



hyperMILL®

hyperMILL®

Release notes 2025 | Update 3.1

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

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

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

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

OPEN MIND Technologies AG

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

Compliance Intelligence

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

(produced on Mon, Oct 13, 2025)



Table of Contents

1. Supported Versions	1
Operating systems and CAD platforms	1
CAD interfaces	1
Interfaces Tool database	3
Supported EDM formats	3
Supported OPTICAM Software Versions	4
Interfaces NC simulations	4
2. New commands and additions	5
CAM	5
User interface	5
hyperMILL and CAD program	5
The hyperMILL browser	5
Turning	6
Roughing	6
Grooving	7
TOOL Builder	7
Introduction	7
CAD	8
Introduction	8
Documentation and help	8
User interface	8
Tabs	9
Default settings	9
Options / properties	9
Sketcher options	9
Points, curves and faces	9
Drafting	9
Shapes	9
Parametric modeling	12
Automatic rebuild on / off	12
Solids, features and meshes	12
Features	12
Solid	14
Meshes	14
Design electrodes	14
Electrode options	14
Create partial	15
Create user defined	15
Modify eroding path	15
Using user-defined tags for electrode documentation	16
Export to EDM	17
Preparing generator programs	17
Analyzing probing results	18
Import probing data	18
3. Release notes	20
Release 2025 Update 1	20
CAM	20
Release 2025 Update 2	20
CAM	20
CAD	26
Release 2025 Update 3	28
CAM	28
CAD	31
Release 2025 Update 3.1	32
CAM	32
CAD	32



1. Supported Versions

Operating systems and CAD platforms

64-bit operating systems	Windows 10, Windows 11 *
64-bit CAD platforms	hyperMILL® Inventor 2024, 2025, 2026 SolidWorks 2023, 2024, 2025
Server operating systems (License server only)	Windows Server 2016, Windows Server 2019 Windows Server 2022
hyperMILL exclusively supports 64-bit operating systems. * Starting with Windows 11 version 24H2, we recommend at least hyperMILL® 2024 Update 3.	

CAD interfaces

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

Product	File type	Up to version	Import	Export
CATIA V4	*.model	4.2.5	x	
	*.exp	4.2.5	x	
CATIA V5	*.CATpart	2025	x	
	*.CATproduct	2025	x	
	*.CGR	2025	x	
CATIA V6	*.3dxml	2025	x	
PTC Creo Parametric	*.prt *.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	No version	x	x
	*.stla			
	*.stlb			
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 - <i>hyperMILL</i> (AME) (iM-HYP) Optional for 3D Tool data transfer: 3D-Solid Converter for <i>hyperMILL</i> (iCHYP)	TDM Systems - Base Installer TDM Systems - Data Installer TDM Application Server Installer TDM GlobalLine Interfaces Installer (for the Smart Interface Client <i>hyperMILL</i>)
Zoller TMS	<i>hyperMILL</i> v2 Interface first license TMS Tool Management Solutions	TMS Tool Management Solutions BRONZE package TMS Tool Management Solutions from version 1.17.0
WinTool AG	WinTool <i>hyperMILL</i> Interface	WinTool 2020 (WT2020.2.1) Microsoft Server 2012 or higher Microsoft SQL Server 2012 or higher <i>hyperMILL</i> Interface (2.13.5)
Hexagon Manufacturing Intelligence	NCSIMUL Tool NCT-CAM-HY (<i>hyperMILL</i> interface)	NCSIMUL Tool NCSIMUL Tool Client NCSIMUL Tool Server NCSIMUL Tool Interface FlexLM

Supported EDM formats

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



Manufacturer	Software	Version	Output 3-point path	Virtual electrode	Rotating electrode	Output 3D path
Exeron	Exoprog	1.0.0.0	x	x	x	
Makino		1.0.0.0	x	x	x	
ONA		1.0.0.0	x	x	x	
OPS Ingersoll	Multiprog	1.0.0.0	x	x	x	
Sodick		1.0.0.0	x	x	x	
Zimmer & Kreim	Alphamoduli	1.0.0.0	x	x	x	x
+GF+HMI	AC FORM HMI	1.0.0.0	x	x	x	

Changes or customization requests must be commissioned.

Supported OPTICAM Software Versions

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

<i>hyperMILL</i> ®	OPTICAM
2025 Update 3	2025.2
2025 Update 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

User interface

hyperMILL and CAD program








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

The hyperMILL browser

Jobs tab

Turning with turret

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

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

Turning

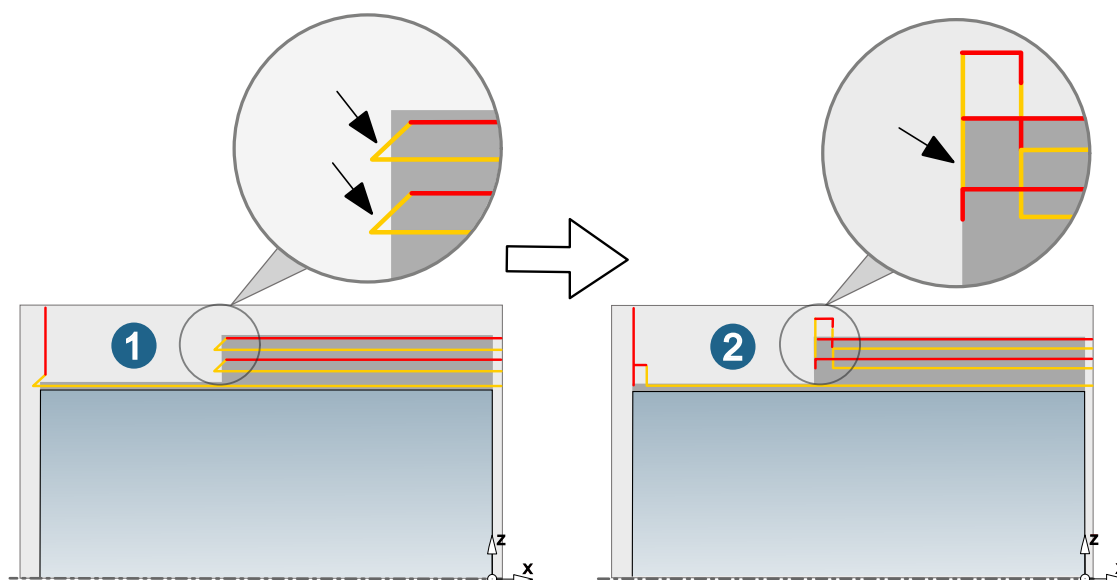
Roughing

Strategy

Options

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

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



High Performance

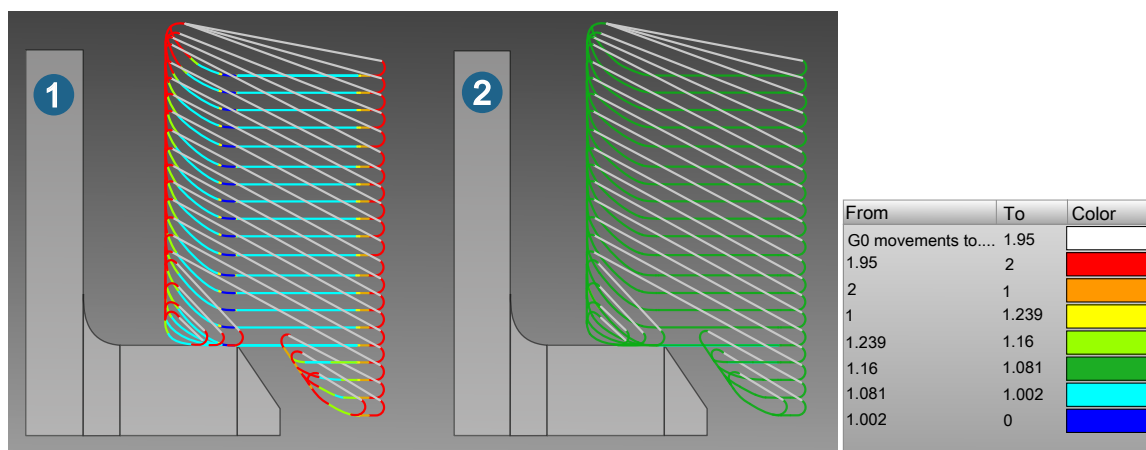
Chip thickness control

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

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

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

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

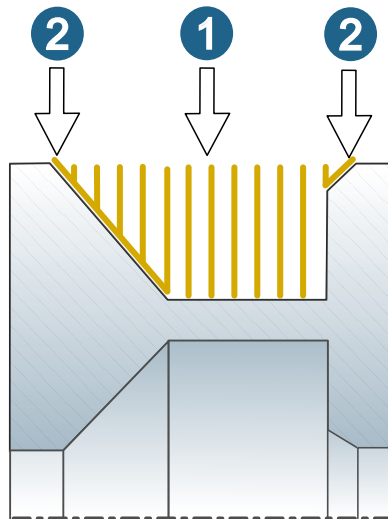


Grooving

Strategy

Options

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



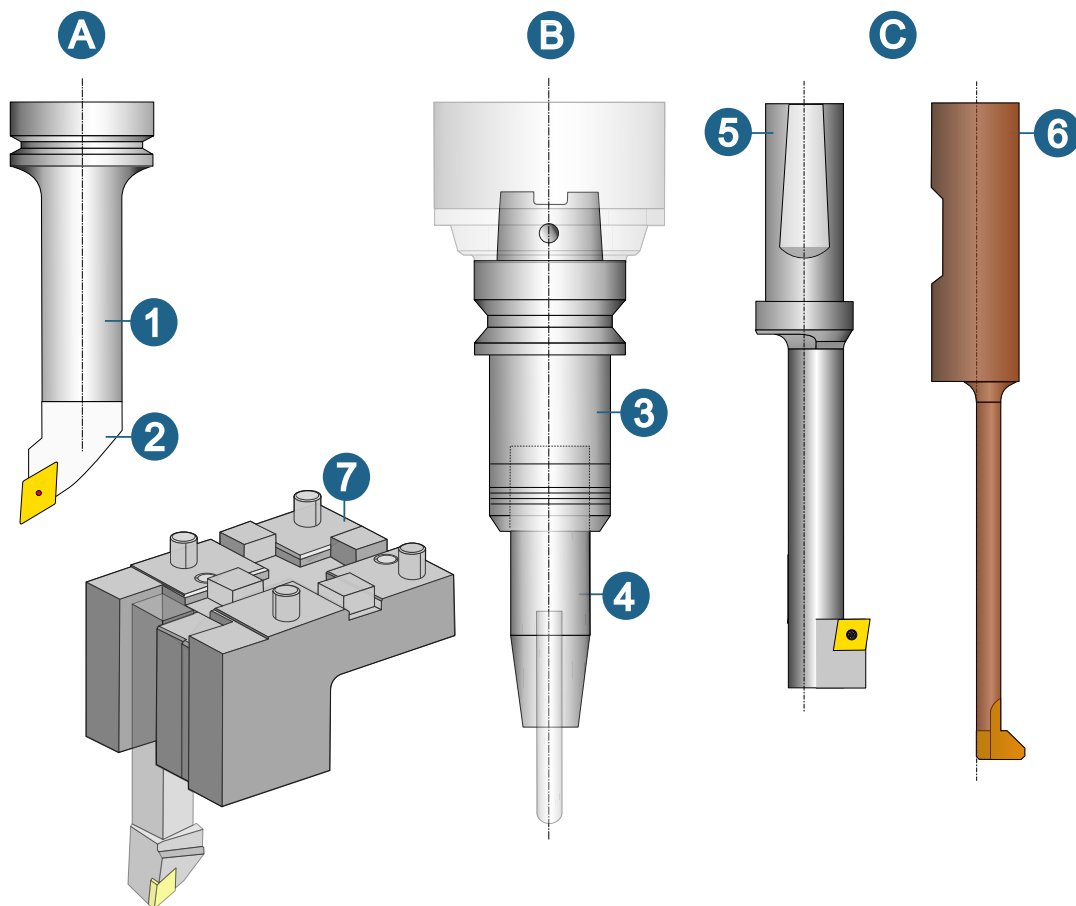
TOOL Builder

Introduction

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

The following tools and tool elements are supported:

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



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



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

CAD

Introduction

Documentation and help

Send to support



Send the current document to support.

Help → **Send to support**

2024

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

User interface



Tabs

Model

Assign entities to current layer

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

Default settings

Options / properties



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

File → Options → Options / properties

Graphic > System > Navigation

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

Sketcher options



Options for the Sketch and V-sketch commands

File → Options → Sketcher options

V-sketch options

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

Points, curves and faces

Drafting

Shapes

Sweep (two contours)



Generate side faces between two contours.

Shapes → Sweep (two contours)

2024

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

First contour

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

Second contour

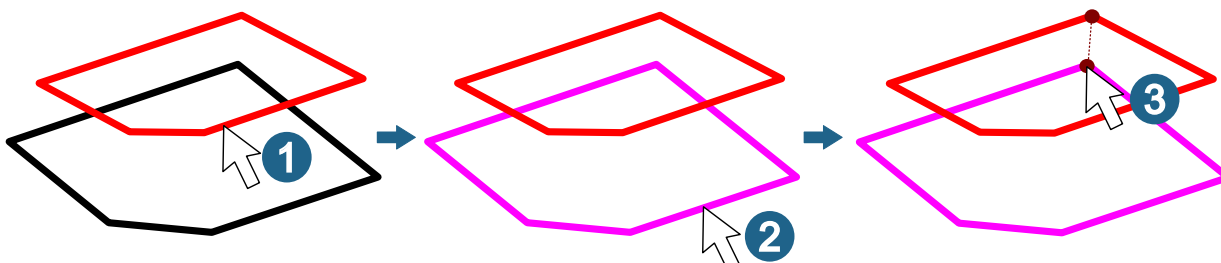
Curves: Select entities. The number of selected entities is displayed. All selected entities must complement each other to form a closed contour.

Synchronization points

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

First: Select point or snap position.

Second: Select point or snap position.



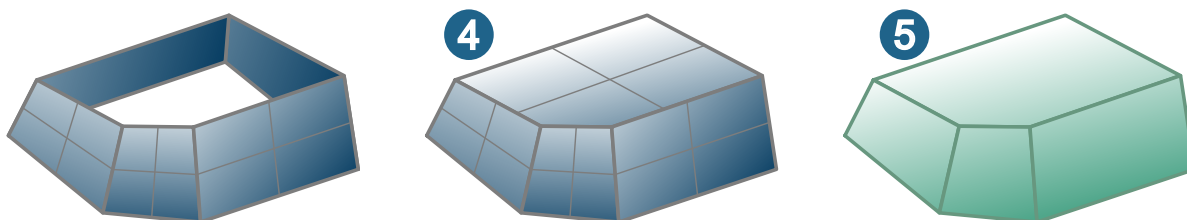
More options

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

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

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

Solid: Collect the faces into a solid.



From grid

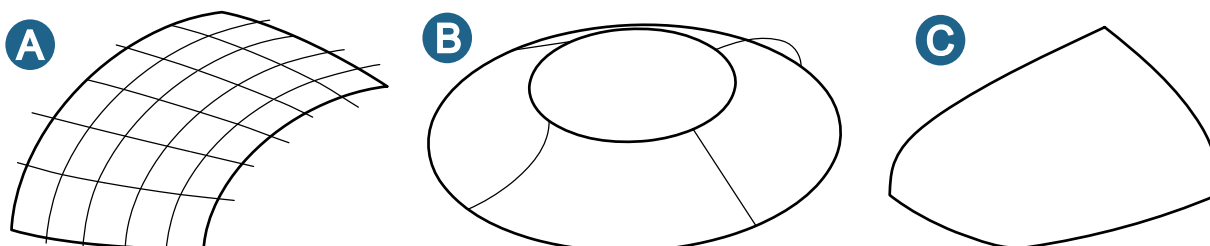


Create a face from several curve chains in one direction plus several curve chains in the other direction.

Shapes → From grid

2024

Create a face from several curve chains in one direction plus several curve chains in the other direction ①. The number of curves per chain can vary. The curve chains in both directions must intersect within a given tolerance. If curve chains protrude, the last intersection point is used for the edge boundary. The resulting protruding curve segments are not included in the face. The face can be created closed in one direction ②. Both directions can converge at the start or end of the face, or the face can just consist of three edge boundaries ③.



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

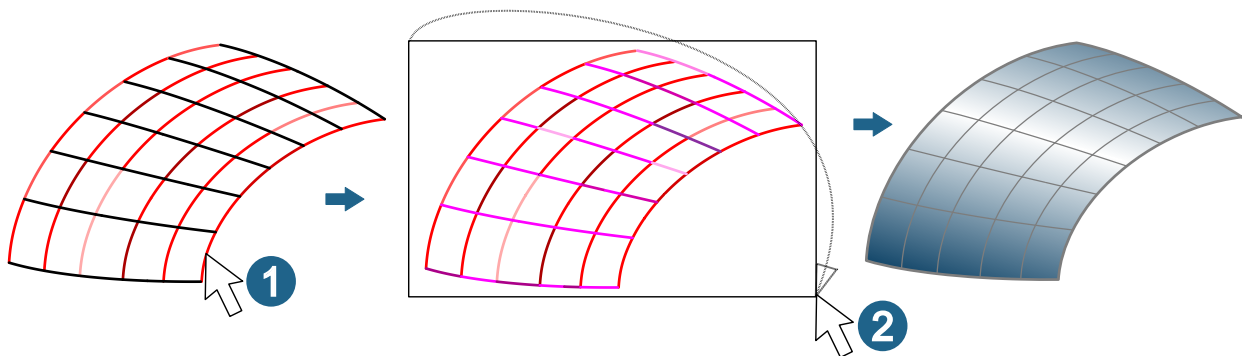
Click the message line for the “Most distant points” message on the **Info** tab to display the positions where this situation occurs.

First direction

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

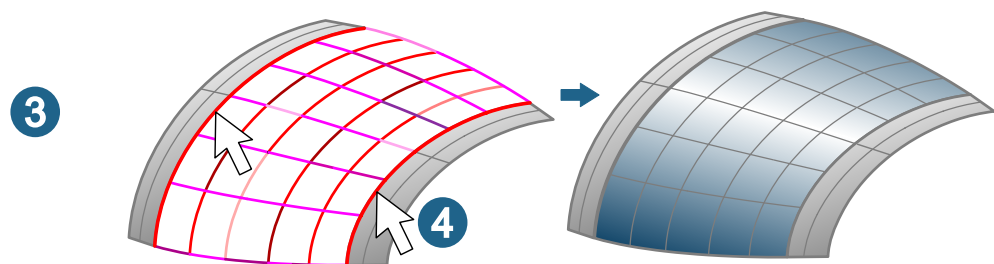
Second direction

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



Continuity

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



Intersecting curves

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

Precision

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

Parametric modeling

Automatic rebuild on / off



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

Edit → Parametric → Automatic rebuild on / off



Solids

2024

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



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

Solids, features and meshes

Features

Protrusion (two contours)



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

Features → Protrusion (two contours)

Solids

2024

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

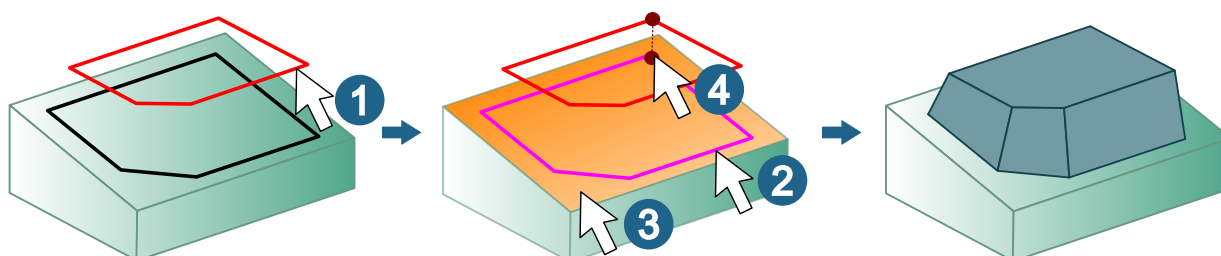
First contour

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

Second contour

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

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



Synchronization points

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

First: Select point or snap position.

Second: Select point or snap position.

More options

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

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

Slot (two contours)



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

Solids

Features → Slot (two contours)

2024

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

First contour

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

Second contour

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

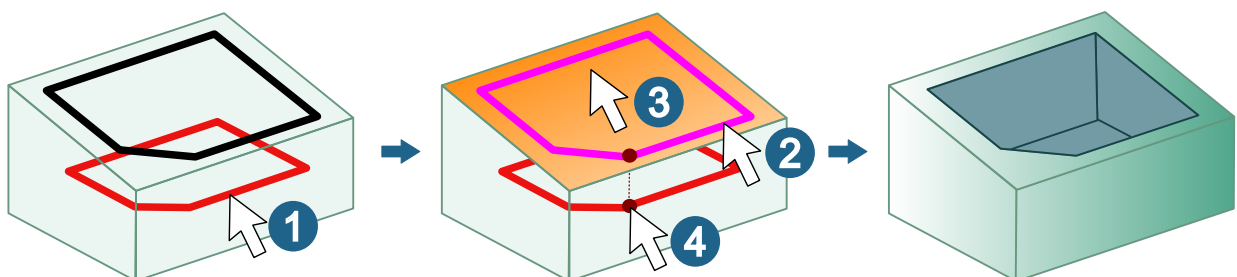
Face: Select a face of the existing solid in which to embed the slot to be generated ③.

Synchronization points

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

First: Select point or snap position.

Second: Select point or snap position.



More options

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

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

Solid

Repair open solid



Repair open solid.

Solids

Modify → Repair open solid

Cover openings

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

Meshes

Mesh from faces



Creates meshes from faces and faces in solids.

Solids

Shapes → Mesh from faces

More options

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

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

Keep attributes: Color attributes are retained.

Design electrodes

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

Electrode options



Select default settings for electrodes.

Electrode

File → Options → Electrode options

Geometry

Prepare

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



Technology

- **Generator program:** Select the required generator program. Displays all generator programs that are entered in the XML file for generator programs selected in the **XML file for generator program** option. Also see [Preparing generator programs \[17\]](#).

Folders

XML file for generator program: Select an `electrode_generator_programs.xml` file with generator programs for performance parameters for eroding. Also see [Preparing generator programs \[17\]](#).

Create partial



Create electrode geometries without an electrode block and electrode holder.

Electrode

Electrode → Create partial

More options

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

Create user defined



Extend existing electrode geometry to create a complete electrode.

Electrode

Electrodes → Create user defined

Mode

Default: Create an electrode with extensions.

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

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



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

Modify eroding path



Make eroding path visible and modify.

EDMconNG

Electrodes → Modify eroding path

2022.1

Blind



Motion

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

Through

Motion

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

Using user-defined tags for electrode documentation

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

For electrode assembly

Preparation in the title block template

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

In the electrode document

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

For single electrode

Preparation in the title block template

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



In the electrode document

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

Export to EDM



Convert electrodes into readable EDM files.

EDMconNG

Electrodes → **Export to EDM**

2024

Convert electrodes into readable EDM files.

If no part number is stored in the electrode options, the file name is used.

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.

EDM export

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

EDM file name

Enter a file name without a file extension.

Electrode list

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

Preparing generator programs

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

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

```
<program name="VDI24GAP10VDI18GAP08" description="EDM_example2" />
</GeneratorPrograms>
```

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

Analyzing probing results

Check the deviations based on the workpiece geometry in the graphics area, in tabular form, and as a trend. For 3D probing jobs, the deviation, meaning the actual state, is determined in the face normal direction of the probing point on the face (target state). In the graphics area, the target and actual states are displayed visually using colored dots with the deviation values. On the **Probing** tab, sort and filter the results by probing job, probing data, and trends. All values are output in relation to the current workplane. Click a probing point in the graphics area to jump to this point on the **Probing** tab.

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



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

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

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

The touch probe must be calibrated.

Standard logs, such as the standard used by CNC control manufacturer Heidenhain, cannot be processed.

Interaction with other commands

Probing data can be selected for other commands.

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

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

Import probing data



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

CAM → **Import probing data**

Probing tab → **Context menu** → **Import probing data**

2024



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

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

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

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

hyperMILL® VIRTUAL Machining / Siemens control

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

hyperMILL® VIRTUAL Machining / Turning

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

hyperMILL® VIRTUAL Machining / Simulation

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

hyperMILL® VIRTUAL Machining / Simulation

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

hyperMILL® VIRTUAL Machining/ Simulation

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

hyperMILL® VIRTUAL Machining / Simulation



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

hyperMILL® VIRTUAL Machining / Simulation

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

hyperMILL® VIRTUAL Machining / Simulation

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

hyperMILL® VIRTUAL Machining / Simulation

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

hyperMILL® VIRTUAL Machining / Post processing

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

hyperMILL® VIRTUAL Machining / Collision check

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

hyperMILL® VIRTUAL Machining / Collision check

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

hyperMILL® VIRTUAL Machining / Collision check

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

hyperMILL® VIRTUAL Machining / Siemens control

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

hyperMILL® VIRTUAL Machining / OKUMA

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

hyperMILL® VIRTUAL Machining / NC Generator

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

hyperMILL® VIRTUAL Machining

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

hyperMILL® VIRTUAL Machining



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

SIMULATION Center

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

SIMULATION Center

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

Tool database

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

Tool database / Hummingbird connection

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

Drilling / Tap

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

Drilling / Thread Milling

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

Turning / Grooving

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

2D Contour Milling on 3D Model

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

2D Contour Milling on 3D Model

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

2D Contour Milling on 3D Model

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

3D Optimized Roughing

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

3D Optimized Roughing



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

3D Optimized Roughing

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

3D Optimized Roughing

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

3D Optimized Roughing

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

3D Optimized Roughing

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

3D Optimized Roughing

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

3D Optimized Roughing

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

3D Optimized Roughing

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

3D Optimized Roughing / 3D Arbitrary Stock Roughing

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

3D Profile Finishing

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

3D Complete Finishing

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

3D Plane Machining

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



3D Plane Machining

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

3D Equidistant Finishing

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

3D Iso Machining

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

5X Profile Finishing

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

5X Rework

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

5X Tube Roughing

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

Stock calculation / Mirrored job list

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

Automatic stock chain

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

General / Consistency check

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

General / Coolant setting

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

General / Job definition

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

Inventor integration



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

Toolpath Edit

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

Edit → Move / Copy

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

General

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

Macro technology

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

General

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

Updating recommended

SIMULATION Center

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

2D Contour Milling

- An issue has been fixed where the 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/Media/Dateien/Downloads/SETUP_OPTICAM_V2025_2_HCNT.zip).

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

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

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

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



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

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

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.

Release 2025 | Update 3

CAM

Additions and notes

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

The **Standard** option follows this order:

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

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

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

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

Bug fixes

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



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



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



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

CAD

Additions and notes

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

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

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

Bug fixes

The following issues have been resolved:

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



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

Release 2025 | Update 3.1

CAM

Bug fixes

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

CAD

Additions and notes



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