



hyperMILL®

2024

hyperMILL 2024

WHAT'S NEW?

 **OPEN MIND**
THE CAM FORCE



hyperMILL® as Innovative CAD/CAM Complete Solution

OPEN MIND has always offered an innovative CAD/CAM solution whose CAD functionalities are seamlessly linked to CAM programming. This leads to enormous time savings in workpiece preparation. This alone makes it clear that CAM without CAD is no longer an option today. From version 2024, hyperMILL® now combines CAD and CAM under one name (instead of hyperCAD®-S), thus consolidating “CAD for CAM” for the future. The familiar CAD functionalities remain the same, only their names change.

By separating topics into the new sections – CAD, CAM, and Technology – we hope to provide an even better overview of our variety of products.

CONTENTS

3-4

CAD

Import model-based definition (MBD)

Create faces from grids

Invert curve normals

Sweep with two contours

Creation of three-dimensional electrode paths

Electrode generation for open surfaces

User-defined title block

Extension for user-defined electrodes

5-11

CAM

Chamfer Milling on 3D Model

Optimized Deep Hole Drilling

3D Automatic Rest Machining

3D Cutting Edge Machining

3D Plane Machining

3D Z Level Shape Finishing

5-axis path compensation for surface strategies

5-axis Multiblade Fillet Milling

5-axis Rest Machining

5-axis Radial Machining

5-axis Halfpipe Machining

Read back measuring points

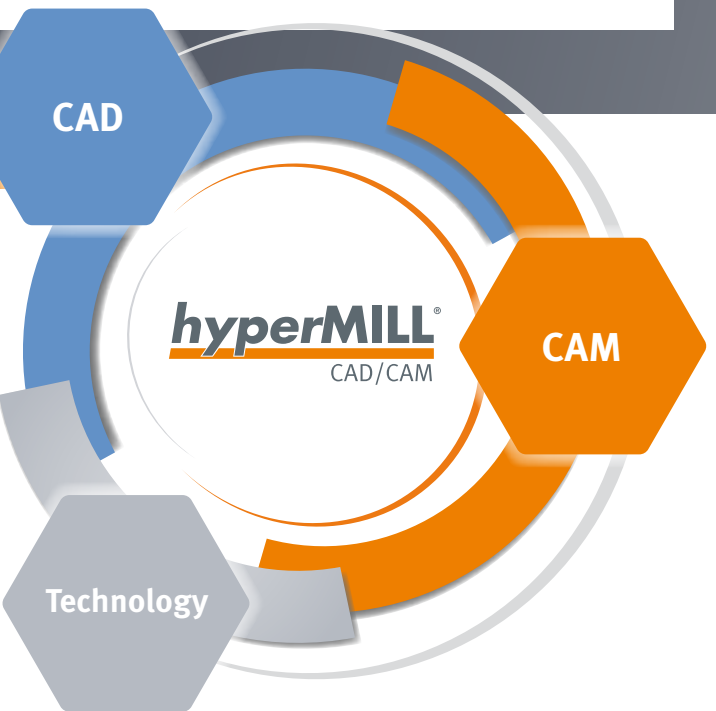
CAD functionalities for turning operations

Finishing path for grooving

Stretch 2D

Roughing – Remove rings

Turret support for lathes



12-15

TECHNOLOGY

Improvements for MILL-TURN machines

Tool breakage check

CONNECTED Machining for Fanuc controls

Milling with rotation axis

3D/5-axis path compensation

Performance improvements

Supported controls

Restrict input during user guidance

Rest material display

Tool synchronization with Hummingbird MES

New tool type: Gun drill

Programming assistance: CAM Plan

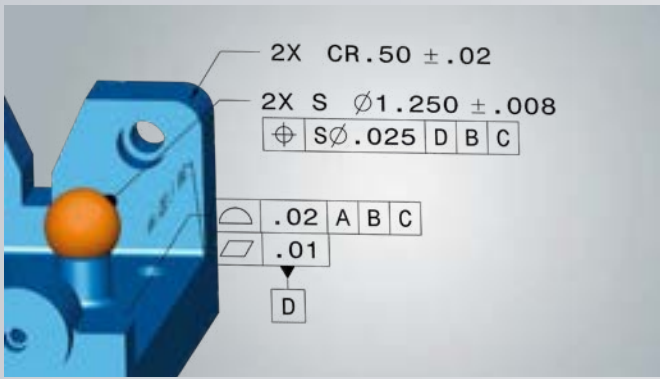
Review system compatibility: To ensure optimal performance and stability, we recommend regularly running our diagnostic program, Systemchecktool.exe.

Note: Windows® may reset the graphics driver or its settings when carrying out updates.

System requirements: Windows® 10/11 (64-Bit) | **CAD integrations:** Autodesk® Inventor®, SOLIDWORKS

Software languages: de, en, es, fr, it, nl, cs, pl, ru, sl, tr, pt-br, ja, ko, zh-cn, zh-tw

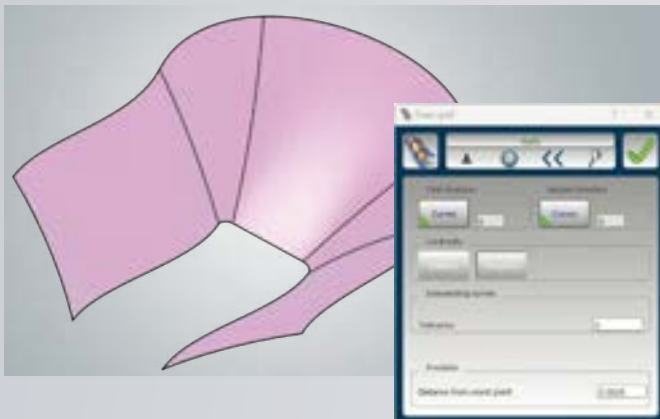
HIGHLIGHTS



Import model-based definition

hyperMILL® supports the import of PMI and MBD data in various formats such as STEP, CATIA V5, SOLIDWORKS, Creo, and Siemens NX. A model-based definition (MBD) is assigned to faces, and PMI symbols are assigned to dimensions, tolerances, and surface finishes. This data can be evaluated with the help of the AUTOMATION Center for faster processing.

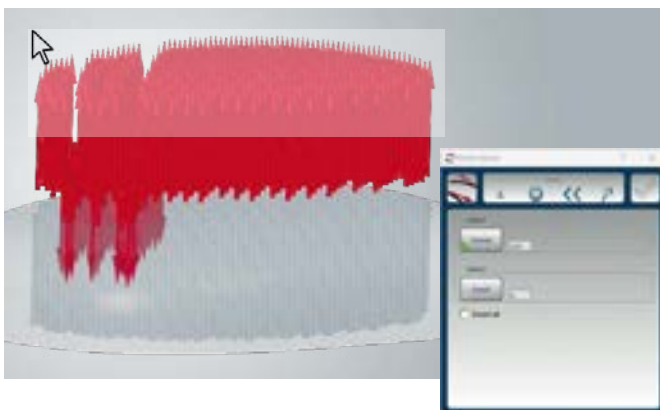
Benefit: Model data available during CAM processing.



Create faces from grids

hyperMILL® now offers the option of generating both open and closed faces from a variety of grid curves. Even non-intersecting curves are taken into account within a tolerance, allowing faces to be generated effortlessly, even in the most complex areas.

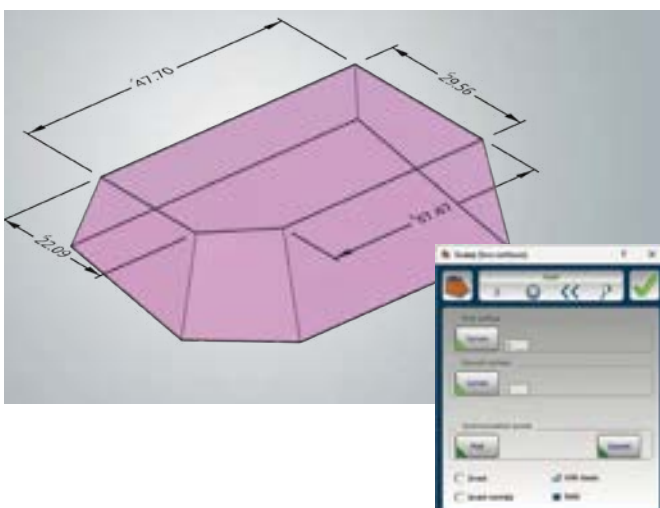
Benefit: Simple face creation for milling or 3D modeling.



Invert curve normals

You can easily select all zoom-independent arrows to reverse the directions of the curves as required. This function saves you a lot of work, especially if a machining direction is specified for thousands of elements.

Benefit: Improved user-friendliness.



Sweep with two contours

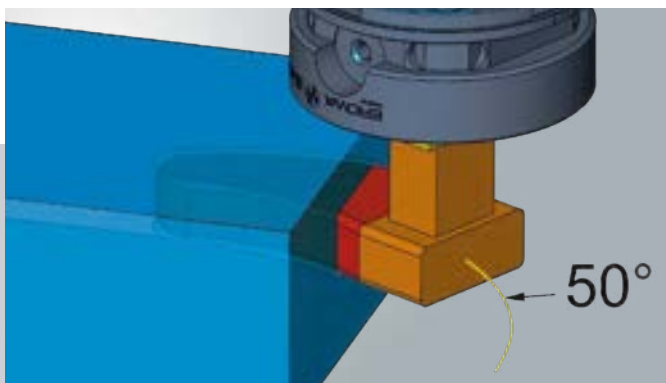
There are now extended design options for faces, solids, and features thanks to the option to select two contours in the sweep operations.

Supported functions:

- Sweeps
- Protrusions
- Grooves

Benefit: Simple design.

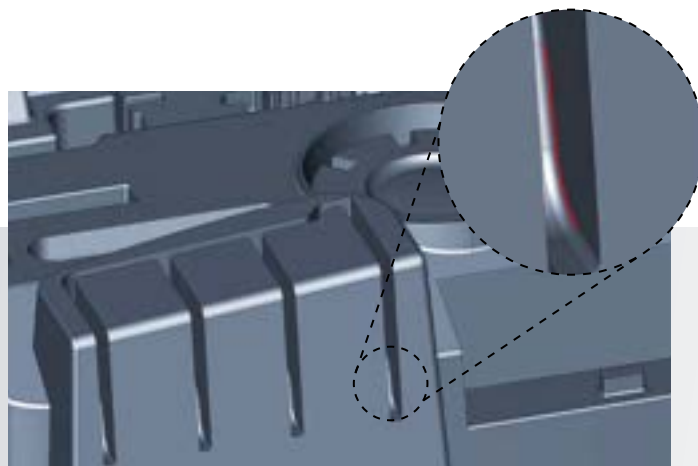
HIGHLIGHTS



Creation of three-dimensional electrode paths

Two-dimensional machining is no longer sufficient for some erosion processes. With *hyperMILL*® Electrode, you can now generate machining paths along a 3D curve with simultaneous C-axis rotation. Retract movements are automatically created in the opposite direction to ensure an efficient machining process.

Benefit: Simple erosion process for complex electrodes.



Electrode generation for open surfaces

Deviations in faces and tolerance values often make it very time-consuming to create electrodes. *hyperMILL*® Electrode simplifies this process by allowing electrodes to be created even with open or overlapping faces.

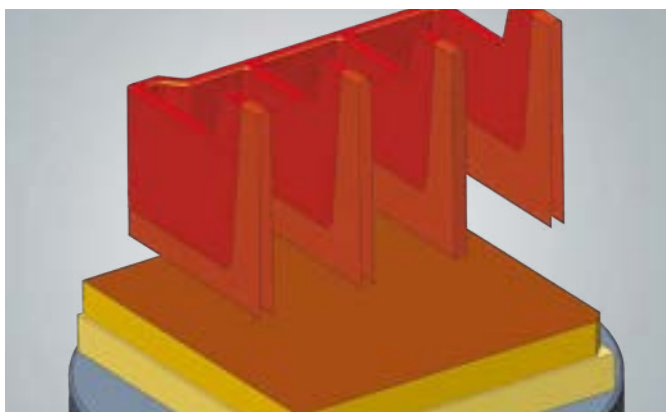
Benefit: Improved user-friendliness.

Company OPEN MIND Technologies AG	Model Standard Tool Holder	Project no. 88871213	Comments Tap stamping
Part number 2024_0002	Description Electrodes for slide	Part number 1.0718.1000	Stocking program E2M E202
hyperMILL Electrode	Manufacturing date 2024-01-25	Version 0002	Stock number 0001
	Created by Mh	Created date 2024-01-15	Released from
	Document name 2012-0332	Can saving date 2024-01-30 15:34	

User-defined title block

hyperMILL® Electrode now offers you the opportunity to customize electrode title blocks. This extension makes it possible to systematically document further relevant manufacturing information, and helps to make processes more reliable.

Benefit: Documentation including user-defined information.



Extension for user-defined electrodes

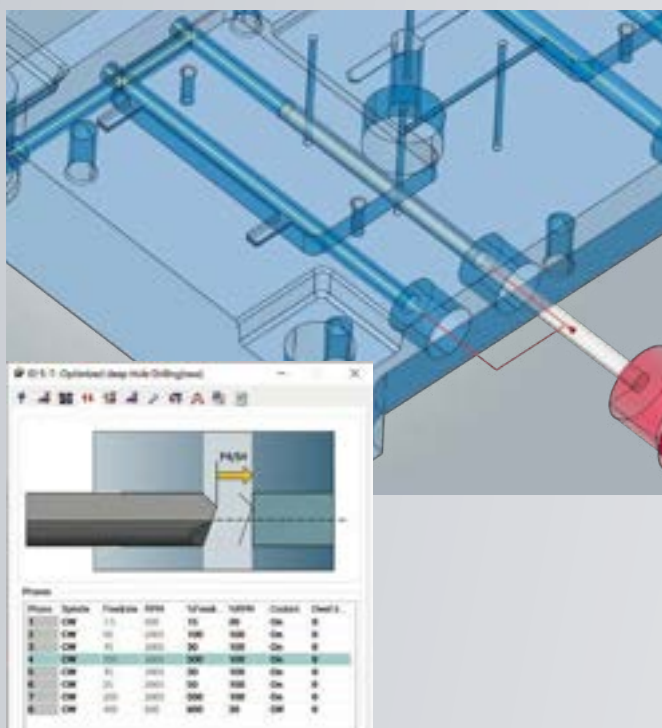
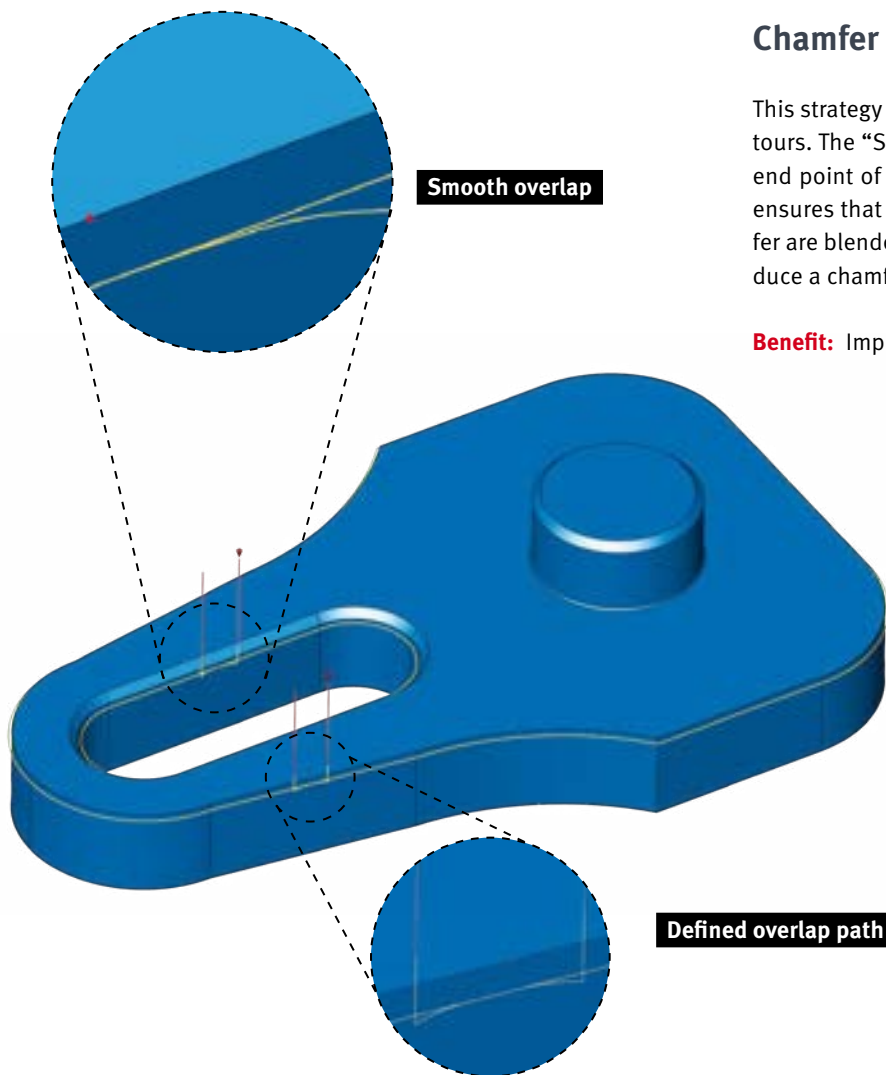
When creating user-defined electrodes, a selected face area can now be adopted as an electrode using the extended “Simplified” function – without any additional machining. This allows you to generate electrodes very quickly and without any unnecessary effort.

Benefit: Creation of electrodes with minimal effort.

Chamfer Milling on 3D Model

This strategy allows an overlap path to be defined for closed contours. The “Standard” option defines an overlap at the start and end point of the chamfer. The second option “Smooth overlap” ensures that the approach and retract movements on the chamfer are blended smoothly. These options make it possible to produce a chamfer without visible approach and retract marks.

Benefit: Improved control and quality during chamfer machining.



HIGHLIGHT

Optimized Deep Hole Drilling

We have revised the deep hole drilling process and developed a new strategy. A user-friendly interface makes programming easier, with all process-relevant parameters now clearly displayed on the new process tab. The new strategy offers all the functions required for safe and reliable deep hole drilling. Coolant and dwell time can now be defined for each stage or phase of the deep hole drilling process. New functions also make it possible to integrate a chip break into the drilling process. Thanks to the various parameters, the drilling process can be perfectly adapted to your machining. This increases process reliability. Optionally, programming can be carried out independently of the stock, which is a particular advantage when there is extensive model data. In parallel with the new strategy, we have introduced a new type of tool, namely, the single-lip gun drill. The simulation includes a precise collision check and detailed visualization of the stock removal.

Benefit: Improved programming of deep hole drilling tasks, reliable machining.

HIGHLIGHTS

3D Automatic Rest Machining

A new algorithm for rest material detection ensures complete detection of all rest material areas. In addition to this detection, we have also optimized the algorithms for path calculation. Toolpaths are now optimally divided to ensure more efficient machining. The detection of intersection areas where paths meet has also been improved. Thanks to a new toolpath layout, the rest material in these areas is machined perfectly.

Benefit: Improved machining of rest material areas.

3D Cutting Edge Machining

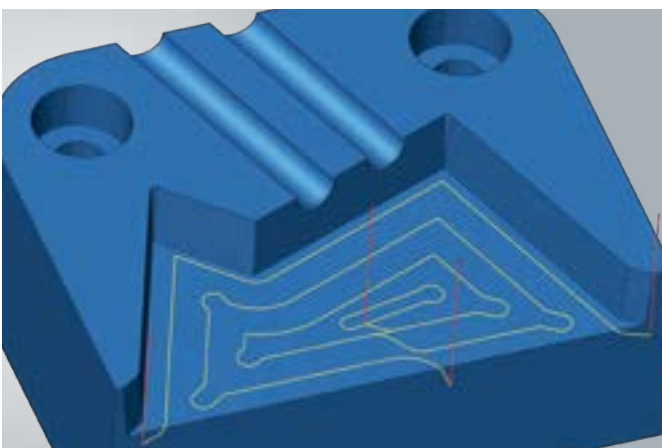
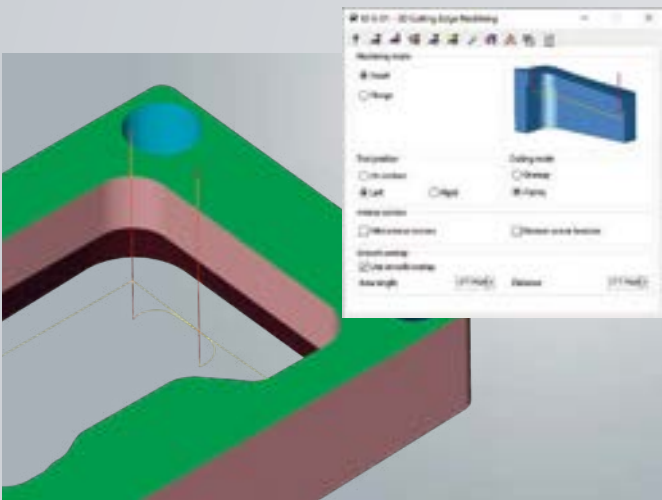
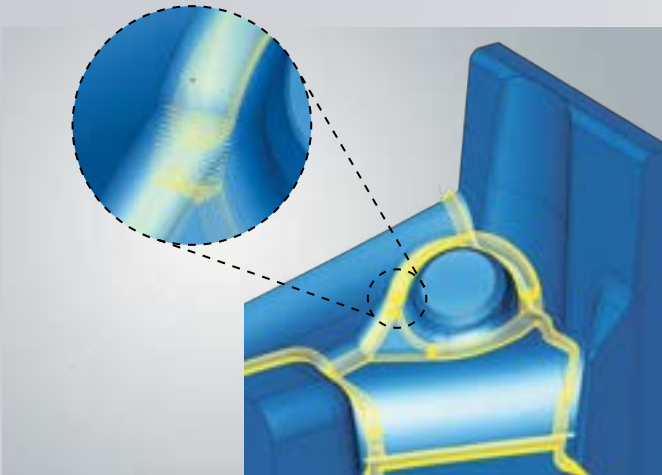
This strategy offers you new functions and a large number of improvements. These include optimized collision avoidance that works on the basis of a reference job during machining. Machining is carried out with as little collision risk as the tool reach allows. The “Smooth overlap” option offers the opportunity to smoothly blend the entry and retract moves with the surface and thus almost completely avoid visible approach and retract marks. Zigzag machining can now be selected for the “Plunge” machining mode, and cutter radius compensation can also be used. We have also revised the user interface for this strategy and arranged all important functions on a new strategy tab.

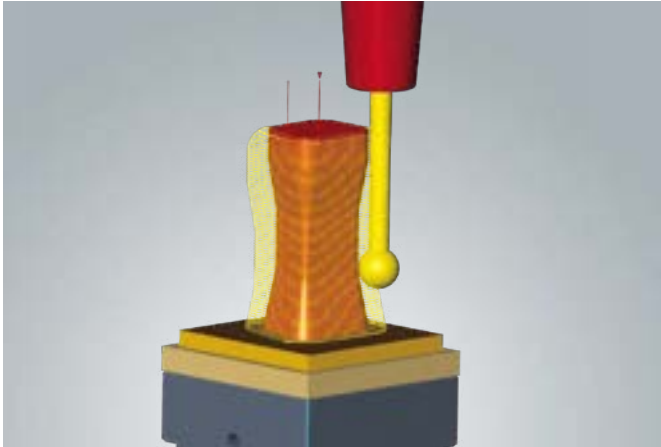
Benefit: More extensive machining options, more user-friendly programming.

3D Plane Machining

A new algorithm ensures improved tool path calculation. The toolpaths are now designed to be smoother, resulting in more uniform and faster machining. This increases tool life and reduces the machining time on the machine.

Benefit: Faster and more tool-friendly machining.





3D Z Level Shape Finishing

The strategy now allows undercuts to be machined with lollipop and woodruff cutters. The entire geometry or just the individual undercut area can be machined. The toolpaths are generated with collision checking, which guarantees reliable machining. This means that 3-axis machines can also be used safely for undercut machining, such as for electrodes or other clearances.

Benefit: Reliable undercut machining on 3-axis machines.

```

56 LN X-32.33945 Y290.99192 Z45.7707 NX0 NY0.9848078 NZ0.1736482 TX0
TY-0.173648 TZ0.984808
57 LN X-32.35927 Y290.92474 Z45.75885 NX0 NY0.9848078 NZ0.1736482
TX0 TY-0.173648 TZ0.984808
58 LN X-32.37593 Y290.86069 Z45.74756 NX0 NY0.9848078 NZ0.1736482
TX0 TY-0.173648 TZ0.984808
59 LN X-32.38948 Y290.79969 Z45.7368 NX0 NY0.9848078 NZ0.1736482 TX0
TY-0.173648 TZ0.984808
60 LN X-32.40002 Y290.74166 Z45.72657 NX0 NY0.9848078 NZ0.1736482
TX0 TY-0.173648 TZ0.984808
61 LN X-32.40761 Y290.68653 Z45.71685 NX0 NY0.9848078 NZ0.1736482
TX0 TY-0.173648 TZ0.984808
62 LN X-32.41233 Y290.63423 Z45.70762 NX0 NY0.9848078 NZ0.1736482
TX0 TY-0.173648 TZ0.984808
63 LN X-32.41426 Y290.58468 Z45.69889 NX0 NY0.9848078 NZ0.1736482
TX0 TY-0.173648 TZ0.984808
64 LN X-32.41346 Y290.53781 Z45.69062 NX0 NY0.9848078 NZ0.1736482
TX0 TY-0.173648 TZ0.984808
65 LN X-32.41003 Y290.49355 Z45.68282 NX0 NY0.9848078 NZ0.1736482
TX0 TY-0.173648 TZ0.984808
66 LN X-32.40402 Y290.45181 Z45.67546 NX0 NY0.9848078 NZ0.1736482
TX0 TY-0.173648 TZ0.984808
67 LN X-32.39553 Y290.41252 Z45.66853 NX0 NY0.9848078 NZ0.1736482

```

5-axis path compensation for surface strategies

5-axis path compensation* allows the machine operator to make fine corrections on the machine control. Vectors for the cutter contact point are written to the NC program. The NC control uses these contact vectors to shift the NC points by an entered correction value during machining. This makes it possible for the first time to correct dimensions – such as fits – during 5-axis movements on the machine.

Available for the following cycles, among others:

- 5-axis Swarf Cutting
- 5-axis Tangent Plane Machining
- 5-axis Tangent Milling

Benefit: 3D path compensation for 5-axis toolpaths.

*Currently, 5-axis path compensation is only supported for Heidenhain controls and requires an adapted postprocessor.



5-axis Multiblade Fillet Milling

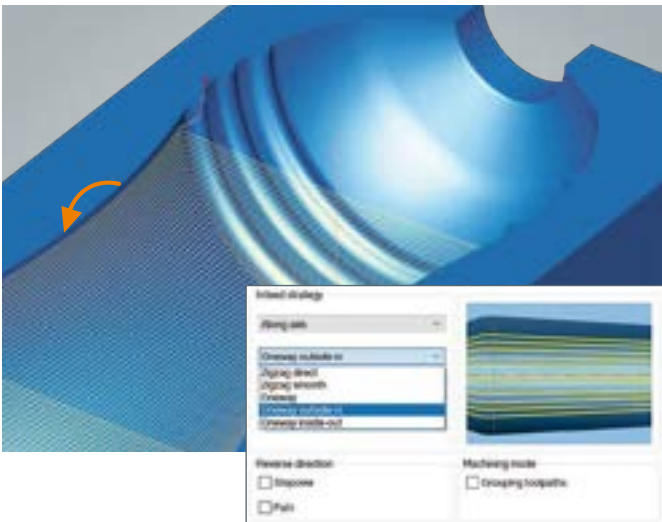
We have revised and improved the calculation mode for this strategy. It is now more stable, especially with strongly inclined and/or curved blades and asymmetrical splitter blades.

Benefit: Improved performance and stability.

HIGHLIGHT**5-axis Rest Machining**

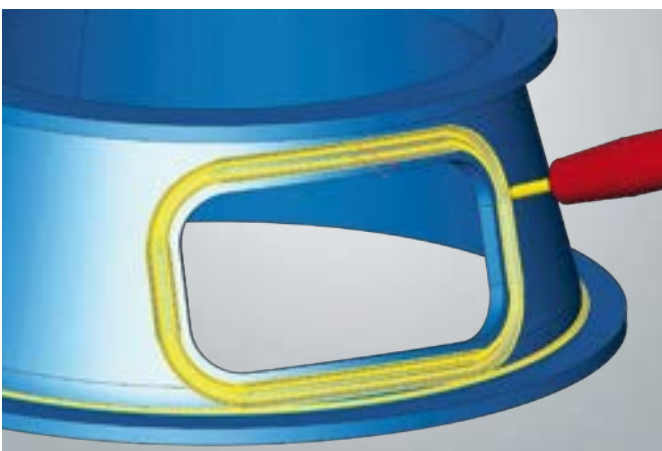
We have revised and improved this strategy from scratch. A new algorithm for rest material detection ensures comprehensive detection of all rest material areas. In addition to the new rest material detection, we have also updated the functions for indexed approach calculation and path calculation. This is reflected in a faster calculation time and better approach calculation for the automatic 5-axis “Indexing” mode. The optimized detection of intersection areas where paths meet, combined with a new tool-path layout, ensures perfect machining of the rest material.

Benefit: Improved machining of rest material areas, simplified 5-axis programming.

**5-axis Radial Machining**

Two new infeed strategies allow machining in a uniform direction across the entire component geometry. The “Direction constant – from outside to inside” and “Direction constant – from inside to outside” options can be used to enable conventional or climb milling for the entire cavity. In addition to simplified programming, direction-constant milling is particularly necessary when machining hard or tough materials, and can now be programmed with just one machining job.

Benefit: Simplified programming and definition of direction-constant machining.

**5-axis Halfpipe Machining**

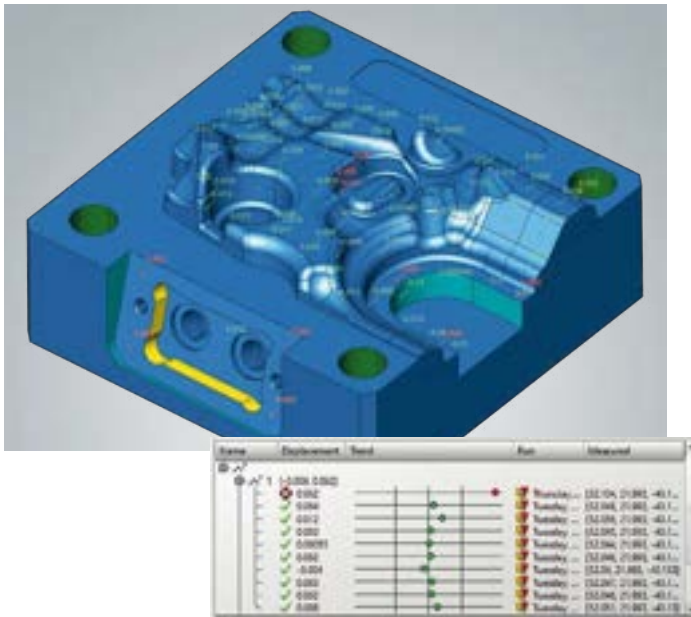
We have added the following functions to 5-axis Halfpipe Machining:

Contact mode: Similar to other machining strategies, the milling path ends as soon as the tool touches the boundary on the surface.

“Endless” geometry type: A new geometry type for endless, strip-shaped geometries is now available with various infeed strategies. This is useful for machining seals, for example, or for circulating rest material machining.

Grouping toolpaths: As with other machining strategies, for example ISO machining, paths can be created with equal distances or along the lower central curve in the case of halfpipes that are open on both sides or are endless.

Benefit: Improved user-friendliness, broader range of applications.

**HIGHLIGHT****Read back measuring points**

Measuring points can now be read back in *hyperMILL*® to ensure and log the component quality. You can see at a glance which measuring points are within or outside the tolerance both on the 3D model and on the “Measured” panel. This allows you to analyze inaccuracies, tool wear, or deviations/trend after milling and compensate for them simultaneously on the CAD and CAM side. This saves time, provides security, and increases quality. This new function can also be used directly on the machine tool with the *hyperMILL*® SHOP Viewer. In addition, reading back the points can be used in conjunction with *hyperMILL*® BEST FIT to visualize the results of the new alignment.

Benefit: Improved quality and process control.

hyperMILL® VIRTUAL Machining postprocessor required. Available controls on request.

Safely Generate, Optimize and Simulate NC Code

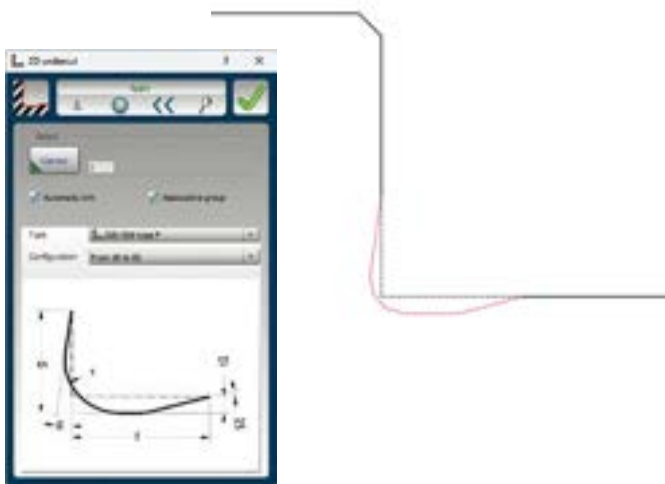
hyperMILL®
VIRTUAL Machining

hyperMILL® VIRTUAL Machining closes the gap between the CAM system and your actual machine environment – providing an unprecedented level of process control and optimization. This is Industry 4.0! Our VIRTUAL Machining technology guarantees reliable CNC machining from programming to the machine.

- Optimization of movement sequences in the NC program
- Perfect machine connection thanks to bidirectional connection
- Two-way data exchange from CAM to machine
- Simplified programming
- Digital twin of your machine
- Simulation based on NC code
- Automatic solution selection

Learn more about
hyperMILL® VIRTUAL
Machining!



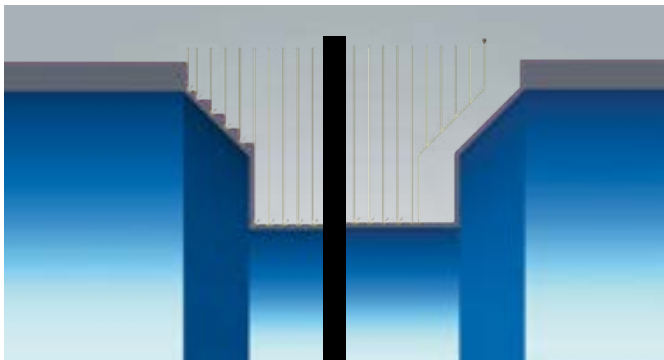


CAD functionalities for turning operations

New CAD-for-CAM functions for turning contours are available to you:

- Simple creation of DIN undercuts on turning contours using drop-down menu
- Globally round the edges of turning contours or apply chamfers to them. A distinction can be made between inner and outer corners. Individual corners can be excluded.
- Restore sharp corners if they have a radius or chamfer

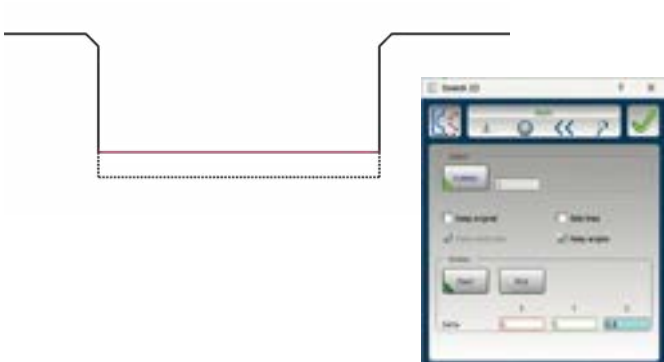
Benefit: Simplified creation of chamfers, radii, and undercuts.



Finishing path for grooving

When grooving, a finishing path can now be used directly afterwards. This ensures a uniform allowance for the subsequent finishing step.

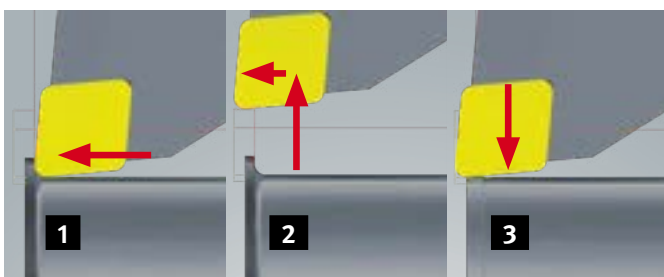
Benefit: Uniform allowance for finishing.



Stretch 2D

The new "Stretch 2D" command allows you to make adjustments to a 2D contour without it having been drawn as a parametric sketch. This allows you to quickly make changes to the component or alter fit tolerances.

Benefit: Simplified adjustment of 2D contours.



Roughing – Remove rings

With the new "Remove rings" function, unwanted ring chips on the component are avoided as soon as you break through the stock with the tool. At the end of the roughing process, the tool travels an additional path to remove any rings that have formed.

Benefit: Reliably remove ring chips.

HIGHLIGHT**Turret support for lathes**

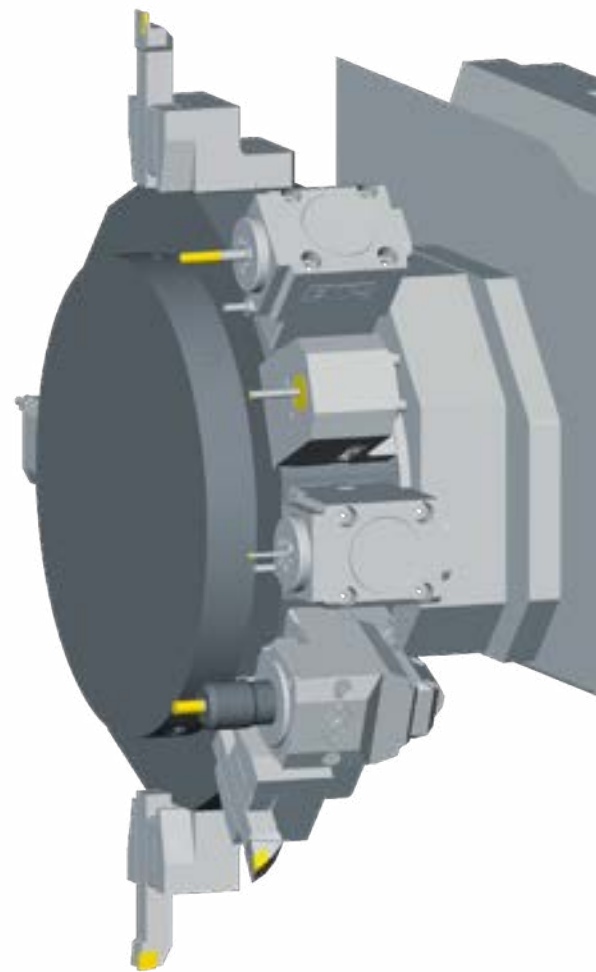
We have taken a big step forward with *hyperMILL*® 2024 and further enhanced our turning functionalities. The turret technology* we have implemented now allows you to program lathes with a turret and a main spindle. Thanks to *hyperMILL*® VIRTUAL Machining, the machine and all tools are mapped in detail and used for NC code simulation. The turret is equipped conveniently with turret holders and tools in the machining planner of the virtual machine.

*Available for machines with a turret and a main spindle with Siemens control system. Further controls to follow.

Convenient use of multiple turret setups

The user can create several setups and select them via the job list. A turret setup is defined as standard. However, several setups can also be exported to the global working space and reused in other *hyperMILL*® projects.

hyperMILL®
TURNING



Job	Name	NC-Start	NC-End	NC-Start	NC-End	NC-Start	NC-End
1	Job1	0	0	0	0	0	0
2	Job2	0	0	0	0	0	0
3	Job3	0	0	0	0	0	0
4	Job4	0	0	0	0	0	0
5	Job5	0	0	0	0	0	0
6	Job6	0	0	0	0	0	0
7	Job7	0	0	0	0	0	0
8	Job8	0	0	0	0	0	0
9	Job9	0	0	0	0	0	0
10	Job10	0	0	0	0	0	0
11	Job11	0	0	0	0	0	0
12	Job12	0	0	0	0	0	0
13	Job13	0	0	0	0	0	0
14	Job14	0	0	0	0	0	0
15	Job15	0	0	0	0	0	0
16	Job16	0	0	0	0	0	0



Learn more about
hyperMILL® TURNING

All tools at a glance

The setup status of each tool is immediately visible in the *hyperMILL*® browser. Two new icons indicate whether a tool is mounted on the turret or not.

Jobs	Tools	Names	Models	Features	Macros
		Cooling Adapter 01			
		1: T2 Roughing			
		2: T2 Roughing			
		5: T4 Linkingjob Turning			
		6: T18196 Grooves Plunging			
		7: T19 Groove Finishing			
		8: T5 Contour Milling			
		11: T8 Drilling			

- ✓ Tool is mounted on the turret
- ✗ Tool is not mounted on the turret



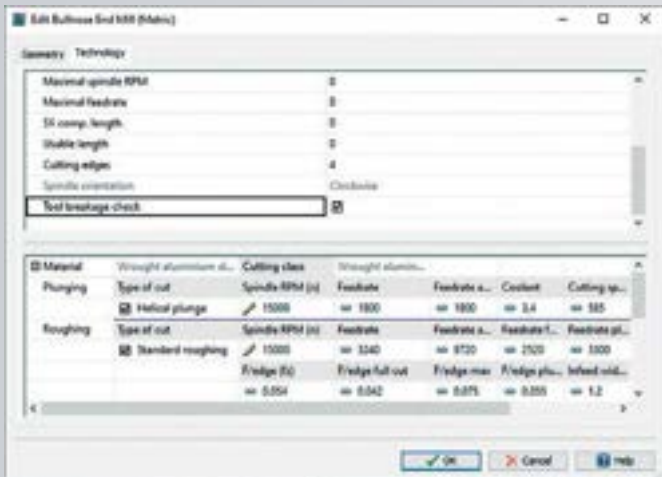
NEW: In the *hyperMILL*® SIMULATION Center and in the virtual machine revolved geometries are displayed as such

Improvements for MILL-TURN machines

In addition to supporting mill turn machines with Siemens controls in the virtual machine, turning programs can also be processed for Heidenhain TNC 640 and TNC 7 controls.

Both the Siemens and Heidenhain controls now support simultaneous turning. In the simulation, the machining is mapped accurately and in detail, and the NC code for these operations is created.

Benefit: Now also supports Heidenhain controls and simultaneous turning.



HIGHLIGHT

Tool breakage check

It is now possible to activate a tool breakage check for tools in the tool database. This information is processed during NC generation with the virtual machine. The generated NC program contains the respective call of the control macro. The breakage check is called before a tool change and at the end of the program. The movements required for breakage control are simulated and checked for collisions. The virtual machine must be adapted to support breakage control.

Benefit: Tool breakage check option, improved process reliability during machining.

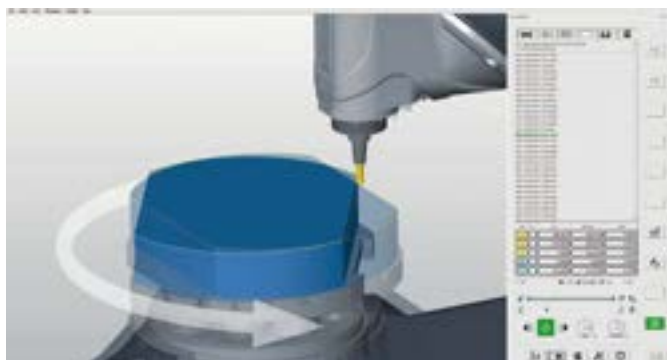


CONNECTED Machining for FANUC Controls

hyperMILL® CONNECTED Machining now also supports FANUC controls. Thanks to the bidirectional connection with the control, data can be sent to and received from the machine. This enables a continuous process chain from the CAM to the machine. Users can do the following:

- Compare tool and zero-point configurations of the machine with the NC program
- Transfer tool information to the control unit
- Transfer NC programs to the control unit
- Display alarm messages from the control unit

Benefit: Direct connection to the machine, more user-friendly, and increased process reliability.

**HIGHLIGHT****Milling with rotation axis**

The Optimizer now offers you the option to transform X- and Y-axis movements into one movement with the rotation axis in the table. By swapping axes, for example, an XY movement is transformed into a simultaneous CX movement. This allows, among other things, machining operations to be generated without rewind movements. This is particularly useful for machines that cannot move over the center of the table, or when machining components that take up most of in the workspace. The axis change can be carried out by the Optimizer for 3D and 5-axis jobs.

Benefit: Simple generation of NC programs with axis change and optimized use of the workspace.

3D/5-axis path compensation

hyperMILL® VIRTUAL Machining now supports the output of 3D and 5-axis path compensation for Heidenhain controls. This allows the machine operator to make fine corrections to tools directly on the control system and perfectly control the accuracy of the machining process.

Benefit: Path compensation also available for 3D and 5-axis machining.

Performance improvements

In *hyperMILL*® version 2024, significant performance improvements in NC code generation and collision checking make our VIRTUAL Machining technology much more efficient. In addition to introducing faster NC program generation for 2D and 3D machining without the Optimizer, we have also optimized collision checking. The collision check performed against the stock during turning and milling operations has been significantly improved.

Benefit: Accelerated NC program generation and checking.



Our *hyperMILL*® VIRTUAL Machining technology already supports a wide range of controls and is continuously being developed to include new controls and functions.

Controls supported by the 2024 version:

- | | |
|--------------------|---------------|
| ■ Brother | ■ Heidenhain |
| ■ D.Electron | ■ Hurco |
| ■ Fagor Automation | ■ Mazak |
| ■ Fanuc | ■ Okuma |
| ■ Fidia | ■ Rödgers Tec |
| ■ Haas | ■ Siemens |

Restrict input during user guidance

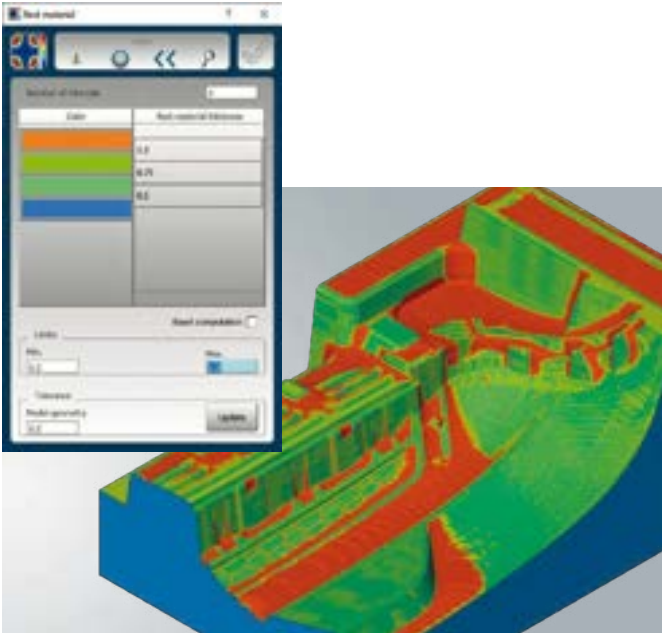
This new function allows user input to be restricted to what makes sense in the given environment. This simplifies operation for inexperienced CAM users and helps to avoid hazards for the operator and machine.

Benefit: Safe and simple operation.

Rest material display

The “Rest material display” function has been revised and now offers a faster and improved display with new technology and a new user interface. Colors, limit ranges, and number of intervals can be freely defined.

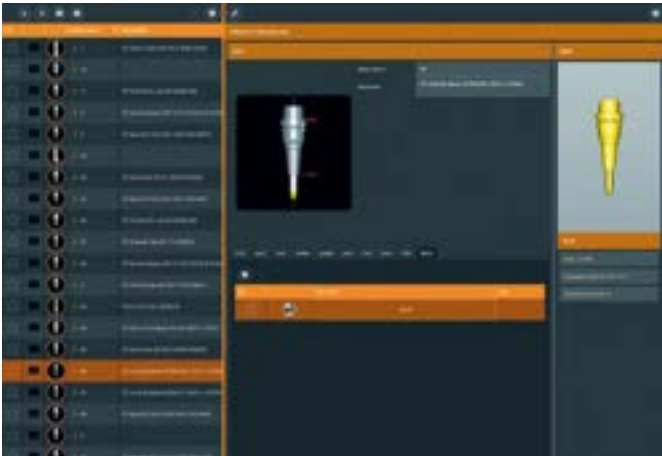
Benefit: Improved user-friendliness and display.



Tool synchronization with the Hummingbird MES

In the new version, tools from the *hyperMILL*® tool database can be synchronized with the Hummingbird MES at the touch of a button. This means that the tools used for CAM programming are automatically transferred to the Hummingbird tool management system. All other processes, such as setting up, measuring, tracking, and use of the tools on the machines, are mapped in the Hummingbird MES. This ensures a continuous exchange of information within the company.

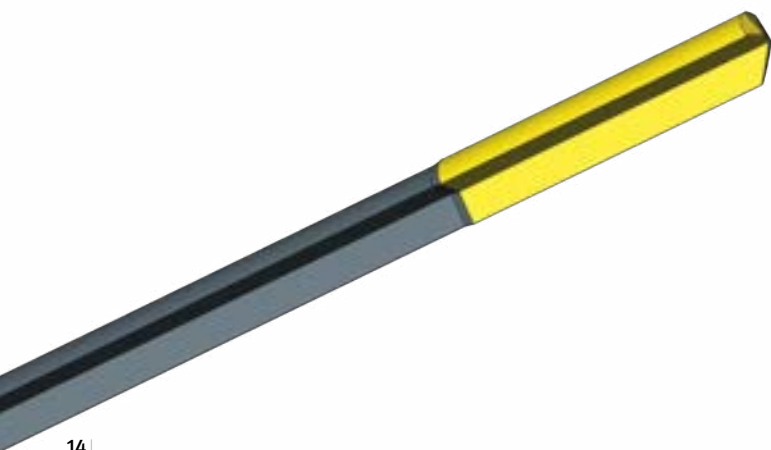
Benefit: Seamless tool management between *hyperMILL*® and Hummingbird MES.



New tool type: Gun drill

A new tool type is supported in *hyperMILL*®. Single-lip drills can now be created in detail in the tool database. The special cutting geometry of the drill bit is reproduced one to one and used to calculate the toolpaths and the simulation. In addition to the precise collision check, the stock removal is also visualized in detail in the simulation.

Benefit: Simple use of single-lip gun drills.



HIGHLIGHT

Programming assistance: CAM Plan

With version 2024 of *hyperMILL*®, we are introducing a new generation of programming assistance called CAM Plan. This takes over various tasks for you during the programming process. With our first release of CAM Plan, we have focused on simplifying daily tasks and eliminating possible sources of error that occur during programming.

Easier programming thanks to automatic process steps

Thanks to predefined workflow steps, you are guided safely through the process, while the geometries and features required for programming are created automatically. One example of this is the fully automatic generation of cover surfaces for drill holes. In addition, after analyzing the geometry data, you are notified of possible errors, such as double surfaces or gaps in the model surfaces.

Assistance with precision machining

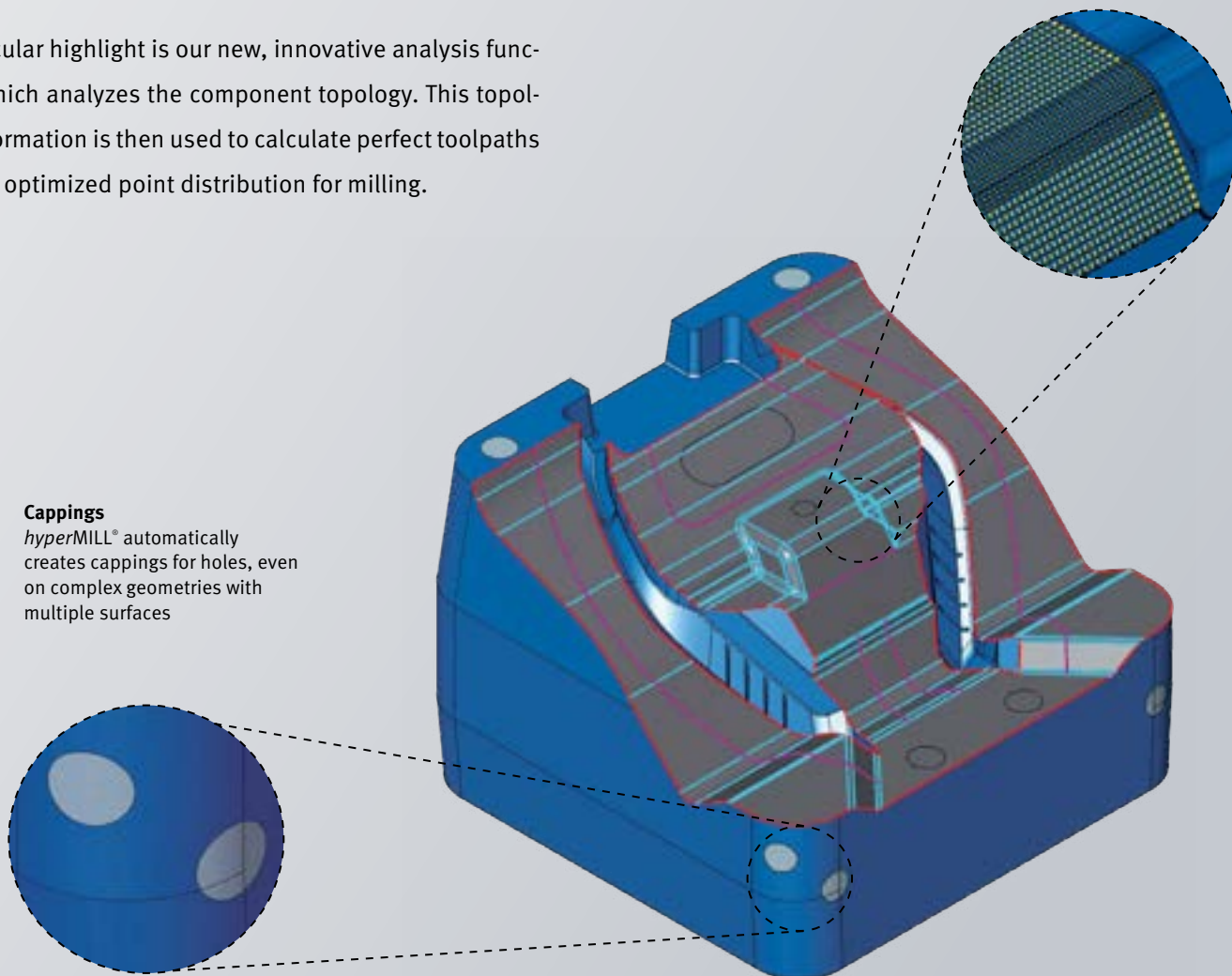
A particular highlight is our new, innovative analysis function, which analyzes the component topology. This topology information is then used to calculate perfect toolpaths with an optimized point distribution for milling.

Toolpaths

hyperMILL® calculates the optimum point distribution based on the topology information for the component

Cappings

hyperMILL® automatically creates cappings for holes, even on complex geometries with multiple surfaces



HEADQUARTERS

OPEN MIND Technologies AG
Argelsrieder Feld 5 • 82234 Wessling • Germany
Phone: +49 8153 933-500
E-mail: Info.Europe@openmind-tech.com
Support.Europe@openmind-tech.com

PORTUGAL

OPEN MIND Technologies Portugal, Unipessoal, Lda
Edifício Centro de Negócios MAPER • Fração N e O
Estr. Nacional 242, Km 9,2 • 2430-074 Marinha Grande
Phone: +351 244 023 359
E-mail: Info.Portugal@openmind-tech.com

SWEDEN

OPEN MIND Technologies Scandinavia AB
Älvhålegatan 34
417 28 Gothenburg
E-mail: Info.Scandinavia@openmind-tech.com

UK

OPEN MIND Technologies UK Ltd.
Unit 3 • Bicester Business Park
Telford Road • Bicester • Oxfordshire OX26 4LN
Phone: +44 1869 290003
E-mail: Info.UK@openmind-tech.com

USA

OPEN MIND Technologies USA, Inc.
1492 Highland Avenue, Unit 3 • Needham MA 02492
Phone: +1 888 516-1232
E-mail: Info.Americas@openmind-tech.com

BRAZIL

OPEN MIND Tecnologia Brasil LTDA
Av. Andromeda, 885 SL2021
06473-000 • Alphaville Empresarial
Barueri • São Paulo
Phone: +55 11 2424 8580
E-mail: Info.Brazil@openmind-tech.com

ASIA-PACIFIC

OPEN MIND Technologies Asia Pacific Pte. Ltd.
MOVA Building, 22 Jalan Kilang • #03-00
Singapore 159419
Phone: +65 6742 95-56
E-mail: Info.Asia@openmind-tech.com

CHINA

OPEN MIND Software Technologies China Co., Ltd.
Suite 1608 • Zhong Rong International Plaza
No. 1088 South Pudong Road
Shanghai 200120
Phone: +86 21 588765-72
E-mail: Info.China@openmind-tech.com

INDIA

OPEN MIND CAD/CAM Technologies India Pvt. Ltd.
No. 610 and 611 • 6th Floor • 'B' Wing
No. 6, Mittal Tower, M.G. Road
Bangalore 560001 • Karnataka
Phone: +91 80 2676 6999
E-mail: Info.India@openmind-tech.com

JAPAN

OPEN MIND Technologies Japan K.K.
Albergo Musashino B101, 3-2-1 Nishikubo
Musashino-shi • Tokyo 180-0013
Phone: +81-50 5370-1018
E-mail: Info.Japan@openmind-tech.com

TAIWAN

OPEN MIND Technologies Taiwan Inc.
Rm. F, 4F., No.1, Yuandong Rd., Banqiao Dist.
New Taipei City 22063
Phone: +886 2 2957-6898
E-mail: Info.Taiwan@openmind-tech.com

OPEN MIND Technologies AG is represented worldwide with own subsidiaries and through competent partners and is a member of the Mensch und Maschine technology group, www.mum.de



 **OPEN MIND**
THE CAM FORCE

We push machining to the limit

www.openmind-tech.com