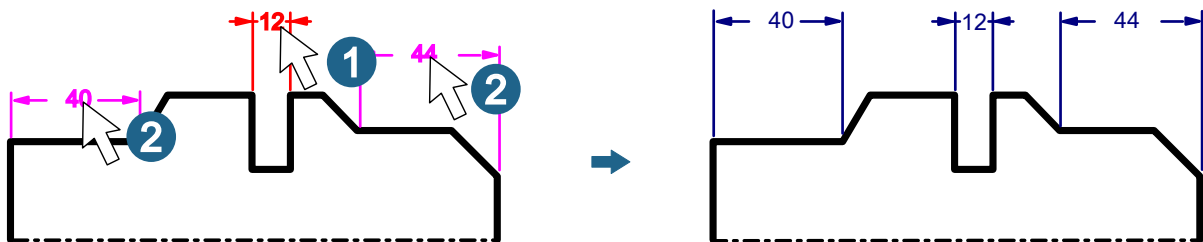


Align multiple dimensions to reference.

Edit → Align dimensions

2024

Align multiple dimensions to reference. The cursor changes. First select the reference dimension to align to ①. Then select the dimensions to be aligned ②. If the dimension to be selected is not permitted due to the direction, feedback about an incorrect selection is displayed.



Stretch 2D



Stretch two-dimensional contour curves.

Edit → Stretch 2D

2024

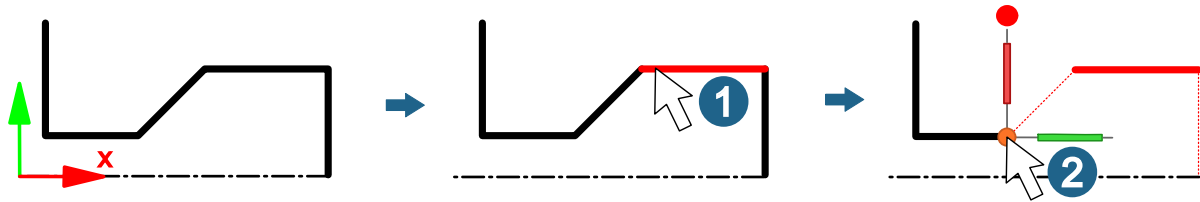
Stretch two-dimensional contour curves. Stretch a turning contour to adjust it to the center of the tolerance.



It is advisable to adjust the turning contours with this command. Use this adapted contour as a 2D model for the turning jobs that is also transferred to the SIMULATION Center and included in the collision check.

Select

Entities: Select curves ①. The number of selected entities is displayed.



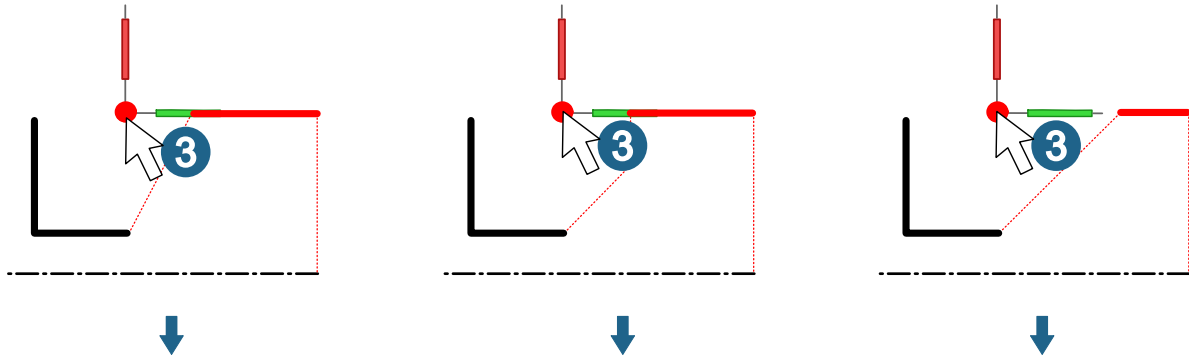
Movement

Movement is limited to the XY plane. Entities can also be moved using the manipulator. When the handles of the manipulator are moved in the graphics area, the values in the dialogue are updated. If you click an axis, the cursor jumps to the associated input.

Start: Select start point ②.

End: Select end point ③.

Delta: Enter or check the linear translation in the X and Y direction.



More options

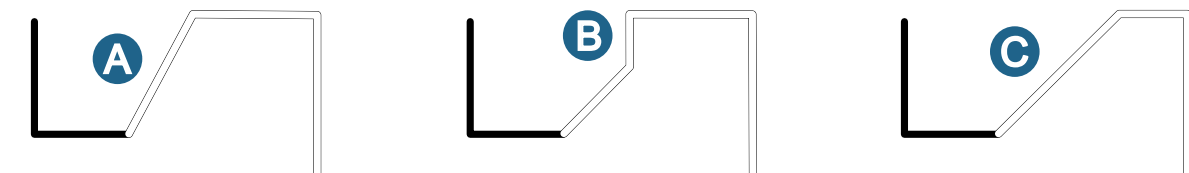
Keep original: Specify whether the selected entities should be retained.

Keep attributes: Color attributes are retained.

Control the dependencies to the adjacent curves:

Side lines: Select whether to stretch the existing, adjacent curves (A), or whether to insert new side lines for stretching (B).

Keep angles: Specify whether you want the angles of the adjacent curves to remain constant (C).



Modify

Invert curves



Invert the orientation of curves.

Modify → Invert curves

Select

The direction arrows can be optionally used to select the curves to which they refer.



When zooming out of the view, the size of the arrows remains constant. This can make it easier to select the arrows of the required direction, for example, by making a selection with **Window** or **Lasso**.

Points, curves and faces

Drafting

Round / chamfer 2D corners



Use criteria to find all required corners of a contour and round or chamfer them.

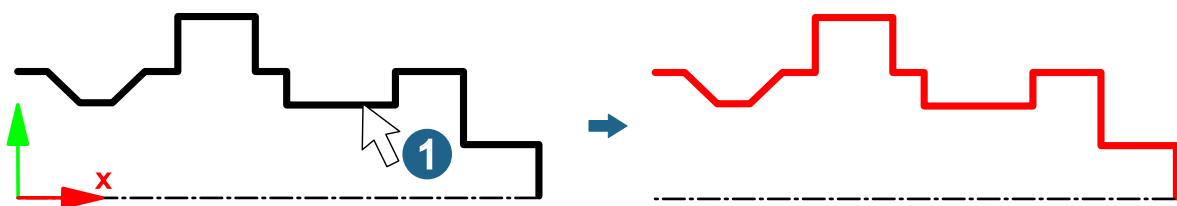
Drafting → Turning → Round / chamfer 2D corners

2024

Use criteria to find all required corners of a contour and round or chamfer them. Distinguish between inner and outer corners. Corners can optionally be left out.

Select

Curves: Select entities ①. The number of selected entities is displayed.



Concave corners

Inner corners ③ are highlighted in color and can be adjusted.

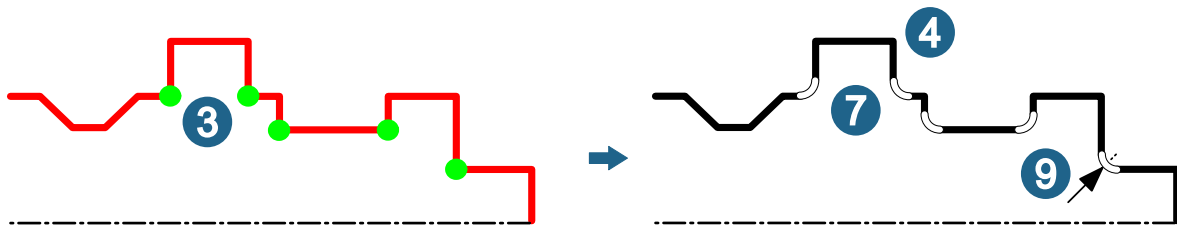
Found: The number of inner corners found is displayed.

Show: Switch the highlighting of the corners found on or off.

Mode: Use **Keep** to not change the corner ④⑤, **Fillet** to insert a fillet ⑦, or **Chamfer** to chamfer the corner ⑥.

Fillet radius: Enter a value ⑨.

Chamfer distance: Enter a value ⑧.



Convex corners

Outer corners ② are highlighted in color and can be adjusted.

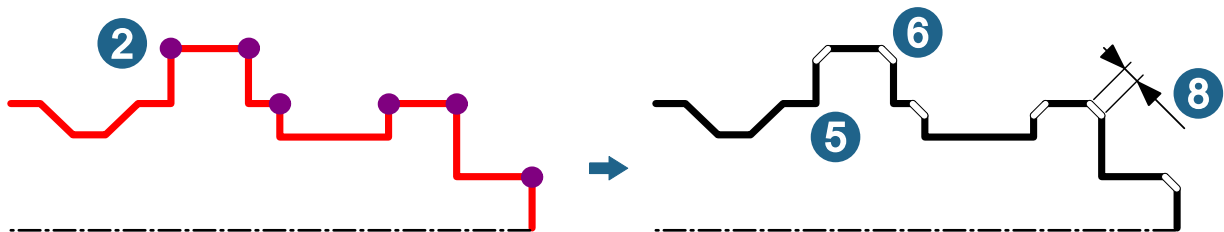
Found: The number of outer corners found is displayed.

Show: Switch the highlighting of the corners found on or off.

Mode: Use **Keep** to not change the corner ④⑤, **Fillet** to insert a fillet ⑦, or **Chamfer** to chamfer the corner ⑥.

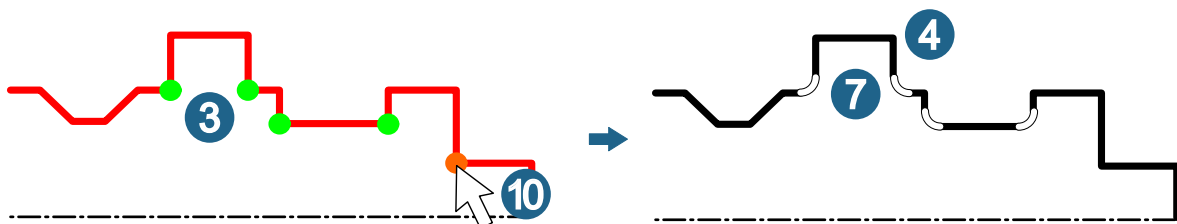
Fillet radius: Enter a value ⑨.

Chamfer distance: Enter a value ⑧.



Skip corners

Select: Select the corners to be ignored ⑩. The number of selected corners is displayed. The omitted corners are highlighted in color.



More options

Min. corner angle: Enter the minimum angle between the two curves forming the corner up to which the corner should be recognized as such.

Total corners found: The total number of corners found is displayed.

Associative: The fillets and chamfers are linked to the adjacent curve entities. When these entities are changed, the linked entities are also updated.

Automatic trim: Trim the contour automatically after inserting fillets and chamfers.

Reset 2D corners



Reset undercuts, fillets, and chamfers on sharp-edged corners.

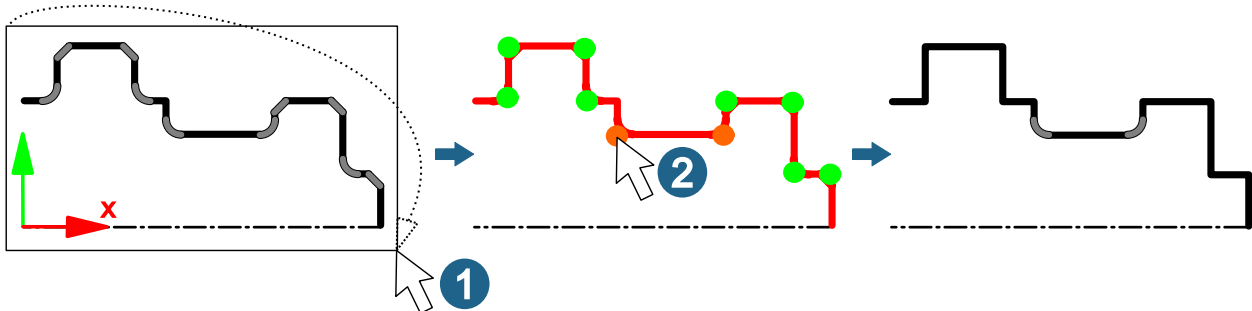
Drafting → Turning → Reset 2D corners

2024

Reset undercuts, fillets, and chamfers on sharp-edged corners as required.

Select

Curves: Select entities ①. The number of selected entities is displayed.



Fillets

Geometric recognition: Recognize fillets based on the geometry.

Max. radius: Specify the radius up to which a curve is to be interpreted as a fillet. Enter a value.

Chamfers

Geometric recognition: Recognize chamfers based on the geometry.

Max. diagonal length: Specify the diagonal length up to which a curve is to be interpreted as a chamfer. Enter a value.

Angular tolerance: Enter an angle value indicating by how much the chamfer may deviate from 45° in order to still be considered a 45° chamfer.

Undercuts

By tag recognition: Undercuts are marked with a tag when they are inserted using the [2D undercut \[71\]](#) command. This tag is used to recognize undercuts.

Skip corners

Select: Select the fillets, chamfers, and undercuts to be ignored ②. The number of selected fillets, chamfers, and undercuts is displayed and highlighted in color.

More options

Total corners found: The total number of corners found is displayed.

Show: Fillets, chamfers, and undercuts are highlighted in color.

2D undercut



Insert predefined 2D undercuts.

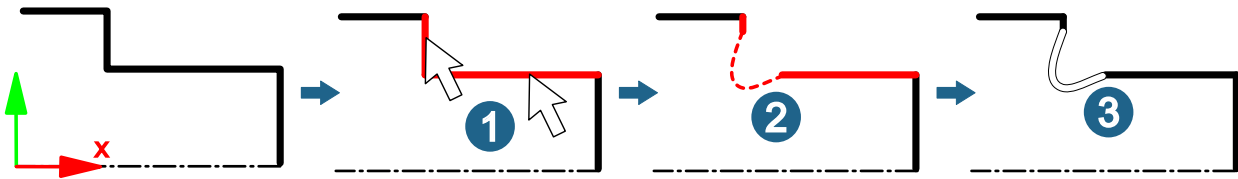
Drafting → Turning → 2D undercut

2024

Insert undercuts for local rewinds into a 2D contour for turning in a form standardized according to DIN 509. The inserted undercut curves are marked with a `T_UNDERCUT2D_TYPE` tag.

Select

Curves: Select the two curves at right angles to each other of a corner where an undercut is to be inserted ①. The number of selected entities is displayed.



Select undercut

Shape: Select the shape of the undercut based on the standard offered (E, F, G, H) ②.

Configuration: The software automatically suggests a size range if the workplane is set for turning the 2D contour. The X axis of the workplane must lie in the rotational axis of the contour. Optionally change the size range.

More options

Automatic trim: Trim the contour automatically after inserting undercuts ③.

Associative group: The curves of the undercut are grouped. The group is given the shape and the selected configuration of the standard as its name. When these entities are changed, the linked entities are also updated.

Trim 2D corners



Trim 2D corners.

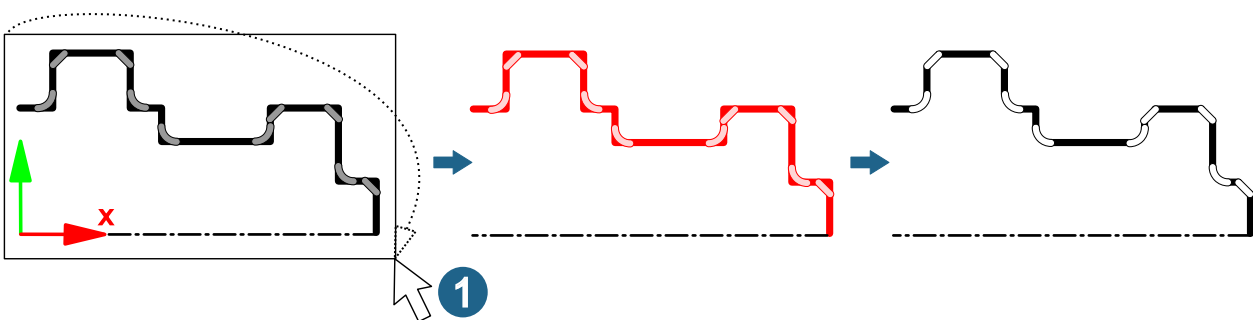
Drafting → Turning → Trim 2D corners

2024

Trim 2D corners in relation to existing, previously added fillets, chamfers, and undercuts for turning that were not trimmed automatically.

Select

Curves: Select entities ①. The number of selected entities is displayed.



Shapes

Sweep (two contours)



Generate side faces between two contours.

Shapes → Sweep (two contours)

2024

Generate side faces between two contours with the same number of segments, optionally for planar contours with top and bottom faces and as a solid.

First contour

Curves: Select entities. The number of selected entities is displayed. All selected entities must complement each other to form a closed contour.

Second contour

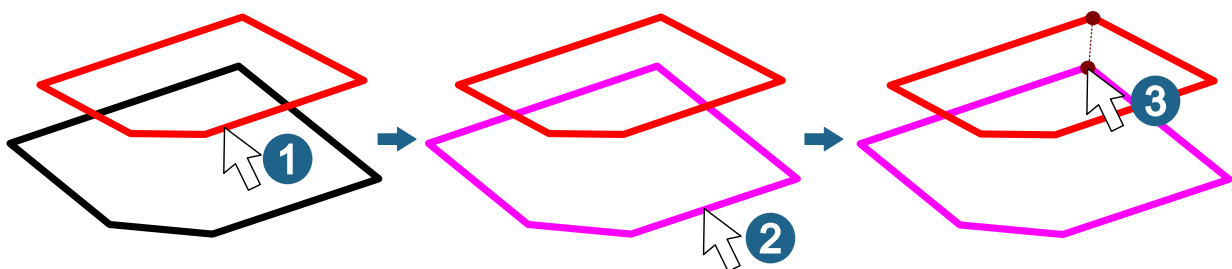
Curves: Select entities. The number of selected entities is displayed. All selected entities must complement each other to form a closed contour.

Synchronization points

Relate a point or a position on the first contour to a point or a position on the second contour to avoid unwanted twisting. The software suggests a solution. A maximum of one synchronization point per contour is possible.

First: Select point or snap position.

Second: Select point or snap position.



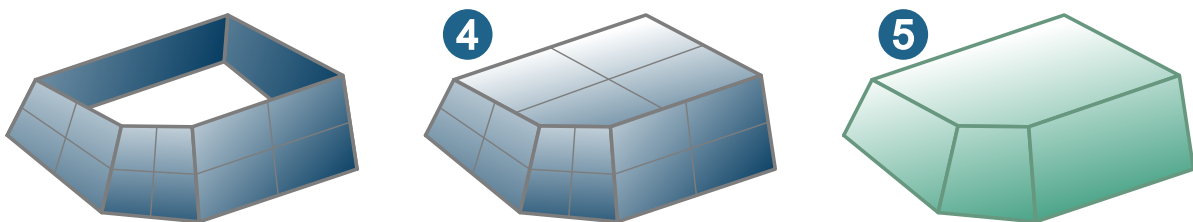
More options

Invert: Correct the orientation of the contours if the faces intersect incorrectly in the preview.

Invert normals: The direction of the face normals of the generated faces is reversed.

With bases: A top face and a bottom face are created if the selected curves and face boundaries result in a closed contour line and are flat.

Solid: Collect the faces into a solid.



From grid

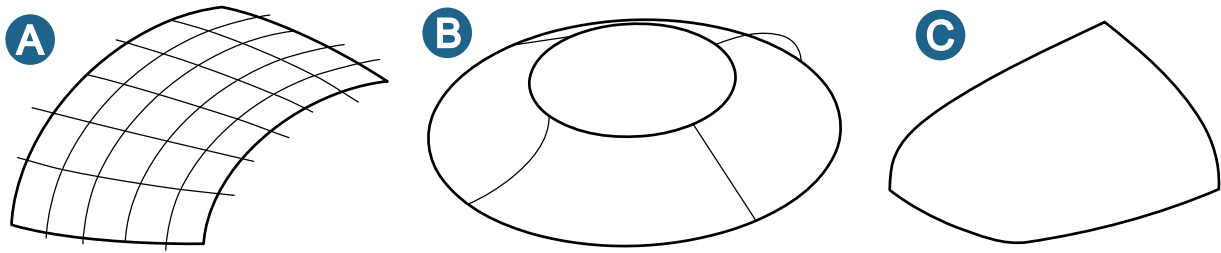


Create a face from several curve chains in one direction plus several curve chains in the other direction.

Shapes → From grid

2024

Create a face from several curve chains in one direction plus several curve chains in the other direction (A). The number of curves per chain can vary. The curve chains in both directions must intersect within a given tolerance. If curve chains protrude, the last intersection point is used for the edge boundary. The resulting protruding curve segments are not included in the face. The face can be created closed in one direction (B). Both directions can converge at the start or end of the face, or the face can just consist of three edge boundaries (C).



Click the message line for the “Intersections missing” message on the **Info** tab to display the positions where this situation occurs.

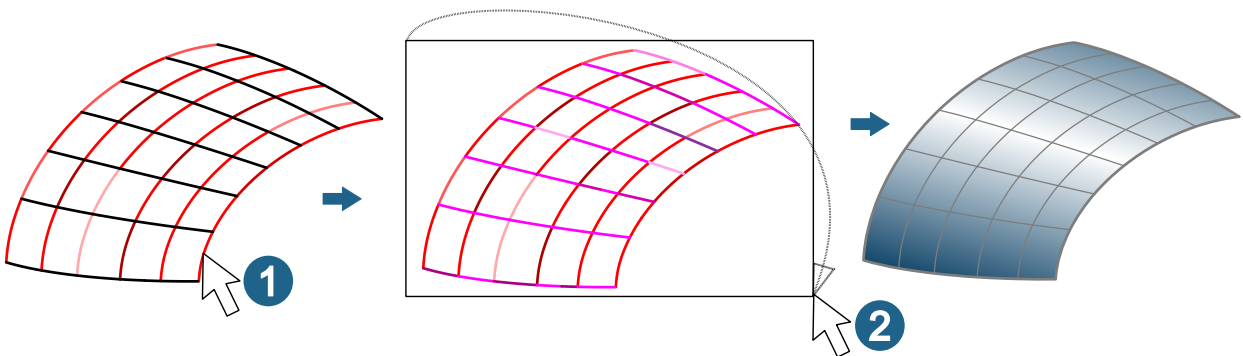
Click the message line for the “Most distant points” message on the **Info** tab to display the positions where this situation occurs.

First direction

Curves: Select curves in the first direction ①. The number of selected entities is displayed.

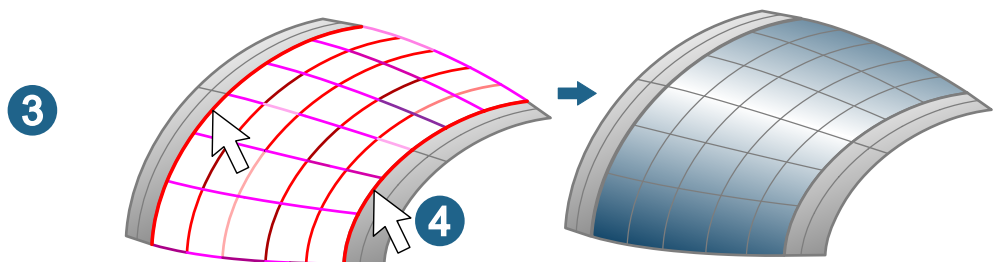
Second direction

Curves: Select curves in the second direction ②. If all curves in the first direction have already been selected, the curves in the second direction can also be selected via the curves in the first direction, for example, using a window selection. The curves in the first direction are not automatically used as curves in the second direction. The number of selected entities is displayed.



Continuity

It is possible to control the continuity of the transition to an existing face ③. To do this, select the boundary of the existing face and no curve at this position ④. Select whether the continuity between the existing face and the face to be created should only be maintained with regard to **Position** G0 or also with regard to **Tangency** G1.



Intersecting curves

Tolerance: Enter a value for the maximum permissible distance at which curves should still be tolerated as intersecting.

Precision

Distance of the furthest point: Outputs the furthest distance between the face to be generated and the curve chains.

Parametric modeling

Automatic rebuild on / off



Switch automatic rebuild on and off after a change in the parametric design.

Edit → Parametric → Automatic rebuild on / off



Solids

2024

Control automatic rebuild after changes in the parametric design as required. For example, you can make several changes in a sequence and only then update the parametric design. While rebuilding is interrupted, most commands are grayed out. The changed values of the dimensions or dependencies are indicated by the symbols < and >. A single Undo/Redo step is created for the entire interruption.



Add the command to the toolbar in the graphics area. To do this, use the **File** → **Options** → **Toolbars and tabs** command.

Solids, features and meshes

Features

Protrusion (two contours)



Create a protrusion between two contours that define the shape of the protrusion.

Solids

Features → Protrusion (two contours)

2024

Create a protrusion between two contours with the same number of segments that define the shape of the protrusion.

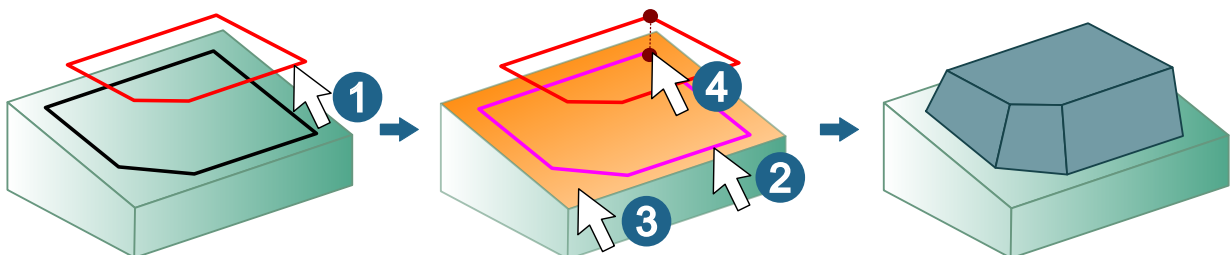
First contour

Curves: Select entities ①. The number of selected entities is displayed. All selected entities must complement each other to form a closed contour.

Second contour

Curves: Select entities ②. The number of selected entities is displayed. All selected entities must complement each other to form a closed contour.

Face: Select a face of the existing solid in which to embed the protrusion to be generated ③.



Synchronization points

Relate a point or a position on the first contour to a point or a position on the second contour to avoid unwanted twisting ④. The software suggests a solution. A maximum of one synchronization point per contour is possible.

First: Select point or snap position.

Second: Select point or snap position.

More options

Invert: Correct the orientation of the contours if the side walls of the protrusion intersect incorrectly in the preview.

Invert normals: The direction of the face normals of the generated faces is reversed.

Slot (two contours)



Create a slot between two contours that define the shape of the slot.

Solids

Features → Slot (two contours)

2024

Create a slot between two contours with the same number of segments that define the shape of the slot.

First contour

Curves: Select entities ①. The number of selected entities is displayed. All selected entities must complement each other to form a closed contour.

Second contour

Curves: Select entities ②. The number of selected entities is displayed. All selected entities must complement each other to form a closed contour.

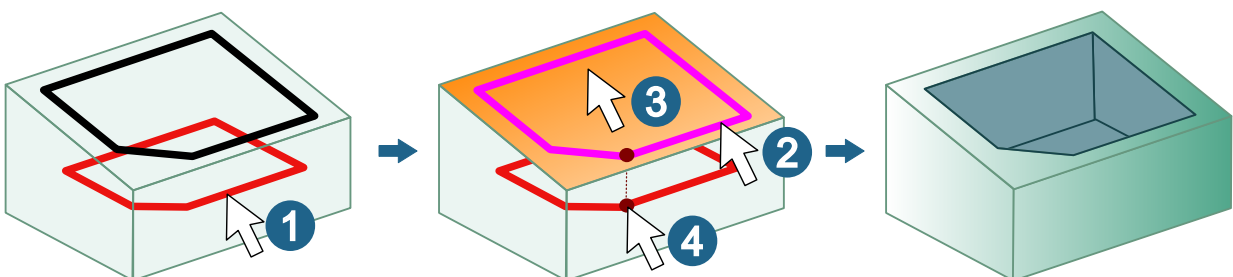
Face: Select a face of the existing solid in which to embed the slot to be generated ③.

Synchronization points

Relate a point or a position on the first contour to a point or a position on the second contour to avoid unwanted twisting ④. The software suggests a solution. A maximum of one synchronization point per contour is possible.

First: Select point or snap position.

Second: Select point or snap position.



More options

Invert: Correct the orientation of the contours if the side walls of the slot intersect incorrectly in the preview.

Invert normals: The direction of the face normals of the generated faces is reversed.

Solid

Repair open solid



Repair open solid.

Solids

Modify → Repair open solid

Cover openings

Create capping: Cover openings caused by removed hole geometries and other openings in the solid. Openings that touch several faces are covered. Planar and non-planar openings are covered. The capping is created as a separate open solid.

Meshes

Mesh from faces



Creates meshes from faces and faces in solids.

Solids

Shapes → Mesh from faces

More options

Create single mesh: Create a single mesh from all selected faces.

Keep original: Specify whether the selected entities should be retained.

Keep attributes: Color attributes are retained.

Design electrodes

Imported CAD geometry data often does not have the quality required to create an electrode. Despite undercut and missing faces, gaps between faces, and overlapping faces, the software allows electrodes to be created on the basis of such CAD geometry data. A warning is only issued if the electrode geometry does not specify a closed shape. It is up to the user to assess whether such a situation will have a negative impact on the generation, NC programming, and production of the electrode, and whether it needs to be corrected in advance.

Electrode options



Select default settings for electrodes.

Electrode

File → Options → Electrode options

Geometry

Prepare

Closing holes from single faces: Features such as holes and breakthroughs inside a *Domain* (with a single continuous face boundary) can be closed automatically. Switch off the option for eroding lettering, for example.



Technology

- **Generator program:** Select the required generator program. Displays all generator programs that are entered in the XML file for generator programs selected in the **XML file for generator program** option. Also see [Preparing generator programs \[80\]](#).

Folders

XML file for generator program: Select an `electrode_generator_programs.xml` file with generator programs for performance parameters for eroding. Also see [Preparing generator programs \[80\]](#).

Create partial



Create electrode geometries without an electrode block and electrode holder.

Electrode

Electrodes → Create partial

More options

Closing holes from single faces: Features such as holes and breakthroughs inside a *Domain* (with a single continuous face boundary) can be closed automatically. Switch off the option for eroding lettering, for example.

Create user defined



Extend existing electrode geometry to create a complete electrode.

Electrode

Electrodes → Create user defined

Mode

Default: Create an electrode with extensions.

Simplified: Create an electrode without extensions. The selected faces of the electrode shape are transferred to the electrode without any adjustments or additions. It is not necessary to repair the CAD geometry beforehand. Enter a minimum distance between electrode and block with the **Distance** value.



Faces for a user-defined electrode (**Electrode shape** → **Shape**) must not have been previously selected as entities in **Create reference system**. This would lead to a collision check taking place within itself.

Modify eroding path



Make eroding path visible and modify.

EDMconNG

Electrodes → Modify eroding path

2022.1

Blind

Motion

In addition to lines, arcs can also be selected as eroding paths. The curves can run three-dimensionally in space and do not have to lie in XY planes parallel to the EDM reference. The eroding path is broken down into individual waypoints.

Through



Motion

In addition to lines, arcs can also be selected as eroding paths. The curves can run three-dimensionally in space and do not have to lie in XY planes parallel to the EDM reference. The eroding path is broken down into individual waypoints.

Using user-defined tags for electrode documentation

Create your own user-defined tags for additional information, add values, and use them in the title block of electrode documentation.

For electrode assembly

Preparation in the title block template

1. In the hyperCAD-S\files\printingtitleblocks folder, switch to the folder of the language set for the software.
2. Open the `assembly_electrode_tb.hmc` file.
3. Create a tag with **Tags** → **Create** → **Quantified tag**.
4. Determine the position within the title block and insert a link to the tag in **Drafting** → **Text**.
5. Select the new tag on the **Tags** tab and select the **Assign** command on the context menu. Select the text there and apply the entries.
6. Save the file.

In the electrode document

1. New tags that have been added to `assembly_electrode_tb.hmc` are shown at the end of the list in the **Electrodes** → **Print** and **Electrodes** → **Modify EDM parameters** commands when the **Document** option is selected.
2. Change the content of the tags.
3. Generate the required electrode documentation.

For single electrode

Preparation in the title block template

1. In the hyperCAD-S\files\printingtitleblocks folder, switch to the folder of the language set for the software.
2. Open the `single_electrode_tb.hmc` file for a new user-defined tag for a single electrode.
3. Create a tag with **Tags** → **Create** → **Quantified tag**.
4. Determine the position within the title block and insert a link to the tag in **Drafting** → **Text**.
5. Select the new tag on the **Tags** tab and select the **Assign** command on the context menu. Select the text there and apply the entries.
6. Save the file.

In the electrode document

1. Use the **Tags** → **Create** → **Quantified tag** command to generate a tag with identical tag names.
2. Assign this tag to the required electrode – as a whole, that is, to the electrode group in the model.
3. Use the **Tags** → **Edit info** command to change the content of the tag.
4. Generate the required electrode documentation.

Export to EDM



Convert electrodes into readable EDM files.

Electrodes → **Export to EDM**

EDMconNG

2024

Convert electrodes into readable EDM files.

Reports for the following eroding machine types can currently be converted.



Manufacturer	Software	Version	Output 3-point path	Virtual electrode	Rotating electrode	Output 3D path
Zimmer & Keim	Alphamoduli		x	x	x	x
OPS Ingersoll	Multiprog		x	x	x	
ONA			x	x	x	
+GF+HMI	AC FORM HMI		x	x	x	
Exeron	Exoprog		x	x	x	

EDM export

Select the eroding machine with the relevant software version from the drop-down list.

EDM file name

Enter a file name without a file extension.

Electrode list

Only selected: Optionally perform an export with selected electrodes only. To do so, select the option. Select the relevant electrodes with the left mouse button. Otherwise, all listed electrodes will be included in the export. The list is sorted by name.

Preparing generator programs

Manage erosion performance parameters using generator programs instead of individual erosion technology settings. For example, transfer existing generator programs from the eroding machine. Write the optional text information for the software to the `Name` attribute. Insert information for easier management of the generator programs in the `description` attribute:

```
<?xml version="1.0"?>
<GeneratorPrograms>
  <program name="VDI33GAP15VDI41GAP10" description="EDM_example1" />
  <program name="VDI24GAP10VDI18GAP08" description="EDM_example2" />
</GeneratorPrograms>
```

The generator programs are offered for selection on the **Electrodes** menu in the commands **Create**, **Create user defined** and in **Modify EDM Parameters** as well as in the **Electrode options**. For this purpose, insert this file into **Electrode options** → **Folders** → **XML file for generator program**. The location and name of this file are freely selectable.

CAM programming

Overview

Toolpath properties



Analyze toolpaths.

CAM → Toolpath properties

Configuration

Toolpath settings → Data filter

The following options appear in the user interface depending on the type of job – if the data is available.

Show tool reference: Select the tool reference point to which the display of the toolpath with ball mill or lollipop cutter should refer.

The globally set color for toolpaths **NC path GO** and **NC path G1** from the **hyperMILL settings** is used to display additional toolpath profiles.

- Tip only
- Center only
- From Job
- Both variants

Rest material display



Display remaining rest material for visual analysis.

CAM browser → Jobs → Shortcut menu → Utilities → Rest material display

2024

Display remaining rest material for visual analysis. A comparison of the pre-calculated current machining status with a workpiece is carried out and the differences are displayed. A current stock chain is required for this.

Color/rest material thickness

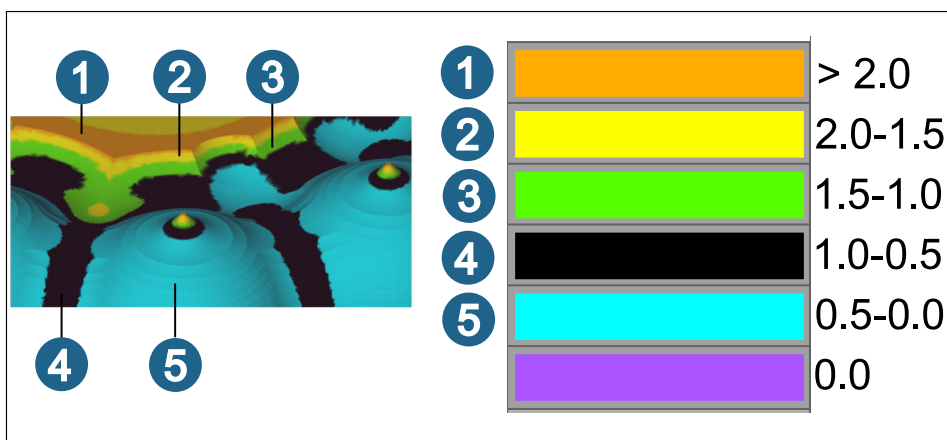
Color: Double-click with the left mouse button on the color field in the table to open the dialog for changing the color of the interval.

Rest material thickness: Double-click with the left mouse button on a table row to change a value of the interval limit.

Limits

Minimum rest material that is taken into account in the rest material display. Maximum rest material that is taken into account in the rest material display. Enter **Min.** and **Max.**

Example: Default values for color and rest material thickness



Tolerance

Model geometry: Value for the accuracy of the rest material display.

Update: Update the display of the rest material.

More options

Number of intervals: The rest material is displayed in five intervals. A higher value increases the fineness of the display.

Exact computation: Refine the calculation of the rest material display. The calculation is performed automatically in relation to the size of the workpiece.

Optimal barrel cutter



Optimize parameters of barrel cutters based on surface and curve information.

CAM → **Optimal barrel cutter**

CAM browser → **Jobs** → **Job** → **Tool**: Select a barrel cutter. A function for optimizing the barrel cutter is displayed in the tool definition.

2024

For machining with barrel cutters, the optimal shape of the barrel cutter must first be found. This ensures that the surface to be machined is completely machined as required. The parameters of the required barrel cutter are optimized on the basis of surface and curve information. A surface and several curves in relation to this surface are included in the analysis. Preset values for the dimensions of the barrel cutter are taken from the tool definition in hyperMILL® when called.

Select

Surface: Select the surface for which an optimal barrel cutter geometry is to be found for machining.

Curve: Select one or more curves in relation to this surface that represent the planned machining path. The number of selected elements is displayed.

Barrel

When calling the dialog from the hyperMILL® job definition (**Tool** dialog page) and selecting a barrel cutter, the **Cutter parameters** of the selected tool are adopted. The parameters can then be changed manually.

For **Base diameter range** and **Barrel radius range**, enter the minimum and maximum values between which the optimal value is to be calculated. All other parameters are considered fixed when the calculation is running.

Both the variables and the fixed parameters are included in the calculation of the optimal barrel radius and base diameter.



If the dialog is started via the **CAM** → **Optimal barrel cutter** menu, preselect an option for a conical, tangential or general barrel cutter for **Barrel type** and an option (ball, radius, sharp) for **Tip type**.

? If the dialog is started via the hyperMILL® job definition (**Tool** dialog page), these parameters are already preset and can no longer be changed.

Options

Contact parameter: Enter how the preview of the tool contact point of the barrel cutter should be positioned in relation to the selected curve. For example, if the value 0.5 is entered, the tool preview is displayed at the center of the curve.

Create 3D: Create a 3D model of the calculated tool geometry in the graphics area.

Verification

- **Single curve**: Select an individual curve to specify where the tool preview should be displayed. Enter a value or vary it using the slider. The determined tool geometry is moved along the curve
- Use **Face offset** to enter a value that enlarges the internal tool shape to allow a preview of the contact area between the tool and the surface. The smaller the value, the smaller the preview.

Further options

Cutting side: Change the tool position in relation to the surface normal of the selected surface (inside or outside).

Infeed direction: Reverse the infeed direction of the tool in relation to the orientation of the selected curve.

Automatic computation: The optimal tool geometry is displayed as a preview for the set values and only in the slider position that defines the contact parameter.

Global mode: If **Global mode** is selected for the calculation, every point in the selected curves is taken into account so that the optimal tool shape is calculated within all selected curves. Depending on the geometry, the calculation may take some time. Otherwise, or if the slider is moved, the calculation is performed continuously for each point currently reached. When the slider stops, the result is updated. Different tool dimensions are obtained for the various points analyzed on the curves.

Result

Optimal barrel radius: The determined maximum possible barrel radius that fits the area is output.

Optimal barrel diameter: The determined maximum base diameter (shaft side) that fits the surface is output.



The function does not carry out a collision check! This is performed in the cycle. The optimum shape is determined using an analytical calculation on the selected surface, taking into account a safe tolerance, in order to obtain the best possible shape.

Current restriction: Currently, the tool is always considered normal to the curve during the calculation. A lead angle is expected to be introduced with one of the next versions.

Adaptive construction

Close holes



Close openings within faces.

Shapes → Close holes

The openings for which a calculation has failed are displayed by clicking the message on the **Info** tab.

Advanced

Max. perimeter length: Maximum length of the circumference of an opening that is still to be closed.

Detect and close multi-faces holes: Detect and close cross-face openings between faces. The selection should be targeted.

Analyzing probing results

Read in and analyze one or more measuring logs for probing jobs. Check the deviations based on the workpiece geometry in the graphics area, in tabular form, and as a trend. The deviation, that is, the actual state, is always determined in the face normal direction of the probing point on the face (target state). In the graphics area, the target and actual states are displayed visually using colored dots with the deviation values. On the **Probing** tab, sort and filter the results by probing job, probing data, and trends. All values are output in relation to the current workplane. Click a probing point in the graphics area to jump to this point on the **Probing** tab.

The probing results are also displayed if they are covered by other entities. In this case, the measure lines appear dotted and the positions are shown as asterisks instead of circles. The points can always be selected, even if they are obscured by a face, while the label with the measured values can be selected if the selection filter does not include the face.



The prerequisite is that the option **Create logs for CAD import** is enabled in **Settings** under **Probing** when creating a job list for probing jobs in the *hyperMILL*® job browser.

The **Create logs for CAD import** command is executed via the VIRTUAL Machining Center. The prerequisite for this is a corresponding virtual machine that is activated in the **Job list** → **Postprocessor** → **Machine** → **Machine administration** dialog and is available for various machine controls.

The permissible tolerances are entered in the CAM browser when the probing job is created.

The touch probe must be calibrated.

Interaction with other commands

Probing data can be selected for other commands.






If a command such as **Deform entities** or **Deformation** is active and the start and target points are selected there, you can left-click the table header of the **Nominal** and **Measured** columns to select all probing point coordinates.

The same procedure is possible if the probing points outside the tolerance are to be transferred for a probing job. To do this, only display the probing points outside the tolerance. In the probing job, on the **Contours** tab, during **Contour selection**, click the table header of the **Nominal** column. Alternatively, you can also select the relevant table cells.

Probing tab

Read in and analyze one or more measuring logs for probing jobs. A lower and an upper deviation is possible for each probing job. All probing points do not appear in the CAD model structure or on layers, and cannot be influenced or filtered by the visibility filter.

Table 3. Filter or delete the probing data using the following commands:



Icon	Commands	Explanation
	Drop-down menu	On the drop-down menu, select whether all or which of the imported files should be displayed.
	Show labels	Display labels with the measured values in the graphics area. Switch the display on and off.
	Nominal values	Display the target states. Switch the display on and off.
	Actual values – in tolerance	Display the probing data within the tolerance. Switch the display of the probed points on and off.
	Actual values – out of tolerance	Display the probing data outside the tolerance. Switch the display of the probed points on and off.
	Slider with two value input fields	Restrict the range of the displayed probing data from below and above. To do this, enter values or move the left or right bar by holding down the left mouse button. Change the restricted area as a whole by moving it with the left mouse button pressed. To do this, click the horizontal bar with the left mouse button and move the mouse.
		Remove all probing data.

The tab has three sections:

Probing jobs

Probing jobs for a workpiece or a series of workpieces with one or more measuring logs are displayed in a tree structure, sorted by probing point and imported measuring logs. The structure from the programming of the probing jobs is used and therefore corresponds to the probing jobs created in the CAM browser whose probing results have been read in.

The information in the tree structure is arranged like a table. The **Name** column contains the tree structure itself with the name of the job list, the names of the probing jobs, and the time at which the respective measuring log was created. The **Status** column summarizes whether the probing data of the substructure is within or outside the required tolerance. This status is displayed as text and graphically:

-  Deviation within tolerance
-  Deviation outside tolerance

The **Tolerance** column contains the values defined for the lower and upper deviation when the probing job was created. Further information may be displayed in the **Description** column.



On the context menu

- **Import probing data:** Select a file with probing results. Existing, previously read-in probing results are retained.
- **Open probing job:** Open probing job. The command only appears if the job linked to the probing data exists in the CAM browser of the open document.
Double-clicking with the left mouse button on the probing point to be examined also opens the probing job.
- **Zoom:** Zoom the graphic section so that the selected probing job or the selected probing points are displayed centered in the graphics area.
- **Activate only this:** Display the selected probing job in the graphics area, but do not display all others.
- **Activate all except this:** Do not display the selected probing job in the graphics area.
- **Expand all:** Fully expand the tree structure.
- **Delete:** Delete selected entry.

Probing data

Displays the results from the measuring log of a probing series for a probing job in tabular form. For this purpose, select a specific probing series in the tree structure based on the time displayed.

For each probing point (**Name** column), the target state from the geometry (**Nominal** column), the measured actual state of the workpiece (**Measured** column), and the calculated value of the deviation (**Deviation** column) are displayed in the table. The deviation is checked to see whether it is within or outside the tolerance specified in the probing job. This status is displayed graphically:

-  Deviation within tolerance
-  Deviation outside tolerance

Click the table header of the **Deviations** column to sort the values by size. If a command such as **Deform entities** is active and the start and target points are selected there, you can left-click the table header of the **Nominal** and **Measured** columns to select all probing point coordinates.

The same procedure is possible if only probing points outside the tolerance are to be transferred for a probing job. For this purpose, only display the probing points outside the tolerance. In the probing job, on the **Contours** tab, during **Contour selection**, click the table header of the **Nominal** column.

On the context menu

- **Open probing job:** Open probing job. The command only appears if the job linked to the probing data exists in the CAM browser of the open document.
Double-clicking with the left mouse button on the probing point to be examined also opens the probing job.

- **Zoom:** Zoom the graphic section so that the selected probing points are displayed centered in the graphics area.
- **Activate only this:** Display the selected probing point in the graphics area, but do not display all others.
- **Activate all except this:** Do not display the selected probing point in the graphics area.
- **Import probing data:** Select a file with probing results. Existing, previously read-in probing results are retained.

Trend

For each probing point, analyze the trend for the change in the accuracy of the machining from several measuring logs of a probing job. To do this, read in several measuring logs. Select a probing job in the probing job tree structure.

The information is arranged in a table-like tree structure. The **Name** column contains the number of the probing point. The **Run** column contains the time at which the respective measuring log was created. The measured values are displayed in the **Measured** column. The calculated value of the deviation is displayed in the **Deviation** column. The deviation is checked to see whether it is within or outside the tolerance specified in the probing job. This status is displayed graphically:



Deviation within tolerance



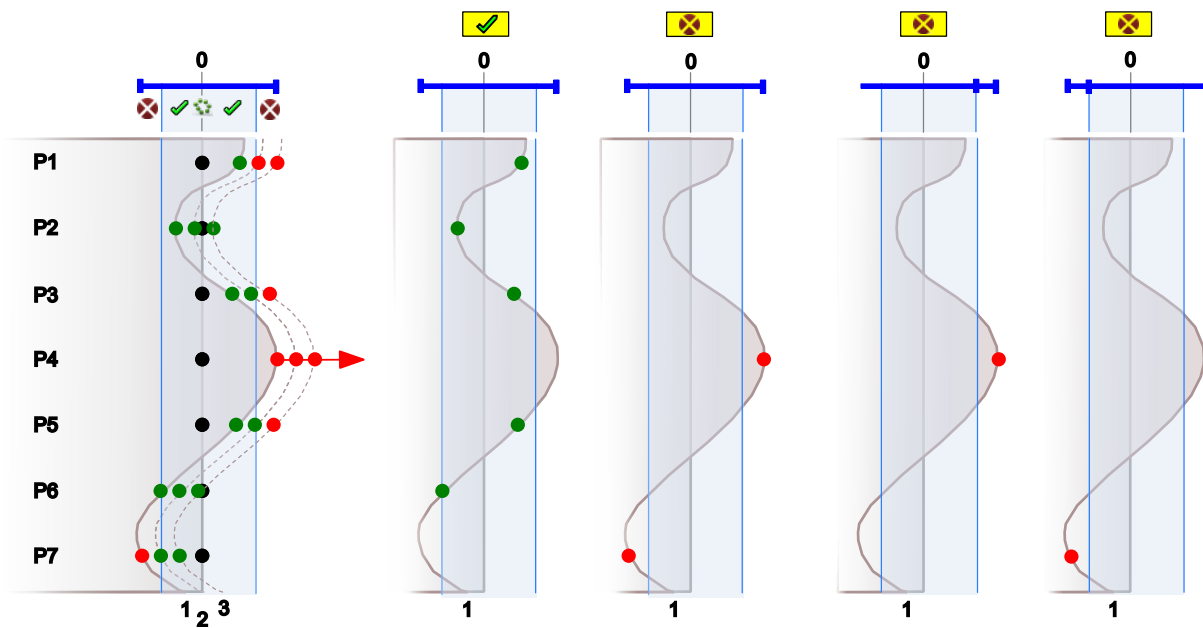
Deviation outside tolerance

Click the table header of the **Deviations** column to sort the values by size.

In the **Trend** column, the deviation is visualized by setting it in relation to the lower and upper deviation. The movement of the point and its color in comparison to the measurements provide visual information on the development of the accuracy of the production of the workpiece(s).

On the context menu

- **Open probing job:** Open probing job. The command only appears if the job linked to the probing data exists in the CAM browser of the open document.
Double-clicking with the left mouse button on the probing point to be examined also opens the probing job.
- **Zoom:** Zoom the graphic section so that the selected probing points are displayed centered in the graphics area.
- **Expand all:** Fully expand the tree structure.
- **Import probing data:** Select a file with probing results. Existing, previously read-in probing results are retained.



Import probing data



Read in the first or a probing result. Add probing result of another probing process (another workpiece).

CAM → Import probing data

Probing tab → Context menu → Import probing data

2024

Select a *.txt, *.log, or *.ompr file with measurement results. Existing, previously read-in measurement results are retained.

Probing options



Options for reading and displaying probing results.

File → Options → Probing options

2024

The settings are stored in the `applicationProbingProperties.xml` file are not document-related.

Options

Layout mode: Select which method to use to avoid overlapping labels. If **Default** is selected, the labels are displayed evenly aligned. If **Opposite** is chosen, they are displayed alternately in the other direction. If **Level of detail** is selected, the label with the greater deviation from the target value is always displayed when two labels overlap.

Color - nominal: Enter a color for the target state.

Color - within the tolerance: Select a color for probing results within the tolerance.

Color - out of tolerance: Select a color for probing results outside the tolerance.

Point size: Select a size for the point that indicates the position of the target state and the probing result.

Leader line tail length: Select the length of the leader line between the point of the probing result and the text of the label.



Font: Select a font for the text of the label.

Text size: Select a size for the text of the label.



3. Release notes

Release 2024

CAM

New default values for filleting toolpaths

- The default value for toolpath filleting (**Fillet radius**) now always refers to the **Diameter** of the tool used. In addition the **Fillet all toolpaths** function is enabled by default. This applies to subsequent cycles:

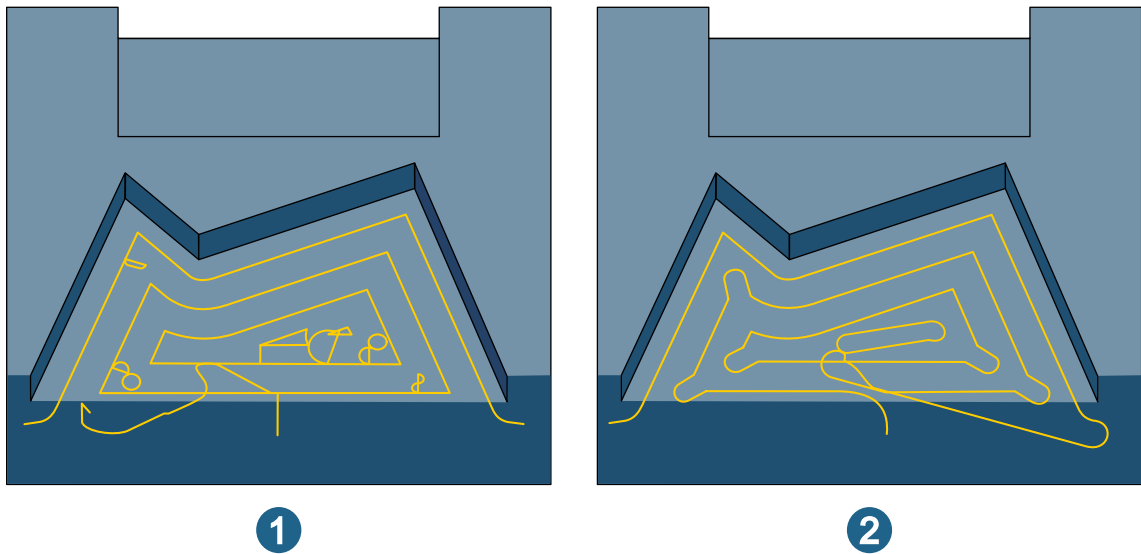
<p>2D Milling</p> <ul style="list-style-type: none"> • Pocket Milling • Contour Milling • Contour Milling on 3D Model • T-Slot Milling on 3D Model • Chamfer Milling on 3D Model • Inclined Pocketing 	<p>3D Milling</p> <ul style="list-style-type: none"> • Profile Finishing • Z Level Shape Finishing • Plane Machining • Arbitrary Stock Roughing • Z Level Finishing 	<p>3D Advanced Milling</p> <ul style="list-style-type: none"> • Complete Finishing • Automatic Rest Machining • Corner Rest Machining • Cutting Edge Machining • Rib / Groove Machining • Form Pocket
<p>5X Cavity Milling</p> <ul style="list-style-type: none"> • Profile Finishing • Z Level Finishing • Rest Machining • Corner Rest Machining • Z Level Roughing • Radial Machining 	<p>5X Surface Milling</p> <ul style="list-style-type: none"> • Top Milling • Iso Top Milling • Shape Offset Finishing • Tangent Plane Machining 	

2D and 3D machining - Improved performance for NC generation

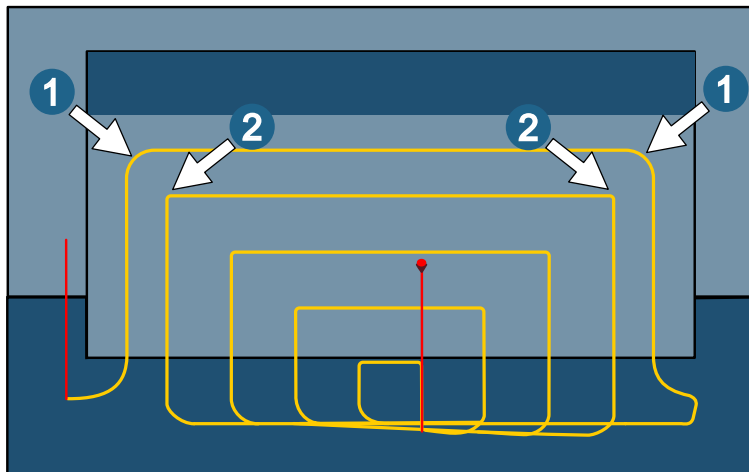
- The time for NC generation for 2D and 3D machining without activated optimizer has been improved.

3D Plane Machining - New pocketing algorithm

- The new algorithm ensures faster machining times on the machine and a more smooth toolpath layout especially with a big stepover (ae).
- (1) Old algorithm, (2) new algorithm



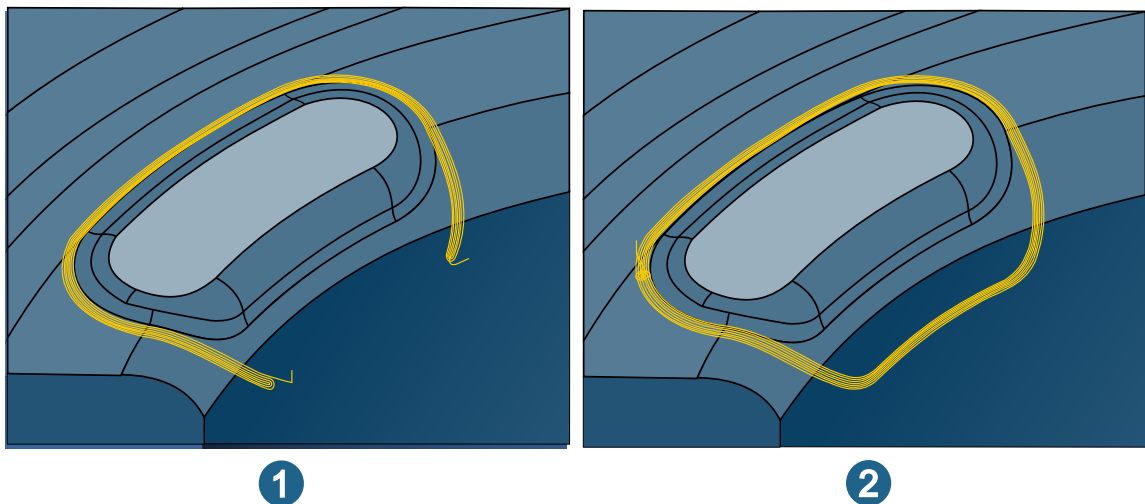
- The logic of how toolpath fillets are applied in the toolpath layout has been enhanced. The fillet radius specified in the user interface is now only used for fillets on the model contour (1). All other toolpaths are filleted automatically (2).



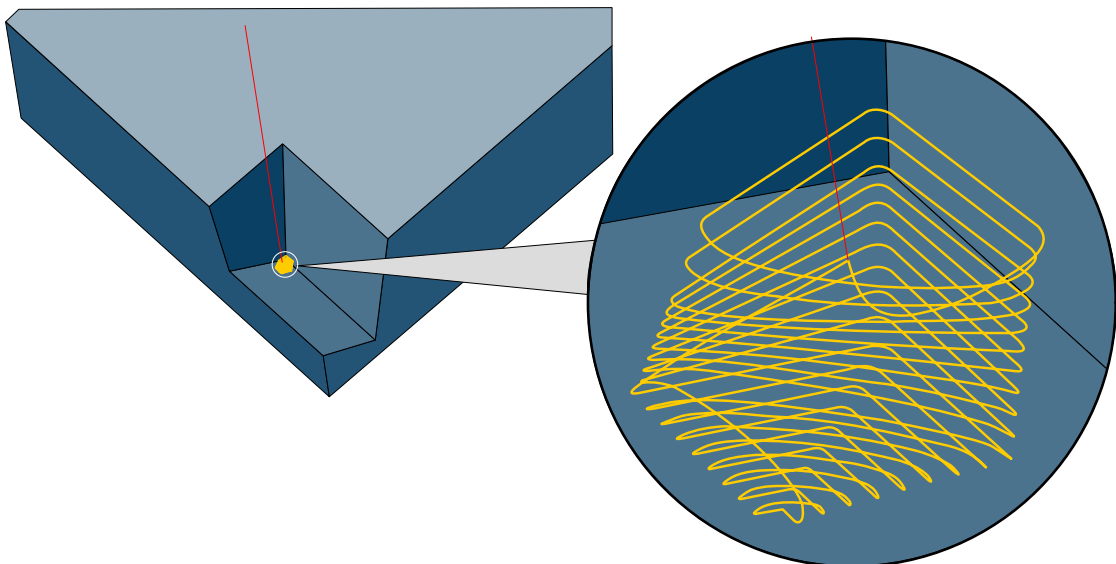
- The new filleting method ensures that the fillet for the toolpaths on the model can be set individually without affecting the fillets on the remaining toolpaths.
- This ensures a more smooth machining on the machine and avoids rest material if fillets in the remaining toolpaths are desired.

3D Automatic Rest Machining and 5X Cavity Milling - Rest Machining

- Several important improvements have been introduced for these cycles:
- A completely new rest material detection algorithm has been introduced and replaces the old. This new detection results in a more reliable rest material recognition and faster computation times.



- A new toolpath pattern removes rest material in certain situations such as corners or the intersection of multiple toolpaths.



- The logic for splitting and connecting toolpaths has been improved. The result is shorter processing times on the machine and a better surface finish.


3D Cutting Edge Machining

- **Collision avoidance in Reference job mode:** Collision avoidance can be activated while rest machining (Reference job mode). The cycle will detect possible collisions automatically and adjust the machining depth within the toolpath calculation to avoid collisions.
- **Infeed mode ZigZag for plunge milling:** The **Zigzag** can now also be combined with the **Plunge** machining mode. This is typically used when finishing contours with plunge milling.
- **Cutter compensation for plunge milling:** Cutter compensation with **Compensated center path** can now also be used for plunge milling (**Plunge** Machining mode).

5X Cavity Milling - Rest Machining

- A new algorithm to find a fixed tool inclination for steep and flat rest material areas. This algorithm replaces the old one and is used when machining with the 5X Mode **Automatic indexing**. Additionally the new algorithm will now try to machine flat areas preferably with a vertical inclination ($A/B = 0^\circ$, $C = 0^\circ$) if no inclination is necessary.

Cycle definition - User interface

The dialogs for job definition are now designed in such a way that all tabs are visible without having to show them using the arrow keys  on the right-hand side of the dialog when using the **Icons** option for the dialog tabs (see **CAM** → **Setup** → **Settings** → **Application**)

Tool database



CHANGED USER GUIDANCE

Since version 2024, the coupling types **Weldon**, **Weldon shank** and **Shrink fit** became standard types and can be directly used in the coupling area of the component (Tool, Extension, Holder, Tool holder). For tools the **Diameter** of the top coupling is automatically adapted to the tool diameter or shank when present. Tools using former coupling classes are automatically converted.

New coupling types

- With version 2024, two new standard coupling types, **Shrink-fit - Safe-Lock** and **Schaft - Safe Lock** are available in the coupling definition.

Tool breakage check

- The tool database has been enhanced with the **Tool breakage check** option. If the Virtual Machine is prepared for this, a breakage check is carried out before the tool is placed if this option is active. Please ask your OPEN MIND contact person if you need this option and your Virtual Machine is not prepared for it.

Tool series and technology templates

- Due to the larger scope of technology parameters of the types of cut, the interfaces to external tool management systems cannot currently be used. In this case, please keep using the default usages for the time being.

VIRTUAL Machining Center / SIMULATION Center

Visualization for turning operations

- The VIRTUAL Machining Center and the SIMULATION Center are now visualizing the part, stock and fixture as rotational geometry in case of turning operations. Also the collision check is using the rotational geometry. For the part and fixture it is therefore now required to define a 2D Contour which is used for the rotational visualization and the collision check.



With this technique it is possible to add undercuts or ISO fit changes to the 2D contour without collision detection for the turning operation.

Reduction of the collision check time

- The calculation time required for the collision check with a stock model when using tools with free defined geometries or indexable cutters has been significantly reduced.

VIRTUAL Machining Center / Toolpath preview

- A toolpath preview is also possible if the VIRTUAL Machining Center has already been started for the relevant job list (see context menu **Generate NC file** / **Generate NC file (batch mode)**).



- The VIRTUAL Machining Center can also be used to generate an NC file if a toolpath preview is already running.
- This applies separately for each job list in the hyperMILL® job browser.

VIRTUAL Machining Center / Heidenhain controls

- It is now possible to check the correct tool radius and corner radius within the NC program in addition to the tool length.
- OPEN MIND has noticed an unexpected behavior of the control on machines with head/head kinematics without an endless rotating C axis in combination with a HEIDENHAIN TNC640 control. The control does not necessarily take the current position of the two rotary axes into account when executing the PLANE command. The 'OM_ROT_AXIS_POS_TEST' is terminated with an error (FN 14: ERROR = 1005 ; Start position not allowed). This enhancement to the VIRTUAL Machining Center contains the following points:
 - PLANE parameter SYM (instead of SEQ)
 - the call of the subroutine WA2_CORRECT_V2.H created by HEIDENHAIN

hyperMILL® VIRTUAL Machining

Changed user guidance

- For better support of turret lathes and milling machines with rotary axes, the C-axis control in the job has been adapted. It is now managed via a drop-down menu. Existing settings are automatically converted according to the following logic.
- Previously: **Use preferred C angle** is not active → New: **Preferred solution C-Axis (A/B = 0)** + drop-down menu **Frame**
- Previously: **Use preferred C angle** is active + input field → New: **Preferred solution C-axis (A/B=)** + drop-down menu **Angle** + input field
- Depending on the selected machine, the additional options **Axis change** and **Transform polar** are available.

3D Milling with rotational axis

- For 3D milling it is now possible to transform a XY milling movement into a rotary axis and linear axis (e.g. CX) milling movement. To activate the function, activate the **Axis exchange option** on the NC dialog page under VM → **Preferred solution C axis (A/B0=0)**.
- Die Position des Werkzeugs durch Ändern des Wertes für den Winkel steuern. Die Grundposition 0° bezieht sich auf die X+ Achse des NCS. Der Wert für den Winkel ist ein Raumwinkel (gegen den Uhrzeigersinn). Falls der vorgegebene Wert für die Fräsposition zu einem Problem führt, zum Beispiel eine Achsbegrenzung, so wird automatisch eine Optimierung durch den NC Optimizer ausgeführt.
- **Voraussetzung:** Die Funktion **NC-Optimizer verwenden** ist aktiviert im **Maschineneigenschaften**-Dialog.

VIRTUAL Machining Optimizer

- The blending length defined for the rewind movement (Rewind) can be automatically reduced by the VIRTUAL Machining Optimizer. The automatic reduction of the blending length is activated in the VM configuration under **Rewind**. The blending length is only reduced if a rewind is not possible without a reduction, e.g. in case of strongly limited rotary axes.
Attention: The blending length can be reduced to the value 0.

CAD

Additions and notes

System check tool

- **Help** → **Automatic profile setting:** The global setting "Workstation App-Dynamic Streaming" for an NVIDIA graphics card is optionally set up automatically if the settings no longer correspond to the recommended setting – for example, if the base profile is set after a Windows operating system update.
In addition, the environment variable `HC_NVIDIA_GLOBAL_PROFILE` can be used to specify an alternative setting (for example, "3D App - Visual Simulation") that is set instead of this default setting. If this environment variable is an empty character string, the setting is not defined.

If the NVIDIA Control Panel is open while the check is running, the NVIDIA Control Panel must be closed and then opened again to see the change to the profile.

User interface

- The buttons on the SpaceMouse® from the manufacturer 3Dconnexion can be assigned CAD commands. The SpaceMouse® based on the SDK4 protocol has been expanded to include the **Rotation center > Use Selected Item** option, which focuses navigation on selected entities. If this option is enabled and entities are selected, the rotation center of the SpaceMouse® becomes the center of the bounding box of the selected entities. The SpaceMouse® buttons **Fit** and **Orientation** use this bounding box.



If the 3DxWare driver version 10-8-16 or an older version is installed, remove the `hyperCAD.xml` from the following directories:

- C:\Programdateien\3Dconnexion\3DxWare\3DxWinCore\Cfg\
- C:\Programdateien\3Dconnexion\3DxWare\3DxWinCore\Cfg\
- C:\Users\[USER NAME]\AppData\Roaming\3Dconnexion\3DxWare\Cfg\

- As part of the rebranding, some commands have been reorganized and renamed, for example, the help calls for CAD and CAM, and e-learning have been combined under **Help**, and the “hyperMILL” menu has been renamed **CAM**.
- On the **Visibility** tab: In **Layer**, a newly created layer is automatically set to current. In **Layer** and **Color**, the commands on the context menu are divided into groups by hyphens.
- On the context menu and on the **Model** tab, the selected entities can be assigned to the current layer using the **Assign entities to current layer** command.

Options / properties in file

- **Graphic** → **System** → **Advanced**: The “Advanced shadow map” and “Clipping plane to all entities” options have been removed as they are no longer necessary.
- CAD Viewer: It is possible to change the unit of measure.

Interfaces

- **Import PMI with MBD**: MBD from STEP AP242, SOLIDWORKS, Siemens NX, CATIA V5, and PTC Creo are imported for dimensions, surface finishes, and tolerances, and faces are assigned as tags. An [explanation \[65\]](#) of this has been added to the software documentation.
- *.omx and *.bnd: Data in the **Inch** unit of measure can be read in.
- **Autodesk® Inventor®**: There are improvements in supporting attribute categorization.
- **PTC Creo**: “Multibody parts” are now also supported.
For models from PTC Creo 7.0: The support of “Multibody parts” changes the naming of the layers when importing models. Previously, the name of the file was used. With the current version, the name of the “part” is used instead.
The file name is still used for older files.

Edit

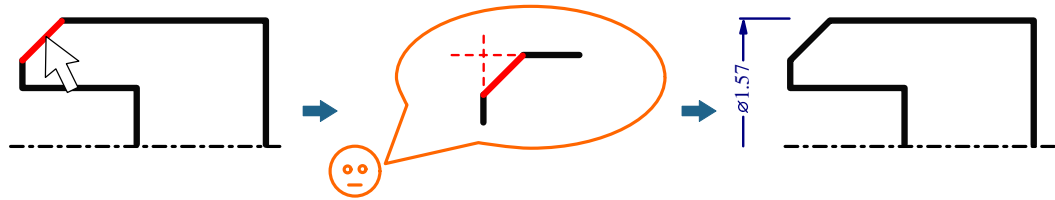
- **Copy**: Properties for copying via the clipboard can be preset in the `copy_defaults.xml` file. For explanations, see [Copy \[66\]](#).
- Commands for parametric modeling are combined on the **Parametric** submenu. The **Spreadsheet** command is also located there.
- Commands for the color libraries are combined on the **Color library** submenu.
- **Align dimension**: Align dimensions with each other. An [explanation \[67\]](#) of this has been added to the software documentation.
- **Stretch 2D**: Stretch two-dimensional contour curves. An [explanation \[67\]](#) of this has been added to the software documentation.

Modify

- **Invert curves**: The orientation arrows can now be (optionally) used to select the curves to which they refer.

Select

- The “Project snap points” command has been renamed **Snap point projection on workplane on / off**.
- The “Pick through transparent color” command has been renamed **Pick through transparent color on / off**.
- **Snap**: The **Curve intersection point** mode also finds the intersection point on (imaginary) extensions of adjacent straight lines for chamfers and fillets. For this purpose, move the mouse pointer closer to the chamfer or fillet. Useful, for example, when dimensioning an edge that is chamfered.



Shapes

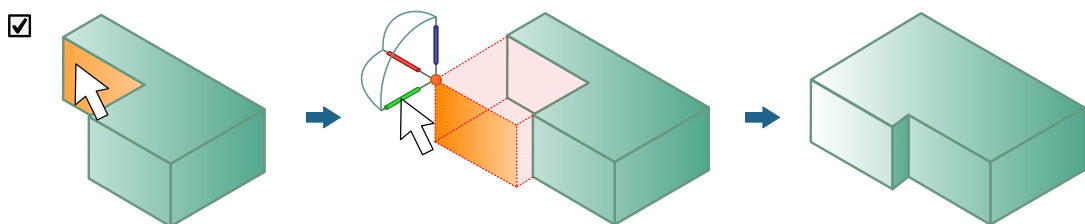
- **Sweep (two contours)**: Generate faces between two contours, optionally with top and bottom faces. An [explanation \[72\]](#) of this has been added to the software documentation.
- **From grid**: Create a face from several curve chains in one direction plus several curve chains in the other direction. An [explanation \[73\]](#) of this has been added to the software documentation.
- **Chamfer**: The “45” option has been renamed **Symmetric**.
- **Fillet**. To create a fillet with a **Face - Curve**, a handle has been added to determine the radius interactively.

Features

- Features can also be inserted on individual faces (in other words, not only in solids). The faces are converted into open solids.
- **Holes**: Several thread and fit tables * .xml saved in the standard paths defined for the general workspace are offered together for selection in one view.
- **Protrusion (two contours)**: Create a protrusion between two contours that define the shape of the protrusion. An [explanation \[75\]](#) of this has been added to the software documentation.
- **Slot (two contours)**: Create a slot between two contours that define the shape of the slot. An [explanation \[76\]](#) of this has been added to the software documentation.

Direct modeling

- Direct modeling of side faces: A preview of the side faces is displayed if the **Side faces** option is enabled for direct modeling.



V-sketches and parametric design

- As already mentioned under **Edit**, some commands for parametric modeling are combined on the **Parametric** submenu. The **Spreadsheet** command is also located there.
- **Strict chirality**: Select the chirality behavior. An [explanation \[64\]](#) of this has been added to the software documentation.
- **Automatic rebuild on / off**: A recalculation is not triggered every time the V-sketch on which a parametric design is based is changed, but optionally only after several combined changes. This saves time. An [explanation \[75\]](#) of this has been added to the software documentation.

Work preparation for turning

Commands for preparing a turning contour have been added to the **Drafting** menu:

- **Trim 2D corners:** Trim 2D corners. An [explanation \[72\]](#) of this has been added to the software documentation.
- **2D undercut:** Insert predefined 2D undercuts based on a library (of model files). An [explanation \[71\]](#) of this has been added to the software documentation.
- **Reset 2D corners:** Reset undercuts, fillets, and chamfers on sharp-edged corners. An [explanation \[71\]](#) of this has been added to the software documentation.
- **Round / chamfer 2D corners:** Use criteria to find all required corners of a contour and round or chamfer them. An [explanation \[69\]](#) of this has been added to the software documentation.

Work preparation

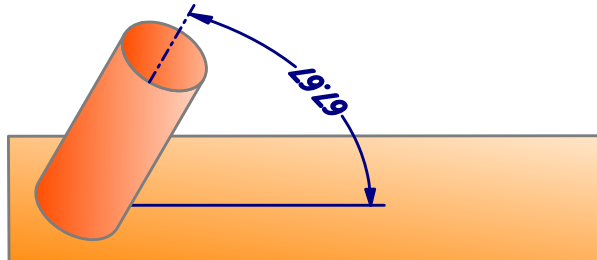
- Close any openings by covering – for this purpose, the **Create capping** option has been added to **Repair open solid**. The functionality of the **Close holes** command has been extended accordingly.

View

- The “Toggle projection” command has been renamed **Perspective view on / off**.
- Under **Stepwise view change**, rotating the view in both directions with **Counterclockwise rotation** and **Clockwise rotation** has been added.
- **Hide / unhide** → ...: The commands for showing and hiding points, faces, solids, meshes, curves, and all entities have been revised. The new commands are around three to four times faster. V-sketches and the sub-entities of groups are also shown and hidden. The user interface does not change and keyboard shortcuts remain the same.

Analysis

- **Two entities info:** The dialog has been expanded to include searching. The angle between a planar face (regarded as endless) and a cylinder (cylinder axis) is determined.



- **Local curvature:** In **Configuration**, it is possible to specify that a point on an entity, the direction of the normal, and the UV directions are generated individually as required.

Electrodes

- Imported CAD geometry data often does not have the quality required to create an electrode. Despite undercut and missing faces, gaps between faces, and overlapping faces, the software allows electrodes to be created on the basis of such CAD geometry data. A warning is only issued if the electrode geometry does not specify a closed shape. It is up to the user to assess whether such a situation will have a negative impact on the generation, NC programming, and production of the electrode, and whether it needs to be corrected in advance.
- Automatically close features such as holes and breakthroughs inside a domain optionally. An [explanation \[77\]](#) of this has been added to the software documentation.



Switch off the option for eroding lettering, for example.

- Information on controlling the generator performance can be managed. For this purpose, the **XML file for generator program** electrode option has been added. Default settings can be specified in this XML file. The values are offered for selection in **Create**, **Create user defined**, and **Modify EDM parameters**, and are taken into account in the completely revised EDM Converter. Also see the explanations in [Preparing generator programs \[80\]](#).
- **Create user defined: Simplified** has been added to the **Mode** bounding box method. An [explanation \[78\]](#) of this has been added to the software documentation.
- Detected collisions between an electrode and the workpiece are also displayed in the graphics area. The first face of the detected collision is always marked. The following collisions are displayed:
 - Electrode between start position and clearance position.
 - Electrode in the start position.
 - Faces
 - Internal joint face in target position.
 - Joint faces between start position and clearance position
 - Joint faces between target and start position.
 - Linear extension faces at the target position
 - Rotated faces with model
 - Stabilization extension faces at the target position
 - Tangent extension faces at the target position
 - Upper solids between target position and start position.
 - Mutually colliding rotated faces
- User-defined tags can be used for electrode documentation. An [explanation \[79\]](#) of this has been added to the software documentation.
- **Modify eroding path**: 3D curves including splines can also be selected for eroding paths. The eroding path is broken down into individual waypoints.
- The position of the holder can be moved freely and is no longer limited to the scope defined by the block size in X and Y and the size of the raw material in Z.

CAD for CAM

- The rest material display has been completely revised. An [explanation \[14\]](#) of this has been added to the software documentation.
- A command has been added that can be used to analyze a face in order to find an optimal barrel cutter for producing the face. An [explanation \[35\]](#) of this has been added to the software documentation.



Current restriction! In this first implementation, the tool is always considered normal to the curve at the face tangent point. We plan to include a lead angle (positive and negative) in the calculation in future extensions.

Bug fixes

The following issues have been resolved:

Updating recommended

- **Feature** → **Hole**: Crossed threaded holes in inches with incorrect thread texture when intersecting with other holes. The matter has been resolved.
- **Boolean** → **Difference**: If solid A is locked and the **Keep original solids** option is selected for solid B, but solid B is not locked, the Boolean operation will not work. The matter has been resolved.



Release 2024 | Update 1

CAM

Bug fixes

Updating highly recommended

VIRTUAL Machining Optimizer

- **Virtual Machine Configuration → Movements → Optimization → Rewind → Allow split G1 movements.**
Fixed a problem where the toolpath was not split despite the option being activated.
- **Virtual Machine Configuration → Movements → Optimization → Rewind → Allow split G1 movements → Allow blending length reduction.**
Previously, the blending length was unexpectedly reduced by the optimizer in special constellations despite the option not being active.
With the new version, the reduction of the blending length is no longer carried out without activation. In rare cases, this can result in no rewind movement taking place. If a reduction is required, it can be activated, and the blending length can then be adjusted to 0 by the optimizer if this is necessary for a rewind movement.

hyperMILL® VIRTUAL Machining

- If the option **Stock contact → Settings for collision check → Allow thin shank contact** was activated, an incorrect collision message was issued during the collision check for a turning operation. This error has been corrected.
- If a machining operation was loaded from **Generate NC file (keep mode)** from hyperMILL® not the updated toolpath was simulated, but the previously simulated toolpath was simulated again. This error has been corrected.

hyperMILL® VIRTUAL Machining / SIMULATION Center

- In the **Turning → Axial Grooving** cycle a change of the reference point was not correctly taken into account, so that the tool was positioned incorrectly. This error has been corrected.

SIMULATION Center

A problem was solved in which - depending on the tool geometry - no material removal simulation was performed in the SIMULATION Center.

3D Optimized Roughing

- An error has been corrected that caused a collision with the fixture, although the fixture surfaces were selected as Avoid surfaces (**Boundary → Avoid areas**).

5X Shape Offset Roughing

- A problem with an aborted toolpath calculation has been fixed.

Tool database

- An error has been corrected that prevented * .hmc file from being opened.

Updating recommended

General

- When machining with a stock, in transparent mode not all holes were displayed when the job dialog was open. This error has been corrected.

2D Contour Milling

- Fixed an issue that caused the update of an NC comment to be ignored when creating the NC file.

3D Profile Finishing

- Fixed a problem with a very long calculation time when the **Strategy** → **Machining mode** → **Slope mode** option was activated.
- If the **3D path compensation** function was activated (**Setup** → **Tolerance parameters** → **3D path compensation**), a vector was not correctly taken into account on a Rödgers machine and processing was aborted. This error has been corrected.
- Fixed a problem when using the options **Boundary** → **Trim to Stock** and **Strategy** → **Infeed strategy** → **Zig zag direct / Zig zag smooth**.

hyperMILL® VIRTUAL Machining - Makino Probing

- The **Workpiece alignment along edge** cycle has been extended, so that the rotary axis can also be aligned with tilted workplane now.
- A bug when setting the origin with activated tolerance check was fixed.

hyperMILL® VIRTUAL Machining / SIMULATION Center

- No removal simulation was performed during a turning operation with activated counter spindle. The error has been corrected.

New commands

3D Optimized Roughing

- The **Finish pass** machining mode (known as **Finish pass all levels** from the **3D Arbitrary Stock Roughing cycle**) is now also available for **Optimized Roughing**.
- If this option is enabled, it ensures the tool moves along the entire defined contour, even if no material is cleared. If there is still some rest material, this is cleared first and then the finish path is performed. You should always enable this option, if minor changes were made to the allowance between two machining passes.

3D Z Level Shape Finishing

- The functions for optimizing the machining tolerances for different machining situations already known from the **3D Profile Finishing** have been implemented also for the **3D Z Level Shape Finishing** cycle. The following functions are available on the **Setup** dialog page in connection with the use of the CAM Plan functions (also introduced with version 2024):

Tolerance parameter

Tolerance use cases: Use the parameters of the tolerance use cases to ensure the best possible tolerances depending upon the individual machining situation. The best settings are automatically used based on the tool diameter and the selected **Use case** (**Semi finishing**, **Finishing**, **Fine finishing**, **Micro finishing**).

For **Mode** specify whether the toolpath is calculated in **Standard** mode or in **High-precision surface mode**. The available use cases represent the typical machining steps during a milling operation.

High precision surface mode: Use when maximum accuracy is required to calculate the toolpath. The calculation of the toolpath is not based on the faceted model geometry, but on the real surfaces of the model. Only available for **Ball mill**.

True Shape point distribution: Activate to redistribute points and guarantee an even and machine control-friendly distribution of points.



To guarantee the best possible point distribution, use the **CAM Plan** function. The machining cycle automatically takes into account the topology information that was created with the **HPM-Milling** task in the CAM plan.

Filter points: Points that are located on straight toolpath sections, for example a planar surface, and are therefore not required, are filtered out.



Please note that no or fewer points are not always the best solution for all machine controls. For more detailed information, please contact the respective machine manufacturer.

Feedrate adaption: Activate to automatically adjust the feedrate depending on the curvature of the part. The feedrate is reduced for concave curvatures (1) and increased for convex curvatures (2). This means that the real contact feedrate of the tool on the part surface remains constant.

3D path compensation: Available for the **Ball mill** tool type. Enables various NC controls to machine the model with a smaller or larger tool than actually programmed in hyperMILL®. In addition to the X, Y and Z coordinates, the direction vectors I, J and K to the surface contact point must also be known.

Protect edges: When using the CAM plan, edges on the part are treated so that they are not damaged during machining. This is achieved by inserting additional points in areas where sharp edges were previously found in the component using the **HPM Milling** task.



The 3D path compensation function is available only with a specially adjusted postprocessor. Without this adjustment, the NC program cannot correct this output and, as a result, both the component and machine may be damaged if the milling geometry used differs from the milling geometry used to calculate the toolpath. To adjust your postprocessor, please contact your OPEN MIND partner.

Tolerance settings

In the tolerance settings, adjust the parameters of the **True Shape point distribution** and **Feedrate adaption** functions.

Activate **True Shape point distribution** to guarantee an even and correct point distribution in toolpaths. The parameters **Calculation tolerance**, **Max. chordal error**, **Max. G1 length**, **Min. G0 distance**, **Reference G1 length**, control the arrangement and distances between the points.

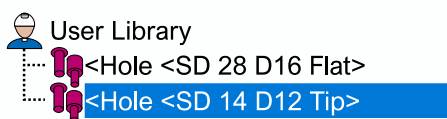
Use the **Adjacent sync. ISO** parameter to achieve asynchronous point distribution between individual toolpath segments by entering a (factor) value less than 1.

Function is not activated (1), Function is activated (2)

The **Min. feedrate (factor)** and **Max. feedrate (factor)** define the upper and lower limit for the feed rate adjustment. The factor refers to the standard feedrate defined for the toolpath. Use **Number of steps** to control in how many intervals the increase or decrease of the feedrate adaption is performed.

Feature technology

- A generic hole that was created in the CAM system and is contained as a list entry in the lower area of the feature browser can now be imported into the standard hole library of the CAD system using the context menu function **Add hole to library**.
- **Features** → **Holes** → **Advanced** → **Library** → **[User Library]**
- **Example:**



hyperMILL® VIRTUAL Machining



DEFINITION OF THE CLAMPING POSITION REQUIRED

The position of the part on the machine table is crucial for correct visualization and testing in the simulation. The clamping position is also used for other calculations such as the Best Fit function and milling with a rotary axis.

The position of the part on the machine table is defined by the clamping position as follows:

Job list → Setup → NCS → Define frame → Clamping position

In one of the next updates, OPEN MIND plans to make the definition of the clamping position for the Virtual Machine mandatory.

We recommend that you define the clamping position in your current projects already now. If you have any questions on this topic, please contact your support team.

Drilling → Back Boring

In the **Back Boring** cycle, the C-axis angle set in the job and a C-axis angle changed by the optimizer are now also supported.

VIRTUAL Machining Optimizer

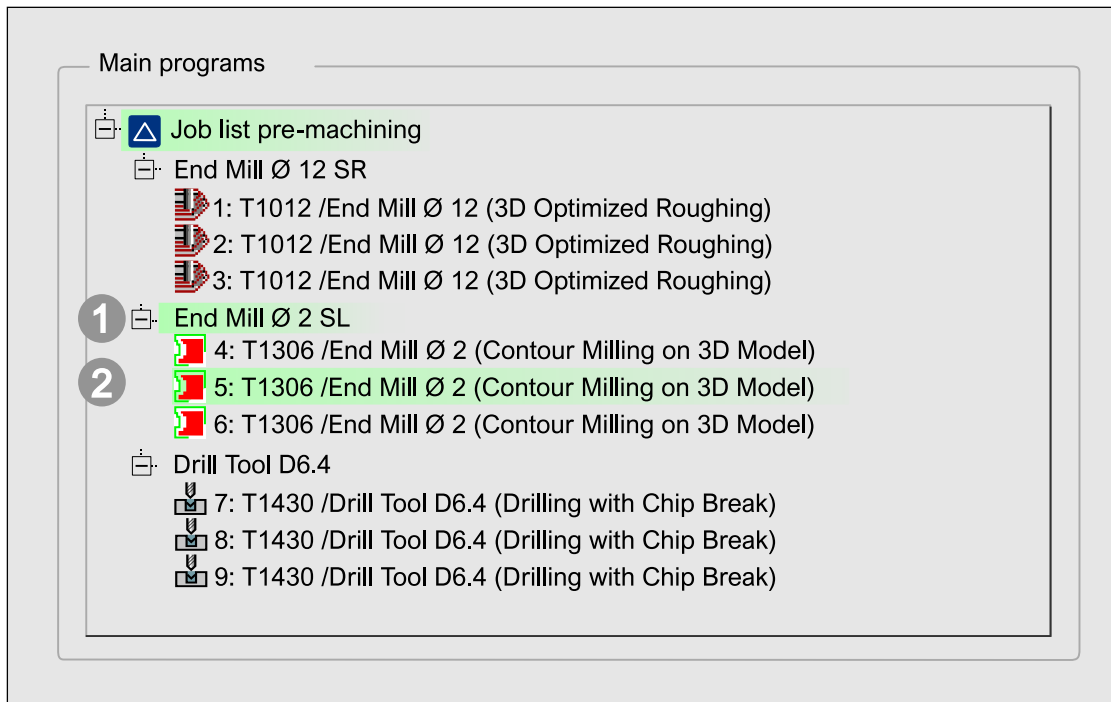
- **Virtual Machine Configuration → NC → Common → Movements → Feedrate for linking**

The defined feedrate for linking movements is now also used for the linking movements within a machining operation during axis exchange (milling with rotary axis).

VIRTUAL Machining Center



When performing a simulation, in **Project administration** operating mode of the VIRTUAL Machining Center the currently simulated job (2) is now also highlighted in green in addition to the tool (1), so that it is clear at a glance which job in the list is being simulated.



Origin check for Fanuc, Mazak, Brother and Haas controls

- For Fanuc, Mazak, Brother and Haas controls, it is now possible to check the correct definition of the origin on the real machine at the beginning of the NC program.
- The value of the origin used in the virtual machine is compared with the value of the origin on the real machine. If this value is outside the specified tolerance, the NC program is aborted.
- This functionality can be used to ensure that the part is positioned on the real machine in the same way as it was tested within the virtual machine. The tolerance is activated and set in the configuration of the Virtual Machine under **NC** → **Common** → **Origin checks**.

Special attention:

- In the case of multi origins, all origins used are checked at the start of the program.
- When an origin is written by probing functions, the origin is checked again.

CAD

Additions and notes



Electrodes → Create user defined:

A user-defined electrode created with the **Simplified** option allows the target position in the Z-axis direction to be set lower than the highest point of the raw material. This allows electrode holders for side electrodes to be positioned freely.

Electrodes → **Export to EDM**: If no part number is stored in the electrode options, the file name is used. For ZK Alphamoduli predefined entries for measuring machines are supported. For ONA the electrode material is now also supported.

Bug fixes

The following issues have been resolved:

Updating is highly recommended



- **CAM** → **Reports** → ...: The tool and job reports have been revised and are now generated more quickly.
- **Curves** → **Offset**: In certain situations, the command causes the software to crash. This issue has been resolved.

Updating recommended

- **CAM** → **Optimal barrel cutter**: If in **Base diameter range** the minimum and maximum values are the same, any calculation is performed. Similarly, any calculation is performed if in the **Barrel radius range** the same values are given. Both issues have been resolved.
- **Electrodes** → **Create user defined**: Creating a user defined electrode with the option **Simplified** crashes if only one face is selected. This issue has been resolved.
- CAD interfaces



For all CAD interfaces, which are based on technology up to 31 December 2019, the **Product and Manufacturing Information (PMI)** option in **File** → **Open** → **Settings** shall remain for import at the **BETA development stage**.

The interfaces have been updated for Autodesk® Inventor® to 2025 and for CATIA V6 (3DXML) to 2024.

