



hyperMILL®

hyperMILL®

Release notes 2025 | Update 5

 **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 Fri, Jan 16, 2026)



Table of Contents

1. Supported Versions	2
Operating systems and CAD platforms	2
CAD interfaces	2
Interfaces Tool database	4
Supported EDM formats	4
Supported OPTICAM Software Versions	5
Interfaces NC simulations	5
2. New commands and additions	6
CAM	6
Basics of CAM editing	6
Specifying the basic settings	6
NC system and frames	6
Tool management	7
Structure CAM project	8
Collision check preparations	13
Feature and macro technology	14
Generate feature	14
Drilling	15
Hole Brushing	15
3D Machining	16
Optimized Roughing	16
Plane Machining	16
5X Machining	17
Cavity Milling	17
Surface Milling	24
Blade Milling	29
Multiblade Milling	29
Tool database	30
Define turning tool	30
Optimal barrel cutter	31
Define NC tool	33
TOOL Builder	33
Turning tool: Creating a static holder for machining with a turret	33
CAD	34
User interface	34
Tabs	34
Default settings	35
Options / properties	35
Sketcher options	36
Data interfaces	36
General interfaces	36
Direct interfaces	41
Select and snap	42
Select entities	42
Snap entities	42
Analysis	43
Check	43
Workplane	43
Create workplane	43
Edit and modify	45
Edit	45
Modify	46
Points, curves and faces	46
Drafting	46
Curves	46
Shapes	46
V-sketch	48
Automatic constraints	48



Geometric constraints	48
Solids, features and meshes	49
Features	49
Design electrodes	50
Modify eroding path	50
CAM programming	51
Overview	51
Adaptive construction	53
Analyzing probing results	59
Probing options	60
PYTHON Toolkit	60
Python script support	60
Python options	61
3. Release notes	62
Release 2025 Update 1	62
CAM	62
CAD	62
Release 2025 Update 2	65
CAM	65
CAD	72
Release 2025 Update 2.1	73
CAM	73
Release 2025 Update 3	74
CAM	74
CAD	77
Release 2025 Update 3.1	78
CAM	78
CAD	78
Release 2025 Update 4	78
CAM	78
CAD	82
Release 2025 Update 4.1	82
CAM	82
Release 2025 Update 5	83
CAM	83
CAD	83



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



hyperMILL® 2025 brings you even more performance and efficiency, both for CAD and CAM. Thanks to the new tessellation algorithm and a new, simplified element display of CAD models as well as improved algorithms for our 3D strategies, you stand to benefit from significantly faster workflows and even more precise machining results.

As a highlight of the new *hyperMILL®* release, three innovative strategies for deburring holes and component edges offer maximum flexibility and unmatched quality. In addition, the new 5-axis automatic mode ensures optimized machining and guarantees efficient and reliable toolpaths, even for the most complex components.



1. Supported Versions

Operating systems and CAD platforms

64-bit operating systems	Windows 10, Windows 11 *
64-bit CAD platforms	<p>hyperMILL®</p> <p>Inventor 2024, 2025, 2026</p> <p>SolidWorks 2023, 2024, 2025</p>
Server operating systems (License server only)	<p>Windows Server 2016, Windows Server 2019</p> <p>Windows Server 2022</p>

hyperMILL exclusively supports 64-bit operating systems.

* Starting with Windows 11 version 24H2, we recommend at least hyperMILL® 2024 | Update 3.

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
CATIA V4	*.model	4.2.5	x	
	*.exp	4.2.5	x	
CATIA V5	*.CATpart	2025	x	
	*.CATproduct	2025	x	
	*.CGR	2025	x	
CATIA V6	*.3dxml	2025	x	
PTC Creo Parametric	*.prt	12.4	x	
	.prt.			
	*.asm	12.4	x	
	.asm.			
	*.neu	12.4	x	
PTC Creo	*.xpr	12.4	x	
	*.xas	12.4	x	
Siemens NX	*.prt	NX2506	x	
SOLIDWORKS	*.sldprt	2025	x	
	*.sldasm	2025	x	



Product	File type	Up to version	Import	Export
Autodesk® Inventor®	*.ipt *.iam	2026	x	
Rhinoceros®	*.3dm	8	x	
Solid Edge	*.par *.asm *.pwd *.psm	2025	x	
PRC (Product Representation Compact)	*.prc	All versions	x	x
Parasolid	*.x_t	38	x	
		14		x
	*.x_b	38	x	
		14		x
JT-Open	*.jt	10.9	x	
		10		x
IGES	*.igs *.iges	5.1, 5.2, 5.3	x	
		5.1		x
STEP	*.stp, *.step	AP 203 E1/E2 ^a	x	x
		AP 214 ^b		x
		AP 242 Edition 2 and Edition 3 ^c		x
AutoCAD	*.dwg *.dxf	2019	x	
		2019		x
Point clouds	*.pt, *.asc *.xyz *.txt	No version	x	
				x
Polygon mesh	*.stl *.stla *.stlb	No version	x	x
Polygon mesh	*.ply2		x	x



Product	File type	Up to version	Import	Export
3MF Reader (3D Manufacturing Format)	* .3mf	1.2.3	x	
ACIS	* .sat	V34-V35 (2024-2025)	x	
	* .sab	5.0		x
Wavefront OBJ	* .obj	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) (iMHYP) 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
Exeron	Exoprog	1.0.0.0	x	x	x	
Makino		1.0.0.0	x	x	x	
ONA		1.0.0.0	x	x	x	
OPS Ingersoll	Multiprog	1.0.0.0	x	x	x	
Sodick		1.0.0.0	x	x	x	
Zimmer & Kreim	Alphamoduli	1.0.0.0	x	x	x	x
+GF+HMI	AC FORM HMI	1.0.0.0	x	x	x	

Changes or customization requests must be commissioned.

Supported OPTICAM Software Versions

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

<i>hyperMILL</i> ®	OPTICAM
2025 Update 5	2025.2
2025 Update 4	
2025 Update 3	
2025 Update 2	
2025 Update 1	
2025	

Interfaces NC simulations

VERICUT from version 9.4
NCSimul from version 2020.0



2. New commands and additions

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

CAM

Basics of CAM editing

Specifying the basic settings

Document dialogue page

Color table

Set the colors for various *hyperMILL* elements.

Toolpath

Define the **Toolpath colors** for G0, G1, points, clipped toolpaths, NC path G0, NC path G1, tool axis, collision, and contact axis.

Tool

For **Tool feedback**, define the colors for the following elements: dimensions, spindle, holder, cutting area extension, tool, tool holder, center line, insert, and insert cutting area

General

Define the colors for the following elements under **hyperMILL® feedback**: clearance plane, top, bottom, contours, boundaries, profiles, NCS, frame, highlight, synchronization lines, cutting direction, and infeed.

Stock model

Define the colors for the **Material feedback** for the following elements: stock model, stock coordinate system, removed material, and material to be removed.

Feature

Define the **Colors for list display**. Activate **Used by Feature Mapping** if the colors should also be used by the feature mapping.

NC system and frames

Corrected output system dialog page

Select 3D Point Probing report

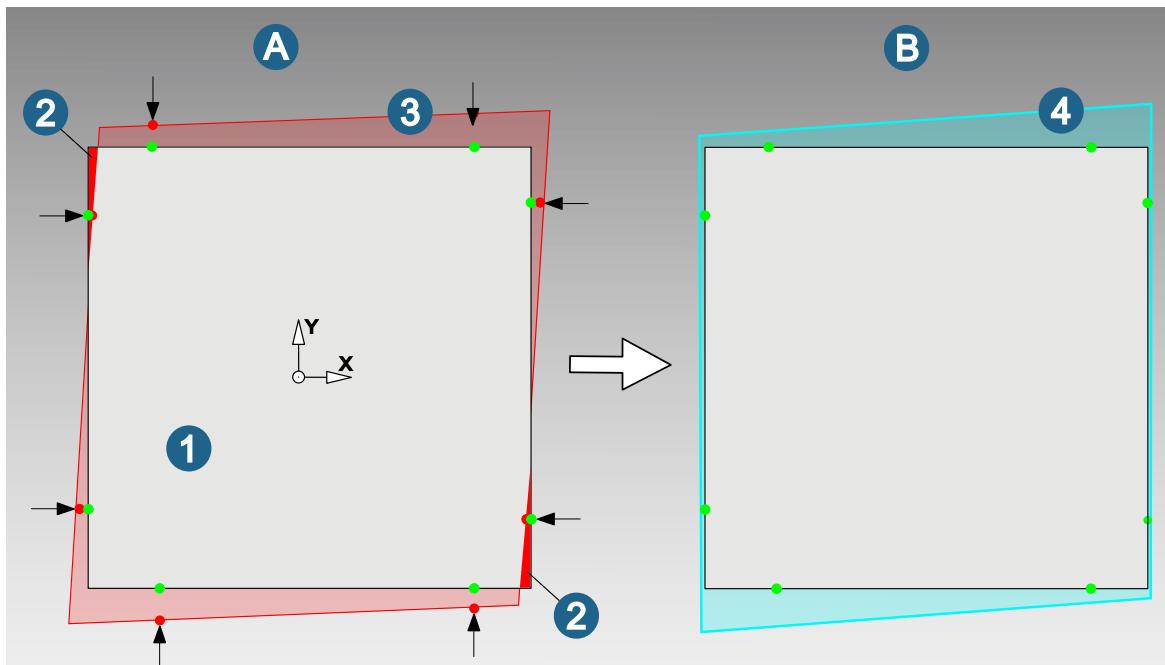
Minimize distance between points

Minimize the distance with the function **Surfaces (distance enclosing normals)** along the surface normals, especially for optimum alignment of components/stock models with uneven, small allowances in different areas.

The deviation of the measuring points from the target dimension is determined in the direction of the surface normals (A). The component/stock model is moved for optimum alignment independently of the surface normals (B).

(1) = Target part, (2) measured undersize of the aligned stock model, (3) stock model to be aligned,

(4) stock model optimally aligned with the function **Surfaces (distance enclosing normals)**.



Alternatively, the distance between the measured values of the points can also be minimized using the function **Points (3D distance)**. To be used if several axes are locked. Only the distances are minimized, the surface normals are not taken into account.



After reading the measuring log file, use the **Best Fit preview** function to visualize the precision of the component alignment. The **Best Fit preview** function shows the virtually corrected alignment (measuring points) taking into account the selected Best Fit calculation method. The selected measuring log file is used (see dialog **Calculate corrected output system**) and the transformed measuring points are automatically displayed on the **Probing** tab in *hyperMILL CAD*.

Tool management

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

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 the tool.

In addition, in profile mode **Manual** it is possible to use the technology of any available type of cut. To do this, select the desired type of cut from the list of cut types.

Edit tool

Meaning of the icons in the tool browser	
	Partially local (document) tool, Current status



Functions on the shortcut menu: Tools



Partial unlink: Partially disconnect the document tool from the external tool database. The parameter **Tool reach**, as well as all parameters relevant to the tool call (NC number, ID, name) can be changed in the document. The other tool parameters remain linked to the external tool database and their status remains current.

Structure CAM project

CAM Plan

Create topology information

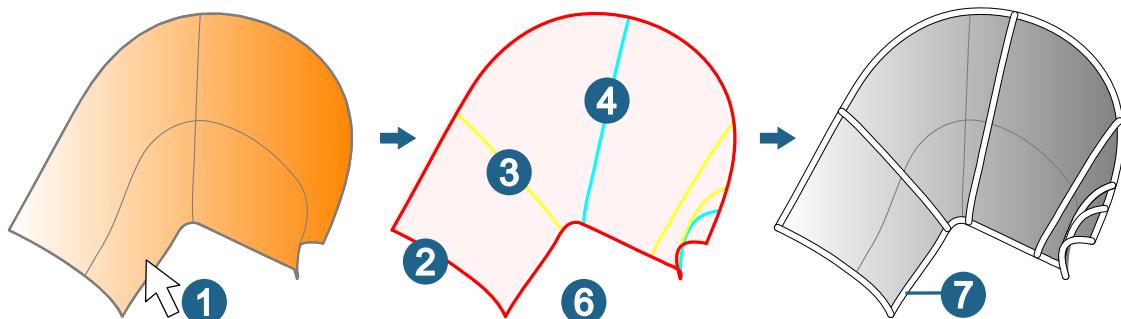


Generate topology curves for 3D finishing.

hyperMILL®-browser → CAM Plan → High precision milling → Create topology information

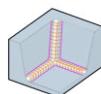
More options

Internal continuities: Visualize continuities within NURBS surfaces ⑥. Both internal C0 and C1 continuities can thus be identified directly. Milling operations can be better prepared and validated.



Rest material

Rest material: Select all surfaces that are relevant as rest material surfaces for machining the part and generate the rest material data. This data is saved internally and used for the toolpath calculation of the cycles **3D Automatic Rest Machining** and **5X Rest Machining**.



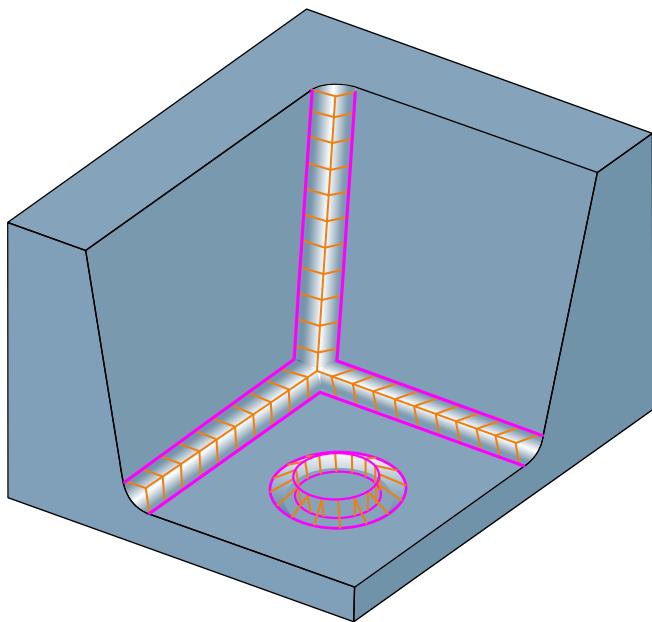
Create rest material data.

hyperMILL® browser → CAM Plan → Rest material → Create rest material data

To calculate the rest material data, select a frame as **Direction**. The calculation is based on the Z direction of the selected frame.

Create a reference tool for calculating the rest material data via the context menu **New** of the **Ballmill list** entry. Enter a value for the tool diameter to be used for calculating the rest material data in the red input field under **Selections → Tool diameter**. Any number of tool diameters can be defined.

Subsequently select the calculated rest material data in the **3D Automatic Rest Machining** and **5X Rest Machining** cycles on the **Tool** dialog page as **Reference → Type → Rest material from CAM Plan**.



Define job

Input dialog: General

Stock chain

Use global resolution: Activate to adopt the accuracy for the stock model calculation from the **Job list** → **Settings** → **Stock chain**. If a different accuracy is required for the job, deactivate the function and define a value for **Resolution**.

Exclude from stock chain: Activate if no stock model calculation is to be carried out for the job.

If the **Job information** → **Generate NC file** function is activated, the **Allow stock without NC** function can be used to perform a stock model calculation for the job.

Input dialog: NC

Optimization

Mode: Set the mode for optimizing the NC program.

Off: The NC program is not optimized.



Solutions

Preferred direction

Closest C angle plus / Closest C angle minus: The solution in the head that is closest to the defined C angle is selected. The solution in the head can switch between plus and minus, which keeps the movement of the C axis small.

Comparison: Preferred direction – Plus / Preferred direction – Closest C angle plus

Preferred direction → Plus (BC table/head kinematics)	Preferred direction → Closest C angle plus → 0 degrees (BC table/head kinematics)
To maintain the plus solution in the head, a large rotation is performed in the C axis. This behavior is shown in the following video:	The solution alternates between plus and minus in the head, which keeps the movement of the C axis small. This behavior is shown in the following video:
https://www.youtube.com/embed/-3lv7j0gSPk	https://www.youtube.com/embed/ZK3Ex2mnCvY

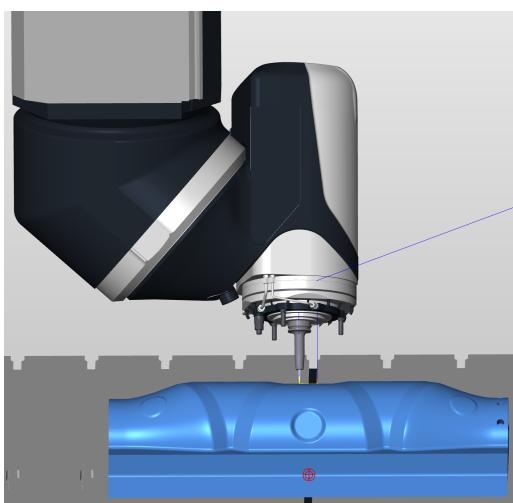
Additional information: Closest C angle plus / Closest C angle minus

If there are two possible solutions with the same distance to the defined C angle, the desired behavior of the machine can be defined via **Closest C angle plus** or **Closest C angle minus**. Here **plus** and **minus** refer to the C-axis solution.

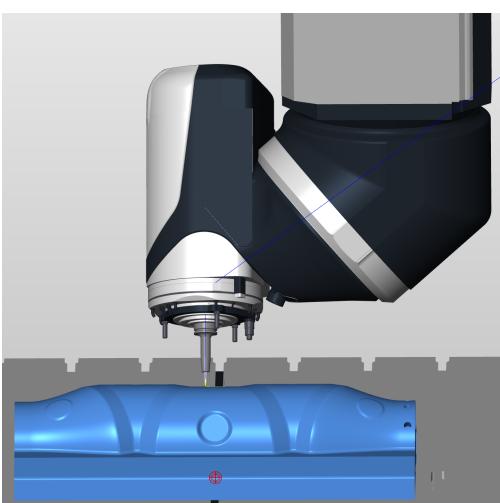
Example: BC head/head kinematics, angle C axis = 0 degrees

- (1) Solution for **Closest C angle plus** = B+32.8239 C+79.2792.
- (2) Solution for **Closest C angle minus** = B-32.8239 C-79.2792.

1



2

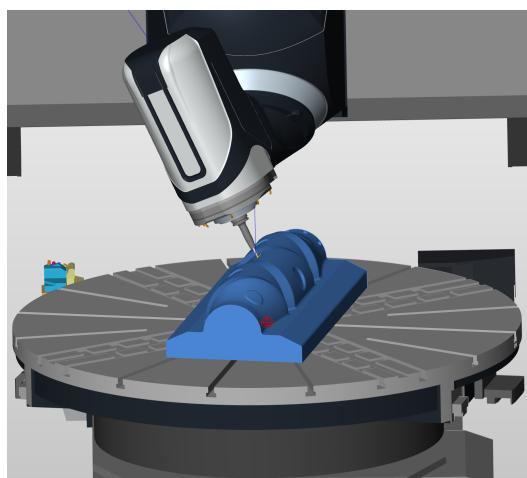


Example: BC head/table kinematics, angle for the C axis = 0

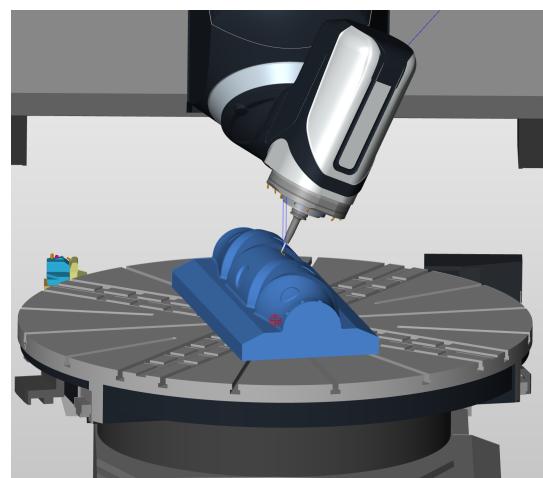
- (1) Solution for **Closest C angle plus** = B+35.6477 C+77.1914.
- (2) Solution for **Closest C angle minus** = B-35.6477 C+282.8086.

Example: BC head/table kinematics, angle for the C axis = 0

1



2



Linking job additive

A **Linking job additive** contains additive manufacturing jobs that are linked with each other in such a way that the approach and retract movements for the entire machining sequence and all linking movements between the individual jobs can be optimized and monitored for collisions.

In addition, the linking job additive offers sorting option which can be used to ease the programming task when several features must be built-up using different technology and/or inclination.

Available sorting strategies:

- No sorting
- Sorting based on the selected frame
- Sorting based on a selected surface
- Sorting by layer number of the reference job

Advantages

- Shortened manufacturing times,
- Reduction of programming times.
- Improved product flexibility.

Requirements

To combine several jobs into one **Linking job additive**, all sub-jobs must use the same additive tool.

Defining a linking job additive

Right-click in the *hyperMILL®* browser and select the **New → Linking job additive** function.

Add job to a linking job additive

Select the job(s) and then select the **Add to linking job additive** function on the shortcut menu or hold down the **SHIFT** key to move the jobs to the previously created **linking job additive**.

Tool dialog page

All jobs that are combined into a linking job additive manufacturing must use the same additive tool.



Parameters dialog page

Sorting strategy

The following sorting methods can be used:

No sorting: Linking job keeps the sequence in each reference job and between the different reference jobs.

Bottom to top - Frame: The linking job sorts the additive toolpaths from bottom to top based on their distance from the job frame.

Bottom to top - Surface: The linking job sorts the additive toolpaths from bottom to top based on their distance from the selected surface.

By layer: The linking job sorts the additive toolpaths based on the layer number and the job sequence within the linking job.

Functions on the shortcut menu: Job lists/Jobs

	<p>NC simulation: Use to load an already generated NC program into the VIRTUAL Machining Center. If the loaded NC file is not up-to-date, the following message appears:</p> <ul style="list-style-type: none"> • Existing NC files are not up to date, continue? Detailed information on all changes that affect the NC file are listed in the dialog. • The function Generate new NC files is activated by default. By confirming the query, new NC files are generated and loaded into the VIRTUAL Machining Center. If the function is deactivated, the VIRTUAL Machining Center is started with the outdated NC files.
	NC file does not exist.
	NC file exists and is up to date.
	NC file exists and is approved.
	NC file exists and is approved but is no longer up to date.
	<p>NC file exists but is no longer up to date.</p> <p>The NC status does not change when the software version is changed.</p>
	 <p>Double-clicking on one of the icons shows the exact status of the NC file with all changes that affect the NC program, such as changes to the settings of the Virtual Machine.</p>
	<p>Show detailed NC Status: Only available if an NC program has already been created. Deviations from the previous status are displayed. The NC status is updated automatically.</p>

Collision check preparations

Define fixture

Turning fixture

Turning fixture:

Select the **Frame for turning axis** from the list of existing frames.

	Define a new frame.
	Edit the selected frame.
	Preselect: Preselect the marked origin or frame. This can then be directly selected from the list in the job definition (see Tool → Frame) by clicking the icon. A preselected frame or origin is displayed in the browser as bold .

Calculate: Create a 2D silhouette from the selected 3D body or the selected surfaces for the turning jobs and the simulation.



Feature and macro technology

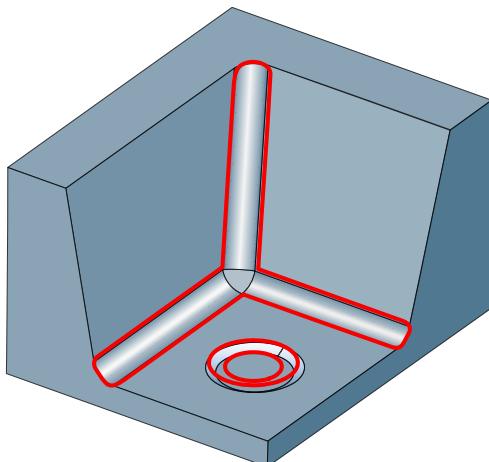
Generate feature

Feature Recognition

Rest material boundary

Create rest material areas to increase flexibility when machining rest material. The boundaries created can be used in all cycles that support boundaries.

Based on the definition of a reference tool, the theoretical rest material is calculated in the defined milling area and provided in the form of boundaries. The boundaries are calculated so that they relate to the center point of the selected machining tool, allowing flexible use for different orientations.



Selection

		Click the respective icon to select or redefine the Milling area and Frame for creating the rest material boundary.
		Click the respective icon Top , Bottom , and Boundary to manually limit the search area for the rest material to be detected for large components. Selected: The number of selected boundaries is displayed.

Tools

Reference tool: The **Diameter** of the reference tool is used to calculate the rest material areas to still be machined.

Machining tool: The creation of the boundaries takes into account the **Diameter** of the selected machining tool.

Allowance in job: Enter the allowance to be used in the job for rest machining and to be taken into account when creating the boundaries.

Result

		Create CAD curves : Click the respective icon to select Color and Layer .
--	--	--

If the function **Create automatic layer** is activated, the layer name is created automatically based on the name and diameter of the reference and machining tool.

Example: Reference tool diameter = 10, machining tool diameter = 6.

Layername: Ref. Ø = 10 // Mach. Ø = 6.

If the **Create automatic layer** function is not activated, enter the layer name directly in the input field (to the right of the layer icon ).

Drilling

Hole Brushing

Brushing, deburring, or polishing holes. This machining step ensures that holes are free of burrs, deposits, or sharp edges resulting from machining processes such as drilling, milling, or reaming. The aim is to achieve the best possible surface quality and dimensional accuracy.

Tools used for brushing do not remove any material apart from small burrs. For this reason the SIMULATION Center and the VIRTUAL Machining Center check the tool for possible collisions with the stock.

Parameters

Brushing area

Safety distance top: (1) Safety that is kept to the top of the hole during the brushing process. This is to make sure that the brushes of the tool are **not** leaving the hole, which potentially can damage the tool. These parameter overwrite the specified **Top**, in case they are defined higher than **Safety distance top**.

Top: (2) Set the upper position of the brushing area. If the checkbox is selected, the mode is switched to feature mode. If feature mode is active, the **Feature top** and the **Feature bottom** are derived from the features active in the Feature Connector. This means, for example, that the thread depth can be automatically transferred from a thread feature to the brushing area.

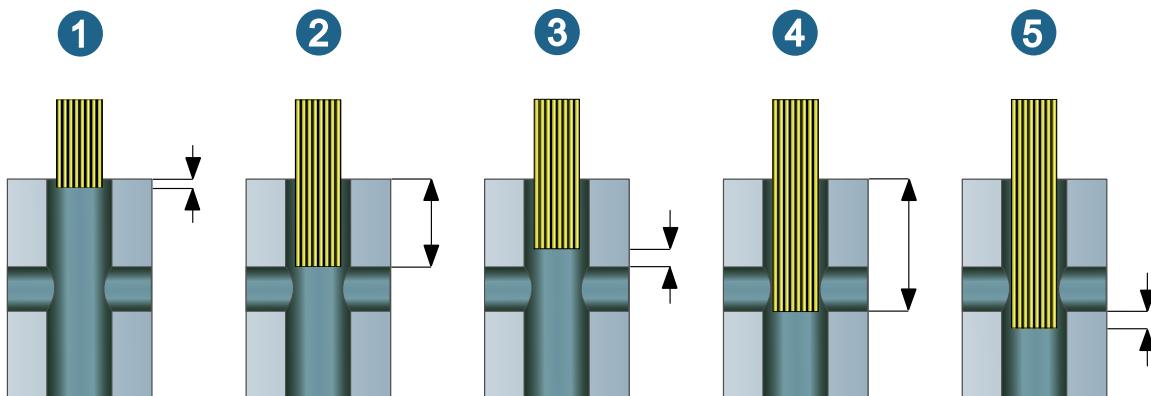
Feature top: The value for **Feature top** is automatically taken from the feature or sub-area of the feature (e.g. chamfer or thread) selected in the Feature Connector.

Extension top: (3) Extend the brushing area by the specified value towards the top.

Bottom: (4) Specify the Bottom position of the brushing area.

Feature bottom: The value for **Feature bottom** is automatically taken from the feature or sub-area of the feature (e.g. chamfer or thread) selected in the Feature Connector.

Extension bottom: (5) Extend the brushing area by the specified value towards the bottom.



Brushing parameters

Brushing direction: Select the desired brushing procedure. Available are:



- Up / Up - Down / Up - Down - Up.
- Down / Down - Up / Down - Up - Down.

Repetitions: Number of repetitions of the brushing process.

Setup

Model

Definition of the collision checked part of the CAD model. For further information, see section [Collision check preparations \[13\]](#).

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.

3D Machining

Optimized Roughing

Strategy

Machining mode



New version: This option activates a new calculation method for the toolpaths. Improved handling of rest material and large side infeeds shorten the machining times on the machine tool. The new method will replace the old one in the long term.



If the **Finish pass** option is used in combination with the **Parameters → Machining area → Manual top** function, the stock model above the defined value is not taken into account in the cycle calculation. This can lead to a collision between the tool and the stock model.

Plane Machining

Strategy

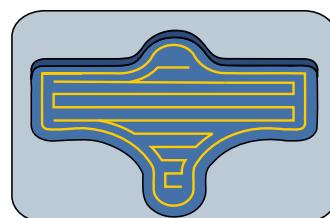
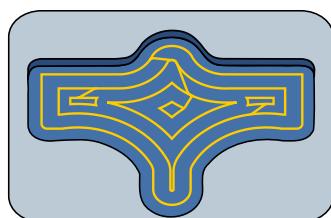
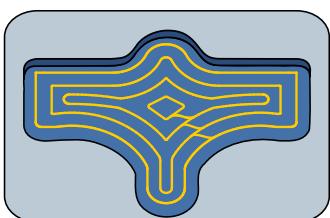
Pocket mode

Use the **Axis parallel mode** (3) to machine in the direction of the XY coordinates of the machining frame.

1

2

3



Infeed direction

Only available for **Axis parallel mode**.

Automatic: The machining direction is automatically set along the longest pocket side or planar surface to be machined.

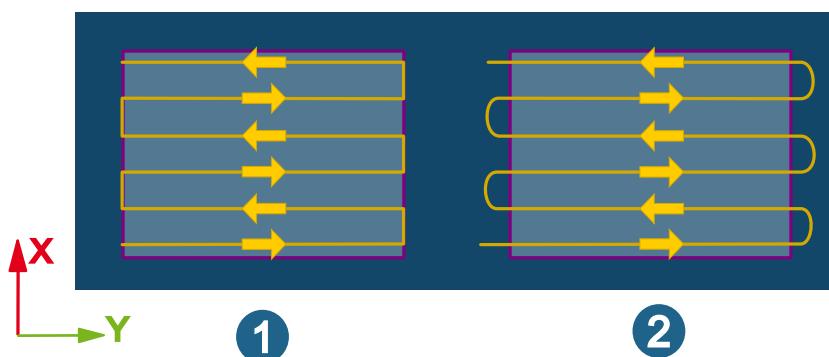
Machining angle: With a machining angle of 0°, the infeed direction corresponds to the X axis of the machining frame.

Infeed mode

Only available for **Axis parallel mode**. Horizontal stepover between the end point of the one milling path and the start point of the following milling path.

Zigzag direct: The machining direction changes per section. The infeed movement follows the shortest path. The horizontal stepover between two adjacent sections takes place with the machining feedrate (G1) (1).

Zigzag smooth: The machining direction changes per section. The infeed takes the form of a gentle movement. This mode is primarily intended for HSC machining (2).

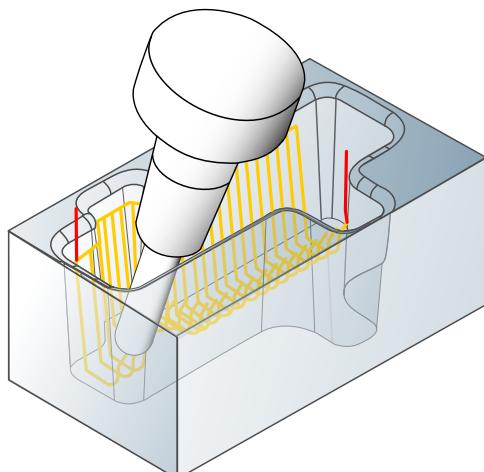


5X Machining

Cavity Milling

Profile Finishing (new)

Finishing of surfaces or areas with various milling strategies. The milling areas can be machined depending on a slope angle. The machining of flat areas is particularly suitable due to the projection direction. Various 5-axis strategies are available to reach areas that are difficult to access.





The 5-axis parameters

Inclination strategy

Automatic: All milling movements in G1 are, if possible, carried out automatically with a fixed orientation in order to achieve maximum speed and stability. When using rotary axes, a constant simultaneous movement of the rotary axes is possible thanks to the pre-analysis and evaluation of the entire toolpath segment.

Preferred tilt angle: Define a starting inclination which – if possible – is maintained for the entire toolpath. If the preferred tilt angle is not used, a vertical alignment is preferred.

Setup

Tolerance parameter

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



Semi finishing: The tolerances are designed for semi-finishing.

Finishing: The tolerances are adapted to general finishing.

Fine finishing: The tolerances are adapted to high-precision finishing (particularly high requirements of the components in terms of accuracy and surface quality).

Micro finishing: The tolerances are adapted to high-precision finishing of very small components. We recommend using this mode only for very small component sizes, as the calculation time and data volume can otherwise increase drastically.

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**.

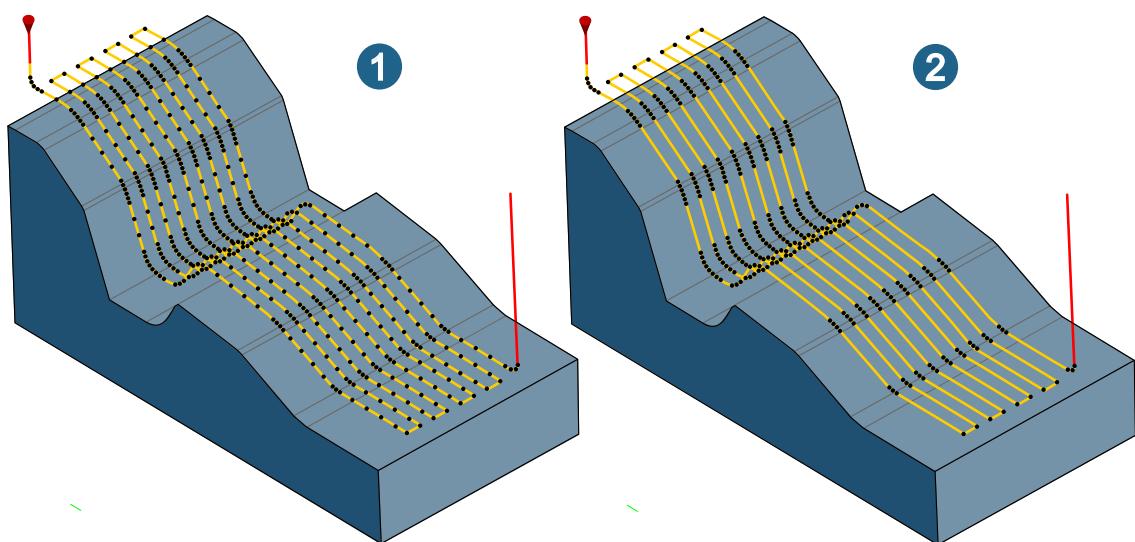
True Shape point distribution: Activate to redistribute points and guarantee an even and machine control-friendly distribution of points.



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.

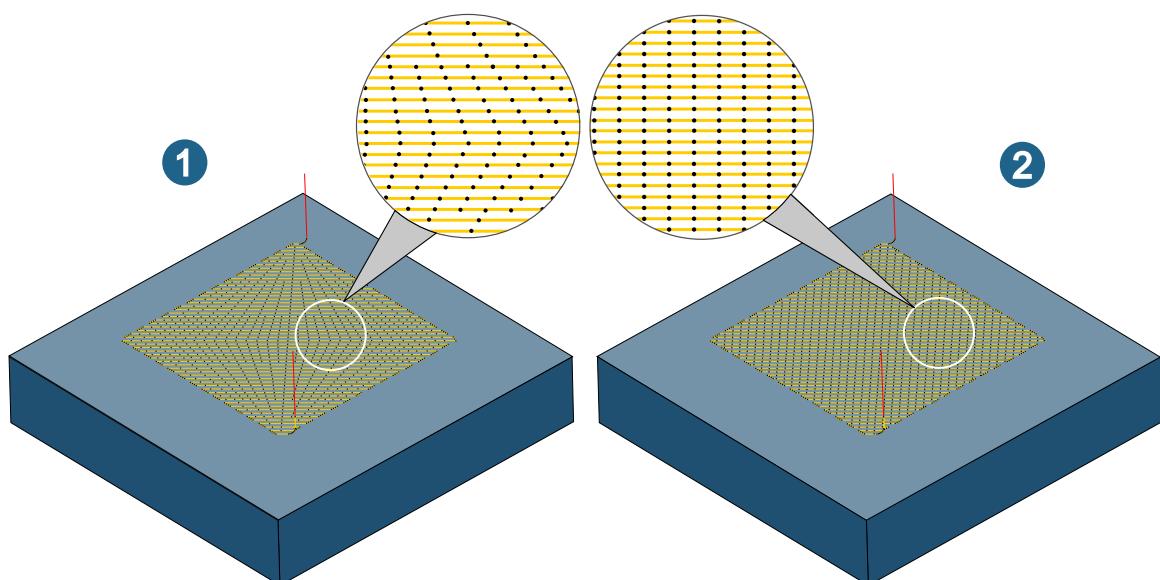
Function is not activated (1), Function is activated (2)



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.

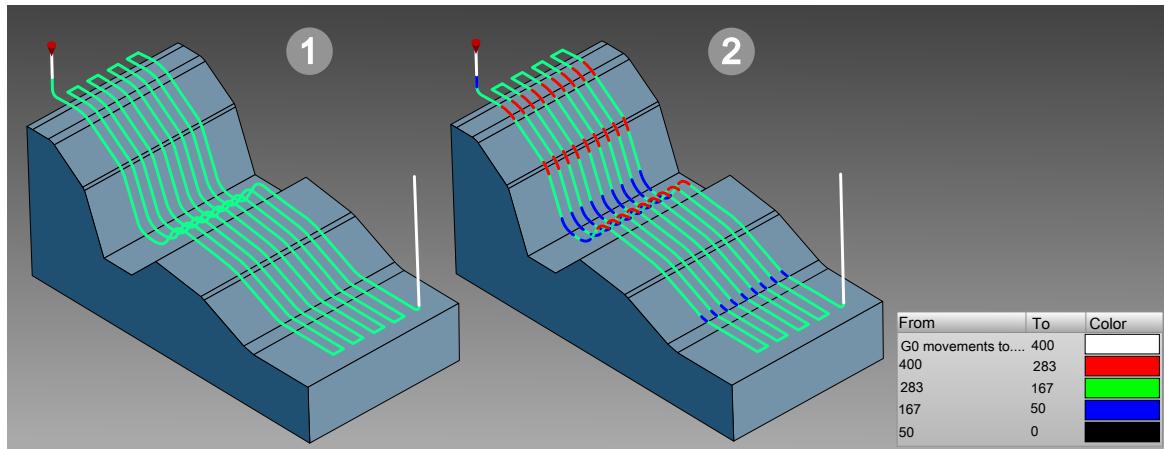
Cross align points: Align the points across the entire toolpath perpendicular to each other (i.e. at right angles to the toolpath direction).

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.

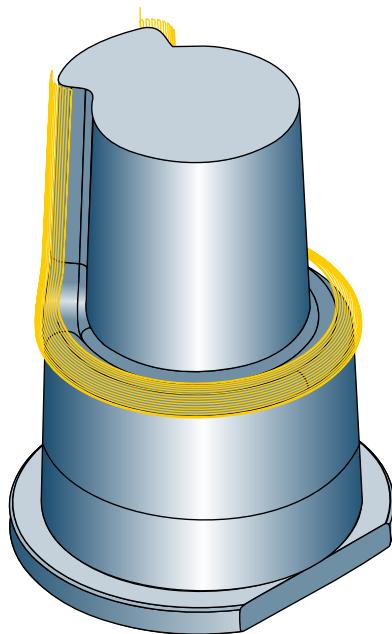
Activate **True Shape point distribution** to guarantee an even and correct point distribution in toolpaths. The parameters **Calculation tolerance**, **Max. chordal error**, **Max. G1 length**, **Min. G0 distance**, **Reference G1 length**, control the arrangement and distances between the points.

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

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.

Iso machining

Finishing of one or more surfaces. The milling paths follow the ISO lines (U, V) so that they are adapted to the surface profile. Various 5-axis strategies are available to reach areas that are difficult to access.



The 5-axis parameters

Inclination strategy

Automatic: All milling movements in G1 are, if possible, carried out automatically with a fixed orientation in order to achieve maximum speed and stability. When using rotary axes, a constant simultaneous movement of the rotary axes is possible thanks to the pre-analysis and evaluation of the entire toolpath segment.

Preferred tilt angle: Define a starting inclination which – if possible – is maintained for the entire toolpath. If the preferred tilt angle is not used, a vertical alignment is preferred.

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).



Semi finishing: The tolerances are designed for semi-finishing.

Finishing: The tolerances are adapted to general finishing.

Fine finishing: The tolerances are adapted to high-precision finishing (particularly high requirements of the components in terms of accuracy and surface quality).

Micro finishing: The tolerances are adapted to high-precision finishing of very small components. We recommend using this mode only for very small component sizes, as the calculation time and data volume can otherwise increase drastically.

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**.

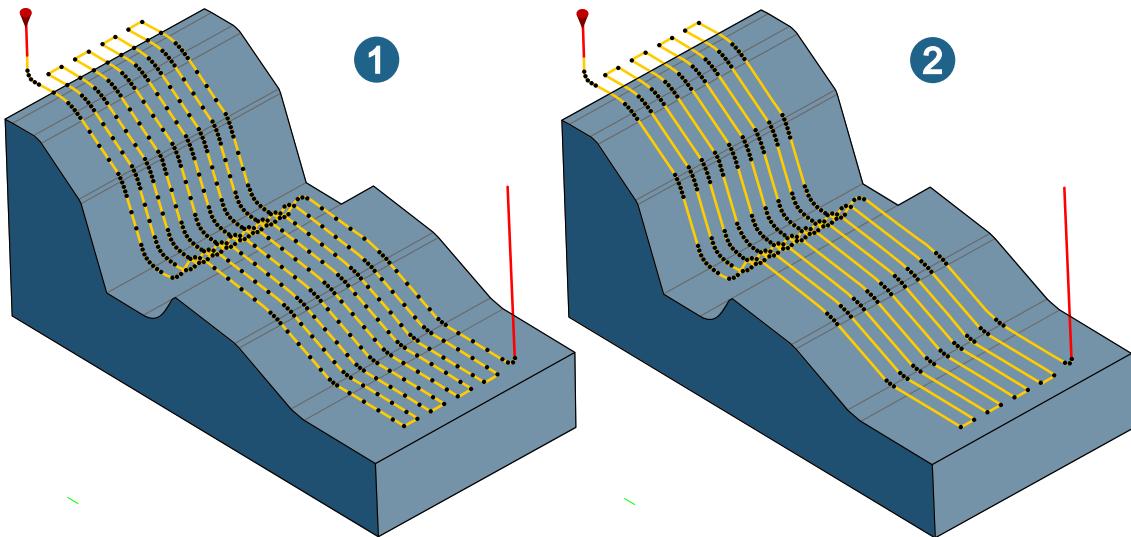
True Shape point distribution: Activate to redistribute points and guarantee an even and machine control-friendly distribution of points.



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.

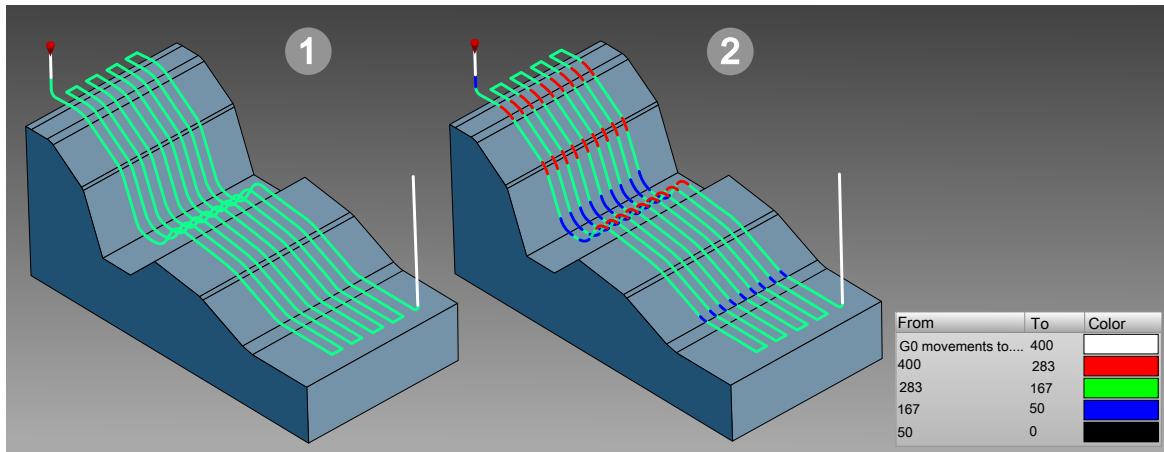
Function is not activated (1), Function is activated (2)



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.

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.



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.

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.

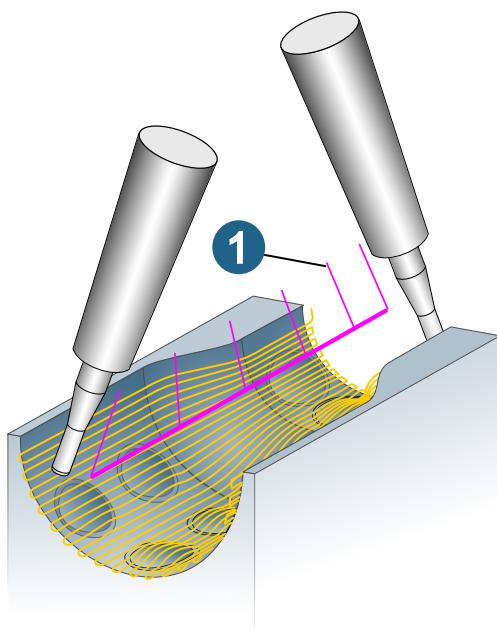


Radial machining

5 Axes

Axial inclination setup / radial inclination setup

Interpolate lines: The orientation of the tool follows the defined synchronization lines (1).

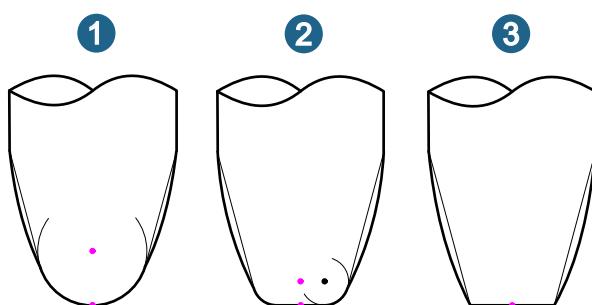


Surface Milling

Tangent Plane Machining

Tool

Supported tool types: Conical barrel tool with ball (1), rounded (2), and sharp (3) tip.



5X Deburring

Contours

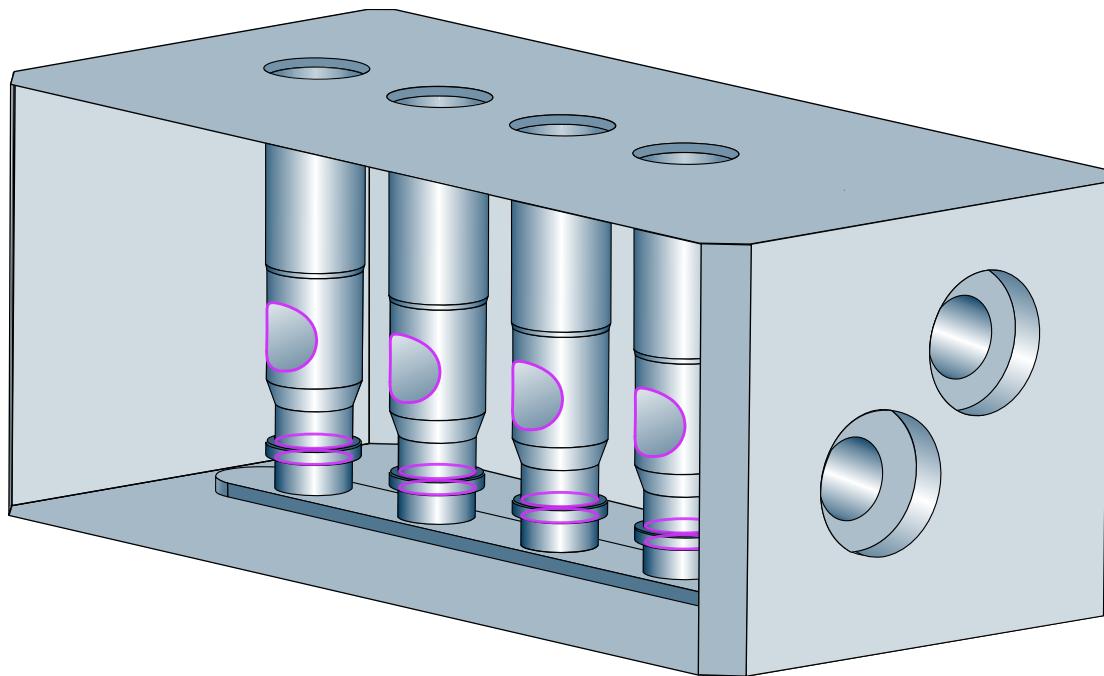
Technology

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.

5X Hole Deburring

Deburring of sharp edges within holes, such as intersections in cross holes. This cycle, which is specialised for machining holes, generates toolpaths for deburring sharp edges using 3-axis or 5-axis mode.



Tool

Supported tool types: Ball mill, Lollipop.

Contours

Define the edges and surfaces to be machined, the contour selection and the contour attributes.



CAM PLAN

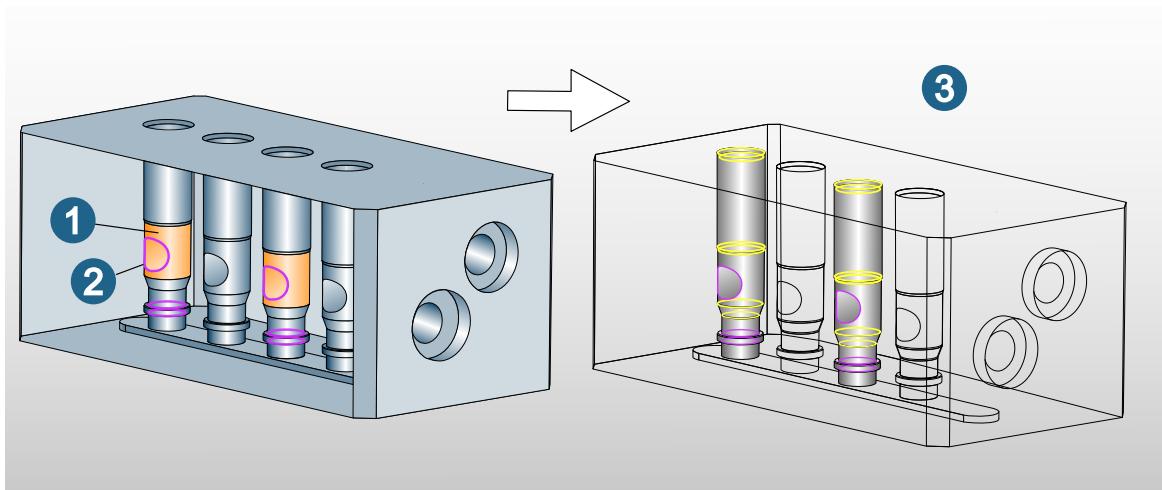
The edges to be machined are defined as preparatory work in the CAM plan. Information on this is provided in section **Basics of CAM editing** → **Structure CAM project** → **CAM Plan**.

Surface selection / Contour selection

Hole faces (1): Select the hole surfaces to be machined.

Contours: (2) By default, all sharp outer edges of the hole are preselected. If necessary, reselect the contours using the **New selection** button or change the selection using the **Edit selection** button.

If the **Display** → **Show selected only** function is activated, the selected surfaces/contours are highlighted and the rest of the model is displayed as a wireframe (3).



Hole attributes

The edges to be deburred, the contours not to be deburred, the hole surfaces and the direction of the hole are displayed.

Geometry / Technology



Freely select the **Start point** for each contour. Set an **End point** if the contour is to be machined only partially.

Reverse: In the default setting the machining direction follows the direction of the selected contour. This can be seen in the graphic preview. If the generated toolpaths do not result in the intended machining direction, select the relevant contours and select the **Reverse** option.

Parameters

Definition of the machining area and the infeed and clearance parameters.

Deburring parameters

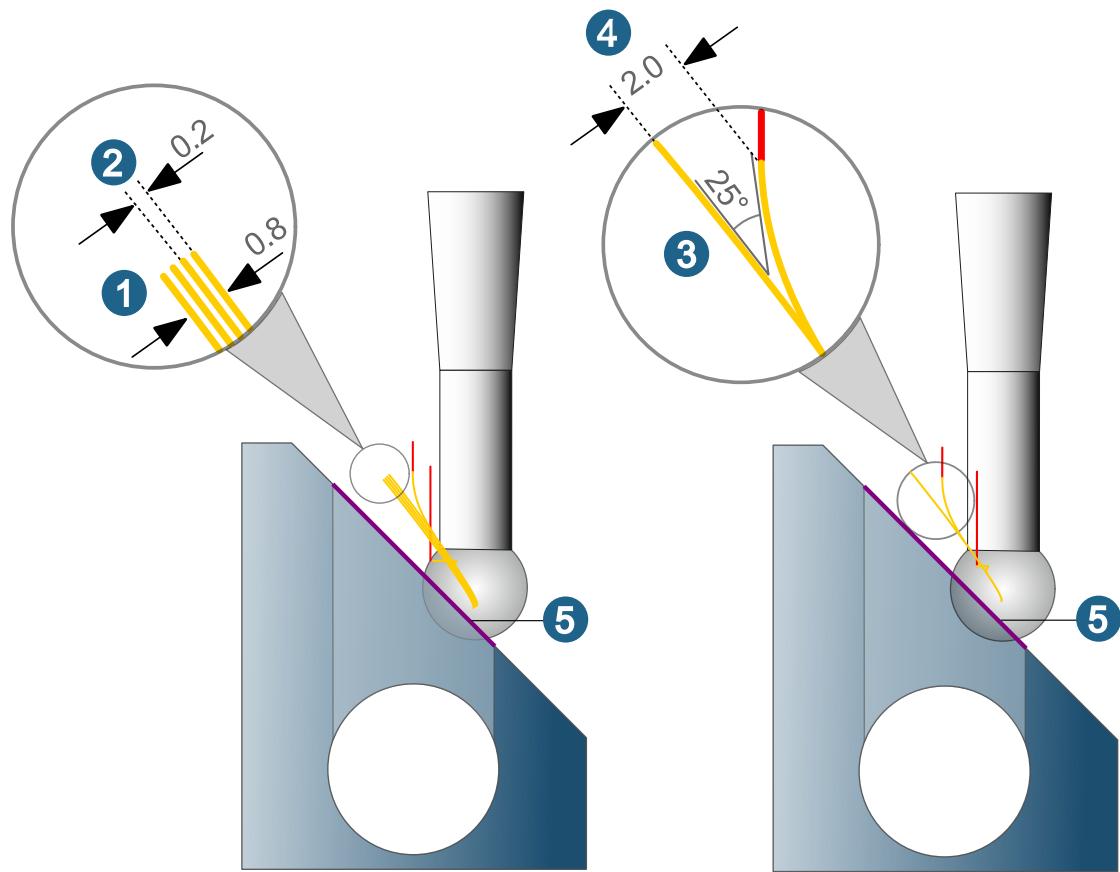
Chamfer distance: (1) Chamfering is defined by a distance from the sharp edge.

Max. axial stepdown: (2) Maximum material removal per tool pass. The number of passes is calculated from the deburring depth.

Ramp angle: (3) The approach to the edge to be deburred is ramped. The form of the approach movement is defined by the **ramp angle** parameter.

Ramp height: (4) Defines the distance above the edge at which the first ramp movement begins.

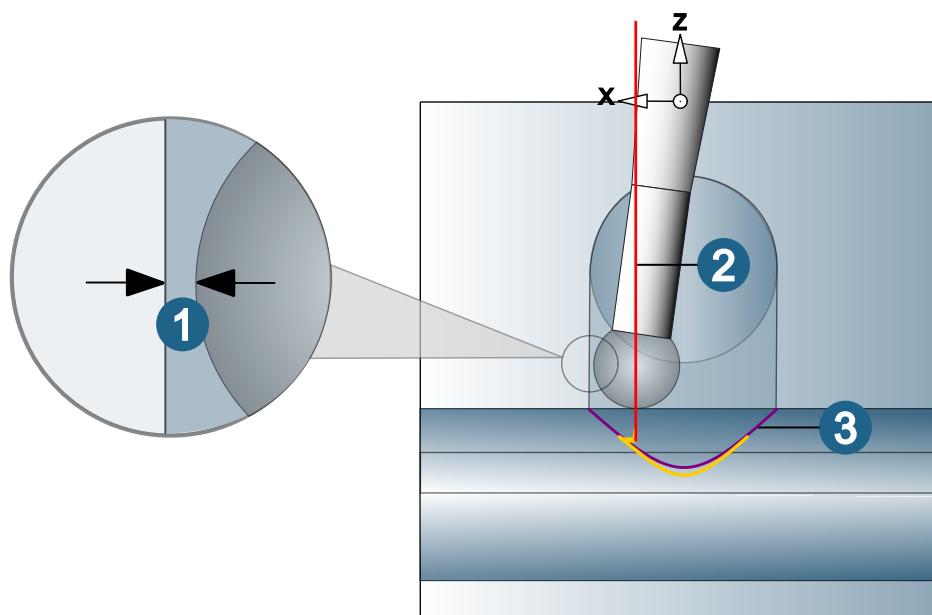
(5) Edge to be deburred.



Hole security

3D hole security: (1) Distance of the tool to the wall of the hole during approach and for movements between the edges to be deburred.

Positioning feedrate: (2) Feedrate for movements within the hole and if the distance between the tool and the hole is smaller than the defined parameter **3D hole security**. (3) = Edge to be deburred.



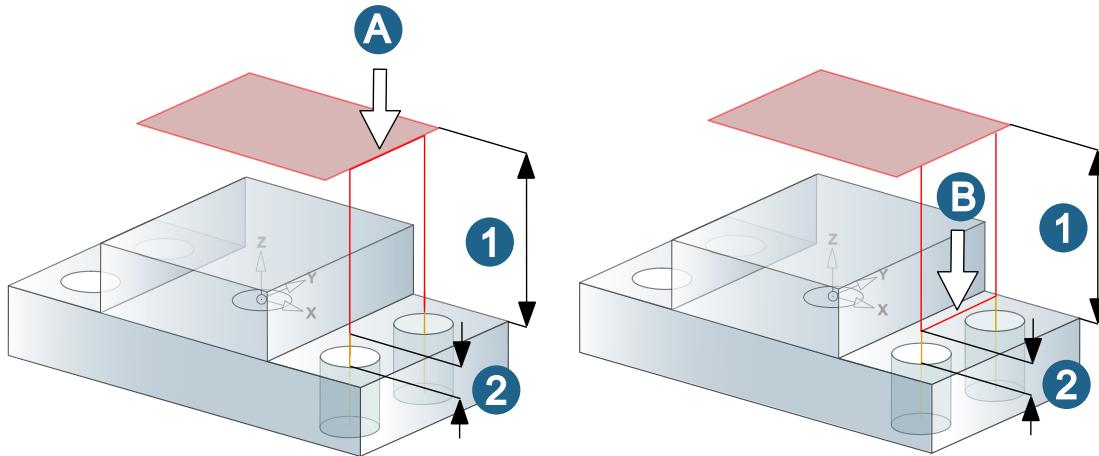


Retract mode

The retract mode defines the Z level where the system executes horizontal infeed movements.

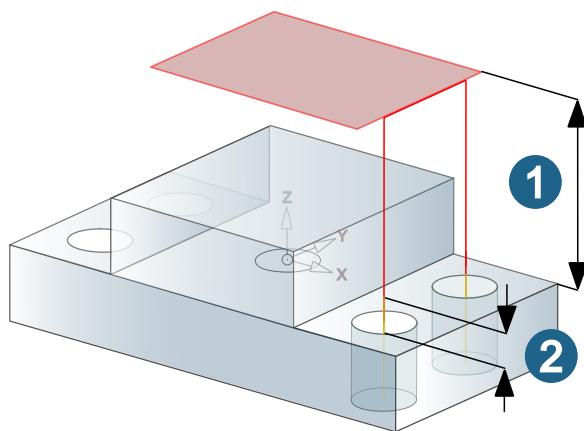
Clearance plane (1): Start and end position of each machining operation is on the clearance plane (A). The linking movements between the machining operations take place at the height of the clearance plane (A).

Clearance distance (2): Start and end position of each machining operation is on the clearance plane (A). The linking movements between the machining operations take place at the height of the clearance plane (B).



Clearance

The clearance parameters **Clearance distance (1)** and **Clearance plane (2)** limit the areas for rapid movement. Above the clearance distance, movements of the tool take place as rapid movements. Below the clearance distance, movements of the tool take place with the machining feedrate.



5 axes

Machine kinematics

5X Machine: The machine used has 5-axis kinematics. As an option, the hole can also be machined simultaneously with 5 axes.

3X Machine: The machine used has 3-axis kinematics. The hole must be machined with 3 axes.

5X

Allow simultaneous: An attempt is made to machine the hole with five axes and with a fixed tool inclination (indexed). If this is not possible, simultaneous 5-axis movements are permitted.

Machine limitation

Max. angle to Z: Can only be enabled if the machine used has 5-axis kinematics. Permissible values for the **Max. angle to Z** are between greater than/equal to 0° and less than/equal to 180°. The default value is 90°.

Blade Milling

Point Milling

Parameters

Technology: final paths

Blades may require step-by-step blade machining. To do this, use the technology of final paths.

Advantageous when

- when working from the platform in the direction of the blade,
- an oversized overlapping area is to be machined (at least partially) with a different strategy,
- the blades must be stabilised to prevent deformation.

For a smooth overlap, specify the number of additional **Overlapping paths** and the **End distance**. The lateral infeed is continued in the same way as when fading in. The paths on the surface must therefore be finished earlier. Use the parameter **Add. side clearance** for this. Calculate this parameter as follows:

at least the number of **Overlapping paths** multiplied by the **Side stepover**.

Top Milling

Parameters

Technology: entry paths / Technology: final paths

Blades can require step-by-step blade machining. Ensure a smooth overlap at the start and end of machining using the technology for entry and final tool paths.

Advantageous when

- the turbine blades must be supported to prevent deformation,
- blade areas cannot be machined with Top Milling.

For a smooth overlap, define the respective number of additional **Overlapping paths** as well as the **Start distance** / **End distance**.

The side stepover is continued. The machining area on the surface must be defined in such a way that the overlapping paths do not cause any collisions with the part.

Multiblade Milling

MB Point Milling

The cycle supports multiple processor cores and therefore shows a significant performance improvement for large calculations.

Parameters

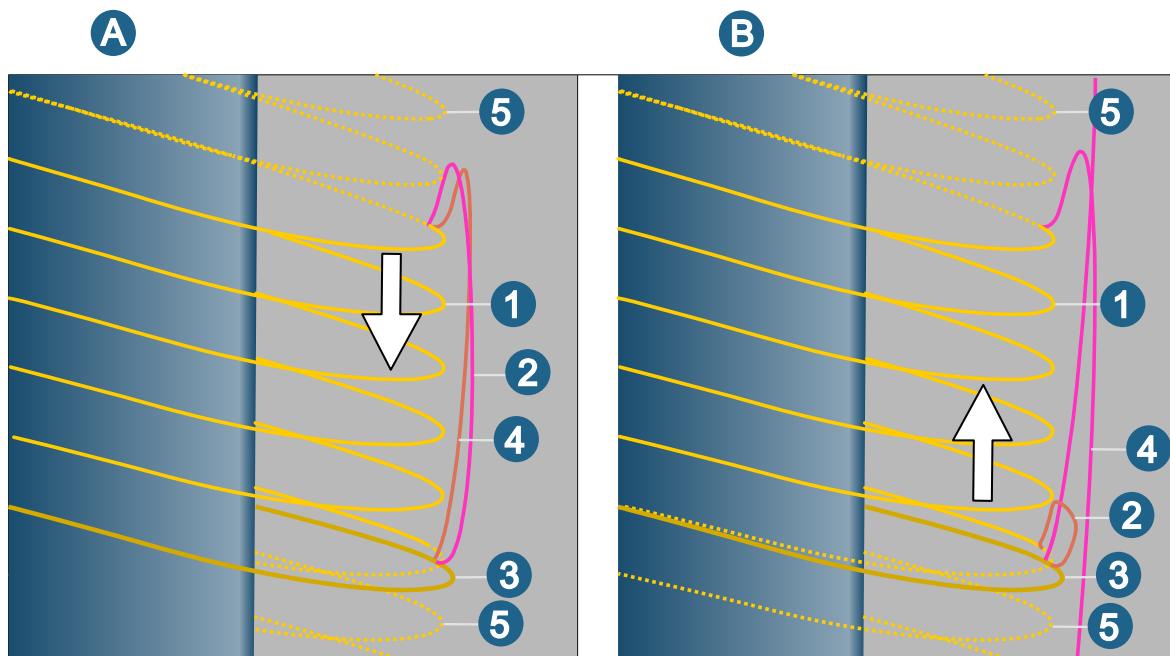
Technology: Stepwise finishing with blending path

Divides the entire finishing toolpath into short steps/areas, with a transition toolpath that reduces the material thickness directly before the finishing path. Blade finishing can therefore be applied to a stronger/thicker pre-machined component, reducing/avoiding deformation and potentially reducing the need for semi-finishing.

Finishing

Contained finish paths: Number of finishing toolpaths for each step.

The **Finishing direction** makes it possible to carry out each finishing step **Bottom to top** or **Top to bottom**.



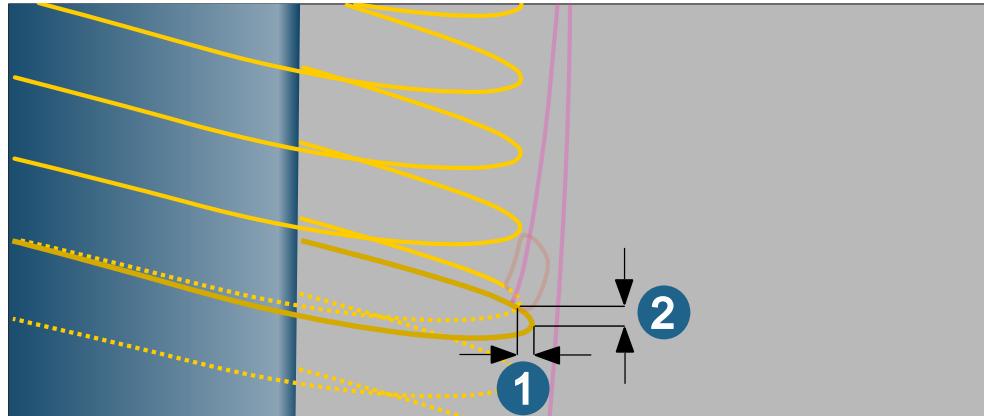
(1) Local finishing toolpaths, (2) linking movement to the finishing toolpaths, (3) blending path, (4) linking movement to the blending path, (5) global finishing toolpaths.

(A) Machining **Top to bottom**, (B) machining **Bottom to top**.

Blending path

Additional allowance: (1) Remaining material thickness after the transition toolpath, which is removed during subsequent finishing.

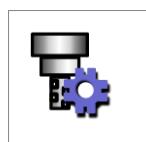
Offset to finishing step: (2) Distance of the transition toolpath before the finishing step.



Tool database

Define turning tool

Define static holder



To define a new mounting position for a turret holder, switch to the **Geometry** area and start the TOOL Builder by clicking the icon.

1.

2. In TOOL Builder use the function **Add new mounting position**. For further information, see section **Turning tool: Create static holder** in the product documentation for the TOOL Builder.
3. Once the new mounting position has been defined in TOOL Builder, it is displayed in the tool database as **Mounting position [x]** in the **Technology** area and in the geometry area as a tooltip of the coordinate system. Enter the value for **Index angle** of the holder manually according to the manufacturer's specifications. If this value is not equal to 0, the entry **Indexed mounting positions** is activated



Version 2025 only supports the first defined **Mounting position** when working with the **Machining planner**. The use of further mounting positions will be implemented with a future update for version 2025.

Optimal barrel cutter



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

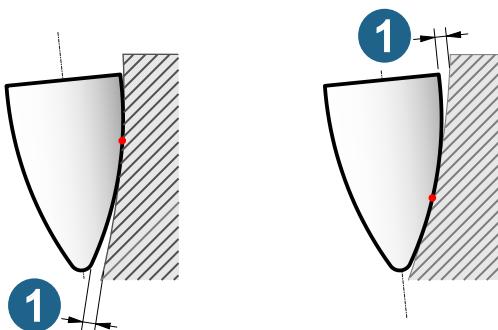
CAM → Optimal barrel cutter

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

2024

Options

Barrel - max. distance: ① Enter a maximum permissible distance between the barrel contour and the surface to be machined. This distance can be either above or below the contact point. Any allowance in between is not taken into account; the tool is always calculated so that it remains in contact with the geometry.

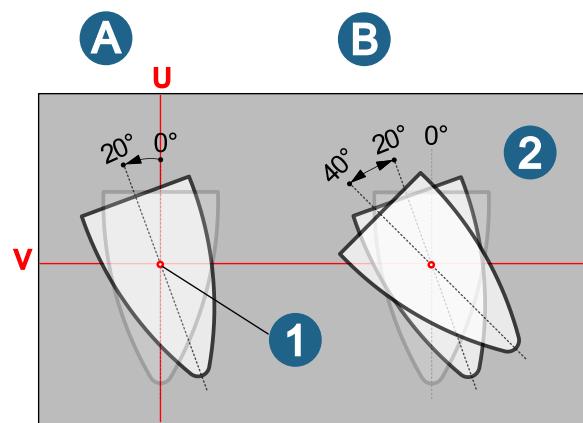


Lead angle

Enter a lead angle as the tool orientation along the selected curve or transfer it from the job settings (**5 Axes → Tilt parameters → Lead angle**) to optimize the calculation of the barrel radius.

The lead angle value is taken from the tool settings of a job, can be changed and is transferred back. In case of an Associative job copy, the associativity is lost if the value is changed.

- **Fixed lead angle:** ①: Enter a value for the lead angle that must be maintained when calculating the optimum barrel shape. With a value of 0°, the alignment of the tool axis corresponds to the U or V direction of the selected surface.
 - **Min. lead angle and Max. lead angle:** ②: Enter a max. and min. value for the lead angle, which must be considered when calculating the optimum barrel shape.
- ① Contact point of the tool with the selected surface, ② Selected surface.



Result

Optimal Lead Angle: The optimal tool lead angle is output.

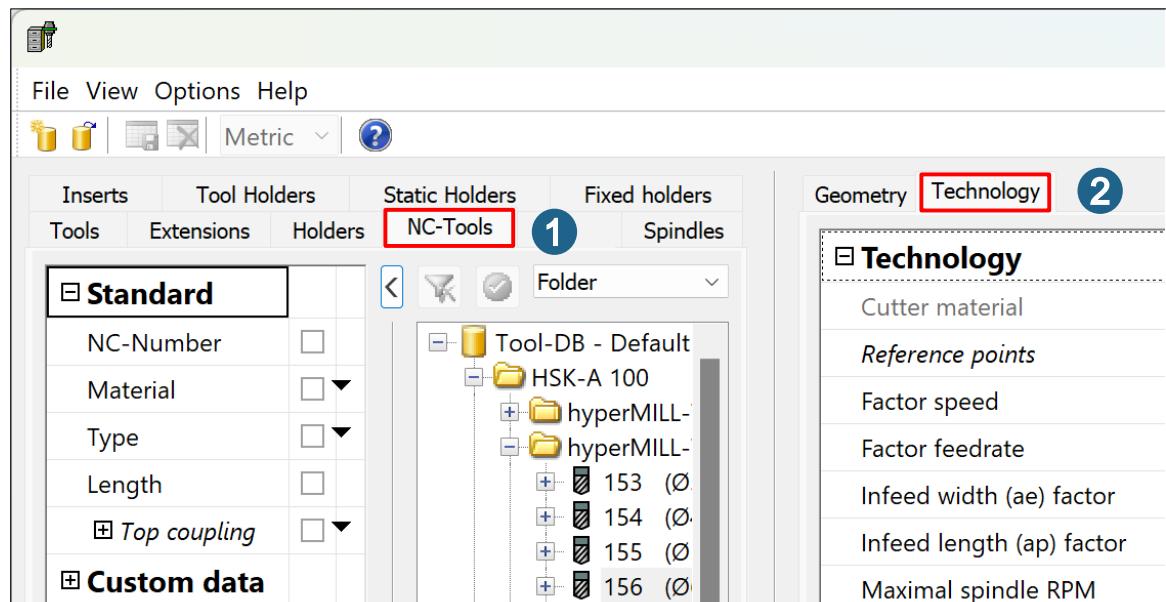
Define NC tool

NC tools and technology data

Usages and types of cut define in the tool for which applications the **Tool** can be used. Depending on the assembly (for example, with holders of different lengths), the **NC tool** may only be used in certain situations or for certain materials.

Adapt technology data for an NC tool

To adjust the technology data for an **NC tool** (1), switch to the **Technology** (2) dialog page in the data view of the overview window.



In the lower dialog area under **Material** (3), deactivate the materials (4) and types of cut/usages (5) that are not to be supported.

<input type="checkbox"/> Material	3	<input type="checkbox"/> Wrought aluminum alloys Si < 6%	4	Cutting class
Plunging		Type of cut		Spindle RPM (n)
		<input type="checkbox"/> Helical plunge	5	10504
		Type of cut		Spindle RPM (n)
		<input checked="" type="checkbox"/> Ramp plunge		8594
				Plunge angle
				8
Roughing		Type of cut		Spindle RPM (n)
		<input type="checkbox"/> Full cut		8594
				Infeed length (ap)
				10
		Type of cut		Spindle RPM (n)
		<input type="checkbox"/> Standard roughing		10504
				F/edge (fz)

TOOL Builder

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



Add a new mounting position

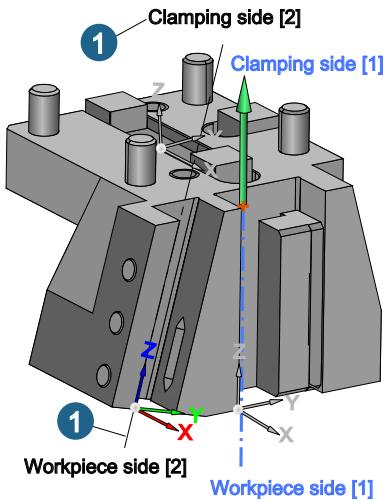


To add an additional mounting position for a turret holder, select the **Add new mounting position** function and carry out the **Define tool axis** and **Define tool coupling** steps again.

Use the **Modify Tool** function on the shortcut menu to adapt a previously defined mounting position.

Then use the **Define tool axis** and **Define tool coupling** functions to correct the selection to be changed.

Use **Remove tool** to delete the mounting position.



CAD

User interface

Tabs

Visibility

Layer

Exclude layers from selection operations

Exclude all entities of a layer or layer container from all selection operations. All entities of these layers are displayed in simplified form.



Switch off the selection of entities of the layer or layer container.



Exclude entities assigned to the selected layer from all selection operations.



Switch on the selection of entities of the layer or layer container.



Entities that are assigned to the selected layer can be selected again.



Some layers of the layer container are excluded from selection.

All entities that change layers automatically assume the status of the target layer. The selection of other entities works as if these entities were transparent. It is not possible to select or make selectable a subordinate entity of a non-selectable entity. A group and its subordinate faces should be on the same layer. If not, the group must be opened to set the layer to not selectable.

The simplified representation eliminates the hierarchy of entities, the boundaries, the texture and various other attributes. This improves rendering performance, but some commands are not available in this state, e.g., changing attributes, tessellation and visibility filtering. In particular, color filtering of faces of a solid is not available.



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

Application

Enable non-selectable layers in visibility tab

Enable non-selectable layers in visibility tab. The display of the icons is updated according to the selection made when the document is reopened.

UI style

Choose between styles. Select the standard appearance with **Dark blue**. **Light grey** matches the color of the CAM part of the software. The slightly larger font is compatible with the Japanese user interface (this font is easier to read than the font in the dark blue style). The label of the button to which the current focus has been set is shown in italics.

Model > Attributes

New layer is always current

A newly created layer is also always immediately set to current.

Model > Entities

Tessellation tolerance

Tolerance for the display of curves, faces, and solids.

For performance reasons, a tessellation tolerance no smaller than 0.001 can be set here. All newly created and imported faces are assigned this tessellation tolerance as a minimum. If a more precise tessellation is required, the tessellation tolerance in the entity properties of selected entities can be refined to a minimum of 0.0001.

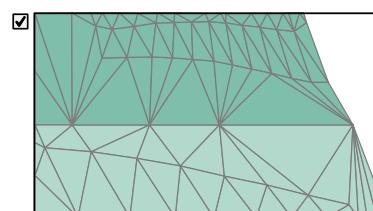
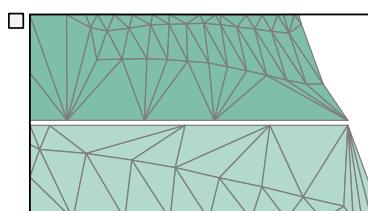


It is recommended that all entities belonging to the same workpiece geometry are tessellated using the same value. This avoids virtual gaps in the geometry. For solids, the **Tessellate connected** option is used to align the boundaries of neighboring faces for display so that no gaps are visible.

Solid

Tessellate connected

Switch connected tessellation on or off. If the topology of a solid is defined as closed between the faces of the solid based on the tolerance, the tessellation is closed – the solid is displayed without gaps. This also applies if there are gaps between the boundaries of individual faces.





Graphic > System > Navigation

Rotation center mode	Determine the navigation behavior when the view is rotated. Select how the center of rotation is to be found. The Enhanced ray cast option defines the standard behavior. A beam is “cast” at the mouse pointer position to find entities. If there are no entities at the mouse pointer position, the center of the view volume is used. The Fit command aligns the center of rotation with the center of the bounding box of the model. The Ray cast option corresponds to Enhanced ray cast , but the Fit command updates the center of rotation in the middle of the view. If the Compatibility option is selected, the previous behavior up to version 2024 is retained for reasons of compatibility with Ranorex tests (for quality assurance of software programming) or for nostalgics. In this case, the behavior depends on the view. In spherical view, the center of rotation is kept in the middle of the view by the Open Inventor graphics engine, while in cylindrical view the center of rotation is found in the same way as with the Enhanced ray cast option.
Update rotation center	The rotation center is updated when Zoom in or when Dynamic rotation starts, using the mouse pointer position.
Show rotation center	Mark the rotation center point around which the display rotates. If Always is selected, the rotation center point is permanently displayed in the graphics area. If Never is selected, the center of rotation point is not displayed. The selection Manually set only is used to display the rotation center point if it was set manually with the Set rotation target command. The manually set center of rotation point is displayed in red, the implicitly set point is displayed in green.

Selection

Mesh edge selection	Switch the selection of mesh edges on or off.
Mesh face selection	Switch the selection of mesh faces on or off.

Sketcher options



Options for the Sketch and V-sketch commands

File → Options → Sketcher options

V-sketch options

Max. diagonal length of chamfers: To identify chamfers, enter a value for the maximum diagonal length of a chamfer.

Show modified curves: Display which curves of the turning contour have changed as a result of entering a tolerance (using the [Modify linear tolerance \[57\]](#) command). A message is displayed in the **Info** tab. Select the message to highlight the changed entities. This makes it possible to detect unwanted changes and prevent these by adding further constraints.

Data interfaces

General interfaces

ACIS file formats

Read or save *.sat and *.sab files.

**Table 1. Options for opening files in ACIS formats**

Property	Description
Entity conversion modes	Merge meshes: When selecting No , no merging will take place; choosing Merge to one activates merging and creates an overall mesh; choosing Merge by colors merges meshes based on their color. Multiple colors will result in multiple meshes.
Layer assignment	Set all layers non-selectable: The entities of all imported layers are set to non-selectable. By setting all layers to non-selectable directly before opening the file, the performance of handling imported data is improved. Individual layers can then be activated later as required for specific processes.
Healing	Remove faces with width less / equal (mm/inch): Enter a value for skipping <i>Sliver faces</i> during import – these are line-shaped faces that have no impact on the model geometry. Entering 0 means that all line-shaped faces are accepted.

Table 2. Options for saving an ACIS file

Property	Description
Save hidden entities	Entities that are marked as hidden can be saved or ignored.
Conversion tolerance	Set the accuracy for the conversion of the entities.
Keep faces orientation	Either retain the existing face orientation or leave it to chance.
Save as binary format	Optionally save the data in .sab ACIS binary format.

The settings are saved in `hewritersettingssatvalues.xml`.

IGES file format

Import or save IGES file `*.iges`, `*.igs` (Initial Graphics Exchange Specification).

Table 3. Options for importing an IGES file

Property	Description
Layer assignment	Set all layers non-selectable: The entities of all imported layers are set to non-selectable. By setting all layers to non-selectable directly before opening the file, the performance of handling imported data is improved. Individual layers can then be activated later as required for specific processes.
Healing	Remove faces with width less / equal (mm/inch): Enter a value for skipping <i>Sliver faces</i> during import – these are line-shaped faces that have no impact on the model geometry. Entering 0 means that all line-shaped faces are accepted.

Table 4. Options for saving an IGES file

Property	Description
Conversion tolerance	Define the permissible tolerance for the conversion of entities to an IGES entity type.
Save solids as faces	Solids are broken down into faces. Faces are saved.
Keep faces orientation	Optionally retain the existing faces orientation or disregard it.



JT-Open file format

Read or save *.jt JT-Open file.

Table 5. Options for opening data via a JT-Open interface

Property	Description
Entity conversion modes	Merge meshes: When selecting No , no merging will take place; choosing Merge to one activates merging and creates an overall mesh; choosing Merge by colors merges meshes based on their color. Multiple colors will result in multiple meshes.
Layer assignment	Set all layers non-selectable: The entities of all imported layers are set to non-selectable. By setting all layers to non-selectable directly before opening the file, the performance of handling imported data is improved. Individual layers can then be activated later as required for specific processes.
Healing	Remove faces with width less / equal (mm/inch): Enter a value for skipping <i>Sliver faces</i> during import – these are line-shaped faces that have no impact on the model geometry. Entering 0 means that all line-shaped faces are accepted.

The settings are stored in the file `cadifoptjtovalues.xml`.

Table 6. Options for saving a JT-Open file

Property	Description
Save hidden entities	You can either save or ignore entities that are identified as hidden.
Conversion tolerance	Define the permissible tolerance for the conversion of entities to an JT-Open entity type.
Keep faces orientation	Optionally retain the existing faces orientation or disregard it.

The settings are stored in the file `hewritersettingsjtovalues.xml`.

Parasolid file format

Read or save the *.x_t, *.x_b Parasolid file.

Table 7. Options for opening data via a Parasolid interface

Property	Description
Entity conversion modes	Merge meshes: When selecting No , no merging will take place; choosing Merge to one activates merging and creates an overall mesh; choosing Merge by colors merges meshes based on their color. Multiple colors will result in multiple meshes.
Layer assignment	Set all layers non-selectable: The entities of all imported layers are set to non-selectable. By setting all layers to non-selectable directly before opening the file, the performance of handling imported data is improved. Individual layers can then be activated later as required for specific processes.
Healing	Remove faces with width less / equal (mm/inch): Enter a value for skipping <i>Sliver faces</i> during import – these are line-shaped faces that have no impact on the model geometry. Entering 0 means that all line-shaped faces are accepted.

**Table 8. Options for saving a Parasolid file**

Property	Description
Save hidden entities	You can either save or ignore entities that are identified as hidden.
Conversion tolerance	Define the permissible tolerance for the conversion of entities to a Parasolid entity type.
Save solids as faces	Solids are broken down into faces. Faces are saved.
Keep faces orientation	Optionally retain the existing faces orientation or disregard it.
Save as binary format	Optionally save the data in the *.x_b Parasolid binary format.

The settings are stored in the file `hewritersettingsparvalues.xml`.

PLY2 file format

Open or save *.ply2 PLY2 file for polygon mesh.

Table 9. Options for opening a PLY2 file

Property	Description
Entity precision	Assign entity tolerance.
Unit of measure	Select Millimeter or Inch as the document unit of measure.
Template	Merge: Insert CAD data to be imported into a hyperMILL® document template. hyperMILL document template name (*.hmct): Select document template.

The settings are stored in the file `meshloadply2filesettingsvalues.xml`.

Table 10. Options for saving a PLY2 file

Property	Description
Save hidden entities	Save or ignore optionally hidden entities.
Unit of measure	Select the Millimeter or Inch unit of measure. Document retains the unit of measure used in the document.
Tessellation mode	Existing: Use the document's currently set tessellation tolerance under Options / properties . Default: Write the data to the file with a specific, default tessellation tolerance set for it. Connected: Combine the tessellation of the individual entities into a connected tessellation of the entire surface of the model data and save it (for example, for advanced processing in a 3D print or in a simulation). This mode is another way of creating the mesh itself: the node points are the same on all edges in order to create a 'connection' between the nearest faces. The arrangement of the triangles is recalculated according to the desired properties.
Tessellation tolerance	Enter the tessellation tolerance for the Default tessellation mode.



Property	Description
Connected Tessellation - tessellation tolerance	Write the data to the file in the Connected tessellation mode with the specified tessellation tolerance.
Connected Tessellation - max edge length	Enter the maximum permissible edge length of the triangles for the Connected tessellation mode.
Connected Tessellation - max boundary edge length	Enter the maximum allowed edge length of the triangles whose edges form part of an entity's boundary for the Connected tessellation mode.

The settings are stored in the file `meshsaveply2filesettingsvalues.xml`.

PRC file format

Read or save `*.prc` PRC file (Product Representation Compact).

Table 11. Options for opening data via a PRC interface

Property	Description
Entity conversion modes	Merge meshes: When selecting No , no merging will take place; choosing Merge to one activates merging and creates an overall mesh; choosing Merge by colors merges meshes based on their color. Multiple colors will result in multiple meshes.
Layer assignment	Set all layers non-selectable: The entities of all imported layers are set to non-selectable. By setting all layers to non-selectable directly before opening the file, the performance of handling imported data is improved. Individual layers can then be activated later as required for specific processes.
Healing	Remove faces with width less / equal (mm/inch): Enter a value for skipping <i>Sliver faces</i> during import – these are line-shaped faces that have no impact on the model geometry. Entering 0 means that all line-shaped faces are accepted.

The settings are stored in the file `cadifoptprcvalues.xml`.

Table 12. Options for saving a PRC file

Property	Description
Save hidden entities	You can either save or ignore entities that are identified as hidden.
Conversion tolerance	Define the permissible tolerance for the conversion of entities to a PRC entity type.
Keep faces orientation	Optionally retain the existing faces orientation or disregard it.

The settings are stored in the file `hewritersettingsprcvalues.xml`.

STEP file format

Import or save STEP file `*.step`, `*.stp` (Standard for the exchange of Product Model Data).

**Table 13. Options for opening data via a STEP interface.**

Property	Description
Entity conversion modes	Set curve and face names: The entity names for curves and faces are transferred. This information can be used, for example, to transfer the original ID, production, or change information that is not defined by dimensions for dimensional and positional tolerances from the source system. Merge meshes: When selecting No , no merging will take place; choosing Merge to one activates merging and creates an overall mesh; choosing Merge by colors merges meshes based on their color. Multiple colors will result in multiple meshes.
Layer assignment	Set all layers non-selectable: The entities of all imported layers are set to non-selectable. By setting all layers to non-selectable directly before opening the file, the performance of handling imported data is improved. Individual layers can then be activated later as required for specific processes.
Healing	Remove faces with width less / equal (mm/inch): Enter a value for skipping <i>Sliver faces</i> during import – these are line-shaped faces that have no impact on the model geometry. Entering 0 means that all line-shaped faces are accepted.

Table 14. Options for saving a STEP file

Property	Description
Conversion tolerance	Define the permissible tolerance for the conversion of entities to a STEP entity type.
Keep faces orientation	Optionally retain the existing faces orientation or disregard it.
STEP AP format	Output the data in accordance with the STEP AP 203, AP 214, or AP 242 standard.

Direct interfaces

Options

Table 15. Options for opening data via a direct interface

Property	Description
Entity conversion modes	Merge meshes: When selecting No , no merging will take place; choosing Merge to one activates merging and creates an overall mesh; choosing Merge by colors merges meshes based on their color. Multiple colors will result in multiple meshes.
Layer assignment	Set all layers non-selectable: The entities of all imported layers are set to non-selectable. By setting all layers to non-selectable directly before opening the file, the performance of handling imported data is improved. Individual layers can then be activated later as required for specific processes.
Healing	Remove faces with width less / equal (mm/inch): Enter a value for skipping <i>Sliver faces</i> during import – these are line-shaped faces that have no impact on the model geometry. Entering 0 means that all line-shaped faces are accepted.



Select and snap

Select entities

Chain

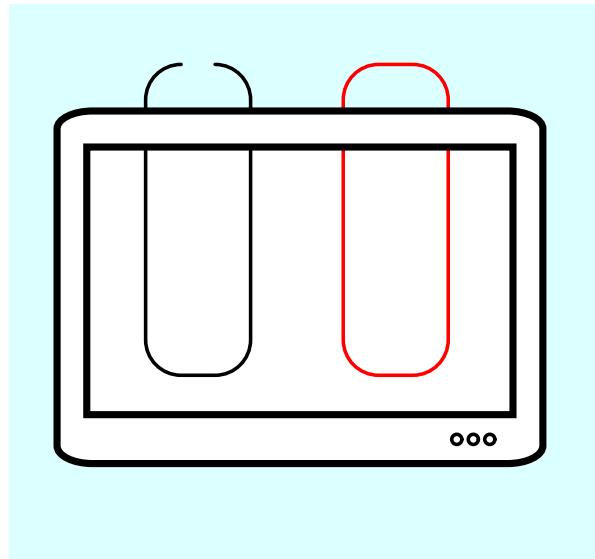
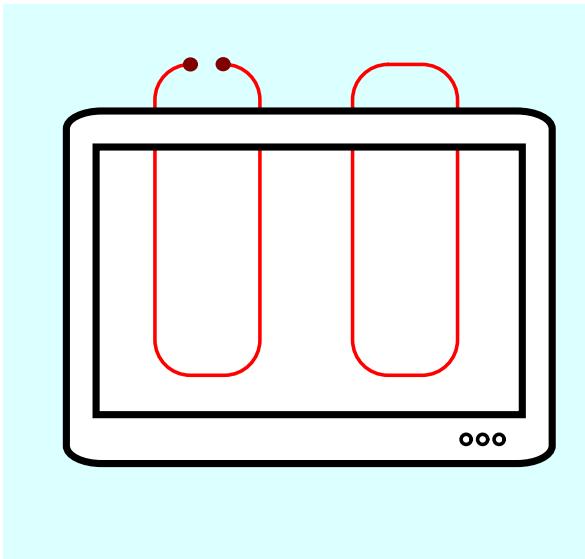


Select a contour or a boundary.

Select → Chain

Select

Only closed: If a chain is selected that extends beyond the graphic area of the screen, it is not possible to see whether the chain is open or closed. This information is important, for example, for the delimitation of a processing area. When using this option, only closed chains are marked ⑦. Other chains are discarded.



Snap entities

Mesh edge on / off



Switch the selection of mesh edges on or off.

Select → Mesh edge on / off



2025

Switch the selection of mesh edges on or off. When switched on, interact directly with mesh edges as selectable entities. If switched off, these entities are excluded from the selection, which simplifies the workflow by focusing on the entire mesh. Preset the selection in **File → Options → Options / properties** under **Selection**.

Mesh face on / off



Switch the selection of mesh faces on or off.

Select → Mesh face on / off



2025

Switch the selection of mesh faces on or off. When switched on, interact directly with network faces as selectable entities. If switched off, these entities are excluded from the selection, which simplifies the workflow by focusing on the entire mesh. Preset the selection in **File → Options → Options / properties** under **Selection**.

Analysis

Check

Shape continuities



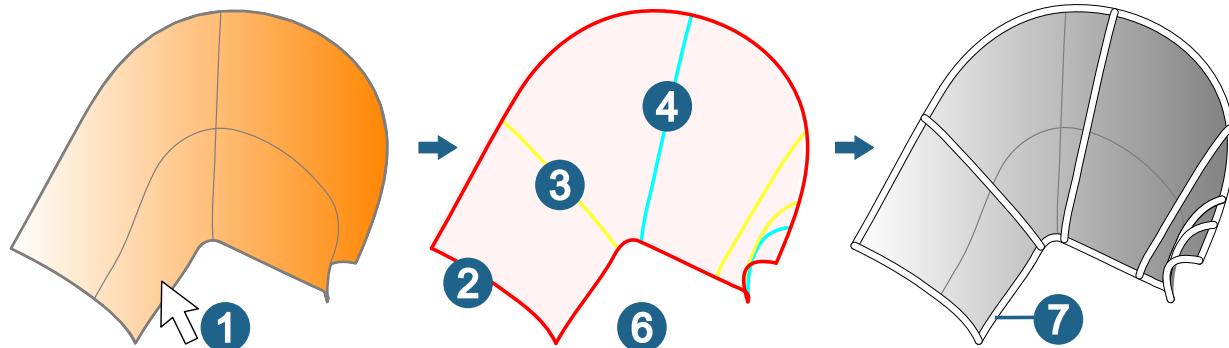
Analyze continuities at edge transitions and internal continuities.

Analysis → Shape continuities

2023.2 | Update 3

More options

Internal continuities: Visualize continuities within NURBS surfaces ⑥. Both internal C0 and C1 continuities can thus be identified directly. Milling operations can be better prepared and validated.



Workplane

Create workplane

Mirror aligned



Create a workplane for NC programming of a geometry with a near mirror-image shape.

Workplane → Mirror aligned

2025

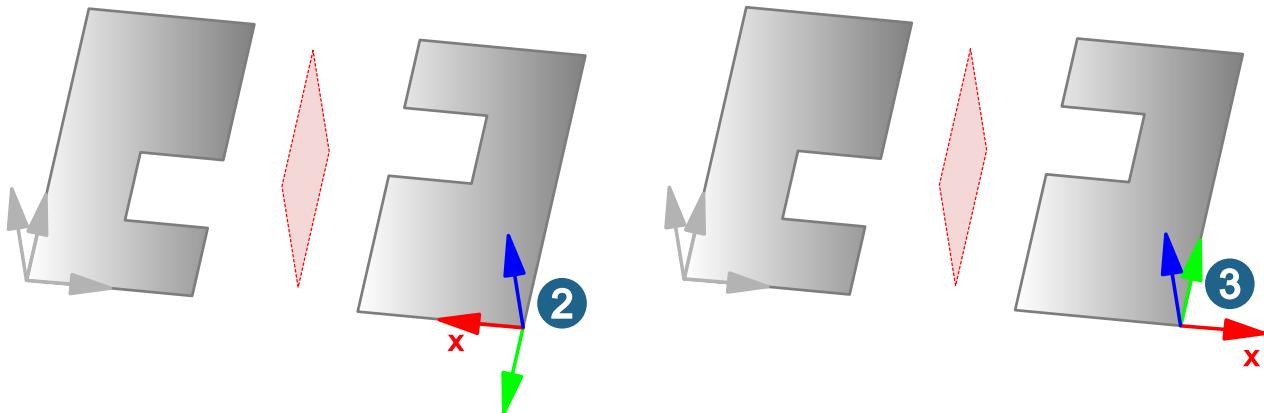
Create a workplane for NC programming of a geometry with a near mirror-image shape. The Z axis is mirrored, and either the X axis or the Y axis is realigned based on the right-hand side. The resulting workplane must not be associative with the original workplane.

NC programming of nearly mirrored workpieces is simplified. For completely mirror-image workpieces, program the “right side” for milling and mirror all cycles with CAM. However, if a workpiece is only very close to

being a mirror image, it is not possible to proceed in this way. How should users proceed in this case? Mirror the workplane used on the “right-hand side” of the near mirror-image workpiece in an aligned manner. This mirrored workplane is then given a new frame, and the cycles can be applied to the new frame.

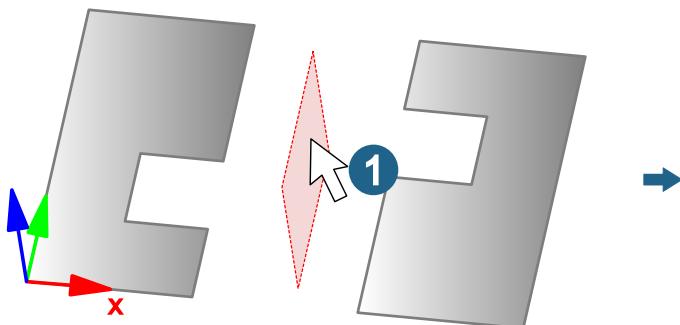
Axis to keep

X or Y: Either the X axis or the Y axis is realigned based on the right-hand side. Select the X or Y axis ②③.



Modes

Select the mirror plane ①.



Entity: Select an entity (e.g., plane curve or face) to define the mirror plane.

3 points: Select three points that are not on a straight line. The mirror plane is created based on these points.

Direction + origin: Define the mirror plane using **Origin** and **Direction**:

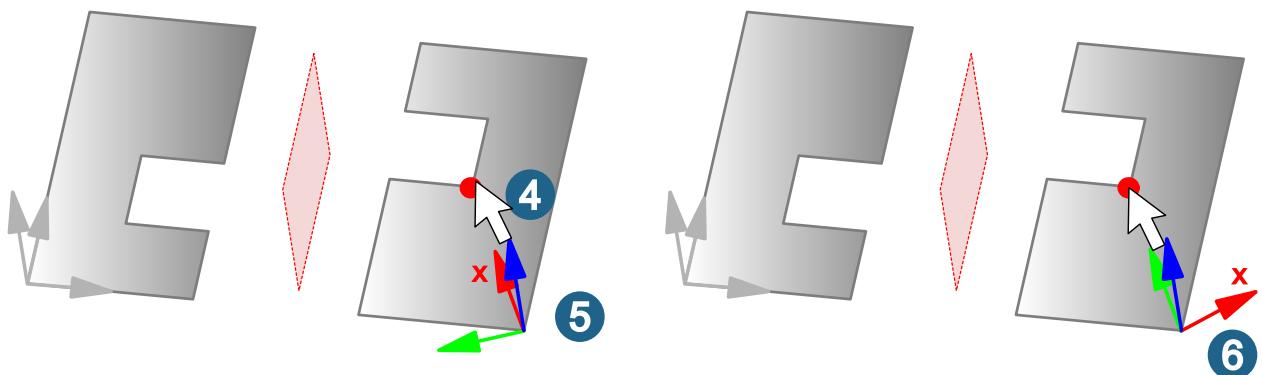
XY axis direction

Automatic: The X and Y axes are aligned based on the result of the mirroring.

X: Align the X axis to an entity ⑤.

Y: Align the Y axis to an entity ⑥.

Entity: Align the X or Y axis to an entity, e.g., a point or parallel to a direction (line). Do so by selecting an entity ④.

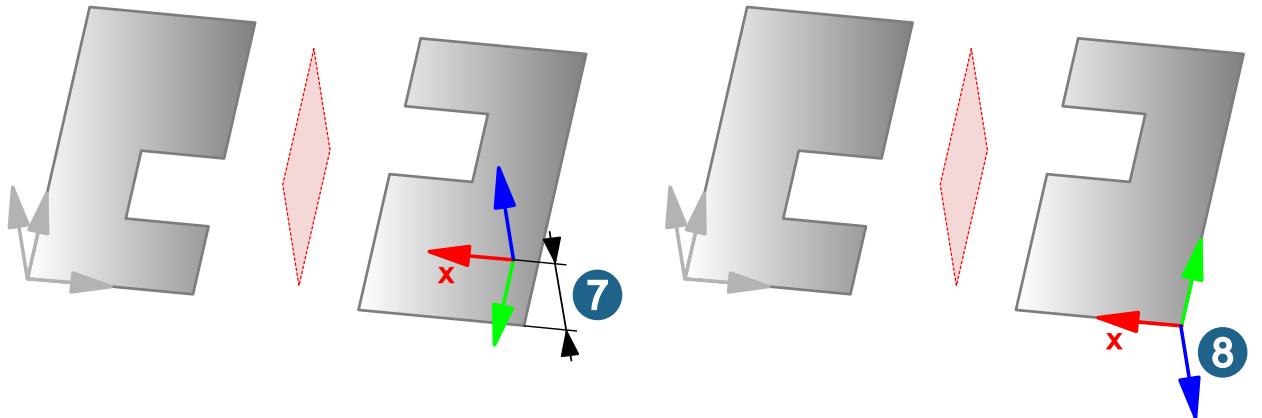


If the selected point or the selected direction (line) is not already on the preset XY plane, the point or line is projected onto the XY plane. The result is used to align the X or Y axis.

Z axis

Invert: Reverse the positive Z axis direction in the opposite direction ⑧. The X direction remains and the Y direction is adjusted.

Offset: Move the workplane in the direction of the Z axis. Enter a value ⑦.



Edit and modify

Edit

Graphic properties



Show graphic properties.

[Edit → Graphic properties](#)

Table 16. Settings that can be made in the graphic properties.

Property	Description
Tessellate connected	Switch the connected tessellation on or off for each entity.



Modify

Extend / shorten face



Modify the length of the faces.

Modify → Extend / shorten faces

Options

Keep domain: The untrimmed initial face is retained.

Split face



Split one or several faces along isoparametric curves.

Modify → Split face

Options

Keep domain: The untrimmed initial face is retained.

Points, curves and faces

Drafting

Curves

Shape contour



Create the contour of a shape as a curve.

Curves → Shape contour

Rotational

More options

Create tag references: Entity ID information is inherited. The generated rotation contours receive references (TAG T_SHAPE_ROTATIONAL_REFERENCES) to the face IDs. This means that model-based definitions, which are referenced to the faces, can also be read via the curves. If a curve is created by more than one face, the entity is defined as a “new entity” and receives the preset color since individual parent entities cannot be assigned.

Keep attributes Without Keep attributes, the properties active for new entities are applied.

Shapes

Linear sweep



Create single curved faces via linear sweeps.

Shapes → Linear sweep

This concerns the handling of penetration at filleted corners with inclined side faces. An edge is created from the penetration to replace the fillet.

Helix



Insert helical face.

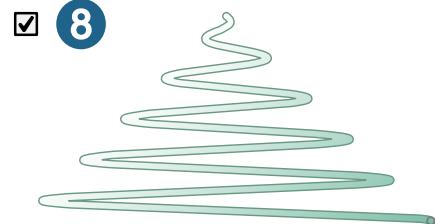
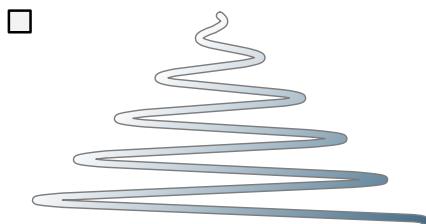
2022.1

Shapes → Helix

More options

Solids

The **Solid** option is used to combine the faces into a solid ⑧.



Fillet



Create a fillet face.

Shapes → Fillet

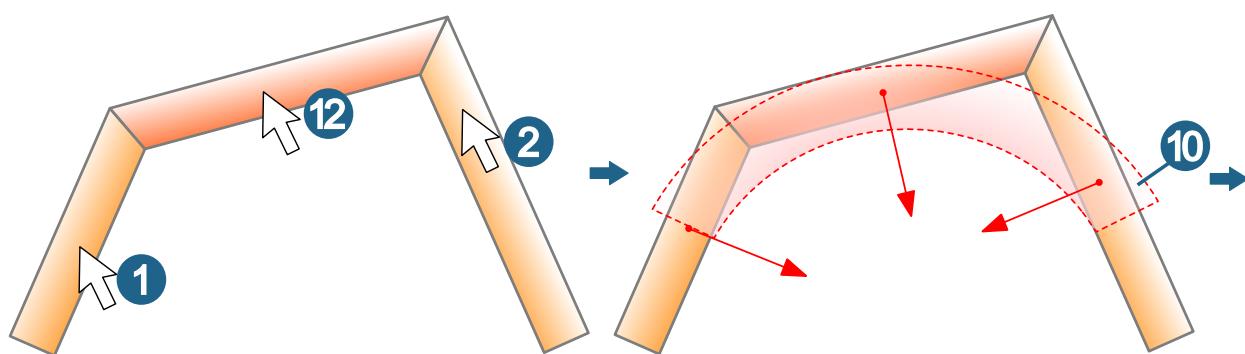
Tri-tangent

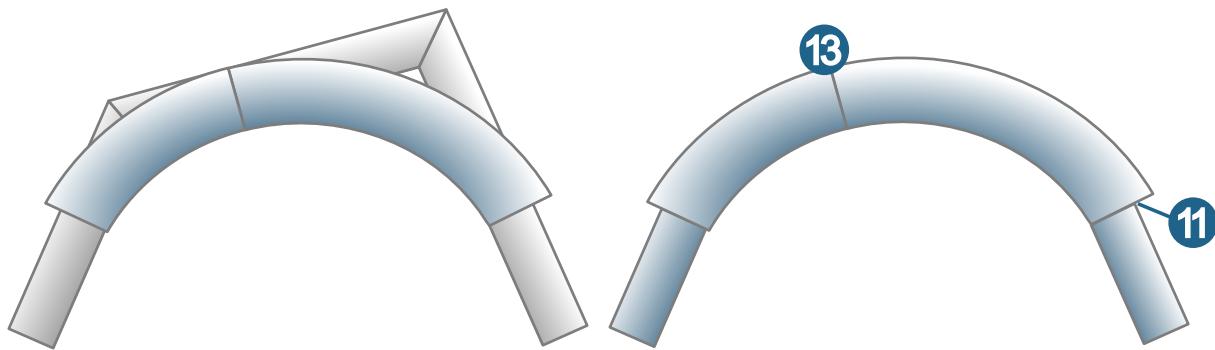
Fillet with tangential transition between 2 side faces and a central face separating these faces – such as a rib or the inflow of an impeller, etc. This ensures seamless transitions free of curvature between the three faces. The radius of the fillet is not constant, but adapts to the angles between the faces to create tangency.

Select the three faces. The face-normal directions are determined automatically.

Mid-face: Select mid-face ⑫.

Delete mid-face: The mid-face is removed from the result ⑬.





V-sketch

Automatic constraints



Add automatically defined constraints.

Drafting → V-sketch → Automatic constraints

Constraints

	Dimension – Chamfer	Insert Dimensional constraints for chamfers.
	Fixed angle	The angle of the entity remains constant during changes.

Geometric constraints



Add geometric constraints.

Drafting → V-sketch → Geometric constraint

Constraints

	Fixed angle	The angle of the entity remains constant if changes are made.
--	-------------	---

Options

Select series: Multiple application of constraints (Parallel, Coincidence XY, Coincidence X, Coincidence Y, Coincidence radius, Coincidence length) in succession without interruption. All constraints assigned in this way are treated as a single constraint and can be changed simultaneously, e.g., with a single dimension.

Solids, features and meshes

Features

Helical protrusion



Create a protrusion via the helical sweep of a contour from curves and boundaries on a face within a solid.

Solid

Feature → Helical protrusion

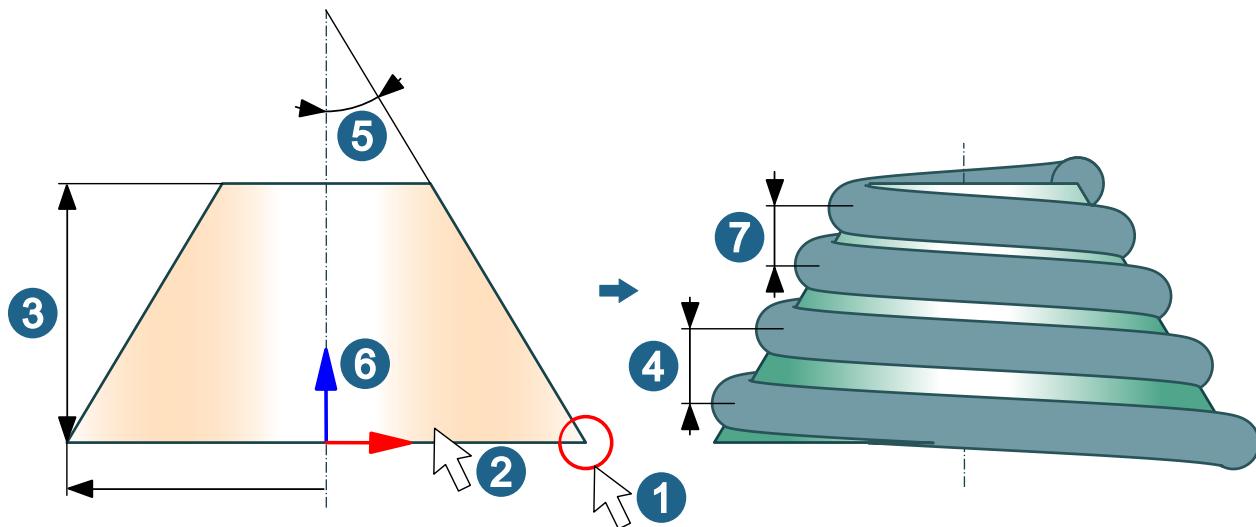
2025

Create a protrusion via the helical sweep of a contour from curves and boundaries on a face within a solid. The curves must form a single closed and plane chain. Parametric modeling is supported.

Select

Curves: Define the starting point and profile of the helix. Select curves ①. The distance between the curve and the origin defines the radius of the helix at its base. The number of selected entities is displayed.

Faces: Select entities ②. The number of selected entities is displayed.



Select the direction and the origin of the rotation axis ⑥.

More options

The **Height** describes the total height of the helix ③. The **Pitch** ④ and **Taper** ⑤ set the angle, further defining the shape of the helix. The direction of rotation is reversed using **Clockwise**.

Variable pitch: Optionally, change the pitch continuously from the beginning to the end of the helix. Enter a value for the **Final pitch** ⑦.

Helical slot



Create a slot via the helical sweep of a contour from curves and boundaries, based on a face within a solid.

Solid

Feature → Helical slot

2025

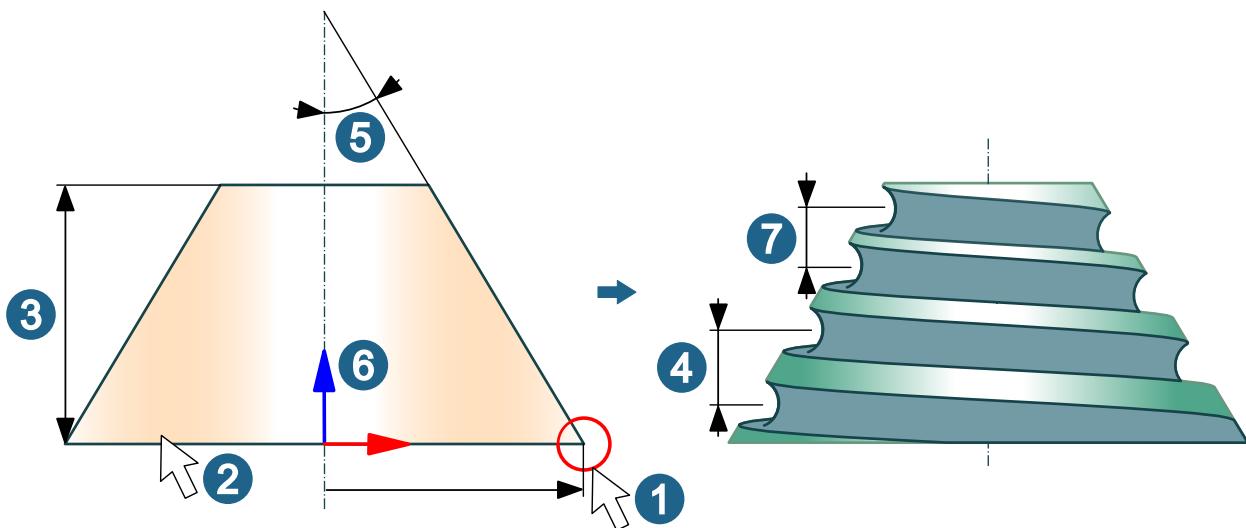
Create a slot via the helical sweep of a contour from curves and boundaries, based on a face within a solid. The curves must form a single closed and plane chain. Parametric modeling is supported.



Select

Curves: Define the starting point and profile of the helix. Select curves ①. The distance between the curve and the origin defines the radius of the helix at its base. The number of selected entities is displayed.

Faces: Select entities ②. The number of selected entities is displayed.



Select the direction and the origin of the rotation axis ⑥.

More options

The **Height** describes the total height of the helix ③. The **Pitch** ④ and **Taper** ⑤ are used to further define the shape of the helix. The direction of rotation is reversed using **Clockwise**.

Variable pitch: Optionally, change the pitch continuously from the beginning to the end of the helix. Enter a value for the **Final pitch** ⑦.

Design electrodes

Modify eroding path



Make eroding path visible and modify.

Electrodes → Modify eroding path

EDMconNG

2022.1

Collision check

Spark gap

Enter a negative value as a negative, user-defined offset for the electrode. When the electrode is forwarded to the simulation center, the electrode is recalculated. Without this step, the simulation of the electrode path would indicate a collision as soon as the electrode comes into contact with the workpiece because the electrode is generated with the same faces as the workpiece without erosion offset. Without the conversion, it would otherwise not be possible to identify actual collisions.

To convert the electrode on the basis of a mesh, the accuracy of the mesh can be influenced with the help of two Windows environment variables. To make the environment variables take effect, restart the software.



Do not select an edge length that is too small. Reduce the tolerance for small fillets. If the SIMULATION Center takes too long, the values selected are too small or the mesh is too large to generate. If the offset cannot be generated, a warning message will be displayed. The electrode is then transferred to the simulation without an offset.

- HC_ELECTRODE_MESH_TESS_TOL: The tessellation tolerance controls the accuracy of the permissible arc approximation. The preset value is 0.005.
- HC_ELECTRODE_MESH_MAX_EDGE_L: The maximum edge length determines the maximum edge length of the resulting triangles. The default value is 1.

CAM programming

Overview

Analyze toolpath



Analyze toolpaths.

CAM → Analyze toolpath

Visibility

Tool contact vector: Display the face normal vectors at the contact point of the tool with the surface for 3D radius compensation. The **3D radius compensation** option must be selected in the job under **Settings → Tolerance parameters**. The ballmill tool type is supported.

Color

Spindle speed: Display the speed values of the toolpaths in different colors ①. Select the settings in the **Color map** tab. The cutting speed is displayed during turning operations.

Limit toolpath

View segments: Display segments of a toolpath (e.g., 3D optimized roughing, additive machining) in order to analyze them individually. If no segments are available, the option is grayed out. The segments are numbered consecutively. To select a segment, enter a value in the "Spinner" UI entity next to the option, or select it by clicking the up or down arrow. Click to increase or decrease the value. Only one toolpath may be displayed. The slider setting only affects the individual segment. When deactivated, everything is displayed again. The segment of a selected toolpath point is displayed in the analysis result under **Segment information**.

Optimal barrel cutter



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

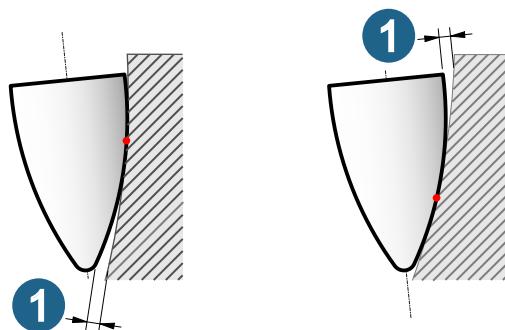
CAM → Optimal barrel cutter

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

2024

Options

Barrel - max. distance: ① Enter a maximum permissible distance between the barrel contour and the surface to be machined. This distance can be either above or below the contact point. Any allowance in between is not taken into account; the tool is always calculated so that it remains in contact with the geometry.



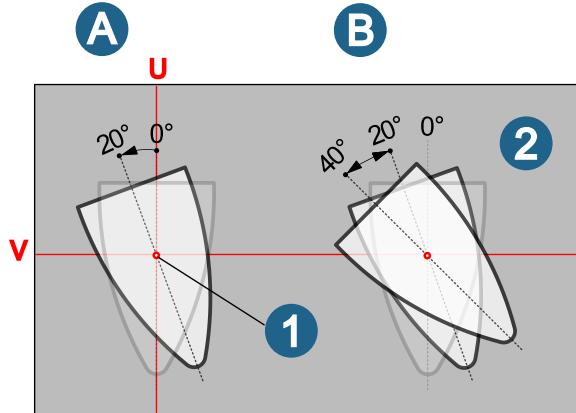
Lead angle

Enter a lead angle as the tool orientation along the selected curve or transfer it from the job settings (**5 Axes** → **Tilt parameters** → **Lead angle**) to optimize the calculation of the barrel radius.

The lead angle value is taken from the tool settings of a job, can be changed and is transferred back. In case of an Associative job copy, the associativity is lost if the value is changed.

- **Fixed lead angle:** (A): Enter a value for the lead angle that must be maintained when calculating the optimum barrel shape. With a value of 0°, the alignment of the tool axis corresponds to the U or V direction of the selected surface.
- **Min. lead angle and Max. lead angle:** (B): Enter a max. and min. value for the lead angle, which must be considered when calculating the optimum barrel shape.

① Contact point of the tool with the selected surface, ② Selected surface.



Result

Optimal Lead Angle: The optimal tool lead angle is output.

Create topology information

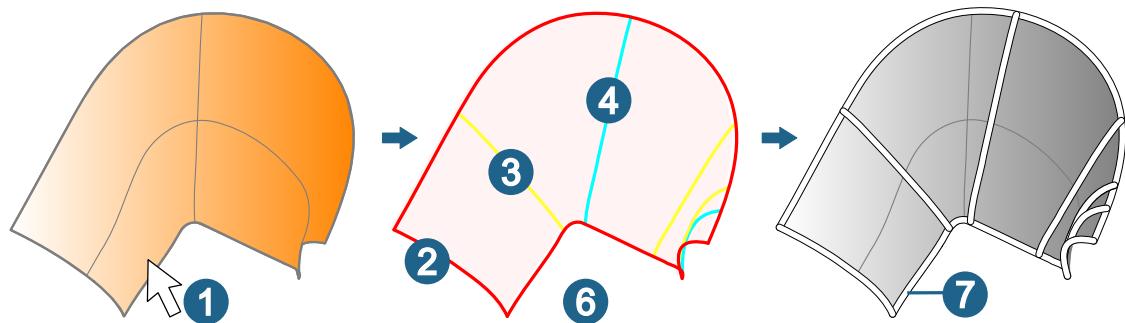


Generate topology curves for 3D finishing.

hyperMILL®-browser → **CAM Plan** → **High precision milling** → **Create topology information**

More options

Internal continuities: Visualize continuities within NURBS surfaces ⑥. Both internal C0 and C1 continuities can thus be identified directly. Milling operations can be better prepared and validated.



Adaptive construction

Prepare turning operations

Create turning V-sketch



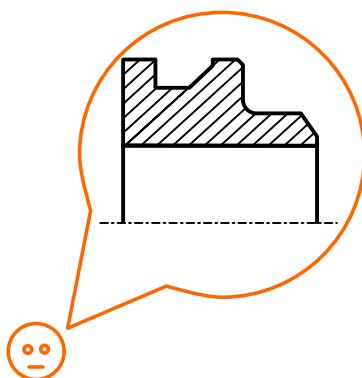
Create or change V-sketch for turning.

Drafting → Turning → Create turning V-sketch

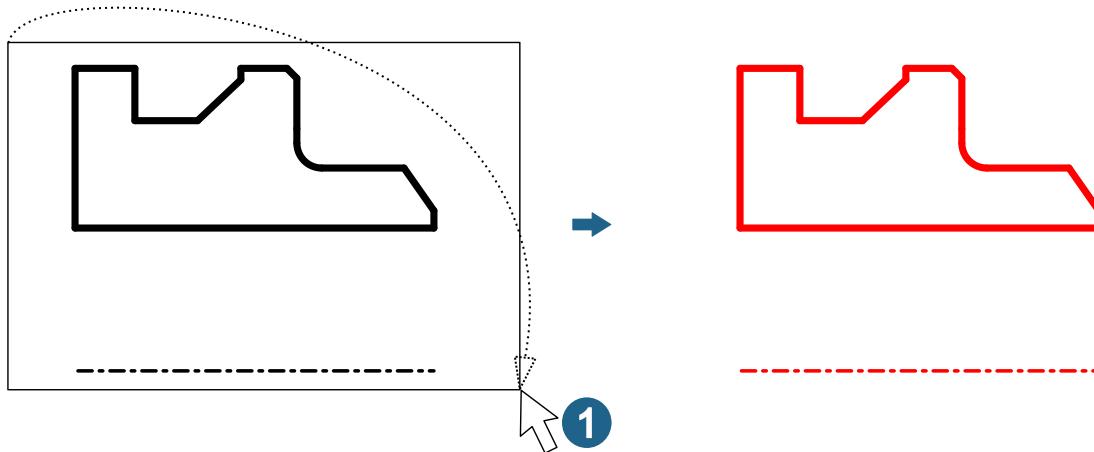
2025

Create or change a V-sketch for turning, including the rotation axis. Constraints (fixed entity, horizontal, vertical, tangential, fixed angle) are added automatically. The dimensional constraints for the diameters of all horizontal lines, the radii, and the chamfers are generated.

Select



Entities: Select all curves of the turning contour ①. Include a separate line for the rotation axis in the selection if it is a turning contour for a tubular workpiece.

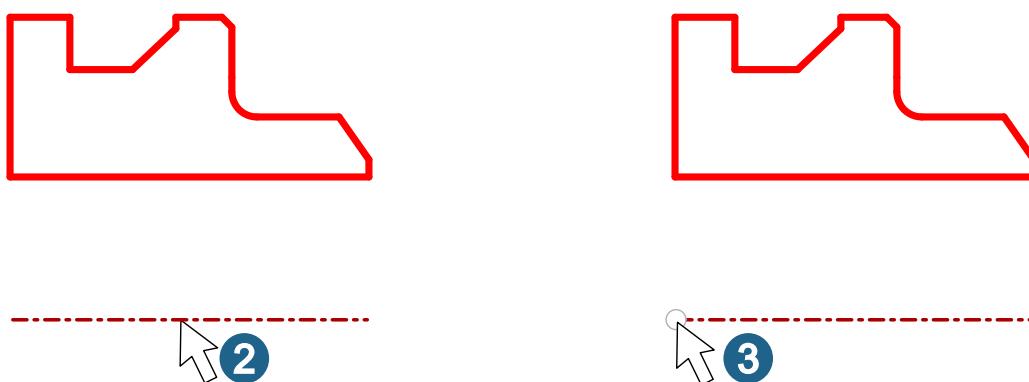


Rotation axis

Select the rotation axis, its orientation, and the origin of the V-sketch (fixed entity constraint). The rotation axis can be part of the turning contour (for a workpiece without a central through-hole) or an additional line next to the turning contour. The length should correspond to the width of the turning contour. The origin should be on the rotation axis:

Line: Select line ②.

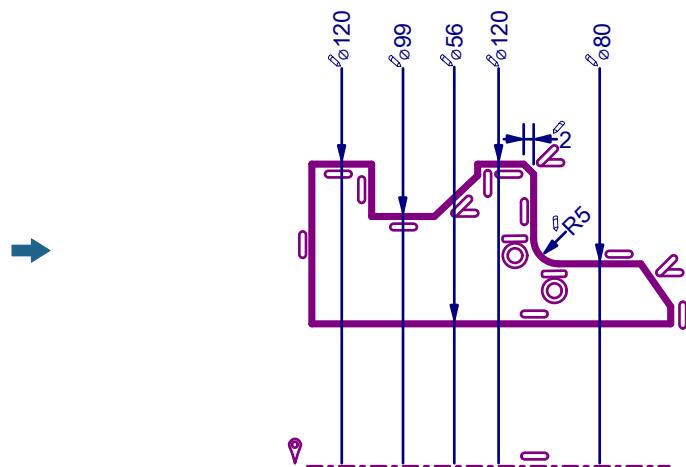
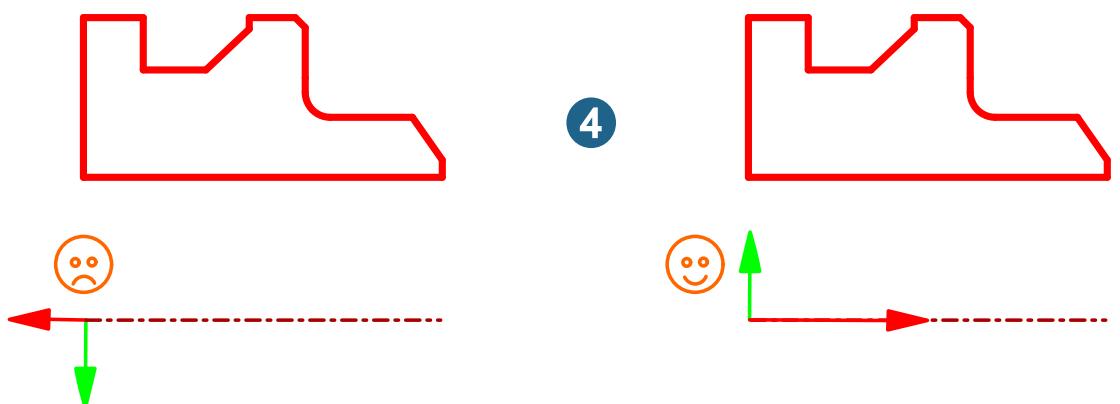
Origin: Snap the start, middle, or end point of the line ③.



The orientation of the rotation axis and the X axis of the temporary coordinate system must match. The positive Y axis must point in the direction of the workpiece contour ④:

Invert (keep Z): The direction of the X axis is reversed. The direction of the Z axis remains unchanged.

Invert (keep Y): The direction of the X axis is reversed. The direction of the Y axis remains unchanged.



More options

Angular tolerance: Enter a value for the limit angle up to which deviation lines are regarded as horizontal or vertical. If lines are within this tolerance, the lines are automatically corrected when the V-sketch is created so that they are actually horizontal or vertical. This value is independent of the angle tolerance entered in **File → Options → Sketcher options** under **V-sketch options**!

Horizontal distance constraints



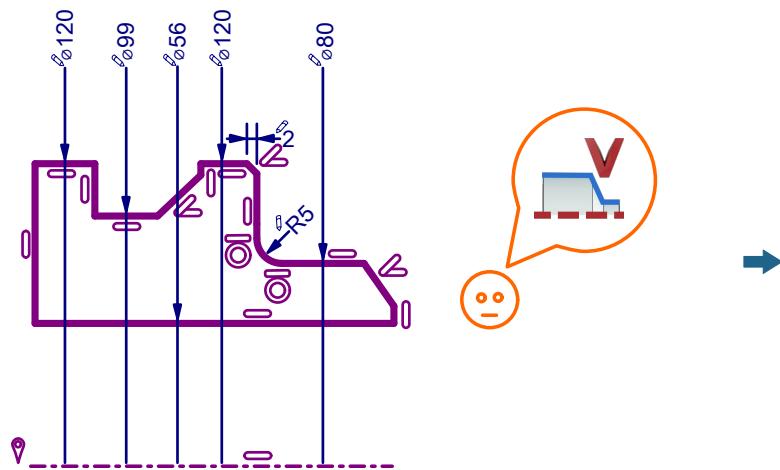
Define horizontal dimensional constraints in a V-sketch for a turning contour.



Drafting → Turning → Horizontal distance constraints

2025

Define horizontal dimensional constraints for a turning contour in a V-sketch. Snap points or positions. Entities of the V-sketch can be used, except for external references.



Select

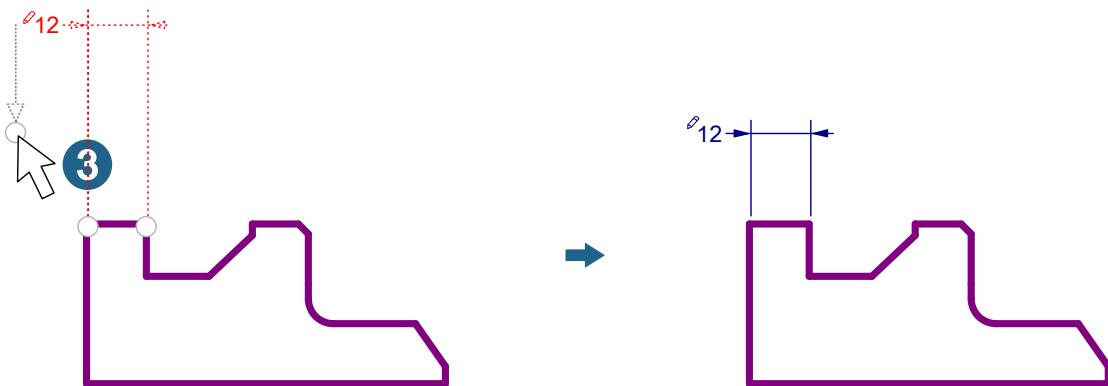
Origin: Snap an origin from which you want the dimensioning of one or more horizontal dimensional constraints (according to the selected mode) to start ①.

Target: Snap the target of the respective horizontal dimensional constraint ②.



More options

Measure position: Specify where a single dimension or the first of several dimensions is placed ③. Position the mouse pointer at the desired position in the graphics area and confirm by clicking the left mouse button.



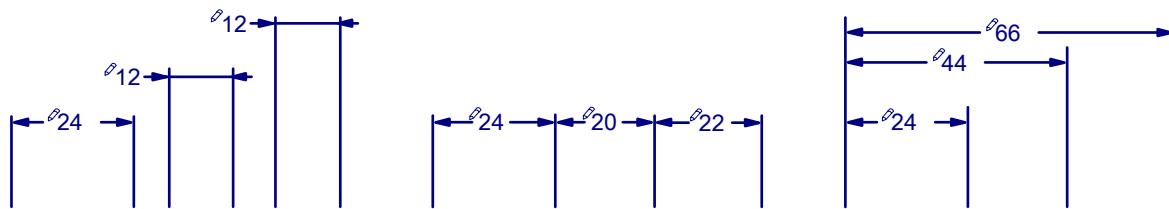
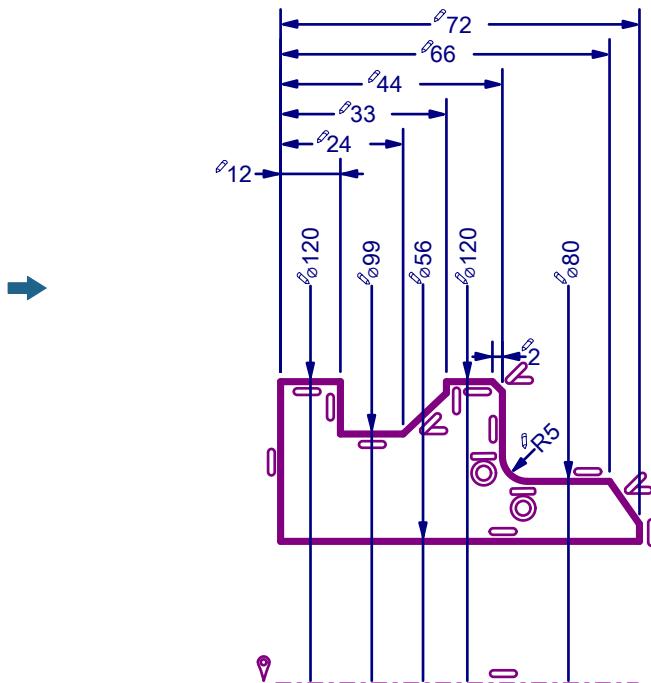
Modes

Select the dimensioning type:

Mono: Create horizontal dimensional constraints as a single dimension ④. Select the origin, target, and measure position individually.

Series: Generate several horizontal dimensional constraints as a chain dimension ⑤. The target of the previous dimension is automatically selected as the origin for the next dimension.

Parallel: Create several horizontal dimensional constraints as zero dimensions. All dimensions start from the same origin and are parallel to each other ④.

**A****B****C**

Modify linear tolerance



Adapt the geometry and display of linear tolerance intervals for turning operations.



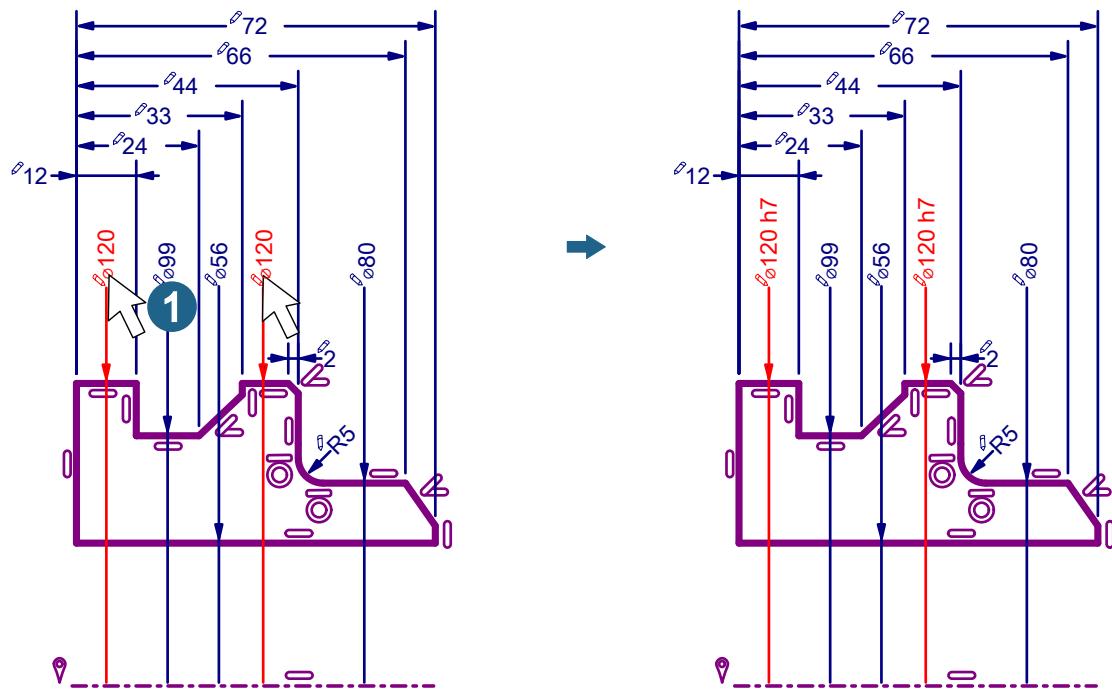
Drafting → Turning → Modify linear tolerance

2025

CAM programmer for turning generally receive models in nominal dimensions. However, a geometry adapted to the tolerance interval is required for machining.

Dimension

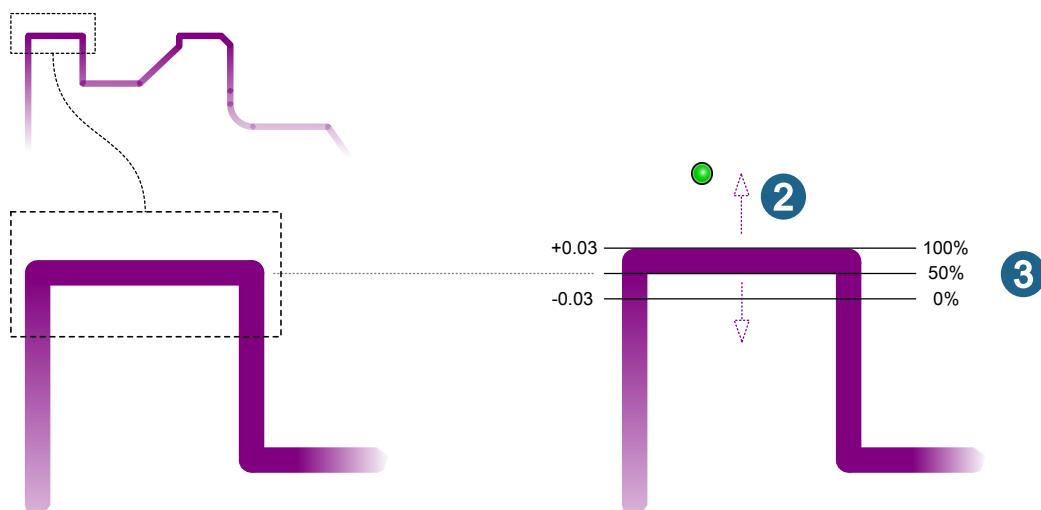
Select: Select one or more linear dimensions of a V-sketch for turning ①. The number of selected entities is displayed.



Apply tolerance: The V-sketch geometry is adjusted to the value specified in **Factor (%)** within the tolerance interval ②.

Factor (%): Adjust the V-sketch geometry to the desired position within the tolerance interval ③. Instead of using the mean value between the two tolerance deviations, any value between the upper and lower deviation can be entered as a percentage. Factor 0% corresponds to the lower deviation, factor 50% represents the middle of the tolerance interval, and factor 100% adjusts the V-sketch geometry to the upper deviation.

The  icon marks the dimensions of the geometry that have been changed due to the tolerance applied.



Representation

Specify the tolerance interval and select whether or how the deviations are to be displayed on the dimension:

None: The dimension does not contain a tolerance specification ④.



ISO: Add an ISO tolerance specification **(B)**. Enter an ISO tolerance specification, e.g. H7. Use the **Deviations** option to select whether the upper and lower deviations defined by the ISO tolerance specification should also be added **(C)**.

+/-: Add a tolerance specification where the lower and upper deviation are the same **(D)**. Enter a value.

Deviations: Add a tolerance specification where the lower and upper deviation are different **(E)**. Enter a value for the upper deviation and a value for the lower deviation.



Attributes

Select the desired values for the properties **Accuracy**, **Leading zero**, and **Trailing zeros**.

Analyzing probing results

Read and analyze one or more probing logs for 2D and 3D probing jobs.

CNC processing machines often provide the option of probing during and after processing. The data is normally output as text. There is no graphical link to the workpiece. If several attributes and possibly also several workpieces have been probed, it is very difficult to match the results to the correct workpieces in the text file, thus making it difficult to draw correct conclusions. It is a complex task to determine a production trend (e.g., regarding tool wear or the effect of heat-related deformations). There is also the additional difficulty that older CNC controls do not provide any functionality for logging a probing process.

As a solution, *hyperMILL*® offers to create a mask file in addition to a probing NC file, if this is necessary for the control type. During probing at the machine, a probing log in **OpenMindProbingEXchangeFileformat** is generated as a text file named ***.log**. This format was designed so that 2D and 3D probing results can be imported to *hyperMILL*® and displayed graphically. A probing log is generated for each probing process. This file with the generic name **OMPREXF.log**, is located in the same folder as the NC file. It is deleted and generated anew when a new probing process is started. This means that the probing log always contains the result of the last probing process. The file is also deleted and newly generated when the process is aborted and restarted. Probing processes can be performed in any order as required.

Supported CNC controls

The following version is the minimum requirement for the supported control systems:

CNC control	Machine manufacturer	Minimum version
Heidenhain iTNC530	-	NC 340 490-06
Heidenhain TNC640	-	NC 340 590-07
Siemens Sinumerik 840D	-	4.5
Fanuc	Mazak	Renishaw F-4013-0114-AN
	Fanuc	Renishaw F-4012-0519-AW
	OKUMA	Renishaw F-4016-1037-AF



Probing options



Options for reading and displaying probing results.

File → Options → Probing options

2024

Options

Caption: Select whether to display the actual probing results as **Absolute values** or if you want to show deviations, **Delta values**.

Trend visualization mode: If **By run** is selected, runs are compared directly with each other. If **Dimensioning** is selected, the same attributes are compared directly next to each other.

Line thickness: Select the width for displaying connecting lines.

Dimension label offset: Enter the distance between the dimension line and the dimension.

PYTHON Toolkit

Python script support

For complex requirements, an externally generated Python script can be executed in the Python programming language.

PYTHON Toolkit

Python tab

2024 | Update 2

Python installation

Installed Python version and Python modules

Module	Version
Python	3.11.6
Tkinter	8.6

Python tab

Workspaces

Active workspaces are displayed as tabs. If several workspaces are available, select a workspace. Define the root folders for the workspaces in the **Python options**. Not all commands are available in all workspaces.

Scripts: Edit and manage the scripts for the active projects in your own user workspace.

Library: The library is a collection of ready-made Python blocks that are intended to provide specific commands. Instead of creating these commands again, reuse these blocks in a Python project using the `import` command. Edit and manage the scripts for these blocks in the library workspace.

Upper section – Python script list

The context menu contains the following commands for a workspace or folder:

Create library: Create a new block of the Python script library (in the **Library** workspace).

Import: Import a script. See [Import \[61\]](#).

Export: Export a script as a `*.zip` archive file. Export script. See [Export \[61\]](#)

Use script as a command

Use a script like a command in the software. Select an option in the context-sensitive menu of a script:



- **Enable command:** Enable the use of the script as a command in the software.
- **Edit command:** Change the use of the script.
- **Edit command:** Switch off the use of the script as a command in the software. First, remove the command from the toolbar.

Command from script



Administrate a command based on a script.

In the **Python** tab in the context menu of a script via **Enable command** and **Edit command**.

PYTHONToolkit

2025

Name: The name is derived from the name of the script.

Icon: Provide and select an icon in *.png format. The recommended size for the icon is X px x X px.

Where does the command appear? In **File** → **Options** → **Toolbars and tabs** in the **Commands** tab, search for the **Python** entry. The scripts that can be dragged into toolbars as commands are listed there.

Import



Import a script.

In the **Python** tab in the context menu of a script folder.

PYTHONToolkit

2025

Script to import: Select a *.zip script archive.

Import library: Import a script archive as a Python library. Select a *.zip script archive.

Export



Export a script as a *.zip archive file.

In the **Python** tab in the context menu of a script.

PYTHONToolkit

2025

Folder where to export: Select a folder.

Python options



Select Python options.

File → **Options** → **Python options**

PYTHONToolkit

2024 | Update 2

Library workspace folder: Select a folder to use as the root folder for the user workspace, for example, in C:\Users\Public\Documents\OPEN MIND\pythonlibrary (the default setting).

Lazy graphic update: The graphics area and the model tree are not rebuilt after every action.



3. Release notes

Release 2025 | Update 1

CAM

Updating highly recommended

- **3D Optimized Roughing**

An unrecognized part violation with tools used by free geometry was fixed.

CAD

Additions and notes

If the connection to the license server is lost, the software no longer terminates. Instead, a warning is issued and the system automatically attempts to establish a new connection. If successful, the file is released again. This means that no unsaved processing progress is lost in the event of network issues. Once the network connection has been re-established, work can continue seamlessly.

CAD Viewer: The following functions are also available: **Break**, **Scale**, **Mirror**, **Stretch 2D**, **Curvature plot**, **Check double entities** and **Check quality / Repair**.

CAD interfaces

- The Open Design Alliance ODA library software library has been updated for the DWG, DXF, and PDF interfaces.
- For importing under **Repair** the option **Remove faces with width smaller / equal (mm / inch)** has been added to skip shard surfaces during import.
- For importing under **Entity conversion modes** the option **Merge meshes** has been added. Selecting **No** means no merging will occur; selecting **Merge to one** activates merging and creates an overall mesh; selecting **Merge by color** merges meshes according to color. Multiple colors result in multiple meshes.
- **STEP**: The export has been revised and accelerated. Three export options have been added. See [STEP file format \[40\]](#).
 - **Assign curve and face names**: Element names for curves and faces are imported and assigned.
- **IGES**: The export has been revised. Three export options have been added. See [IGES file format \[37\]](#)
- The export of formats **JT-Open**, **ACIS**, **Parasolid**, and **PRC** has been added. A layer description cannot be exported.
- Support for the ***.ply2** polygon file format for importing and exporting meshes has been added. See [PLY2 file format \[39\]](#).

User interface

- In the **Visibility - layer** tab, switch entities of a layer on / off for selection. This allows excluded entities from selection operations. In the **Model** tab, affected entities are marked - the icon is displayed in gray. Additionally, an option has been added in **Options / Properties**. When importing CAD data, there is the option to set all layers as non-selectable.
- **Layer tab**. In contrast to the new feature in version 2024, there is now a separate **New layer and set current** command for creating and immediately setting to current.

Options / properties

- **Graphic → System → Navigation**: The navigation behavior when rotating the view has been revised. The options have been restructured. The **Enhanced ray cast** mode has been added for finding the center of rotation. The term “rotation target” has been renamed “rotation center”. This also applies



to the relevant functions in the **View** menu. See also the revision in the software documentation in **Graphic > System > Navigation** [36].

- **Model → Attributes:** The option **New layer is always current** has been added. This eliminates a lot of work when dealing with numerous layers, avoiding the need to find the new layer, scroll to it, and activate it separately.
- **Modell → Entities:** The tessellation tolerance settings for curves, faces, and solids are combined into the single value **Tessellation tolerance**.
- **Modell → Entities → Solid:** The **Tessellate connected** option has been added. See **Model > Entities** [35].
- **Application:** The **Enable non-selectable layers in visibility tab** option has been added. See **Application** [35].
- **Application → User Interface Style:** Added option to select between different styles. See **Application** [35].
- In selection with the options Mesh edge selection and Mesh face selection activate the selection of the edges and faces of a mesh for the command entity properties and Info two entities commands.

Edit

- **Graphic properties:** The **Tessellate connected** option has been added. See **Graphic properties** [45].

Select

- **Chain:** The **Only closed** option has been added. See **Chain** [42].
 - The snapping function has been added within a mesh triangle structure. This allows measuring between the workpiece geometry and raw parts, within machining states, or STLS. If the cursor moves onto a mesh triangle, the corresponding face is used; if the cursor is near an edge of a face, that corresponding edge is used. If the snap modes Stock model point and Mesh vertex point are selected, they are prioritized.
- Mesh edge on / off** and **Mesh face on / off**: Enable or disable snapping within the triangular mesh structure. See **Mesh edge on / off** [42] and **Mesh face on / off** [43].

Curves

- **Shape contour:** The contour curve generated by the rotated faces is linked to its generating faces to assign PMI information. The color of the generating faces is adopted. The curves receive a TAG named **T_SHAPE_REFERENCES_SCSV** with the value of the entity ID of the generating face. If a contour curve is generated by more than one face, the entity is defined as a “new entity” and receives the preset color since individual parent entities cannot be assigned. See **Shape contour** [46].

Shapes

- **Linear sweep:** This concerns handling of penetration at rounded corners with inclined side faces. An edge is created from the penetration to replace the fillet. See **Linear sweep** [46].
- **Fillet:** Create a fillet with tangential transitions between an initial side face and a central face as well as another side face. See **Fillet** [47].
- **Helix:** Optionally create the helix as a solid. See **Helix** [47].

V-sketches and parametric design

- **Geometric constraints:** The **Select series** option makes it possible to apply the conditions Same radius / Coincidence radius, Same length / Coincidence length, Parallel, Coincidence XY, Coincidence X, Coincidence Y, Coincidence radius, Coincidence length multiple times in succession without interruption. The constraint assigned in series can then be changed simultaneously using a single dimension. This is useful if several entities within a V-sketch are to be assigned the same length and radius. The constraint **Fixed angle** has been added. See **Geometric constraints** [48].
- **Automatic constraints:** The maximum diagonal length of the chamfers can be entered in the default settings. See **Sketcher options** [36].



The automatic dimensioning for chamfers and the constraint **Fixed angle** have been added. See [Automatic constraints \[48\]](#).

Features

- **Helical protrusion:** Create a protrusion via the helical sweep of a contour from curves and boundaries on a face within a solid. See [Helical protrusion \[49\]](#).
- **Helical slot:** Create a slot via the helical sweep of a contour from curves and boundaries, based on a face within a solid. See [Helical slot \[49\]](#).

Booleans

- A progress bar has been added to provide feedback when performing complex calculations.

Modify

- **Extend / shorten faces:** The **Keep domain** option has been added. If an area is shortened or extended, the untrimmed initial face is reduced to the altered size of the trimmed face. The loss of the area negatively impacts milling, as the faces in many cases need to be larger or extended as C2 for the milling area. Alternatively, it is possible to prevent the entire untrimmed output face from being created several times when this is not needed for the desired action.
- **Split faces:** The **Keep domain** option has been added. If a face is divided, the untrimmed initial face is reduced to the trimming area. The loss of area negatively impacts milling, as faces in many cases need to be larger or need to be extended as C2 for the milling area. Alternatively, it is possible to prevent the entire untrimmed output face from being created multiple times when it is not needed for the desired action.

View

- “Rotation target” has been renamed to **Rotation center**.

Workplane

- **Mirror aligned:** Create a workplane for NC programming of a geometry with a near mirror-image shape. See [Mirror aligned \[43\]](#).

Analysis

- **Shape continuities:** The **Internal continuities** option for visualizing continuities in NURBS faces has been added.
- **Entity properties:** For meshes, determine the properties for facets and their edges.
- **Two entities info:** A function for determining the angle between two stock facets (STL), the angle between a stock facet (STL) and a CAD face (analytical), the distance between two stock facets (STL) and the distance between a stock facet (STL) and a CAD face (analytical) has been added.

Electrodes

- The variable `%documentname%` has been added to directly use the document name without file extension for **Electrode name** and **Part number**.
- **Electrode options:** In **Technology** the processing steps have been renamed. They now clearly indicate the respective stage of the process. This allows both the user and the EDM converter to select the correct method for each electrode. This is essential as the method choice is directly transferred to the EDM converter. The first electrode is the roughing electrode, the second electrode is the pre-finishing electrode, the third electrode is the finishing electrode, and the fourth electrode is the electrode for fine finishing.

A different electrode material can be selected for each erosion machining step. For machines that do not support this functionality, only the first material is communicated in the EDM export.



To round: The value for rounding the target position of an electrode has been increased to 10.

- When mirroring electrodes, it is automatically considered that the electrode holder must not be used in a mirrored manner.
- **Simulation with spark gap:** When the electrode is forwarded to the simulation center, the value is calculated as a negative user-defined offset for the electrode. See [Modify eroding path \[50\]](#).
- The EDM export for Zimmer & Kreim eroding machines supports the output of the technology (R=Rough, PF=Prefinish, F=Finisch, P=Polish). A counter has been added. However, only the work-piece material can be output, as only a single material is supported.

CAM

- V-sketch commands for turning have been added:

- **Drafting → Turning → Create / modify V-sketch (turning):** Create or change a V-sketch for turning. See [Create turning V-sketch \[53\]](#).
- **Drafting → Turning → Horizontal distance constraints:** Define horizontal distance constraints in a V-sketch for a turning contour. See [Horizontal distance constraints \[55\]](#).
- **Drafting → Turning → Modify linear tolerance:** Adapt the geometry and display of linear tolerance intervals for turning operations. See [Modify linear tolerance \[57\]](#).
- Optionally, a warning can be issued to indicate which entities have been altered by changing the constraint or applying tolerances. See [Sketcher options \[36\]](#).
- In the **Shape continuities** and **Create topology information** commands, "Angular tolerance (degrees)" is now called **Angular tolerance**; in the **Create edge geometry** command, "Max. deburring angle (degrees)" is now called **Max. deburring angle**.
- **Analyze toolpath:** The usability has been improved, and the display of the approach vectors or face normal vectors in the tool contact point has been added. See [Analyze toolpath \[51\]](#).
- **Optimal barrel cutter:** Under Options, set a value for **Barrel max. distance** and search for an optimum lead angle. See [Optimal barrel cutter \[31\]](#). If the **Global mode** option is selected, the **Auto. computation** option is automatically switched off if it is activated. To calculate, this option must be switched on again! This allows full control even for more complex calculations.CFC-6037

Probing

- Now probing results from 2D probing cycles can also be read for analysis. This upgrade is a seamless complement to the analysis of 3D probing results. See [Analyzing probing results \[59\]](#).

Bug fixes

The following issues have been resolved:

Updating recommended

- **Drafting → Intersection points:** In certain constellations, it is not possible to create intersections between plane faces and a curve. This matter has been resolved.

Release 2025 | Update 2

CAM

Additions and notes

3D Z Level Shape Finishing

- The toolpaths have been improved so that the tool does not touch the wall and floor at the same time when machining cavities. This ensures that no (machining) marks are left on the part.



Bug fixes

Updating highly recommended

hyperMILL® VIRTUAL Machining / Sinumerik ONE / newer Sinumerik 840D

- When using CYCLE832 in combination with active VMLink, a warning message may appear on the machine due to the output sequence in the NC program: "Transformation not active: TRAORI/ CYCLE800 before CYCLE832" or a similar message. In order to take the new Siemens specification into account and avoid this message, please get in touch with your contact person so that the required setting can be activated for you.

hyperMILL® VIRTUAL Machining / Siemens control

- The VM configuration parameter "S_TSA" is now also considered in the toolpath representation within the simulation when CYCLE998 "Align Edge" is used.

hyperMILL® VIRTUAL Machining / Turning

- VIRTUAL Machining now also supports **Stop before execution** with turning cycles.

hyperMILL® VIRTUAL Machining / Simulation

- An incorrect simulation when using the option **Allow multiple origins** and Axial cycle 800 output has been fixed.

hyperMILL® VIRTUAL Machining / Simulation

- A Problem has been solved which leads to a "Tool id is not unique" message.

hyperMILL® VIRTUAL Machining/ Simulation

- Fixed an issue where tool and spindle collided with the part during simulation.

hyperMILL® VIRTUAL Machining / Simulation

- An issue has been resolved that caused the virtual machine to crash when collision control and simulation were executed simultaneously.

hyperMILL® VIRTUAL Machining / Simulation

- A problem has been fixed that led to an incorrect machining position when using a rotated clamping position.

hyperMILL® VIRTUAL Machining / Simulation

- An issue has been fixed where the stock was displayed incorrectly in simulation when selecting a solution in Turning, specifically if two consecutive jobs used the same tool but different solution selections.

hyperMILL® VIRTUAL Machining / Simulation

- An issue has been fixed where incorrect rotation caused collisions on a Grob G352T VM machine.

hyperMILL® VIRTUAL Machining / Post processing



- In order to avoid incorrect calculation of the plane function, COORD RED instead of TABLE RED is now output for Grob machines with Heidenhain control up to version TNC640 340590 07 in the case of A0 B0.

hyperMILL® VIRTUAL Machining / Collision check

- An issue has been fixed where a collision was incorrectly detected between the cutting insert and the model, which was caused by a coarse tessellation of the model.

hyperMILL® VIRTUAL Machining / Collision check

- An issue has been fixed where the VM could crash during collision checks if the check was started before the program was fully loaded.

hyperMILL® VIRTUAL Machining / Collision check

- An incorrect collision message when using a rotated clamping position has been fixed.

hyperMILL® VIRTUAL Machining / Siemens control

- An issue has been fixed where the tool number in the CYCLE979 command during measuring and correction was not enclosed in quotation marks, resulting in a syntax error on the machine.

hyperMILL® VIRTUAL Machining / OKUMA

- For OKUMA NC output a parameter naming issue was fixed for tilted workplane cycle OO88.

hyperMILL® VIRTUAL Machining / NC Generator

- An issue was resolved where incorrect NC output statements for GPR occurred during axis-dependent measuring in TCH PROBE 427, caused by an error in the calculation formula.

hyperMILL® VIRTUAL Machining

- With 2.5° indexed head machines, the pole situation is now recognized correctly. This enables the use of the preferred C-axis position and optimization by the Optimizer.

hyperMILL® VIRTUAL Machining

- With horizontal MillTurn machines, the stock was rotated incorrectly. This error is fixed.

SIMULATION Center

- An issue has been fixed where the SIMULATION Center used the probe tip instead of the probe center as the tool reference.

SIMULATION Center

- An issue has been fixed where a milling area of a SolidWorks model is not displayed in the Simulation Center.

Tool database

- An issue was resolved where XML imports failed if a NC-Tool ID already existed.

Tool database / Hummingbird connection



- An error with the database connection to Hummingbird was fixed so that opening parts and displaying job lists and tools now works correctly again.

Drilling / Tap

- Resolved an issue where collisions with the shank of tap tools were not properly detected during the calculation, but only appeared in the simulation.

Drilling / Thread Milling

- An issue has been resolved in connection with variable pitch, ensuring that the entered pitch values are now automatically checked against the tool's defined range. If a pitch value falls outside the tool's specifications, a warning will be generated, preventing potential errors.

Turning / Grooving

- An issue was fixed where a collision was incorrectly reported during calculation when using Coscom tools. The reason was a misalignment in the tool holder contour positioning, which has now been corrected by updating the machine coordinate system.

2D Contour Milling on 3D Model

- An issue was resolved where incorrect feedrates were applied when the **Fit step** option was enabled, which could lead to tool breakage.

2D Contour Milling on 3D Model

- An issue has been fixed where collisions could occur during 2D Contour Milling on 3D models when the "Break edge" option was enabled, without any error message being shown.

2D Contour Milling on 3D Model

- An issue has been fixed where incorrect feedrate was applied during vertical plunge when "Fit steps" was active, which could cause tool breakage.

3D Optimized Roughing

- A calculation error is resolved, so that affected jobs can now be calculated correctly again.

3D Optimized Roughing

- An issue was resolved where the boundary was not being maintained and **High Performance Mode** was not activated as expected.

3D Optimized Roughing

- An issue was fixed where the tool was keeping an unnecessarily large distance from avoidance surfaces.

3D Optimized Roughing

- A bug caused the **Machining tolerance** to be incorrectly added to the **Allowance**. This issue has been fixed.

3D Optimized Roughing



- An issue has been fixed where too much rest material has been left after machining. The pocket cycle has been improved to ensure complete material removal.

3D Optimized Roughing

- An issue was fixed where calculation could fail in the latest software version, while they continued to work correctly in the previous version.

3D Optimized Roughing

- An issue has been resolved where incorrect calculations occurred on plan surfaces when a negative allowance matched the tool's corner radius.

3D Optimized Roughing

- An issue has been fixed where the NC file output an incorrect feedrate value of F-1. Additionally, resetting the job previously triggered an error message and changed the plunge feedrate, which has now been corrected.

3D Optimized Roughing

- An issue has been resolved where calculation failed with an internal computation error caused by closely positioned points in the input data.

3D Optimized Roughing / 3D Arbitrary Stock Roughing

- An issue has been fixed where a collision could occur when using the **Use free tip geometry for calculation** option.

3D Profile Finishing

- An issue has been fixed where the allowance value in CFG parameters did not update correctly when a job with "3D Profile Finishing" was copied and replaced.

3D Complete Finishing

- An issue was resolved where tool collisions could occur during simulation, specifically when inclination-dependent processing was turned off.

3D Plane Machining

- An issue was fixed where selected planes were not processed correctly when using "inch" mode.

3D Plane Machining

- An issue has been fixed with an incorrectly reported collision.

3D Equidistant Finishing

- An issue has been fixed where tool paths were incomplete when processing multiple areas in certain jobs.

3D Iso Machining

- An issue has been fixed where an unnecessary toolpath was generated in the 2025 version, which did not occur in the 2024 version. The toolpath generation now matches the expected behavior from previous versions.



5X Profile Finishing

- A bug was fixed where applying a 45° angle in the **5X Profile Finishing** cycle using the **Fixed** mode also unintentionally affected the C angle.

5X Rework

- An issue that caused an occurrence of F0 (Feedrate=0) in the NC file, resulting in a machine stop, has been fixed.

5X Tube Roughing

- An issue was resolved in connection with the function **Away from central curve**, which led to the toolpath calculation being aborted. The fix also includes a correction of the retract movement.

Stock calculation / Mirrored job list

- An issue was fixed where using stock models in the mirrored job list could incorrectly cause a collision by referencing the wrong stock. As a correction, the **Continue stock chain** function has also been set to **read-only**. The function is now automatically deactivated if it was previously activated.

Automatic stock chain

- An issue has been fixed where for some cycles in the joblist no resulting stock was created. If necessary the software now automatically closes a stock and generates the resulting stock for all jobs.

General / Consistency check

- An issue was resolved where .hmc files could not be opened due to Windows file permission errors. The software now automatically adjusts file permissions to ensure consistent access.

General / Coolant setting

- An issue has been fixed where the coolant setting was not retained in manual profile mode for tool series, due to incorrect handling of non-linkable values. The consistency check has been adjusted to ensure that coolant settings are now properly retained.

General / Job definition

- An issue has been fixed where replacing a job within a compound job incorrectly generated a new job ID. The job ID now remains unchanged when a job is replaced.

Inventor integration

- An issue has been fixed where system reactors not being properly removed, leading to crashes on exit and potential memory leaks.

Toolpath Edit

- An issue has been fixed where enabling G2/G3 output in **3D Optimized Roughing** after using the **Edit toolpath** function caused unexpected behavior and tool gouging.

Edit → Move / Copy

- An issue has been fixed where *hyperMILL®* could crash when using the **Edit → Move / Copy** function.

General



- An issue has been fixed where displaying toolpaths in *hyperMILL®* caused excessive memory usage, leading to performance problems.

Macro technology

- An issue has been fixed where macros were not automatically selected when changing material groups in the macro database, resulting in a red "X" indicator.

General

- An issue has been fixed where incorrect collision detection with the vice could occur during machining, depending on toolpath and position.

Updating recommended

SIMULATION Center

- An issue has been fixed where a "Kinematic deviation" error message appeared when no machine model was in use.

2D Contour Milling

- An issue has been fixed where the **Front / Back chamfered cutter** tool type caused a collision with the clamping device. The calculation now processes correctly for this tool type.

3D Optimized Roughing

- An issue was resolved when using the **High Performance mode** where a cut tolerance of 0.005 could cause tool breakage due to a missing toolpath segment. The problem has been fixed by refining the toolpath calculation.

3D Plane Machining

- A calculation error when using automatic infeed direction and axis parallel mode has been fixed.

5X Deburring

- Fixed an issue with a calculation error that occurred when using a tool with a collar and the setting for the **Max. stepdown** parameter was smaller than the **Chamfer distance**.

5X Rework Machining

- An issue has been fixed where the *HOLDCHECK_MODE setting changed from 3 to 0 after reopening a project, causing unnecessary recalculations.

Tool Database / Connected Tool Technology

- Fixed an issue where the XML mapping was corrected to ensure proper parameter assignment in the cycles **5X Swarf cutting with one Curve** und **5X Swarf cutting with two Curves**.

General / Stock chain

- An issue was resolved affecting an error message when using a stock chain in the **2D Contour Milling on 3D Model** and **Playback Milling** cycle and with activated stock.

Analysis / Toolpath properties



- An issue was fixed where toolpath vectors were not displayed unless specifically activated in the job settings. This fix also addresses a related crash that could occur when clicking on the tool contact vector.

Hummingbird integration

- Fixed an issue where a programming element was missing for the **Optimized deep Hole Drilling (new)** cycle in Hummingbird.

General / Edit tool dialog

- An issue has been fixed where the **Show used only** field was not permanently displayed in the **Edit tool** dialog due to a property editability check error.

General / Mirrored job list

- An issue has been fixed where mirrored job lists could not be deleted.

General / Global editing

- An issue has been fixed where it was previously not possible to globally edit the turning area for turning jobs.

CAD

Additions and notes

A separate note on a slight change in behavior since version 2025: Saving to the local network and creating backup copies now behave as they did in previous software versions.

The feature **Assign material** is now also available in CAD Viewer.

Opticam users can now download the latest version 2025.2 (https://www.camtek.de/assets/template/Medien/Dateien/Downloads/SETUP_OPTICAM_V2025_2_HCNT.zip).

File → Options → Electrode options: In anticipation of the 2026 release, a **Customized EDM converter** folder can be defined in which a user-defined electrode converter is searched for.

CAM → Optimal barrel cutter: The default value for the option **Barrel max. distance** is changed from 0 to 0.02 mm. To make this change take effect, the dialog must be reset to the default settings once. To do this, select the icon  in the dialog.

As early as Update 1, the behavior of the user interface in global mode was modified to enhance the user experience. Previously, every parameter change in activated global mode automatically resulted in a calculation, which could lead to unwanted waiting time. With the new behavior, however, you can now decide when to carry out a calculation.

If global mode is deactivated, everything remains the same: Each parameter change causes the calculation results to be updated directly.

However, if global mode is activated, the automatic calculation is initially deactivated. As a user, you must actively select the option **Auto. computation** to calculate the optimal barrel cutter.

The calculation has been carried out successfully, this option is automatically switched off again. After that, the buttons **OK** and **Apply** are available and the calculated values for the optimal barrel cutter are displayed. However, if no new calculation has been carried out after a parameter change, the option **Auto. computation** remains deactivated. The **OK** and **Apply** buttons are not available. The display for optimal values is set to zero.



- **Create CAM plan** and **Activate CAM plan** added as template commands for users with an Advanced license.
- **Analyze outer curve** added as a template command for users with an Advanced license.

Bug fixes

The following issues have been resolved:

Updating is highly recommended

- CAD interfaces

PTC Creo: The threads from a sketch are only partially recognized during feature recognition. The issue was resolved. The fact that the origin of the pattern axis is set incorrectly in the feature data was also resolved.

Updating recommended

- When a solid is moved to a layer whose elements are in the “not selectable” state, there is a problem displaying the solid. The issue was resolved.

- **File → Merge**: If special characters such as ö, ä, or ü appear in the pathname, a *.pt file cannot be opened. The issue was resolved.

- **Curve → Shape contour**: An incomplete rotational silhouette is created due to geometry that has not been fully modelled. The issue was resolved.

- The tooltip does not display the lower PMI tolerance value for H7 tolerances. The issue was resolved.

- CAD interfaces

The limit for **Starting layer for faces** has been increased.

SOLIDWORKS: ISO fit data is missing from the SolidWorks file. The issue was resolved.

STEP: Hidden solids are exported even though the **Save hidden entities** option is disabled. The issue was resolved.

IGES: Hidden solids are exported even though the **Save hidden entities** option is disabled. The issue was resolved.

- AUTOMATION Center

Milling area: A solid should be selected as the milling area. The type face is set. Twice the number of faces is inserted into the milling area. The issue was resolved.

- AUTOMATION Center Reports

When running a tool report, an error message appears if the Front/Back Chamfer Cutter tool type is used. The issue was resolved.

A tool name defined as value 9 in the tool report template is no longer output. The issue was resolved.

When outputting a report in PDF format, a dialog for saving the Excel file that is not brought to the foreground prevents the report from being displayed. The issue was resolved.

Release 2025 | Update 2.1

CAM

Updating highly recommended

- **3D Optimized Roughing**

An issue has been fixed where High Performance mode in Optimized Roughing could generate a full cut, which could lead to a possible tool breakage.



Release 2025 | Update 3

CAM

Additions and notes

- **Feature Technology / Customized Process Feature (CPF):** It is now possible to add images and comments to parameter groups in a CPF feature.
- **hyperMILL® VIRTUAL Machining/ Post processor configuration:** For machines with a fixed, parallel rotary axis, the new function **Movements → Optimizing → Fixed / Parallel axes → Rotary solution order** is now available. This function controls the sequence of solution finding for the fixed rotary axis using the two options **Standard** and **Optimized**.

The **Standard** option follows this order:

1. User-defined position.
2. Position of the previous job.
3. Search for a new position.

The **Optimized** function uses the following order:

1. Position of the previous job.
2. User-defined position.
3. Search for a new position.

- **Linking job / Optimized deep Hole Drilling:** The option to add an Optimized deep Hole Drilling job to a Linking job has been disabled because it could lead to undesirable behavior.
- **Tool database:** The tool type **Front/Back chamfered cutter** is supported.
- **TOOL Builder:** The import of 3D data for generating the tool type **Angle head** is supported.
- **Feature Technology:** A fine thread M25x1.5 has been added to the **omThreadCatalog**, along with corrections to the M11x1 core diameter and additional thread sizes.
- **5X Deburring / 5X Hole Deburring:** The following new functions are available: **Sort contours:** The contours are automatically sorted by the cycle. If the option is deactivated, sorting is performed in the order defined by the user (No. 1 → No. 2). In addition, machining is performed uniformly either in Climb milling or Conventional milling. If **Sort contours** is activated and **Path direction** is set to **Manual**, the order of the contours is determined by the cycle, but the direction defined by the user is retained. Additionally, functions for limiting the machining area are available in the **5X Deburring** cycle. The options **Manual bottom** and **Manual top** (Parameter dialog page) refer to the defined frame, and it is also possible to define a boundary.

Bug fixes

- **hyperMILL® VIRTUAL Machining / Siemens control:** An incorrect toolpath display for the circular pocket cycle has been fixed.
- **hyperMILL® VIRTUAL Machining / Simulation:** An incorrect movement after a tool change on 3X machines with Fanuc control has been fixed.
- **hyperMILL® VIRTUAL Machining / Simulation:** An issue has been fixed where unexpected Z-axis movement occurred in Fanuc machine simulations after a tool change due to a missing G49 command.



- **hyperMILL® VIRTUAL Machining / Simulation:** An issue causing incorrect turning paths with invalid arcs and collisions on the counter spindle during combined milling and turning jobs has been fixed.
- **hyperMILL® VIRTUAL Machining / Simulation:** An issue causing software crashes when starting collision check and material simulation at the same time, without generating a crash log, has been fixed.
- **hyperMILL® VIRTUAL Machining / Simulation:** Fixed an issue that could cause part damage during 5X machining with negative tool inclination.
- **hyperMILL® VIRTUAL Machining / Simulation:** A problem has been fixed that caused unnecessary head rotations in connection with a fixed axis on a Bimatec machine.
- **hyperMILL® VIRTUAL Machining / Simulation:** An issue has been fixed where tool paths were displayed incorrectly when using polar coordinate transformation, particularly affecting origins shifts and kinematics on slant bed machines.
- **hyperMILL® VIRTUAL Machining / Toolpath preview:** An issue has been fixed where toolpaths were displayed in incorrect positions in the Toolpath preview (SIMULATION Center), while appearing correctly in the Virtual Machine view.
- **hyperMILL® VIRTUAL Machining / Collision check:** An issue has been fixed where incorrect collision detection occurred when using the Circular Pocket strategy, caused by an improperly generated simulation path for later operations.
- **hyperMILL® VIRTUAL Machining / Collision check:** An issue has been fixed where the tool contour could not be modified.
- **hyperMILL® VIRTUAL Machining / Collision check:** An issue has been fixed where a false collision message was generated when drilling with a retract value of 0.
- **hyperMILL® VIRTUAL Machining / Collision check:** An issue has been fixed where inconsistent collision check results occurred when using tool series and manual pre-drill diameter changes were made.
- **hyperMILL® VIRTUAL Machining / Optimizer:** Fixed an issue where the Optimizer produced unexpected movements on 7-axis machines when switching between multiple origins.
- **hyperMILL® VIRTUAL Machining / Optimizer:** An issue has been fixed in the NC optimizer that could cause collisions with parts when smooth linking was enabled due to an incorrect safety distance check.
- **hyperMILL® VIRTUAL Machining:** An error in the NC output for **Turning → 3X simultaneous Finishing** on a GROB machine has been fixed.
- **SIMULATION Center:** An issue has been fixed where the machine limits were not considered, causing the C rotary axis to rotate incorrectly and resulting in a gouge on the part.
- **SIMULATION Center:** An issue has been fixed where a job could not be simulated due to a mismatch between the machine model and post-processor.
- **Probing cycles:** An issue has been fixed where measurement positions were incorrectly displayed in the graphical view after saving and reopening .hmc files, specifically affecting the Z-axis orientation of measurement areas.
- **Probing cycles:** An issue has been fixed where measurement positions were incorrectly displayed on the Z-axis after saving and reopening files.
- **Probing cycles / 5X Rework Machining:** An issue causing *hyperMILL®* to crash when switching jobs in non-modal mode has been resolved.



- **Turning cycles:** Fixed an issue where the toolpath for the cutting insert types **W-shape** and **Triangular** was calculated incorrectly, resulting in areas that could actually be machined not being machined.
- **Turning / Finishing:** An issue has been fixed where toolpath calculation failed when using large tools with complex geometries, caused by unrecognized island areas.
- **Turning / Face Groove Plunging:** When a machine with a machine model was activated in a job list, an incorrect collision message appeared about collision of the holder, even though no holder was defined. This error has been corrected.
- **Turning / Face Groove Plunging:** An issue has been fixed where the cycle produced a calculation error when the insert diameter matched a groove width of 1.5 mm.
- **Turning cycles / Probing cycles:** An issue has been fixed where offset values defined by variables in the **Turning → Finishing** cycle were not sent to the **Probing cycles**, which could lead to incorrect probing behavior.
- **Groove Plunging:** An issue has been fixed where the cycle failed to generate a toolpath in Inch units.
- **Drilling / Centering:** A problem has been fixed where a part collision of the extension of a tool was not detected during calculation.
- **2D Helical Drilling:** Fixed a calculation error related to the clearance plane that caused a program stop.
- **Chamfer Milling on 3D Model:** An issue has been fixed where milling with a negative XY allowance caused a calculation error.
- **3D Optimized Roughing:** An issue has been fixed where machining in High Performance mode and full cut could lead to collisions.
- **3D Optimized Roughing:** An issue has been fixed where using the **Adaptive pocket → Use adaptive pocket → Adaptive pocket only** option caused gouges.
- **3D Iso Machining:** An issue that caused a crash when calculating tool paths for the **3D ISO Machining** cycle has been fixed.
- **3D Iso Machining:** An error in the consideration of stop surfaces has been fixed.
- **3D Equidistant Finishing:** An issue has been fixed where the cycle ignored model surfaces because of tolerance settings, resulting in missing toolpath areas.
- **3D Arbitrary Stock Roughing:** An issue has been fixed where a collision occurred when using a 3 mm radius cutter.
- **3D Cutting Edge Machining:** An issue has been fixed where the tool infeed length was not available as a vertical stepdown option.
- **5X Tube Finishing:** The trimming behavior on virtual surfaces has been improved.
- **5X Rest Machining:** Unwanted B-axis movement with a manual curve strategy has been fixed. The axis now behaves correctly when **Rigid guiding** is used.
- **5X Radial Machining:** An issue has been fixed where the boundary was not mirrored correctly, causing the mirrored boundary to be ignored.
- **5X Radial Machining:** An issue has been fixed where the software crashed during machining when accessing the **Parameter** tab.



- **5X Debugging:** An issue has been fixed where incorrect or missing toolpaths were generated during calculations.
- **Tool database:** An issue has been fixed where a formatting problem in material parameters (**mm²** vs. **mm³**) caused calculation errors.
- **Tool database:** An issue has been fixed that caused inconsistencies in the processing of upper/lower case letters in an SQL database. The fix also applies to SQLite databases, but not to umlauts such as ä, ö, or ü.
- **Tool database / Report designer:** A problem has been fixed where numerical values in the tool report for NC tools were exported with incorrect formatting. The reports are now formatted correctly.
- **Tool database / TDM Systems interface:** A new version of the interface is being released, that corrects a missing Italian translation.
- **Linking job:** Corrected an issue that occurred when a **3D Z-Level Finishing** job was added to a linking job.
- **General:** A performance issue when opening frames in projects with CAD workplanes has been fixed.
- **General / CAM Plan:** An issue has been fixed where the CAM Browser lost its customized layout when switching between active and inactive CAM Plan due to incorrect handling of stored window and docking configurations.
- **General / Stock calculation:** An issue has been fixed where incorrect stock generation in a turning job could lead to collisions due to misaligned axis handling.
- **General / Stock calculation:** An issue has been fixed where using a Front/Back Chamfered Cutter with an Automatic Stock Chain led to incorrect material display and visible steps in 2D contour milling.
- **General / Stock calculation:** An issue has been fixed where transformed stock model names were not unique after splitting.
- **General / Mirror stock:** An issue has been fixed where mirrored job lists could not be calculated due to a corrupt internal flag in the **Mirror stock** function.
- **Converter / hyperMILL® for SOLIDWORKS:** An issue has been fixed that caused jobs to be marked with a red X after calculation when the 3DF data (milling area) was defined as a file and not as a surface.
- **Feature Technology:** An issue has been fixed in the CPF feature where the NCS orientation was missing in deburring cycles.

CAD

Additions and notes

File → Save As → Settings: As a preview of Release 2026, the entities for image files can be colored dark.

CAM → Optimal barrel cutter: The option **Explore lead angle** allows for the calculated values to graphically assess the effects of changing the tilt angle.

As a preview of Release 2026, the commands **Set WP to origin** and **Link origin to WP** added.

Bug fixes

The following issues have been resolved:

- **File → Open:** When opening a specific *.hmc file, an error occurs and the software close with a critical message.



- **File → Options → Options / Properties:** Loading predefined settings in the CAD options and properties may cause the software to crash.
The software crashes when **Reset** or **Load settings** is selected. This behavior only occurs when certain license settings are combined.
- **File → Options → Tooltip content:** The software crashes when the area of a complex face is to be displayed in the tooltip by moving the mouse pointer over the face.
- **File → Options → Layout manager:** The software crashes when **Restore configuration settings** from previous version is selected. This behavior only occurs when certain license settings are combined.
- **Curves → Shape contour:** Due to a specific Bézier curve and its insufficient tolerance, a contour is not recognized as planar, e.g., for turning operations.
- **Modify → Break shapes:** The software crashes when breaking a solid into faces. For example, this occurs when the locked entities are unlocked for editing purposes.
- **CAD interfaces**
When importing *.e3 data from ThinkDesign with *hyperMILL®* information, job lists are not generated correctly and several messages are displayed.
Autodesk Inventor: An *.ipt file from Autodesk Inventor 2024 cannot be imported.
STEP: A specific file cannot be opened. The software crashes during import.
- **AUTOMATION Center**
An error occurs when using the **Origin from Workplane** command with the **Workplane** option if the data does not contain a joblist. The origin will not be generated correctly.
- **AUTOMATION Center Reports**
Customer-specific data is no longer transferred to the tool report.

Release 2025 | Update 3.1

CAM

Bug fixes

- **Tool database:** An issue has been fixed where jobs showed errors and parameter changes when material names contained tab characters in the external database.
- **Inventor / Installation:** An issue has been fixed where an error message occurred when opening files in Inventor, caused by a property type conversion problem in the CAD-Shell Inventor module.

CAD

Additions and notes



Due to a necessary adjustment in the electrode simulation, when the folder C:\Program Files\OPEN MIND\hyperCAD-S\34.0\hyperCAD-S\files\electrode has been copied for custom settings, the folder Simulation must be replaced. Otherwise, there will be an incorrect rotation of the electrode.

Release 2025 | Update 4

CAM

Additions and notes

- **5X Hole Deburring / 5X Deburring: Parameter → Deburring parameters → Max. stepdown:** The variable $J:ap$ is now available to define the **Max. stepdown** parameter in both deburring cycles.



- **3D Z Level Shape Finishing:** The **Check** tool → **Calculate length** → **Extend** option on the **Setup** dialog page is now also available for the **3D Z Level Shape Finishing** cycle.
- **OPEN MIND Features → Pocket** can now also be used in the **Gun Drilling**, **Optimized deep Hole Drilling** and **Boring** cycles.
- **Feature Technology / Pocket Milling / Optimized deep Hole Drilling (new):** In the pocket milling cycle, it is now possible to select the Optimized deep Hole Drilling cycles (new) as the Plunge point on the Contours dialog page, as was previously possible with the Optimized deep Hole Drilling cycles.
- **Gun Drilling / Feature Technology:** The **Gun Drilling** cycle can now also be used with Pocket Feature. The functionality has been extended to support the parameters **Spindle RPM approach** and **Feedrate approach** with indexable cutters.

Bug fixes

- **Vericut interface:** UPDATE 4 supports Vericut from version 9.4 onward. For Vericut versions earlier than 9.4, please use UPDATE 3 or an earlier update.
- **Vericut interface:** An issue was resolved where Vericut 9.6 generated errors due to outdated and unsupported tool library versions because the interface incorrectly output older TLS versions instead of the required version 8.2 or higher.
- **hyperMILL® VIRTUAL Machining:** When generating NC using NCEvent between operations with same frame and tool for Waldrich Coburg machines with Siemens controller the tilted workplane option was not recalled after NCEvent. This issue has been fixed.
- **hyperMILL® VIRTUAL Machining:** A problem has been resolved where inconsistent tool names within tool series caused incorrect tool IDs during import, post-processing, and simulation.
- **hyperMILL® VIRTUAL Machining:** A problem has been resolved where invalid or misaligned tool geometries led to incorrect coupling detection in hyperMILL, causing errors during tool import, loading, and simulation particularly when using **hyperMILL® VIRTUAL Machining**.
- **hyperMILL® VIRTUAL Machining:** An issue has been fixed where the material removal simulation for **Thread cutting** was incorrect when using inverted **Threading tools**, caused by the **Threading insert** not being properly rotated during inversion.
- **hyperMILL® VIRTUAL Machining / Post processing:** A bug was fixed where linking movements lost their feedrate when the optimizer converted G0 to G1 moves and incorrectly interpreted the **Feedrate for linking** value "MAX" as 0.0.
- **hyperMILL® VIRTUAL Machining / NC Generator:** An issue has been resolved where the Q379 value in the Heidenhain CYCLE241 did not correctly account for the clearance distance, which could lead to inaccurate tool positioning.
- **SIMULATION Center:** An issue has been fixed where the SIMULATION Center generated incorrect tool paths and axis directions when using multiple origins, particularly in cases where the frame's Z-axis did not match the machine coordinate system and no machine model was used.
- **SIMULATION Center:** Fixed an issue where jobs with multiple origins were not handled correctly when no rotations were involved.
- **SIMULATION Center:** An issue has been fixed where tools were displayed on the wrong side in the SIMULATION Center when running without a machine model, caused by unsupported head kinematics with multiple origins.



- **SIMULATION Center:** An issue has been fixed where the SIMULATION Center showed discrepancies when working with multiple origins, causing differences compared to the internal machine simulation.
- **Tool Database:** An issue has been fixed where copying tools could cause conflicts with predefined cutting materials due to incorrect material identification. The behavior has been corrected to prevent duplicate names and ensure proper recognition of predefined materials.
- **Tool database:** An issue has been fixed where jobs showed errors and parameter changes when material names contained tab characters in the external database. The problem was resolved by ensuring that tab characters in material names are correctly handled.
- **Tool database:** An issue has been fixed where coupling points in the Tool Database were not correctly updated when switching between metric and imperial units in tool holders.
- **Tool database:** An issue has been fixed where the NC tool gage length in the tool database was not correctly converted to inches, ensuring accurate scaling when switching measurement units.
- **Tool database:** An issue was resolved where measurements and angle labels in the tool geometry overlapped or were difficult to read. The display of measurements was repositioned, and angle texts were adjusted for improved visibility and reduced overlap.
- **Tool database:** An issue has been fixed where geometry distortions occurred when importing DXF files defined in imperial units, caused by incorrect scaling and contour generation.
- **Tool database / TOOL Builder:** A licensing issue has been resolved where *hyperMILL®* Turning was not correctly recognized and therefore could not be locked by the required modules.
- **Tool management:** An issue has been fixed where modifying the cutting profile factor on the **Tool** dialog page of the job definition was not possible for values such as 0.25 due to incorrect user interface behavior.
- **Turning cycles:** An issue has been fixed where the missing of the **Use model original colors** option affected automated generation of turning areas using model colors.
- **Turning / Thread cutting:** An issue has been fixed where parameters of the second diameter were not retained after reopening the dataset, which caused an error message indicating changed job parameters.
- **Drilling / 2D Milling:** An issue has been fixed where the tolerance cycle was incorrectly applied for 2D drilling on Okuma machines. This fix ensures that user can customize behaviour for 2D drilling and 2D compensated milling in Virtual Machine configuration.
- **2D Contour Milling on 3D Model:** A collision was not detected despite being visible in the SIMULATION Center and NC path display.
- **2D Contour Milling:** An issue has been fixed where recalculating a job with the tool type **Front/Back chamfered cutter** triggered an incorrect warning message due to inconsistent parameter handling.
- **Chamfer Milling on 3D Model:** A collision during chamfer milling in the Simulation Center has been resolved by correcting an error in the contour offsetting process.
- **2D Chamfer Milling on 3D Model:** An issue has been fixed where false collision messages occurred. The cause was a collision check, which has been corrected by adjusting the detection logic and clearance distance.
- **3D Optimized Roughing:** An issue has been fixed that caused instability during calculation.



- **3D Optimized Roughing:** An issue has been fixed where toolpaths could extend beyond the defined boundary when the **Adaptive pocket** option was active.
- **3D Optimized Roughing:** An issue has been fixed where a collision occurred during High Performance machining at a high feedrate due to incorrect tool movement through the part during repositioning.
- **3D Optimized Roughing:** An issue has been fixed where a core diameter error message occurred when the calculated core diameter exceeded 90% of the tool radius.
- **3D Plane Machining:** An issue has been fixed where jobs without stock could not be calculated due to an access violation in the STL file generation.
- **5X Iso Machining:** An issue has been fixed where the cycle did not recognize tilt curves in CPF mode. The update ensures that tilt curves are now properly supported in CPF mode.
- **5X Profile Finishing:** A model violation that occurred when applying a side allowance in profiling has been corrected, ensuring that the machining process no longer compromises the part when the allowance is set.
- **5X Shape Offset Roughing:** An issue has been fixed where the cycle caused crashes during machining simulations due to self-intersections in the STL stock model. The problem was resolved by improving stock handling.
- **5X Rework Machining:** A bug that caused *hyperMILL®* to generate NC output with a feedrate value of F0 has been corrected.
- **Installation / Inventor:** An issue has been fixed where *hyperMILL®* 2025 Update 3.1 could not be properly installed or integrated with Autodesk Inventor 2026 due to missing compatibility settings.
- **Installation / Inventor:** An issue has been fixed where an error message occurred when opening files in Inventor, caused by a property type conversion problem in the CAD-Shell Inventor module.
- **General / Stockmodel dialog:** An issue has been fixed where the **Frame as center** setting (**Bounding geometry**) was not saved as default, causing inconsistent behavior.
- **CAM Plan / Inch mode:** A defect was corrected that prevented cappings from being created in Inch mode, although they were generated correctly in metric mode.
- **CAM Plan:** A crash issue that occurred when using the CAM plan under certain model view conditions has been resolved.
- **User interface:** An issue causing incorrect text display in the NC input field on certain Windows 11 versions - where deleted text was not properly refreshed - has been fixed by replacing the affected Windows control with a custom control.
- **User interface:** A crash has been resolved that occurred when switching between jobs with the **Define name pattern** dialog open, which could lead to application failure and data loss.
- **Installation:** In the *hyperMILL®* 2025 Update 3 installer, formatting inconsistencies in the Japanese license text, such as unnecessary spaces and section-numbering discrepancies, have been corrected.
- **Macro Technology / General:** A defect causing jobs to fail when opening in the Macro database has been resolved.
- **Cycles / General:** A collision issue has been resolved where split toolpaths incorrectly reused the same solution file due to identical UCIDs being assigned to all subjobs.



CAD

Additions and notes

Electrodes: Material marked with the attribute `electrode="yes"` in the material list for workpieces and for eroding with electrodes is now only offered for selection as electrode material. It no longer appears in the selection for the workpiece.



Reuse an electrode

When reusing an electrode, there is a change in the procedure. To copy, the EDM reference systems in project A and in project B must be currently set instead of the world workplanes.

Bug fixes

The following issues have been resolved:

- The change of the tessellation of a solid using the command in the graphics area toolbar does not work if the solid was previously selected as a closed solid using the inside/outside selection option.
- **Drafting → Turning → Modify linear tolerance:** When changing the linear tolerance of a V-sketch for turning, the "green dot" is no longer displayed above a certain tolerance.
- There is a typographical error in the description of the electrode colors.
- **CAM → Analyze distance toolpath-shape:** The software crashes as soon as the Automatic computation option is checked for a specific file.
- CAD interfaces
 - Supported versions updated for PTC Creo, Siemens NX and Parasolid.
 - A Siemens NX file cannot be opened in the software. This issue occurs after adding the new Siemens NX surface attribute "Powder Coating".
- AUTOMATION Center Reports
 - The job listing selection does not open when exporting the report.

Release 2025 | Update 4.1

CAM

Bug fixes

- **General:** Under certain circumstances, milling surfaces selected in the job were not considered in the cycle calculation.
This applies to the following cycles:
3D Equidistant Finishing
3D Z Level Shape Finishing
3D Rib / Groove Machining
5X Swarf Cutting one Curve
5X Swarf Cutting two Curves
5X Prismatic Fillet Finishing
5X Tangent Machining
5X Shape Offset Finishing
5X Shape Offset Roughing
5X Tangent Plane Machining



Release 2025 | Update 5

CAM

Additions and notes

Bug fixes

hyperMILL® VIRTUAL Machining	An issue has been fixed that caused errors when no machine folder was used. The correction ensures proper processing in this scenario and prevents the problem from recurring.
hyperMILL® VIRTUAL Machining	A matter in the MillTurn simulation has been fixed, where the cutting tool was incorrectly positioned and rotated by 90° in particular in combined turning and drilling operations with round axes.
hyperMILL® VIRTUAL Machining	A bug has been fixed where a disabled table rotation led to incorrect pole recognition and thus to inaccurate kinematic behavior.
hyperMILL® VIRTUAL Machining / Simulation	An issue was resolved where faulty angle calculations during 5-axis drilling with multiple origins led to incorrect simulation behavior and invalid NC code at tilted origins.
2D Milling	A bug has been fixed where collision warnings were missing in the SIMULATION Center.
Feature Macro Technology	An issue was resolved where tools in the Macro DB could not be opened after being used in the Back Boring job due to missing required 3D geometry data.
3D Optimized Roughing	A defect causing unintended stock gouging during link movements has been resolved.
Macro Technology (Job macro)	An issue has been fixed where NC-Event names in macro outputs were incorrectly prefixed with "T", causing misleading NC event naming.
Additive Manufacturing / Linking job additive	An issue has been resolved where the Linking Job Additive did not retain the feed-related technology parameters from subjobs, resulting in incorrect feeds.
hyperMILL® VIRTUAL Machining	An issue has been fixed where the processing time calculation in the VIRTUAL Machining Center was significantly too high because non-existent tool movements were included in the calculation.
General	An issue was resolved where changes to coolant settings within a job were not correctly taken into account in the status or NC output, leading to faulty NC files.
General	An issue was resolved where navigation and text selection in input fields did not function correctly due to custom processing.
5X SWarf Cutting 1 Curve	An issue was resolved where a crash occurred during processing in the virtual machine.

CAD

Additions and notes



A note for OPTICAM users: When updating to a newer *hyperMILL®* version, the OPTICAM version should also be updated to the latest available version. OPTICAM has minor changes with each update from *hyperMILL®*, but the version number remains the same. These are solely changes that ensure that the new functionality is functional in the API.



Bug fixes

The following issues have been resolved:

- CAD interfaces

PTC Creo

A copied M4 threaded hole has been incorrectly recognized without a thread.

In a component, the threaded holes are mirrored twice within the original data. After the import, the two holes were only recognized as threads for the initial side, and incorrectly only as holes for the other sides of the model.

A file of version 11 could not be opened.

ACIS

The supported version has been updated to V34-V35 (2024-2025).

A file could not be read because it was not recognized as *.sat.