*hyper*MILL®

Release notes 2025 | Update 2

OPEN MIND
THE CAM FORCE

This document is intended for users and administrators. It applies to *hyper*MILL®, *hyper*MILL® SHOP Viewer, *hyper*MILL®CAD, CAD Viewer, *hyper*MILL® for SOLIDWORKS, and *hyper*MILL® 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 *hyper*MILL®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 *hyper*MILL® data (such as postprocessors and simulation tools), you should consider the following: The format of all data generated by *hyper*MILL® 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 Thu, Jul 3, 2025)

Table of Contents

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

*hyper*MILL® 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 *hyper*MILL®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 und CAD platforms

64-bit operating systems	Windows 10, Windows 11 *
64-bit CAD platforms	*hyper*MILL® Inventor 2023, 2024, 2025 SolidWorks 2023, 2024, 2025
Server operating systems (License server only)	Windows Server 2016, Windows Server 2019 Windows Server 2022
*hyper*MILL exclusively supports 64-bit operating systems. * Starting with Windows 11 version 24H2, we recommend at least *hyper*MILL® 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` `*.prt.*`	11	x	
	`*.asm` `*.asm.*`	11	x	
	`*.neu`	11	x	
PTC Creo	`*.xpr`	11	x	
	`*.xas`	11	x	
Siemens NX	`*.prt`	NX2412	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	37.1	x	
		14		x
	*.x_b	37.1	x	
		14		x
JT-Open	*.jt	10.9	x	
		10		x
IGES	*.igs	5.1, 5.2, 5.3	x	
	*.iges	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	2019	x	
	*.dxf	2019		x
Point clouds	*.pt, *.asc *.xyz *.txt	No version	x	
	*.pt			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`	2023	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 - *hyper*MILL (AME) (iM-HYP) Optional for 3D Tool data transfer: 3D-Solid Converter for *hyper*MILL (iCHYP)	TDM Systems - Base Installer TDM Systems - Data Installer TDM Application Server Installer TDM GlobalLine Interfaces Installer (for the Smart Interface Client *hyper*MILL)
Zoller TMS	*hyper*MILL 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 *hyper*MILL Interface	WinTool 2020 (WT2020.2.1) Microsoft Server 2012 or higher Microsoft SQL Server 2012 or higher *hyper*MILL Interface (2.13.5)
Hexagon Manufacturing Intelligence	NCSIMUL Tool NCT-CAM-HY (*hyper*MILL 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 *hyper*MILL® version:

*hyper*MILL®	OPTICAM
2025 | Update 2	2025.2
2025 | Update 1	
2025	
2024 | Update 5	2025.1
2024 | Update 4	
2024 | Update 3	
2024 | Update 2	2024.2
2024 | Update 1	
2024	

Interfaces NC simulations

VERICUT from version 7.0
NCSimul from version 2020.0

2. New commands and additions

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

CAM

Basics of CAM editing

Specifying the basic settings

Document dialogue page

Color table

Set the colors for various *hyper*MILL 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 ***hyper*MILL® 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

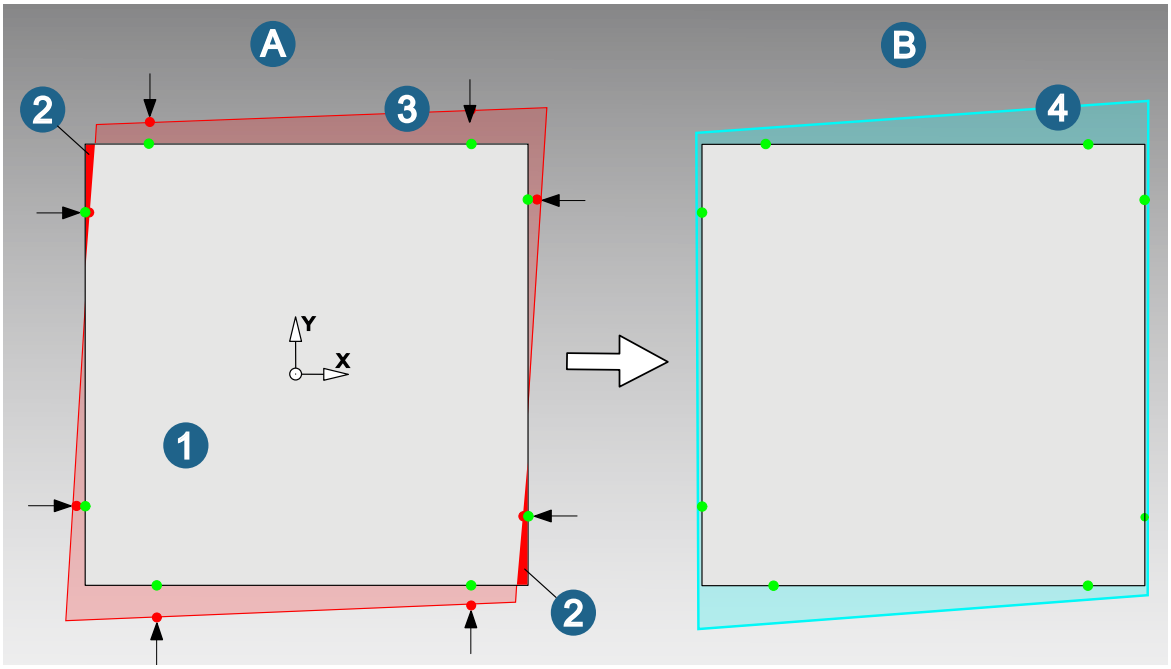
Minimize distance between points

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 *hyper*MILL 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

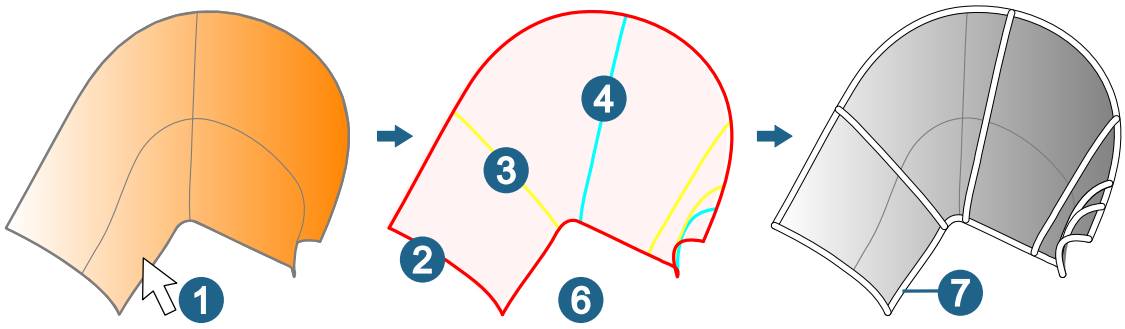
Create topology information

Generate topology curves for 3D finishing.

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

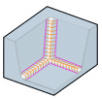
More options

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**.



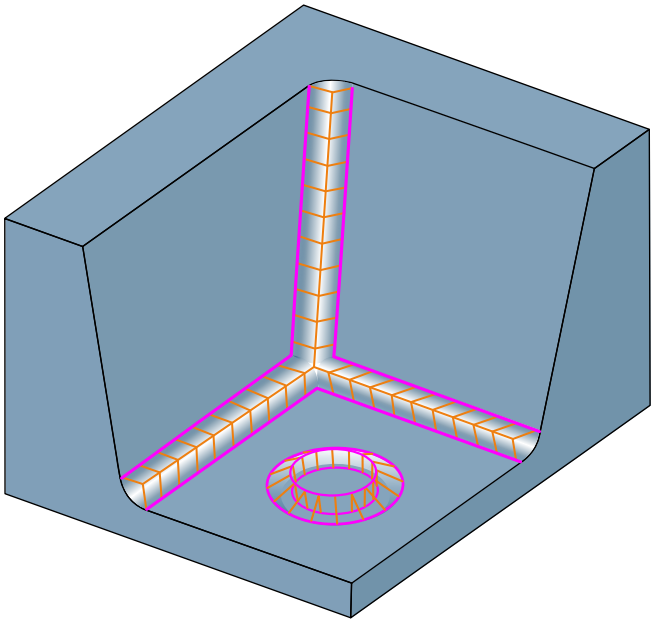
Create rest material data.

***hyper*MILL® 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

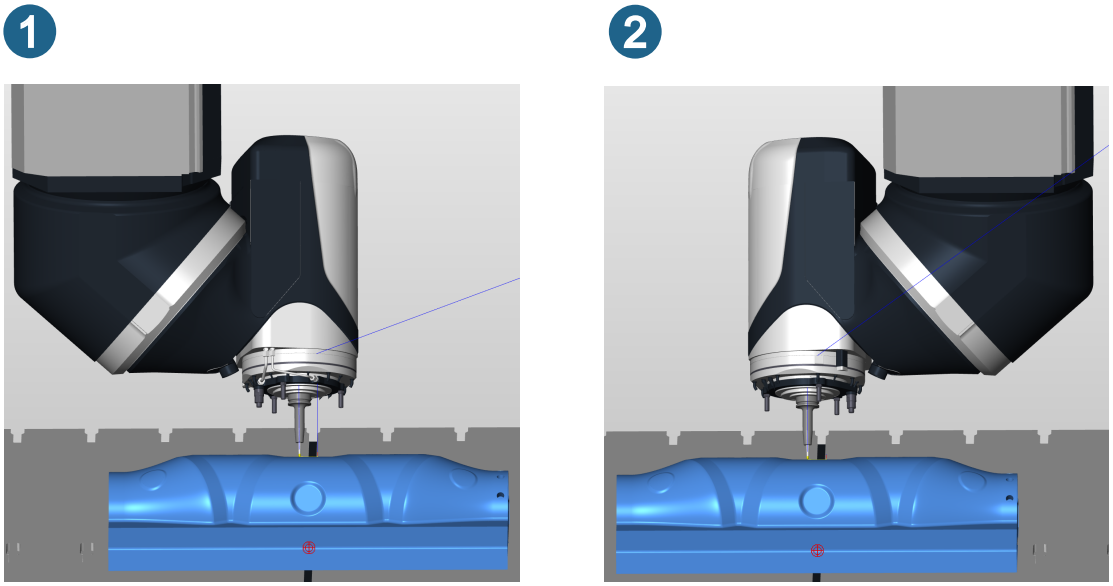
Additional information: Closest C angle plus / Closest C angle minus

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.

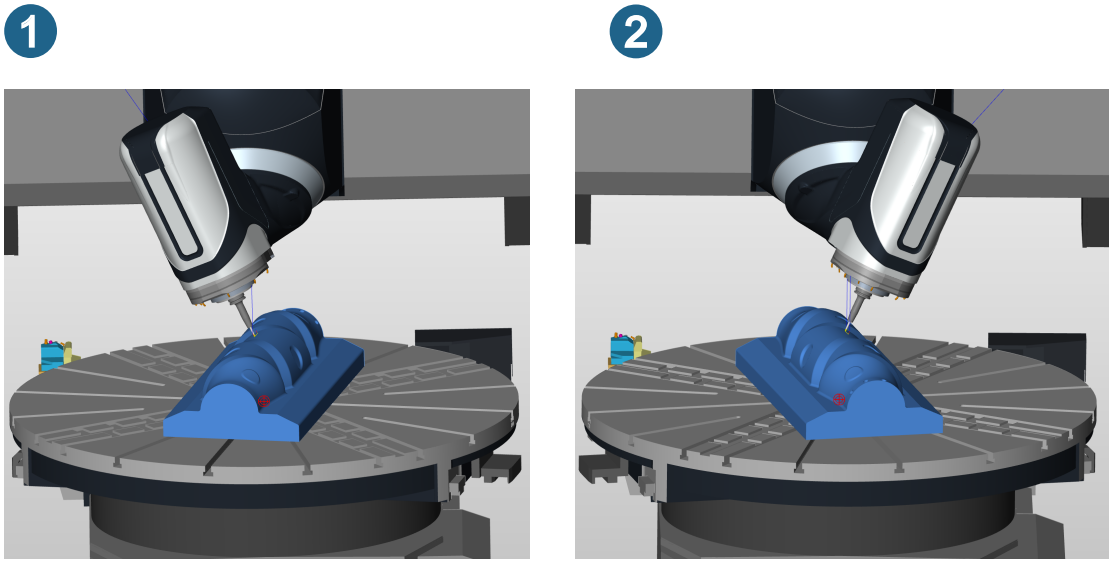


Example: BC head/table kinematics, angle for the C axis = 0

(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



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 use to ease the programing 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 *hyper*MILL® 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

	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: • `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.

	NC file exists but is no longer up to date. The NC status does not change when the software version is changed.
	 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.
	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.

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 selected directly from the list in the job definition (see **ToolFrame**) by clicking the icon. A preselected frame or origin is displayed in **bold** in the browser.

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

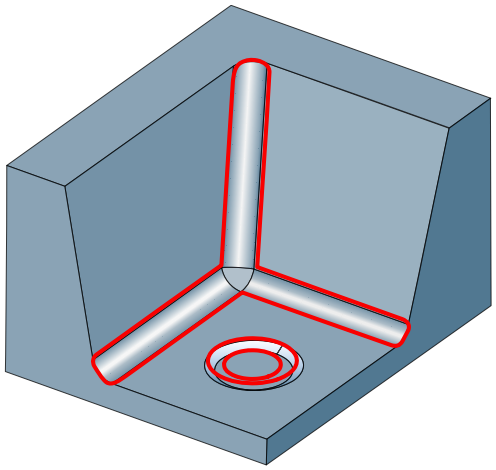
Generate feature

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.

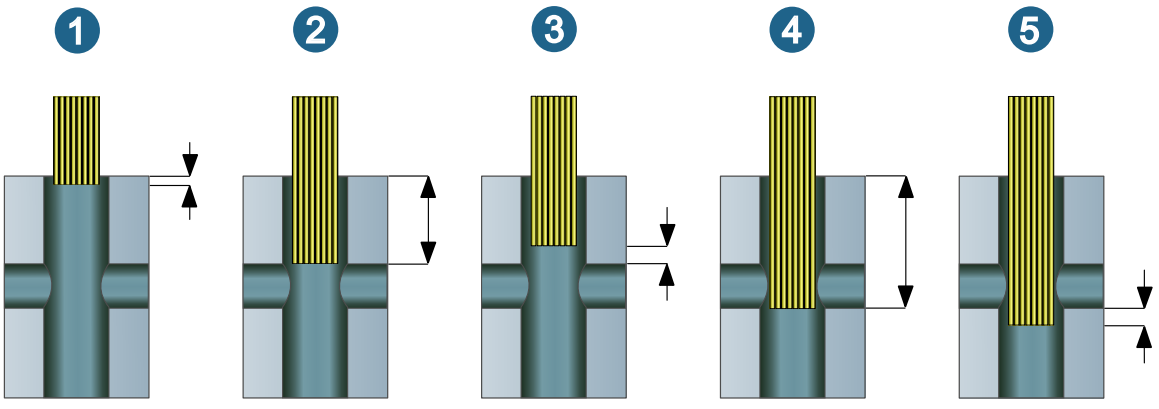
Feature top: The value for **Feature top** is automatically taken from the feature or sub-area of the feature (e.g. chamfer or thread) selected in the Feature Connector.

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 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.

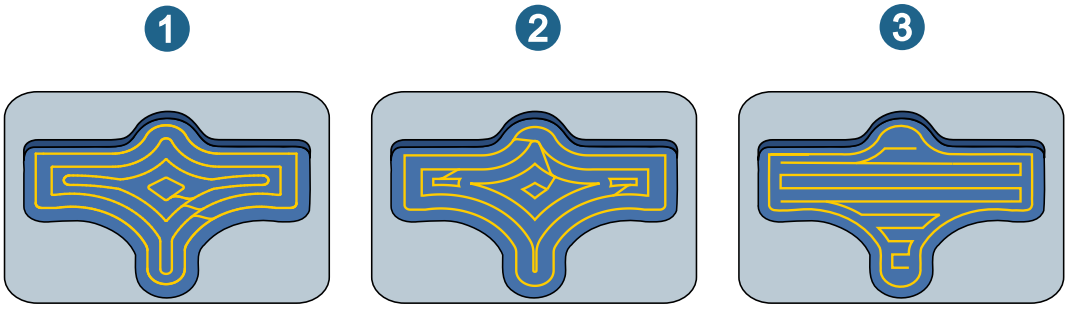
If the **Finish pass** option is used in combination with the **Parameters** → **Machining area** → **Manual top** function, the stock model above the defined value is not taken into account in the cycle calculation. This can lead to a collision between the tool and the stock model.

Plane Machining

Strategy

Pocket mode

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



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.

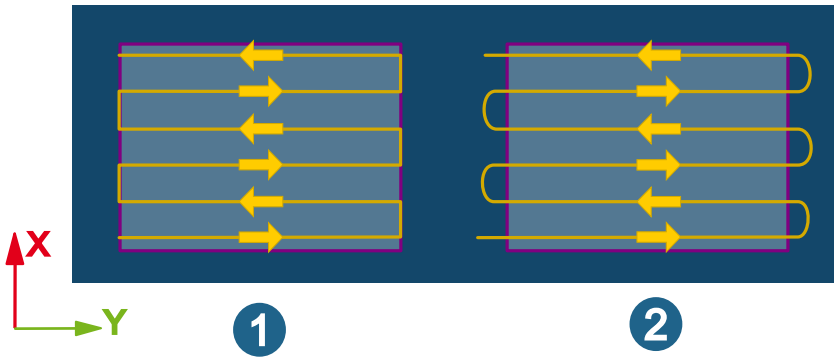
Machining angle: With a machining angle of 0°, the infeed direction corresponds to the X axis of the machining frame.

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).

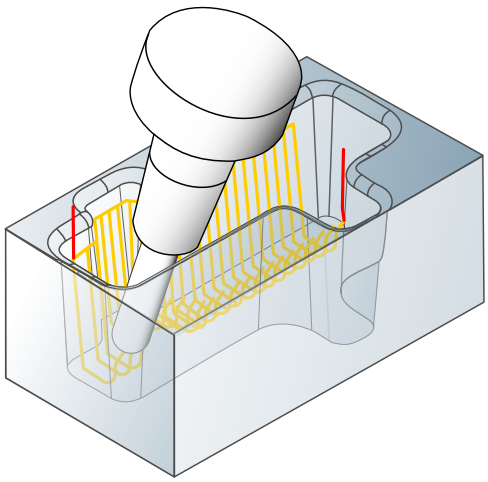


5X Machining

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.

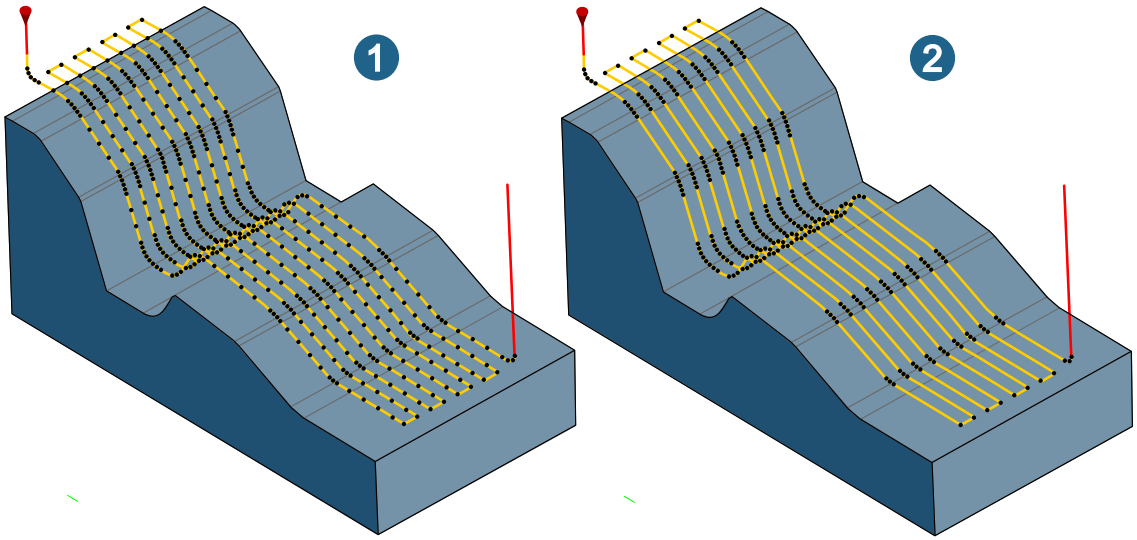
High precision surface mode: Use when maximum accuracy is required to calculate the toolpath. The calculation of the toolpath is not based on the faceted model geometry, but on the real surfaces of the model. Only available for **Ball mill**.

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.

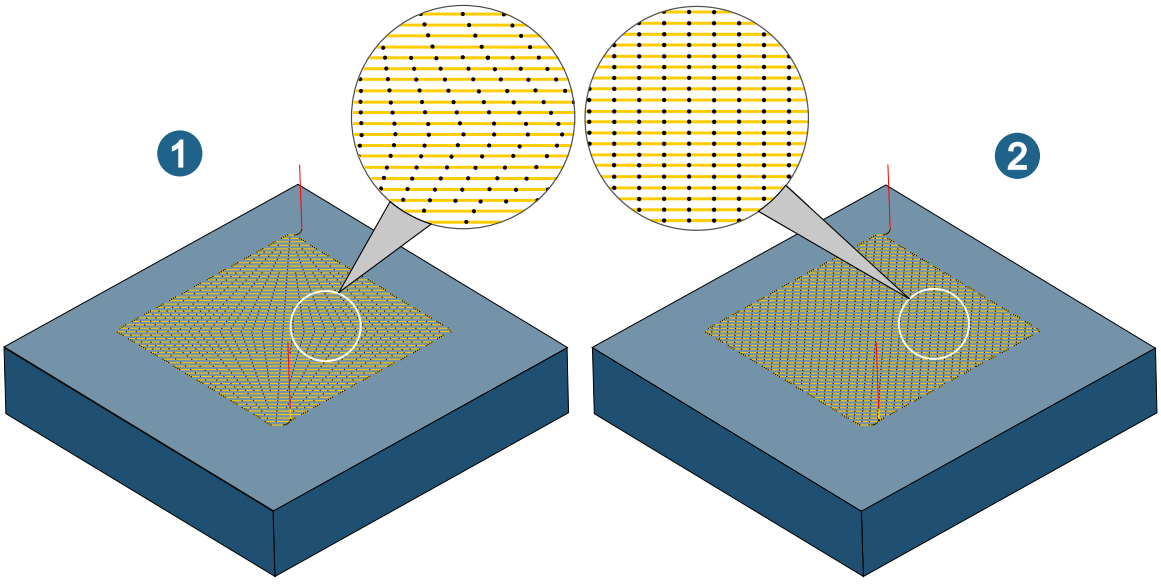
Function is not activated (1), Function is activated (2)



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.

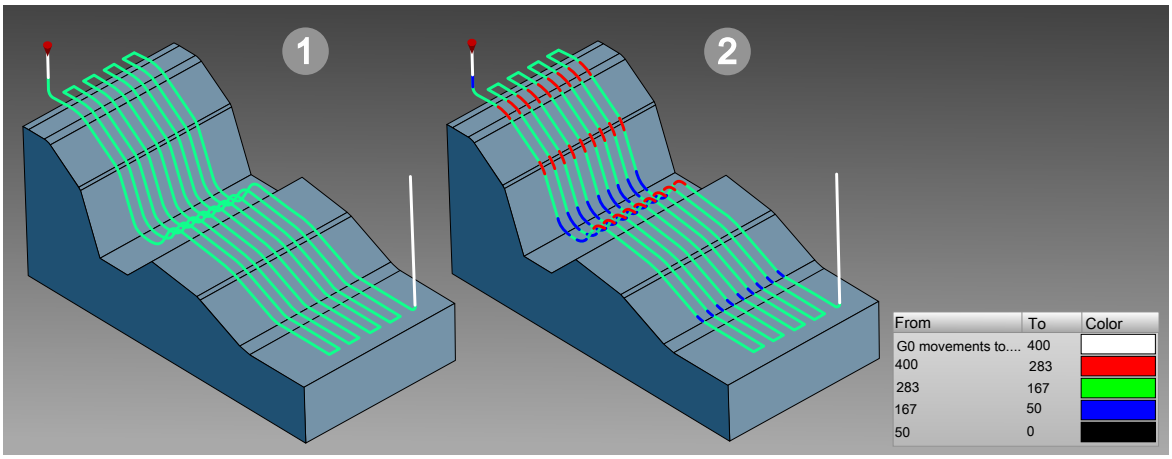
Cross align points: Align the points across the entire toolpath perpendicular to each other (i.e. at right angles to the toolpath direction).

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 *hyper*MILL®. 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.

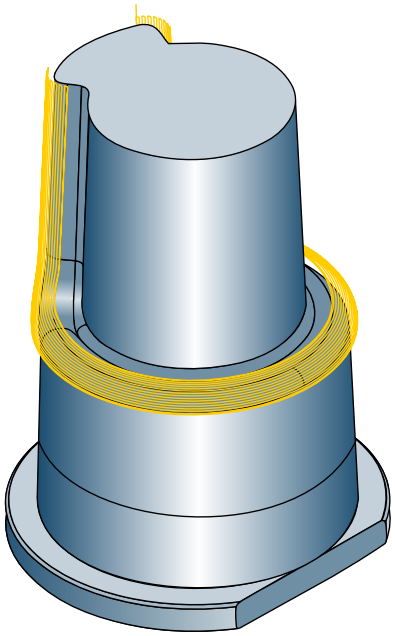
Activate **True Shape point distribution** to guarantee an even and correct point distribution in toolpaths. The parameters **Calculation tolerance, Max. chordal error**, **Max. G1 length**, **Min. G0 distance**, **Reference G1 length**, control the arrangement and distances between the points.

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.

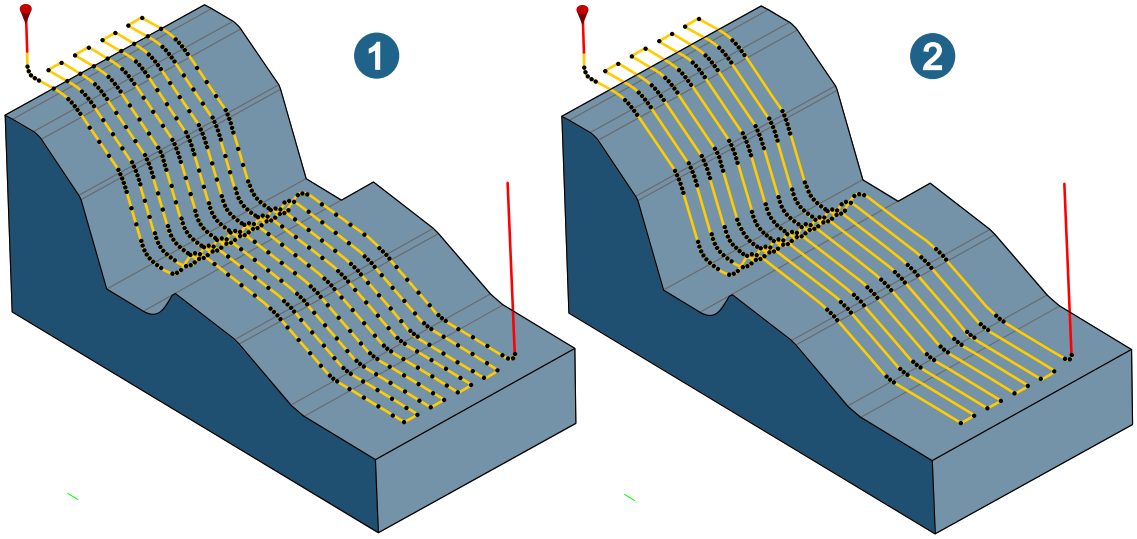
High precision surface mode: Use when maximum accuracy is required to calculate the toolpath. The calculation of the toolpath is not based on the faceted model geometry, but on the real surfaces of the model. Only available for **Ball mill**.

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.

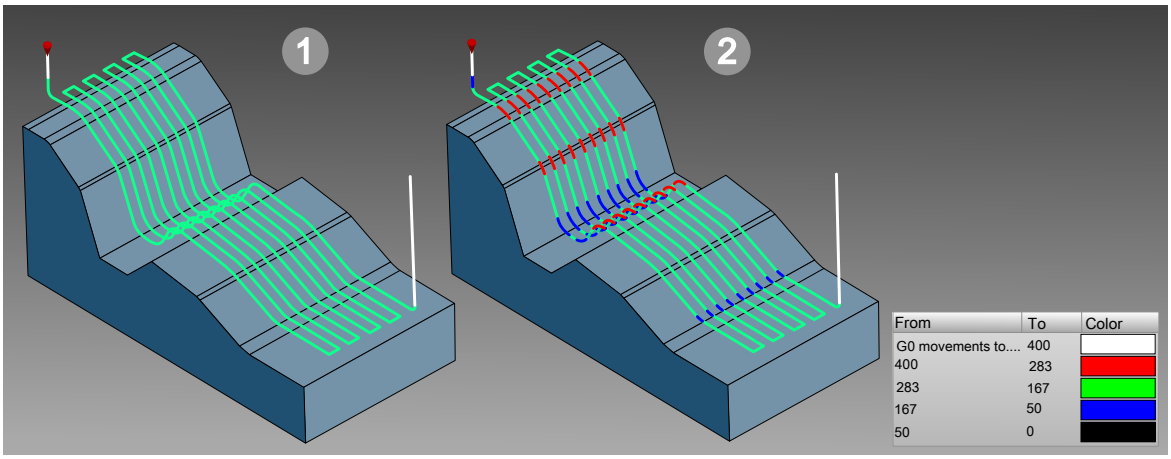
Function is not activated (1), Function is activated (2)



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 *hyper*MILL®. 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.

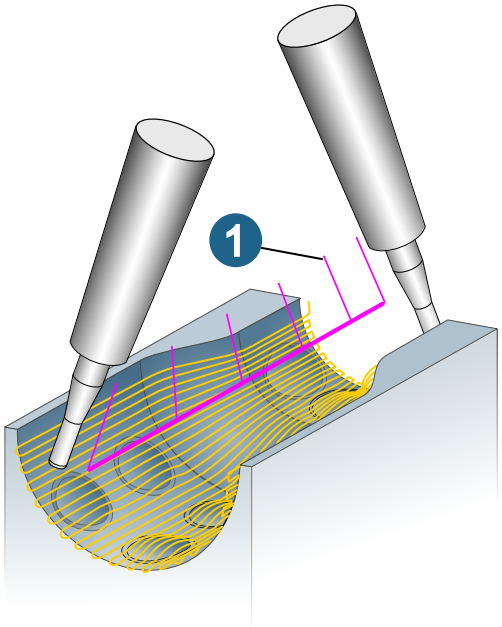
The **Min. feedrate (factor)** and **Max. feedrate (factor)** define the upper and lower limit for the feed rate adjustment. The factor refers to the standard feedrate defined for the toolpath. Use **Number of steps** to control in how many intervals the increase or decrease of the feedrate adaption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

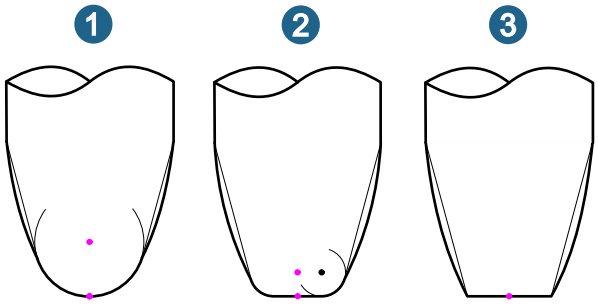


Surface Milling

Tangent Plane Machining

Tool

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



5X Deburring

Contours

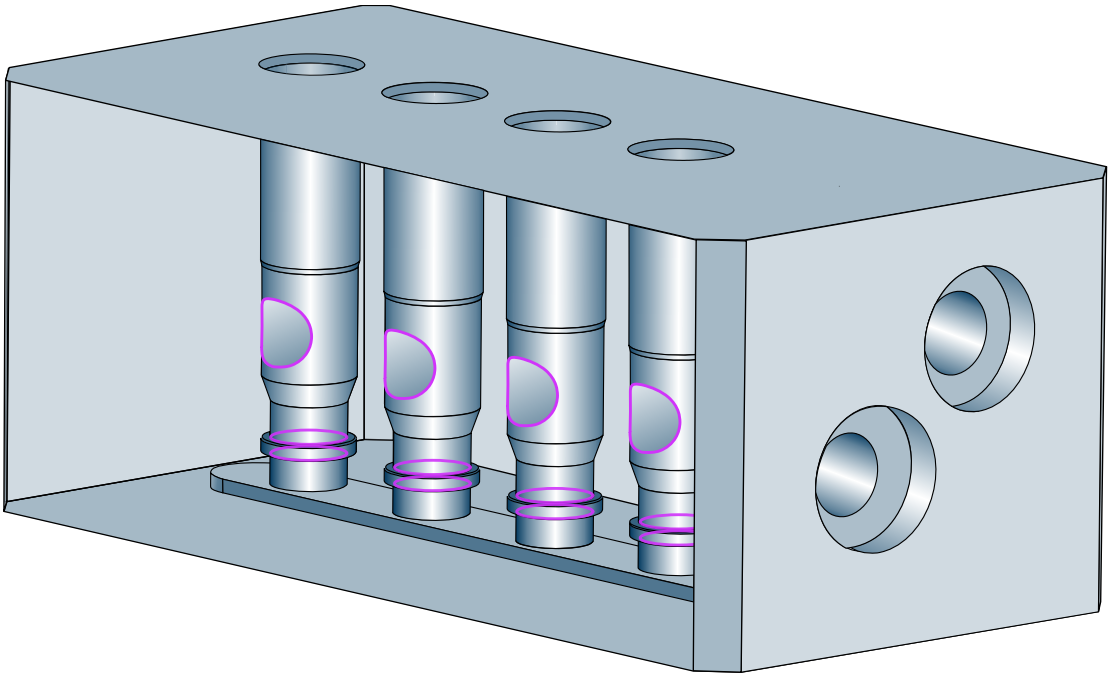
Technology

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

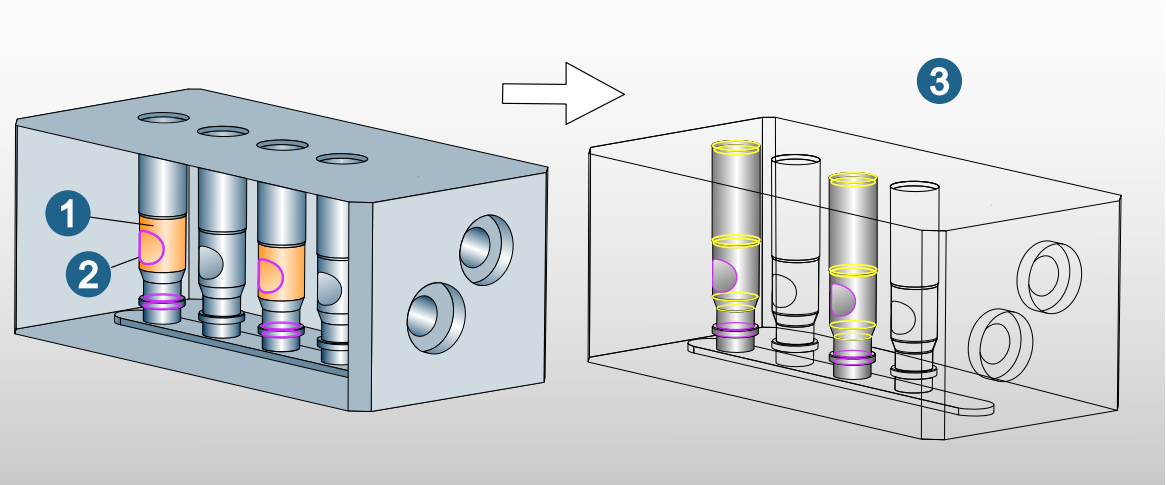
The edges to be machined are defined as preparatory work in the CAM plan. Information on this is provided in section **Basiscs of CAM editing** → **Structure CAM project** → **CAM Plan**.

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

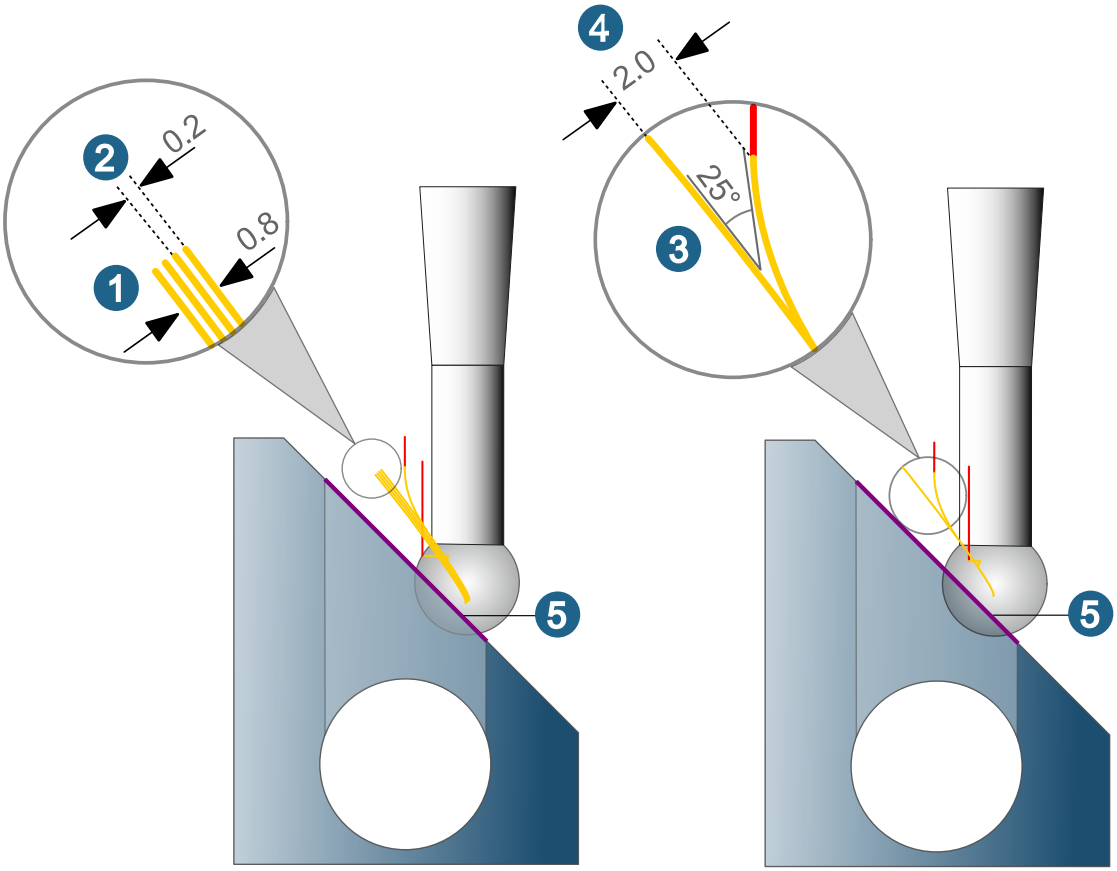
Chamfer distance: (1) Chamfering is defined by a distance from the sharp edge.

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.

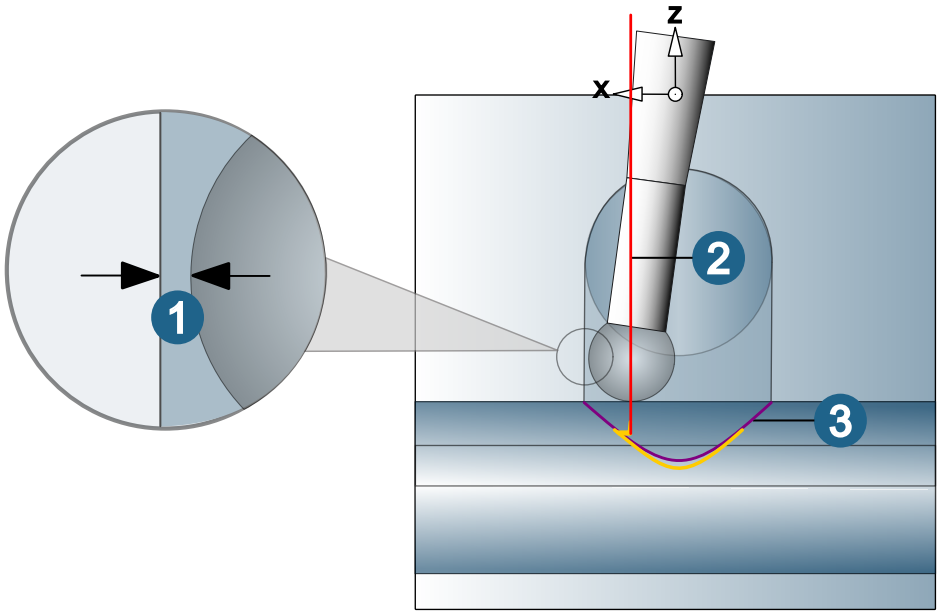
(5) Edge to be deburred.



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.

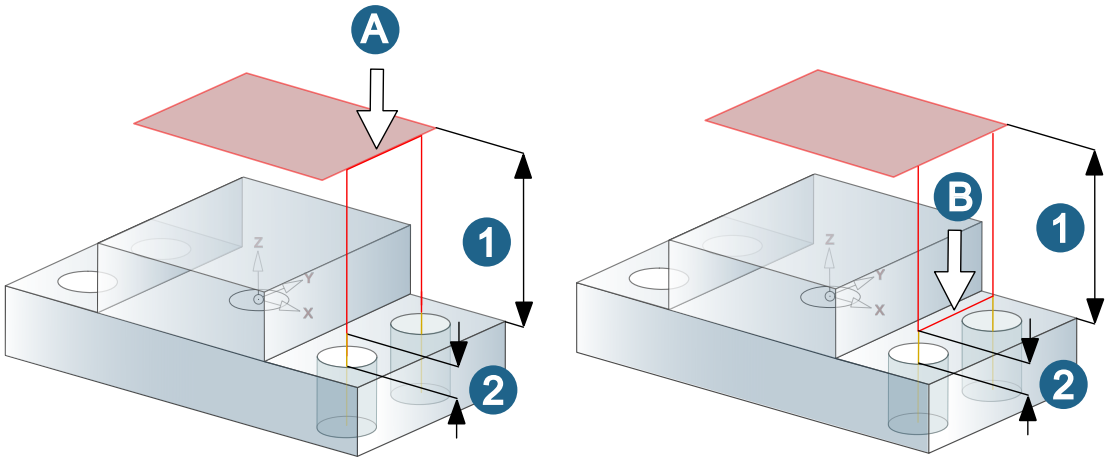


Retract mode

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

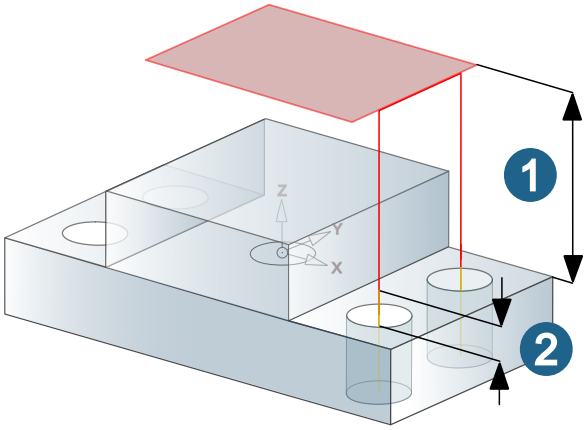
Clearance plane (1): Start and end position of each machining operation is on the clearance plane (A). The linking movements between the machining operations take place at the height of the clearance plane (A).

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 **Overlaping 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

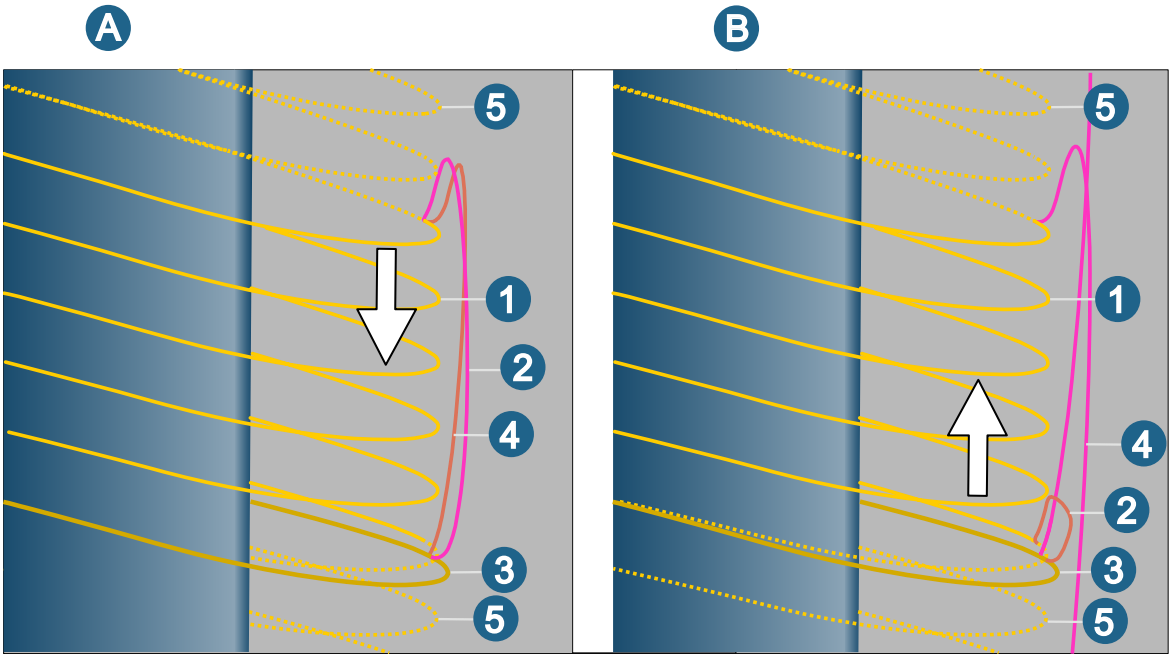
Technology: Stepwise finishing with blending path

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**.



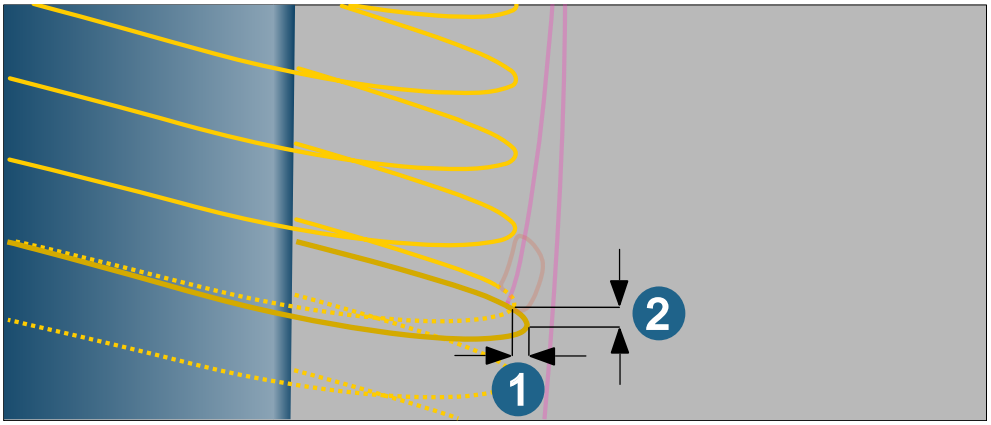
(1) Local finishing toolpaths, (2) linking movement to the finishing toolpaths, (3) blending path, (4) linking movement to the blending path, (5) global finishing toolpaths.

(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

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

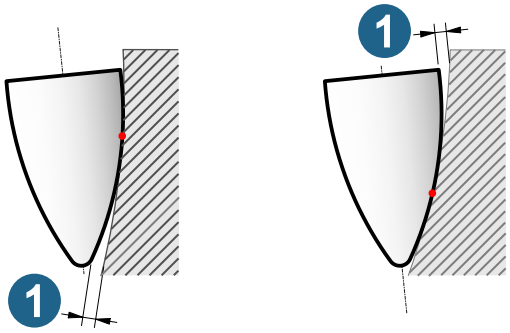
Optimize parameters of barrel cutters based on surface and curve information.

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.

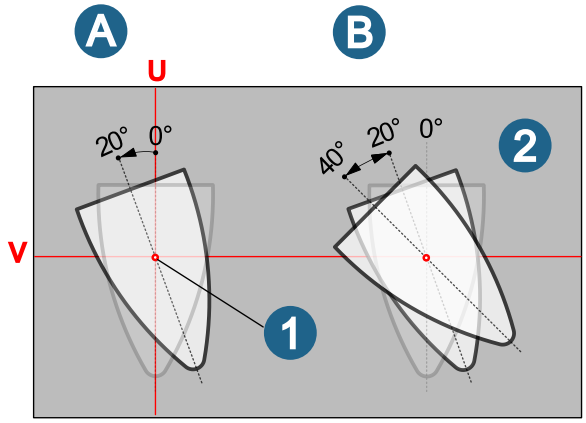


Lead angle

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**: Ⓐ: 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**: Ⓑ: 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.

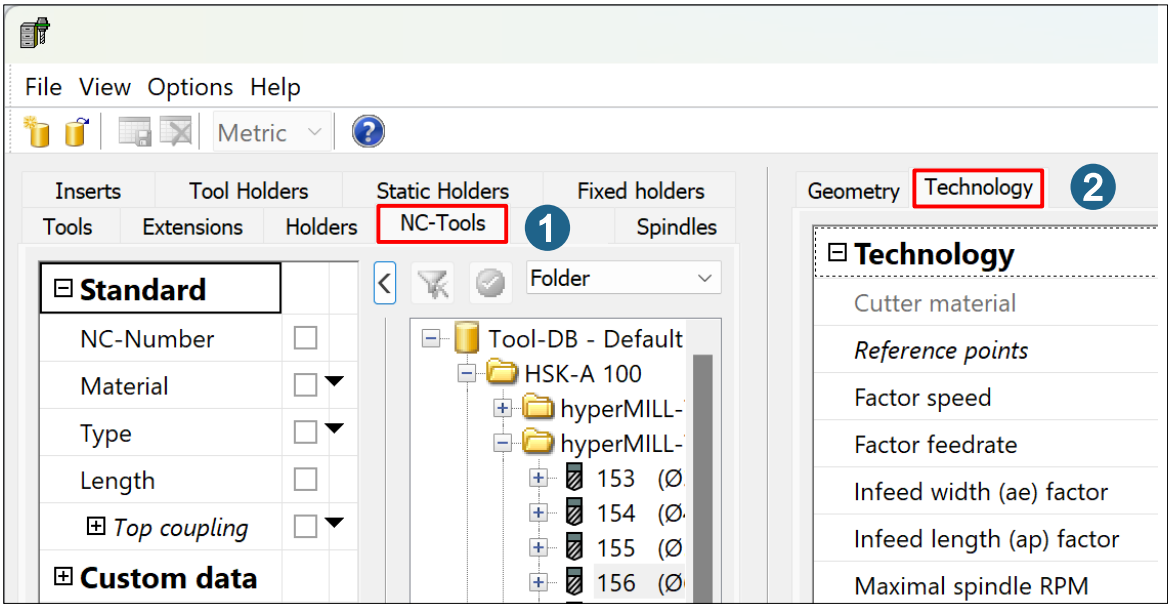
Define NC tool

NC tools and technology data

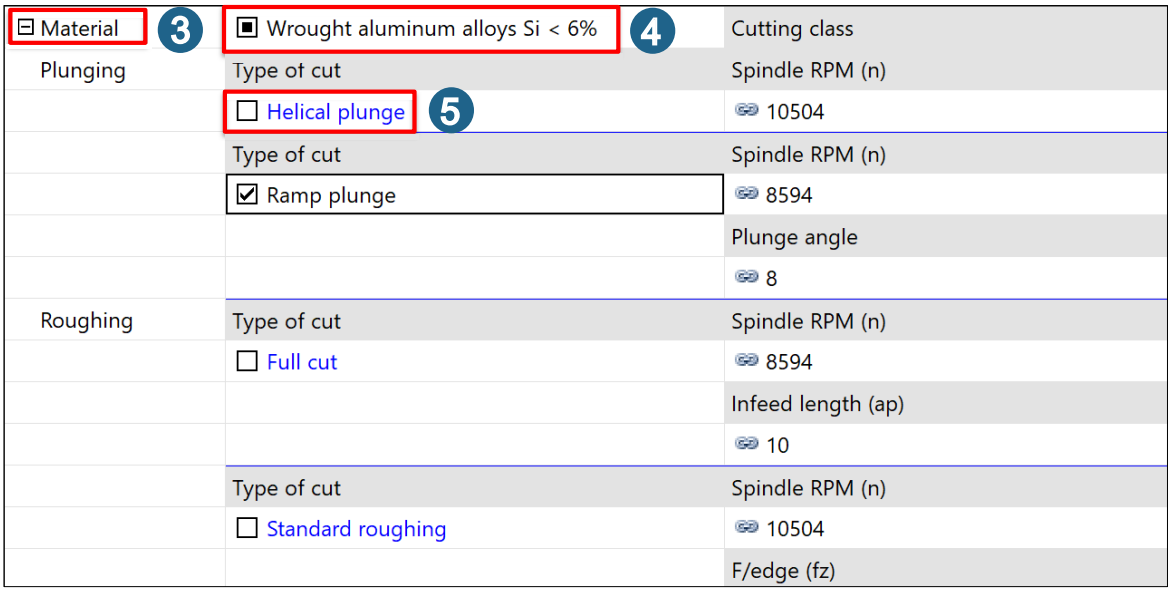
Usages and types of cut define in the tool for which applications the **Tool** can be used. Depending on the assembly (for example, with holders of different lengths), the **NC tool** may only be used in certain situations or for certain materials.

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.



TOOL Builder

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

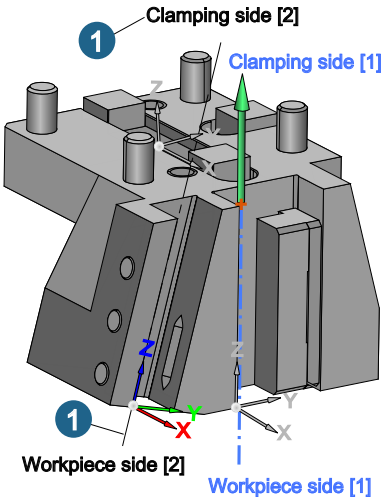
Add a new mounting position

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.

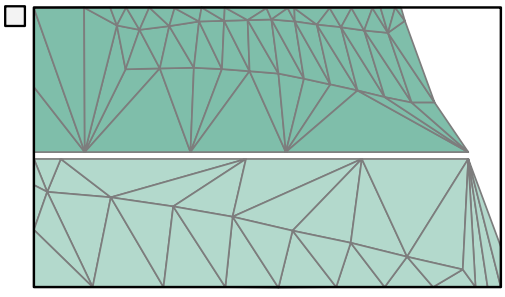
Model > Entities

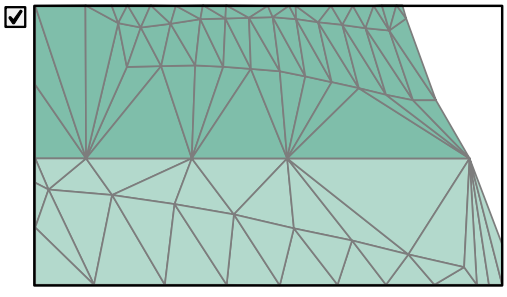
Tessellation tolerance: Tolerance for the display of curves, faces, and solids.

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 [56] 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 *Sliver faces* 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 *Sliver faces* 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.

Property	Description
Healing	**Remove faces with width less / equal (mm/inch)**: Enter a value for skipping *Sliver faces* 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 *Sliver faces* 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 *hyper*MILL® 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.
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 *Sliver faces* 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 *Sliver faces* 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 *Sliver faces* 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

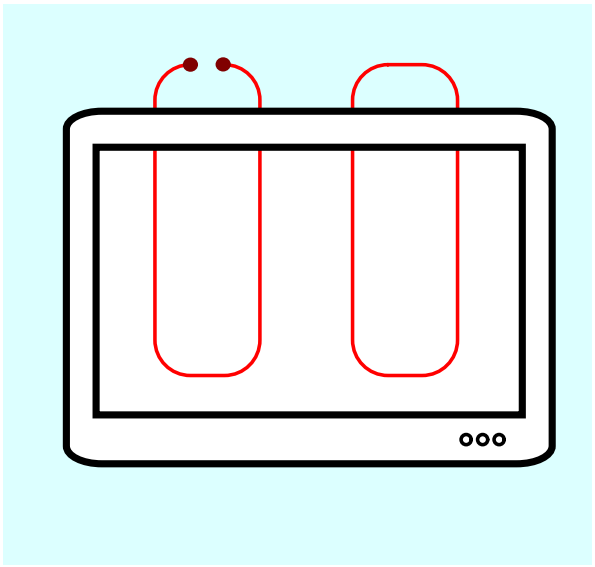
Select a contour or a boundary.

Select → **Chain**

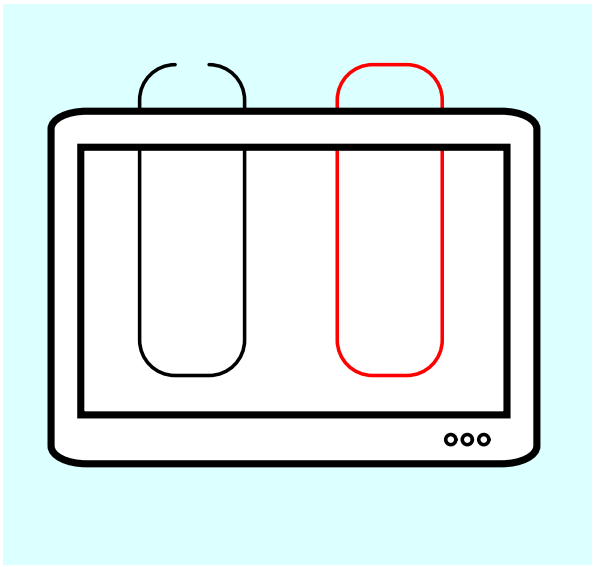
Select

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.

☐



☑ 7



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

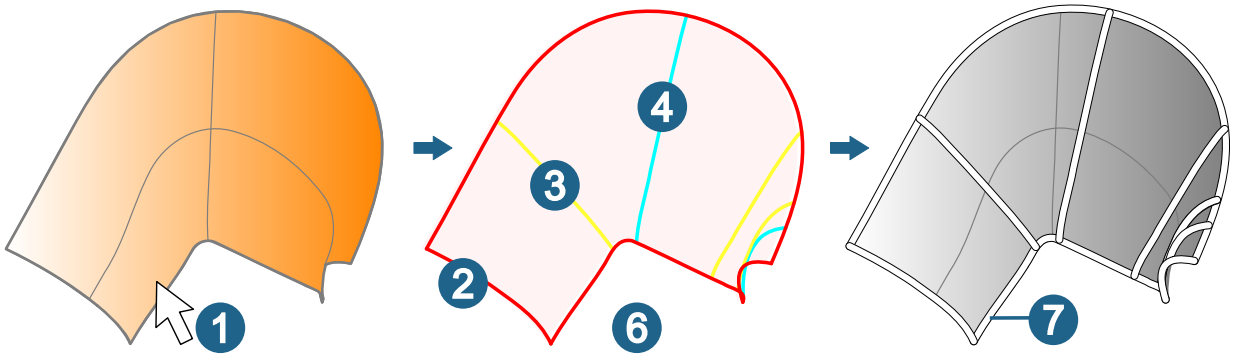
Analyze continuities at edge transitions and internal continuities.

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

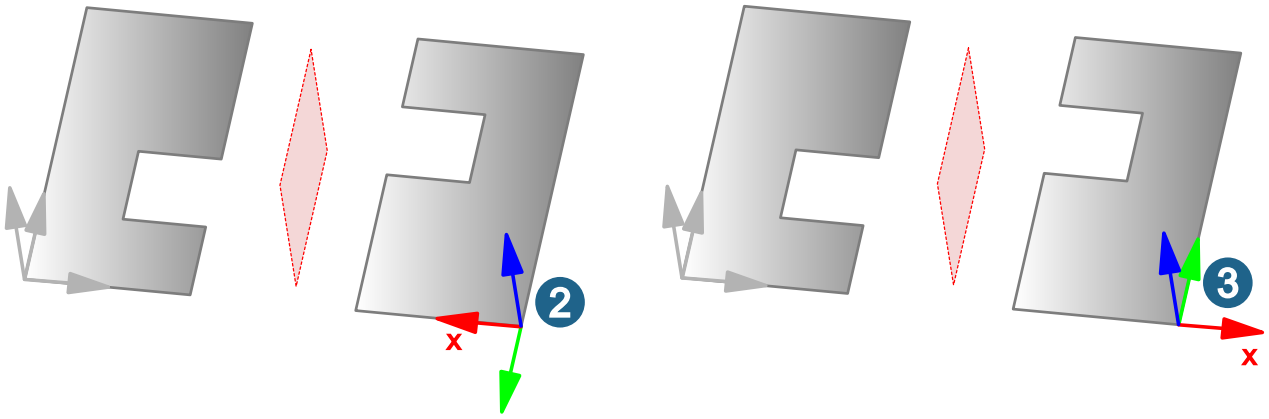
2025

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.

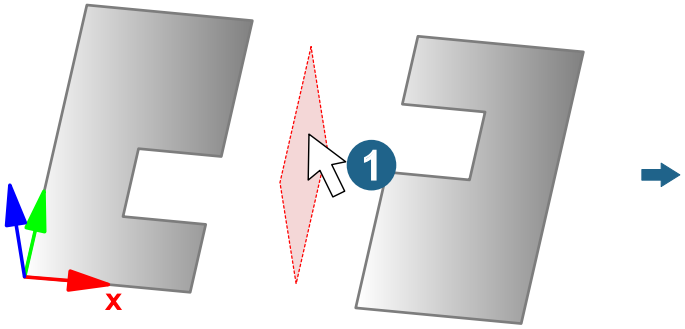
Axis to keep

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**:

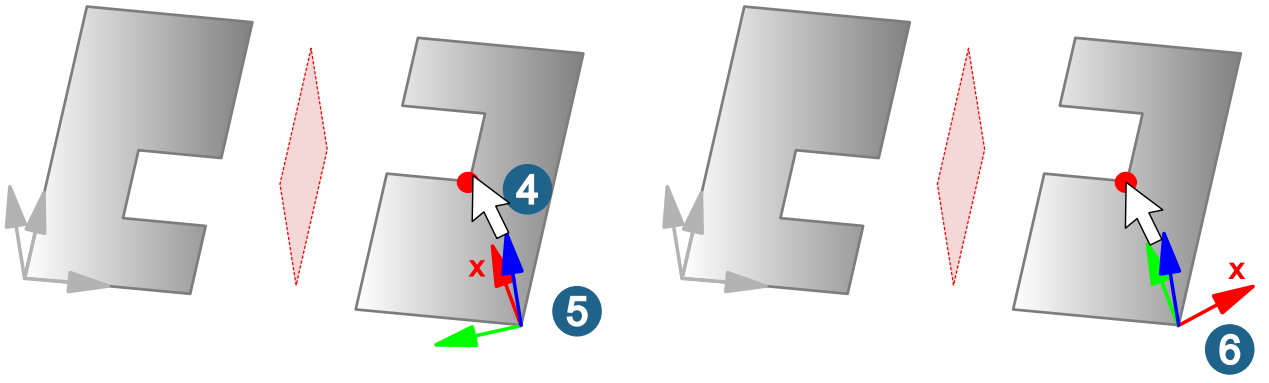
XY axis direction

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.

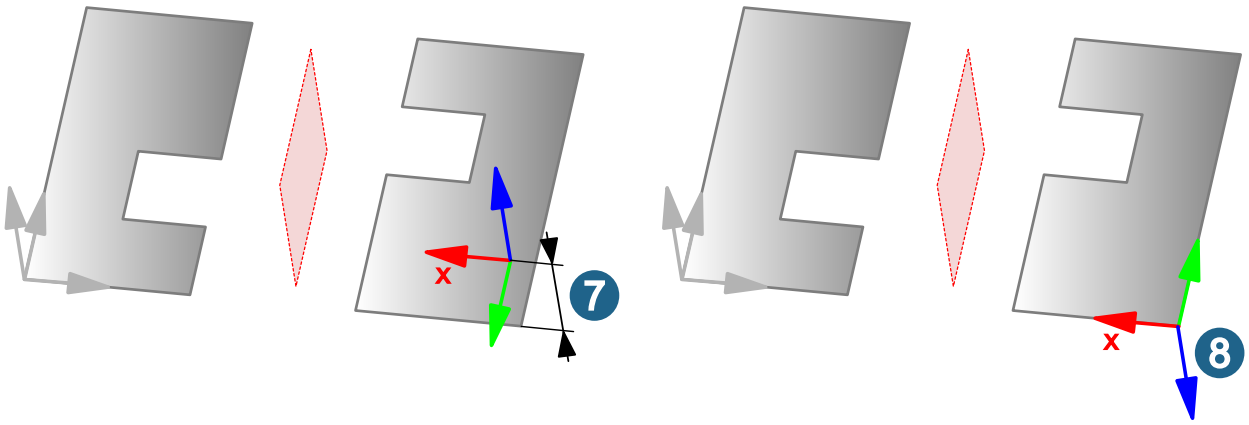


If the selected point or the selected direction (line) is not already on the preset XY plane, the point or line is projected onto the XY plane. The result is used to align the X or Y axis.

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**

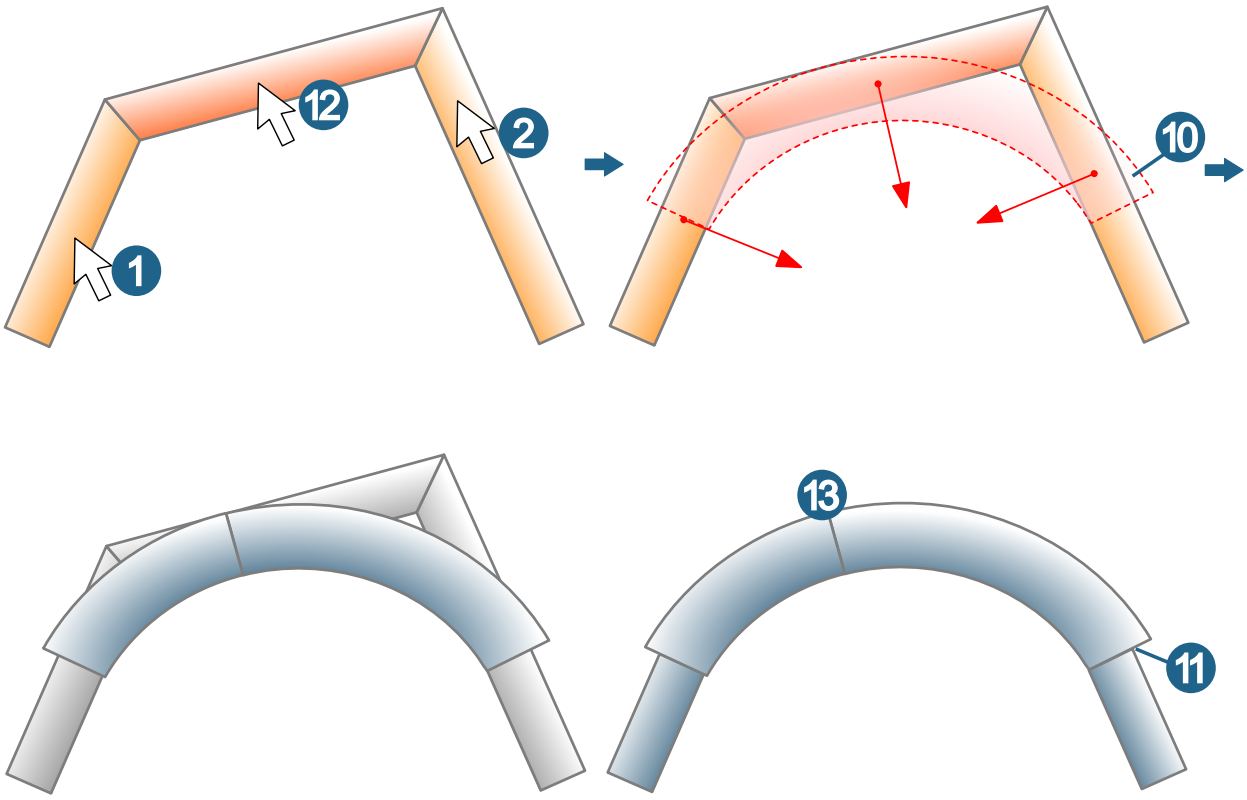
Tri-tangent

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**

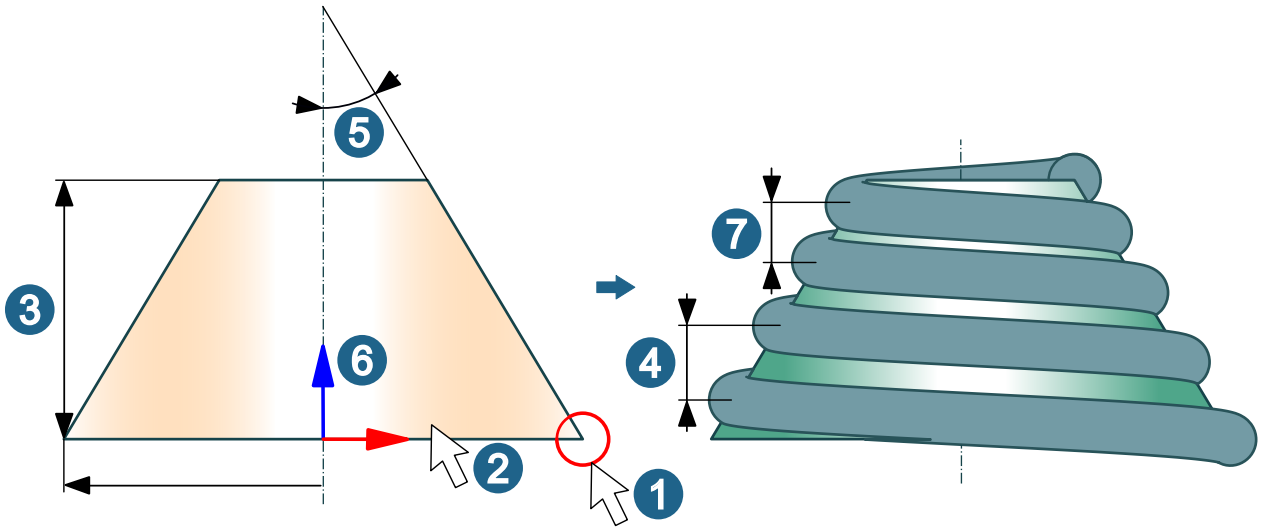
2025

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

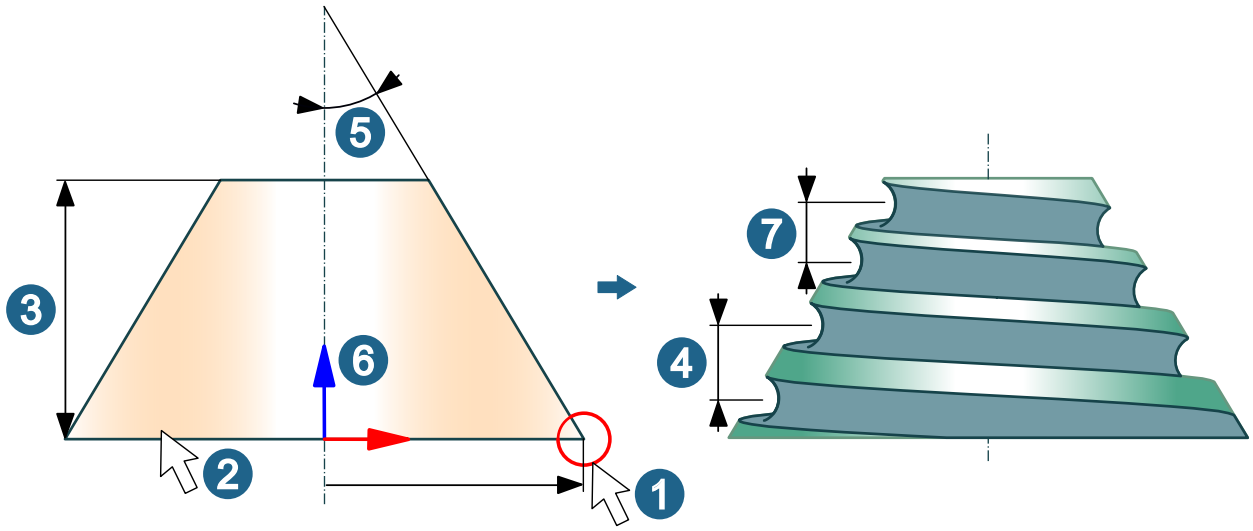
2025

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.

EDMconNG

Electrodes → **Modify eroding path**

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

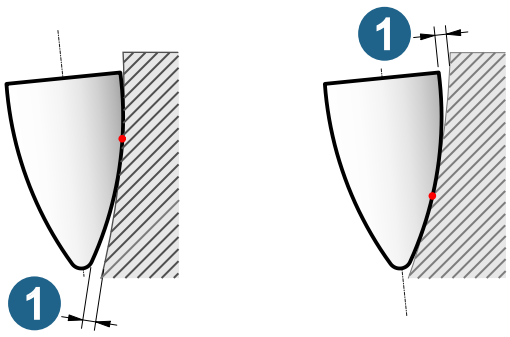
Optimize parameters of barrel cutters based on surface and curve information.

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.

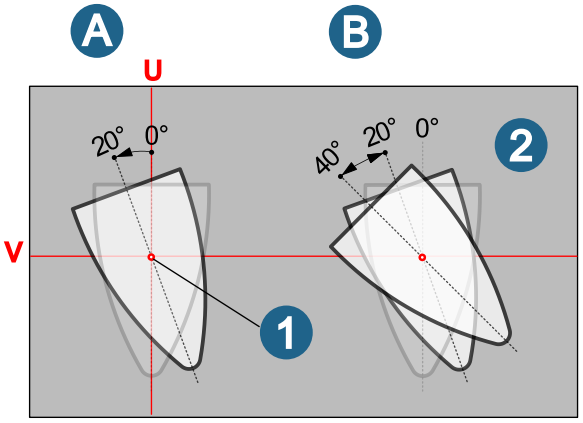


Lead angle

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**: Ⓐ: 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**: Ⓑ: 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.

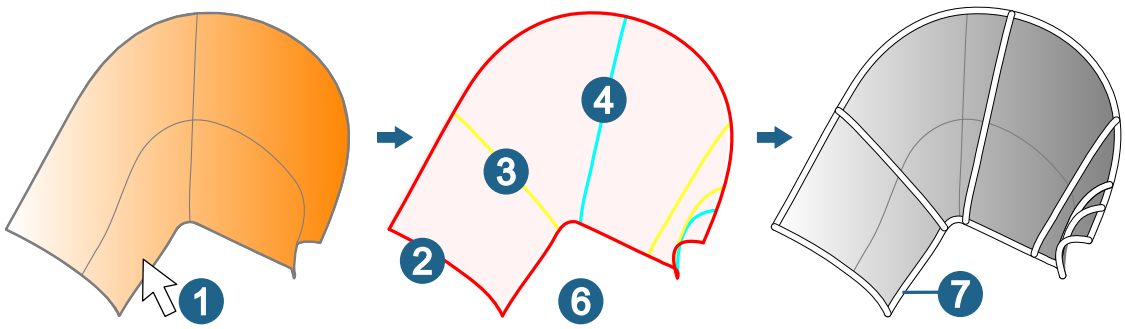
Create topology information

Generate topology curves for 3D finishing.

***hyper*MILL®-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

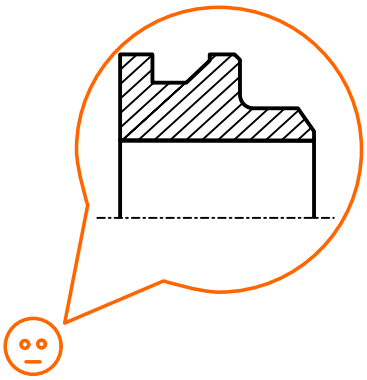
Create turning V-sketch

Create or change V-sketch for turning.

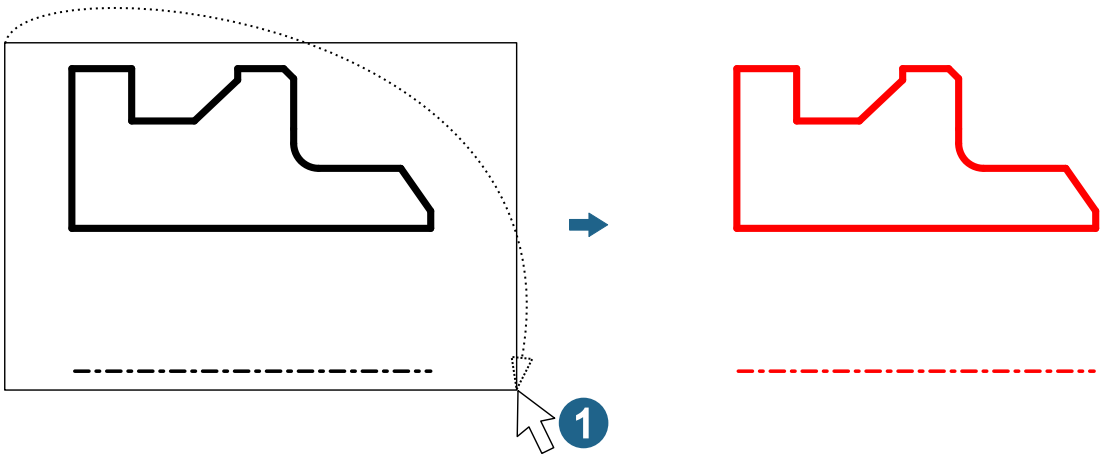
Drafting → Turning → Create turning V-sketch 2025

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.

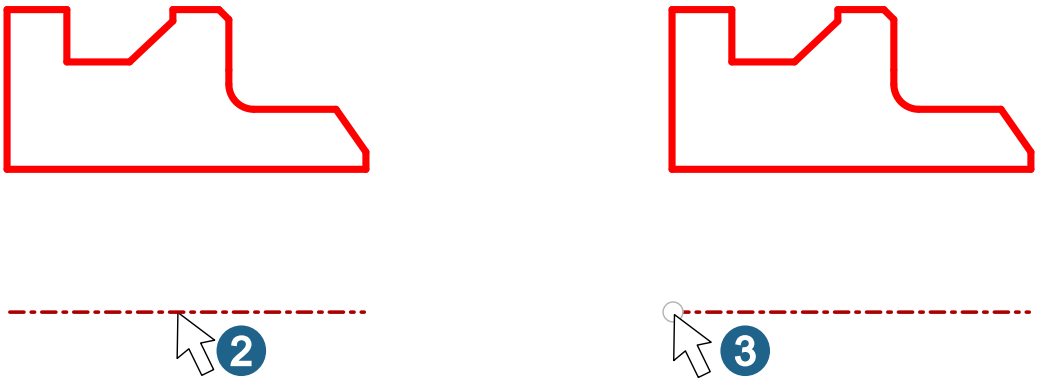


Rotation axis

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.

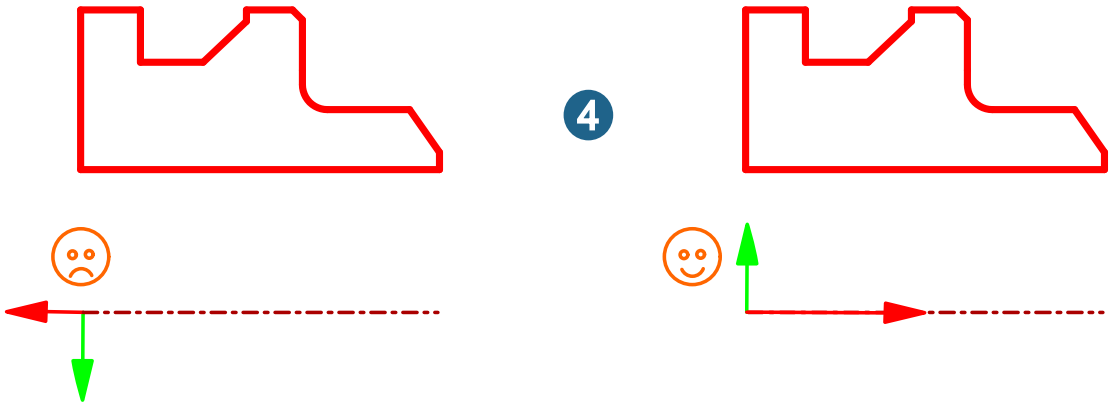
Origin: Snap the start, middle, or end point of the line ③.

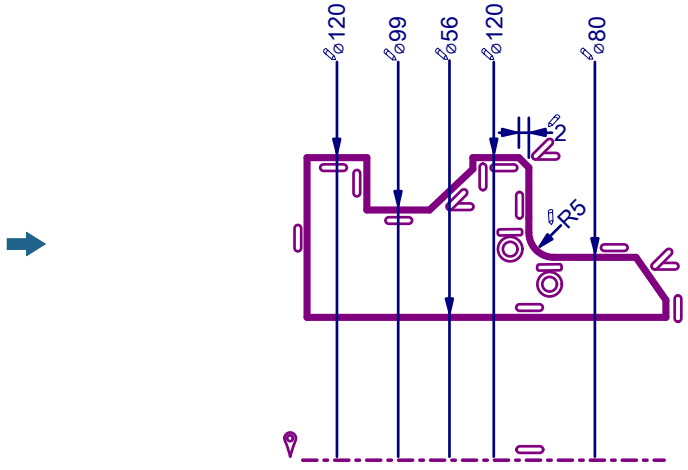


The orientation of the rotation axis and the X axis of the temporary coordinate system must match. The positive Y axis must point in the direction of the workpiece contour ④:

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

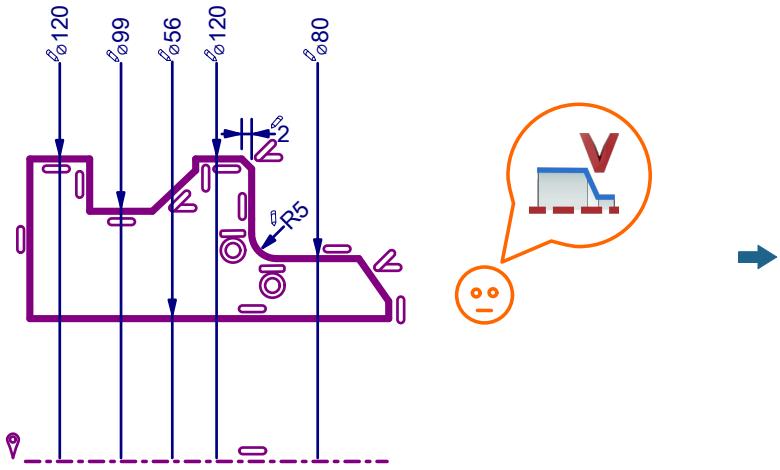
Angular tolerance: Enter a value for the limit angle up to which deviation lines are regarded as horizontal or vertical. If lines are within this tolerance, the lines are automatically corrected when the V-sketch is created so that they are actually horizontal or vertical. This value is independent of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



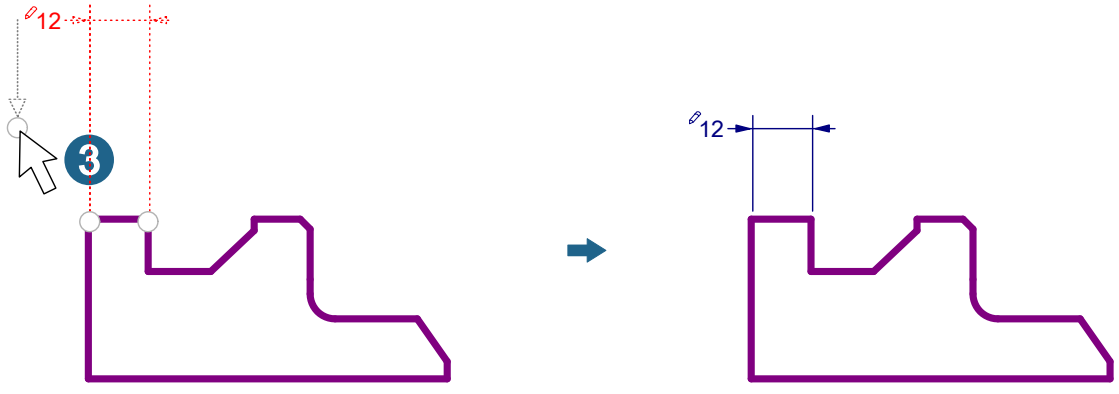
Select

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.



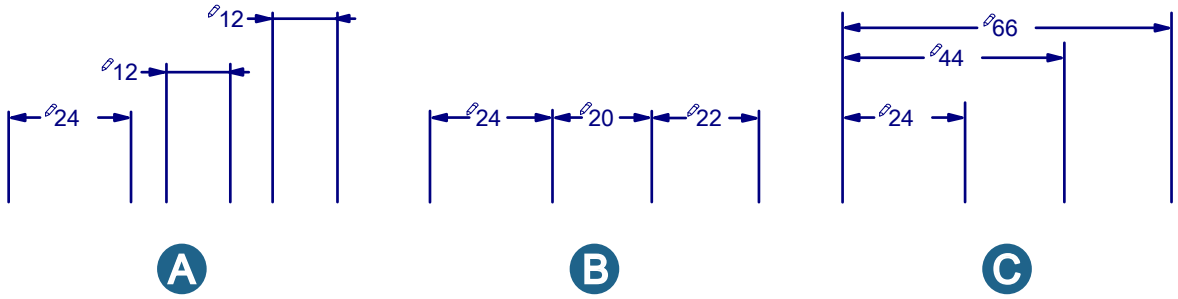
Modes

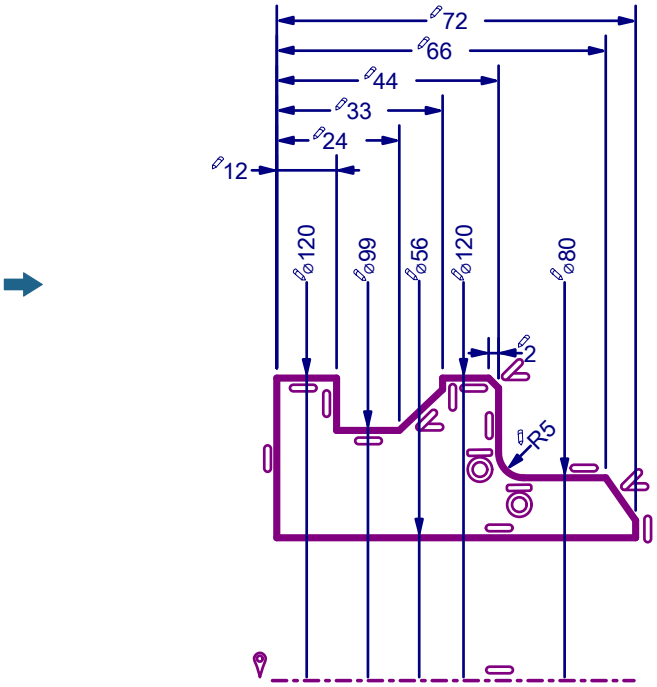
Select the dimensioning type:

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

Series: Generate several horizontal dimensional constraints as a chain dimension Ⓑ. 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 Ⓒ.





Modify linear tolerance

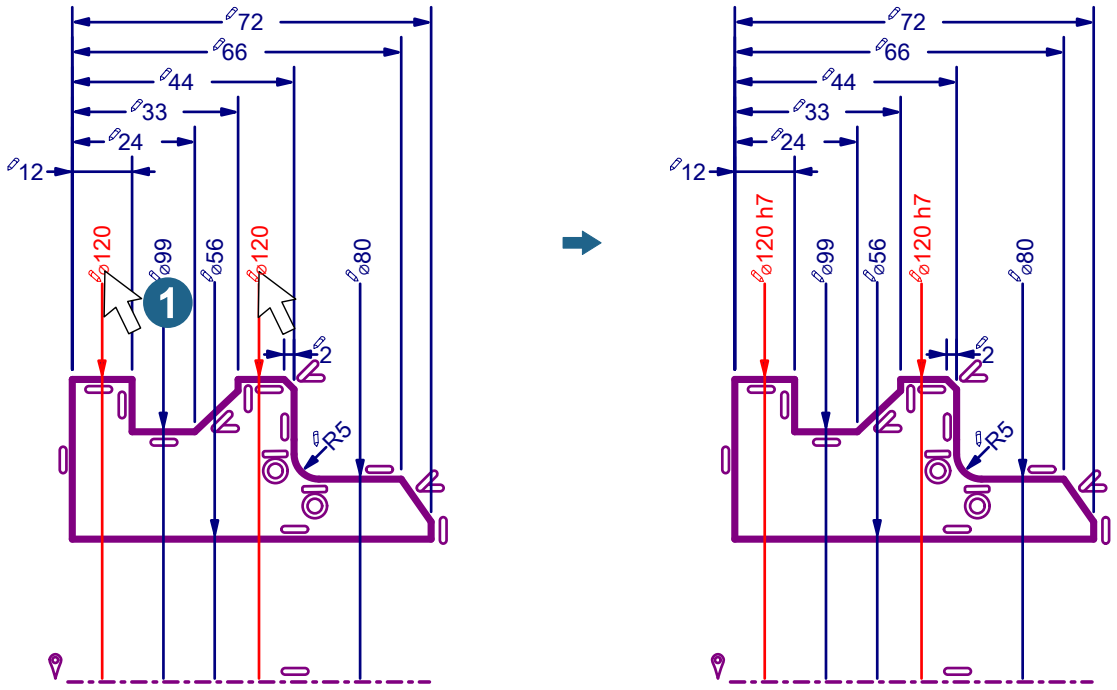
Adapt the geometry and display of linear tolerance intervals for turning operations.

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

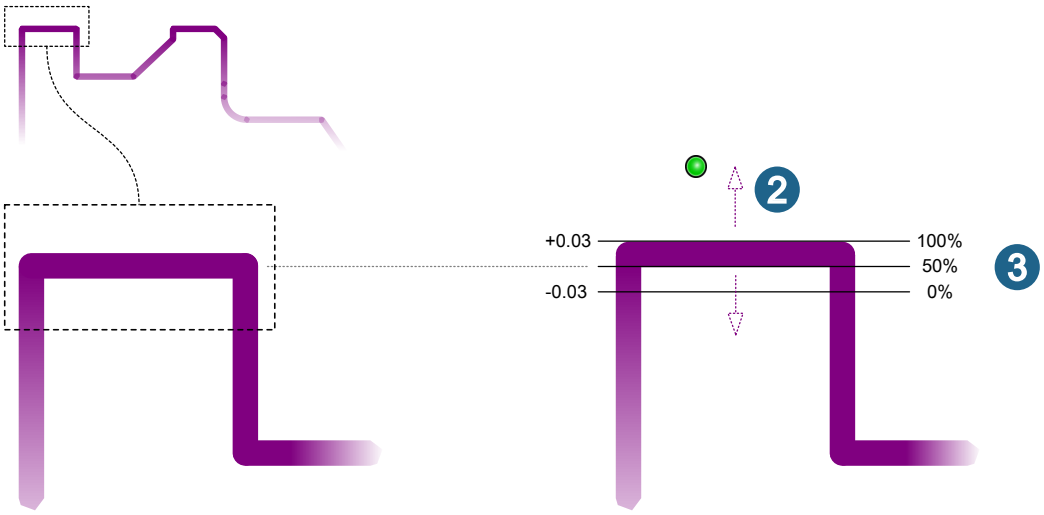
Select: Select one or more linear dimensions of a V-sketch for turning ①. The number of selected entities is displayed.



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

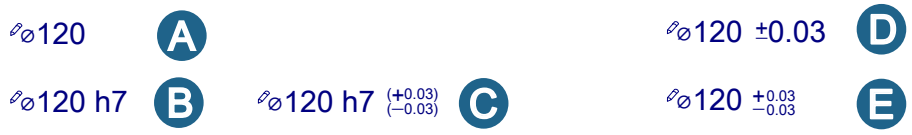
Specify the tolerance interval and select whether or how the deviations are to be displayed on the dimension:

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

ISO: Add an ISO tolerance specification Ⓑ. 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 Ⓒ.

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

Deviations: Add a tolerance specification where the lower and upper deviation are different Ⓔ. 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 date 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, *hyper*MILL® 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 **O**pen**M**ind**P**robing**EX**change**F**ileformat is generated as a text file named `*.log`. This format was designed so that 2D and 3D probing results can be imported to *hyper*MILL® 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.

PYTHONToolkit

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 [60].

Export: Export a script as a `*.zip` archive file. Export script. See Export [60]

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.

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

***hyper*MILL® 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.

***hyper*MILL® 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.

***hyper*MILL® VIRTUAL Machining / Turning**

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

***hyper*MILL® VIRTUAL Machining / Simulation**

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

***hyper*MILL® VIRTUAL Machining / Simulation**

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

***hyper*MILL® VIRTUAL Machining/ Simulation**

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

***hyper*MILL® VIRTUAL Machining / Simulation**

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

***hyper*MILL® VIRTUAL Machining / Simulation**

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

***hyper*MILL® 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.

***hyper*MILL® VIRTUAL Machining / Simulation**

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

***hyper*MILL® 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.

***hyper*MILL® 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.

***hyper*MILL® 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.

***hyper*MILL® VIRTUAL Machining / Collision check**

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

***hyper*MILL® 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.

***hyper*MILL® VIRTUAL Machining / OKUMA**

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

***hyper*MILL® 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.

***hyper*MILL® 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.

***hyper*MILL® 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 *hyper*MILL® could crash when using the **Edit** → **Move / Copy** function.

General

- An issue has been fixed where displaying toolpaths in *hyper*MILL® 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 calculation with 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 die 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.

AUTOMATION Center

- **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.