Copyright © 2020 OPEN MIND Technologies AG

This document applies to hyperMILL and hyperMILL SHOP Viewer.

It contains notes about recent changes that are not described in the manual. All rights reserved. Since we continuously work on further developments, we reserve the right to implement changes.

Last update: 2020-08-04T12:28:53+02:00.
## Table of Contents

1. Supported software ................................................................................................................... 4  
   Operating systems ..................................................................................................................... 4  
   64-bit CAD platforms ................................................................................................................ 4  
   32-bit CAD platforms (Windows 7 only) ..................................................................................... 4  
   Server operating systems (Licence Manager only) ................................................................. 4  
   NC interfaces ............................................................................................................................ 5  
   Interfaces Tool database .......................................................................................................... 5  
   Use of third-party software ..................................................................................................... 5  

2. Installation and licensing ......................................................................................................... 6  
   Supported languages ............................................................................................................... 6  
   Parallel installations .............................................................................................................. 6  

3. Notes on functions ..................................................................................................................... 7  
   hyperMILL SHOP Viewer ....................................................................................................... 7  
   5X Strategies on machines with Non Endless Rotational Axes .............................................. 7  
   Negative allowances .............................................................................................................. 7  
   5X Simultaneous Machining .................................................................................................. 8  
   Supported tool types ............................................................................................................. 8  
   Feedrates ............................................................................................................................... 8  
   3D and 5X Rework Machining - Linking job - Compound job .............................................. 8  
   hyperCAD 2009.3: Select surfaces ......................................................................................... 8  
   TDM Systems Integration ..................................................................................................... 8  

4. Notes on older hyperMILL versions ....................................................................................... 9  
   Postprocessors ....................................................................................................................... 9  
   Thread Milling / Helical Drilling ............................................................................................... 9  
   2D Contour Milling ................................................................................................................ 9  
   Toolpath segments with a Z component ................................................................................ 9  

5. Additions to the manual 2020.2 ............................................................................................ 10  
   2D Chamfer Milling on 3D Model ......................................................................................... 10  
   2D Contour Milling on 3D Model - Macros .......................................................................... 10  
   3D Optimized Roughing - Adaptive pocket ......................................................................... 11  
   Tool database - improved filter options ............................................................................... 12  
   Feature Macro Technology ................................................................................................. 13  
   Job list – NC status display ................................................................................................. 13  
   Generate NC file .................................................................................................................. 13  
   Tool definition and collision check .................................................................................... 14  
   Parametric ........................................................................................................................... 14  
   TDM Systems Integration .................................................................................................... 14  

6. Solved problems and changes 2020.2 ServicePack1 .......................................................... 19  
   5X Multiblade Roughing ..................................................................................................... 19  
   3D Automatic Rest Machining ............................................................................................ 19  
   3D Z Level Shape Finishing ................................................................................................. 19  
   hyperMILL VIRTUAL Machining Center ............................................................................ 19  
   Mirror - 3D Optimized Roughing ....................................................................................... 19  
   hyperMILL in Inventor ......................................................................................................... 19  

7. Bug fixes - Hotfix 1 ................................................................................................................. 20
1. Supported software

*hyperMILL* exclusively supports 64-bit operating systems: For the latest information on hardware requirements and supported operating systems, please visit the website www.openmind-tech.com.

**Operating systems**

- Windows 7
- Windows 8.1
- Windows 10

**64-bit CAD platforms**

- *hyperCAD-S*
- Inventor 2018, 2019, 2020, 2021
- SolidWorks 2018, 2019, 2020
- thinkdesign 2018.1, 2019.1, 2020.1

A mixed installation of thinkdesign 32- and 64-bit on one machine is not supported. *.e3* files based on thinkdesign 64-bit can not be imported in *hyperCAD-S*.

**32-bit CAD platforms (Windows 7 only)**

> **CAUTION**
The release *hyperMILL 2020.2* will be the last one to provide support for Microsoft Windows 7 and OPEN MIND’s *hyperCAD® 2009.3*.

If you have not already, please take this opportunity to upgrade your working environment. Please note the *hyperMILL* Batch Converter utility, to update many files and the programming metadata to the current format. Should you have any questions, please contact OPEN MIND or your local OPEN MIND dealer.

- *hyperCAD 2009.3*

**Server operating systems (Licence Manager only)**

- Windows Server 2008 R2
- Windows Server 2012 R2
- Windows Server 2016
- Windows Server 2019
NC interfaces

- VERICUT from version 7.0
- NCSimul from version 9.0

Interfaces Tool database

<table>
<thead>
<tr>
<th>OPEN MIND Technologies AG</th>
<th>tdm systems</th>
</tr>
</thead>
<tbody>
<tr>
<td>hyperMILL 2017.1 SP1 or higher</td>
<td>tdm 4.8 Hotfix 9 or higher and tdm 2017 Hotfix 5 or higher</td>
</tr>
<tr>
<td>License – Interface Walter TDM</td>
<td>CAM Mapping to hyperMILL Microsoft SQL Server</td>
</tr>
<tr>
<td>OPEN MIND Technologies AG</td>
<td>Zoller TMS</td>
</tr>
<tr>
<td>hyperMILL 2014.1 SP1 or higher</td>
<td>Version 1.14.6.0 or higher</td>
</tr>
<tr>
<td>License – Interface Zoller Tool Management</td>
<td>Interface for data exchange (one interface for each hyperMILL license). Microsoft SQL Server</td>
</tr>
<tr>
<td>OPEN MIND Technologies AG</td>
<td>WinTool AG</td>
</tr>
<tr>
<td>hyperMILL 2014.1 SP1 or higher</td>
<td>The compatibility of the version must be checked with WinTool.</td>
</tr>
<tr>
<td>License – Win tool Interface</td>
<td>Interface for data exchange (one interface for each hyperMILL license). Microsoft SQL Server or Access database (*mdb)</td>
</tr>
<tr>
<td>OPEN MIND Technologies AG</td>
<td>Hexagon Manufacturing Intelligence</td>
</tr>
<tr>
<td>hyperMILL 2019.2 SP1 or higher</td>
<td>NCSIMUL</td>
</tr>
</tbody>
</table>

Use of third-party software

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.
2. Installation and licensing

CAUTION
As a ServicePack not only fixes errors, but also adds new functions backward compatibility between the release version and a ServicePack or Hotfix is no longer supported. This applies to version 2020.1 or later and means that files saved with version 2020.1 SP1, for example, can no longer be opened with version 2020.1.

Due to necessary technical updates of our license management it is mandatory to always use the current license manager on the license server and the workstations in the future! You should therefore install the latest version immediately on your license servers and on all workstations on which OPEN MIND products are used. Otherwise, licensing will no longer be possible.

Please install the hyperMILL software according to the installation guide. Direct any questions you may have about the installation to our support service.

Please note that the installation requires that you have administrator rights.

The hyperMILL software is protected by an dongle or license server. Each licence is unique. Secure the licence against loss!

For further information, please refer to the installation guide, which can be found on your product DVD or on the OPEN MIND website.


Configuration License Manager
The following notes on configuring the License manager are relevant to you if you already have hyperMILL installed and use a network license server.

By changing the dongle driver it is necessary to reconfigure the network server configuration of the licensing. Proceed as follows.

1. After installation, open License Manager As administrator and switch to the Options tab.
2. Remove the name of the license server and reenter it.
3. Close the License Manager and restart the computer to complete the configuration.

Supported languages
hyperMILL is available in the following languages: German, English, Italian, French, Dutch, Japanese, Chinese (Traditional and Simplified), Korean, Spanish, Czech, Russian, Polish, Slovenian and Portuguese (Brazilian).

Parallel installations
If several versions of hyperMILL are installed, it is possible to switch between versions using hyperMILL Switch.

Run SwitchUi.exe from the C:\Program Files\OPEN MIND\Settings Wizard directory.

Select the required hyperMILL version, the CAD system and the language. Afterwards, click Switch. Please close the software first!
3. Notes on functions

**hyperMILL SHOP Viewer**

Reusing SHOP Viewer data (also called SHOP Viewer Pack & Go data) in hyperMILL is intended for OPEN MIND support and analysis purposes only. If Pack & Go data are changed in hyperMILL and then saved with the Save as function, the usability of the resulting file cannot be guaranteed.

**5X Strategies on machines with Non Endless Rotational Axes**

OPEN MIND recommends defining the toolpath in such a manner that no tension-release rotations are necessary. It is possible that the geometry to be machined will need to be broken up into multiple jobs, or it may be necessary to define alternative toolpaths. This may increase the complexity of the toolpath definition.

**WARNING**

OPEN MIND recommends proceeding with extreme care with these toolpaths if tension-release rotations cannot be avoided. The entire sequence for the tension-release toolpath – from the retract movement from the workpiece up to the re-approach – is not collision checked!

**WARNING**

If a postprocessor run with stored solutions is performed, the NC file contains no rewind.

OPEN MIND additionally recommends taking appropriate measures to first successfully test and execute the tension-release toolpaths, such as for example test runs above or next to the part, or without the part in place. Jobs which include tension-release rotations should be run separately on the machine if possible (not as part of an NC program with multiple operations). Using additional verification software for NC programs can help increase safety.

These tips are intended to help successfully execute the defined toolpaths and prevent damage to the CNC machine. Please contact your OPEN MIND partner if you have further questions on this topic.

**Negative allowances**

The following restriction applies to all cycles: If the sum of the allowances is negative, the sum total must be less than the Tool radius - Machining tolerance.

**Example:**

Tool radius = 5.00 mm

Machining tolerance = 0.01 mm

Allowance = -3.0mm

Additional allowance XY must be greater than -1.99 mm, e.g. 1.98, because

\[-3.00 \text{ mm } +(-1.98 \text{ mm})\] < 5.00 mm -0.01 mm

In this context, the area-based allowance is also taken into account as an allowance in the Milling area > Definition dialog. If several values are defined here, only the smallest value is taken into account.
5X Simultaneous Machining

Important note for 5X simultaneous users with controls without great circle interpolation:

The hyperMILL cutter path calculations for all 5 axes cycles (except 5X Multiblade) are based on a great circle interpolation between the tool orientations. If your control works with a different interpolation type and cannot be modified for great circle interpolation, please contact your OPEN MIND partner to have your hyperMILL configured accordingly. To see the interpolation type with which your control works, refer to the documentation of your control.

Supported tool types

hyperMILL generally only supports the tool types listed in the documentation. Other tool types, in particular, angle tools or angle heads are not supported.

Chamfered cutters

Specifying a nominal diameter of less than 0.001 mm or less than 0.0001 inches may prevent the activation of a tool radius correction. Please avoid specifying such diameters.

Feedrates

All feedrates in hyperMILL refer to the respective tool reference point. Therefore, feedrates can occur on single axes that deviate from the value of the tool reference point. A parametric feedrate output is generally not supported, particularly not for linking jobs if the underlying sub-jobs contain feedrate changes.

3D and 5X Rework Machining - Linking job - Compound job

Initial situation

Reference job: Generate NC file is disabled.
Rework job: Generate NC file is enabled.

1. Reference job and related Rework job are used within a Linking job. Generate NC file is automatically disabled in both jobs.
2. Reference job and related Rework job are moved from the Linking job into a Compound job. By doing so, the Generate NC file option is automatically enabled in both jobs. It has to be manually disabled within the Reference job again.

hyperCAD 2009.3: Select surfaces

For the 2D Contour Milling on 3D Model cycle the surface selection function in hyperCAD 2009.3 / thinkdesign 2009.3 not available.

TDM Systems Integration

To provide features associated with feature / macro technology (intelligent macros), a synchronization service has been implemented to exchange data between TDM Systems and OPEN MIND. More information in section TDM Systems Integration (page 14).
4. Notes on older hyperMILL versions

Information in this section is relevant for you if you have not updated from the previous release of hyperMILL to the latest version. This may be the case if

- you do not have a maintenance contract or
- you have purchased an update again after a longer period of time.

Postprocessors

Use the Frame ID

If your postprocessors require you to define a Frame ID or a frame Comment, your postprocessors are incompatible with toolpaths that were created using the hyperMILL Transformation function. For this reason, please use only toolpaths without transformations for these postprocessors.

Thread Milling / Helical Drilling

The Path compensation option must be made available by OPEN MIND separately and requires a prior check and possible update of your postprocessor (subject to a charge). Please contact your OPEN MIND partner. In the event of use without a postprocessor check, OPEN MIND cannot assume any guarantee for the output of a correct NC program.

2D Contour Milling

Spiral stepdown: If, while using this option, you see an error message on the Parameters tab during the postprocessor run, your postprocessor is not yet ready for this output and requires a prior check and, perhaps, update (for a fee). Please contact your OPEN MIND partner.

Toolpath segments with a Z component

All circular toolpath segments with a Z component (such as circular macros or spiral stepdowns) require a separate postprocessor adjustment. Please contact your OPEN MIND partner.
5. Additions to the manual 2020.2

2D Chamfer Milling on 3D Model

With the new option Parameter → Additional options → Prefer spiral the infeed to the next machining level is performed in a spiral form when the contours are closed.

Example: (A) closed contour with spiral shaped infeed, (B) open contour with contour-parallel infeed, Chamfer mode = modeled chamfer, (1) = infeed step.

2D Contour Milling on 3D Model - Macros

With the new options Macros → Approach macro / Retract macro → Length / Height / Radius options you can control the length, height and radius of the approach and retract movements and thus allow the tool to smoothly plunge into the material. Machining marks in adjacent levels are avoided.
Example: A) Approach macro = Tangential, B) Approach macro = Quarter circle, (1) height, (2) length, (3) radius.

3D Optimized Roughing - Adaptive pocket

The Adaptive pocket algorithm has been improved in such a way that, considering the plunging capability of the tool, it detects and fits the largest possible adaptive pocket. This results in technologically optimized tool paths with high removal rates. (1) Previous technology, (2) current technology.
When processing rectangular adaptive pockets, the pocket shape is now automatically detected by default, regardless of the orientation of the pocket in relation to the job frame. (1) Previous technology, (2) current technology.

Elongated, open areas (pockets) are now automatically recognized even if the *Use adaptive pocket* option is not activated. *Advantage:* less G1 movements, improved performance. (1) Previous technology, (2) current technology.

**Tool database - improved filter options**

The newly designed filter improves the display of database elements according to individual filter criteria.

- If the *Select NC tool* dialog is opened from the job definition, suitable tool properties are already preselected. *Examples.*
• Adaptor: Defined in the machine management (see Joblist → Postprocessor → Machine Administration → Machine Properties) and preselected in the filter as Standard → Top coupling.

• Material: Defined in the joblist (see Job list → Part data → Material) and preselected in the filter as Standard → Material.

Further improvement: All tool types allowed for machining can be changed directly in the filter without having to close the Define NC tool dialog end and switch to the job definition.

Feature Macro Technology

Optimize macro jobs without grouping: The new function Optimization (without job grouping) allows you to group the macro jobs of a job list according to the type of processing (for example centering, drilling, etc.). However, the individual jobs are not grouped together in one step (as with the Optimization function).

Requirement: The macro jobs to be optimized were created by using a feature-based macro.

Advantage: It is possible to subsequently change, move or delete all macro jobs.

<table>
<thead>
<tr>
<th>Without Optimization</th>
<th>Optimization</th>
<th>Optimization (without job grouping)</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image1" alt="Top of part" /> <img src="image2" alt="1: T1 Centering 1" /> <img src="image3" alt="2: T2 Drilling 1" /> <img src="image4" alt="3: T1 Centering 2" /> <img src="image5" alt="4: T2 Drilling 2" /></td>
<td><img src="image6" alt="Top of part" /> <img src="image7" alt="1: T1 Centering (1+2)" /> <img src="image8" alt="2: T2 Drilling (1+2)" /></td>
<td><img src="image9" alt="Top of part" /> <img src="image10" alt="1: T1 Centering 1" /> <img src="image11" alt="2: T1 Centering 1" /> <img src="image12" alt="3: T2 Drilling 2" /> <img src="image13" alt="4: T2 Drilling 2" /></td>
</tr>
</tbody>
</table>

Job list – NC status display

The current progress of calculating the NC file is now indicated by an icon. The icon colours mean: Gray - No NC file has not been created yet, black - an NC file has been created, red - an NC file has been calculated, but due to changes in the job list, it is no longer up to date and therefore needs to be calculated again.

NOTE
This function is available for the hyperMILL VIRTUAL Machining Center and the corresponding postprocessor technology.

Generate NC file

For post-processing, mark the desired jobs or job lists (right mouse button) and select one of the two following options in the context menu:

• Generate NC file (or button P)
• Generate NC file (batch mode) (or button B)

The Generate NC file function starts a new instance of the hyperMILL VIRTUAL Machining Center each time, machining takes place in the same instance of the hyperMILL VIRTUAL Machining Center.
Tool definition and collision check

![WARNING]
---

**WARNING**

For tools whose geometry contains a [Free tip geometry](#) (option **Tool → Tip → Free tip geometry**), this part of the tool geometry is **not** checked against collisions.

---

**Parametric**

With the release 2020.2, parametric modeling is introduced in *hyperCAD-S* for the first time. OPEN MIND is continuously developing this technology. *hyperMILL* supports parametric model changes in *hyperCAD-S*, but please note the following information:

- To ensure the associativity of the geometry selected in *hyperMILL* to parametrically executed model changes, please use the **V-Sketch / Faces** function as selection basis.
- Surface edges / Snap points do not follow the parametrically executed changes and must be selected again after these changes.
- The functions **Roll back / Roll forward** resp. **Enable / Disable** of generated geometric elements can lead to undesired tool paths during programming.
- For release 2020.2, OPEN MIND recommends that parametric components should first be completely designed and **after** the *hyperMILL* NC programming should be started.
- Feature recognition should also be performed **after** parametric design (exception: Feature Mapping (Hole), CAD-Feature Recognition).
- Please check the correctness of the generated tool paths after parametric changes.

**TDM Systems Integration**

To provide features associated with feature / macro technology (intelligent macros), a synchronization service has been implemented to exchange data between TDM Systems and OPEN MIND.

For this purpose, a local service was developed by OPEN MIND, which can be controlled via a command line with different options. The program **hmTDMSyncService.exe**, that controls the service, is located under the following path (the path may vary depending on the installation and version):

```plaintext
C:\Program Files\OPEN MIND\hyperMILL\2019.2\addins\hmTDMSystems\hmTDMSyncService.exe
```

**Set up the synchronization service**

To set up the service after the installation of *hyperMILL*, follow these steps:

1. Open a command line with administrator rights. To do this, enter `cmd` in the Windows start menu, open the context menu on the **Command prompt** and select **Execute as administrator**.
2. Use `cd` to change to the directory `<C:\Program Files\OPEN MIND\hyperMILL\Version \addins\hmTDMSystems>` and call the synchronization service **TDMSyncService.exe** with the `/H` option.
3. A list of possible options for setting up the synchronization service appears:

**Table 1. Options synchronization service**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>/i [nstall]</td>
<td>Installs and starts the service.</td>
</tr>
<tr>
<td>/r [emove]</td>
<td>Stops and uninstalls a running service.</td>
</tr>
<tr>
<td>/h [elp]</td>
<td>Display of available options.</td>
</tr>
<tr>
<td>/f [older]</td>
<td>Path to the folder where TDM Systems exports tools to. The folder must be empty and must not be a directory of a private user account.</td>
</tr>
<tr>
<td>/d [atabase]</td>
<td>Path to hyperMILL database to be synchronized. Possible database formats are: *.DB, *.MDB or *.DSN (SQL Server).</td>
</tr>
</tbody>
</table>
4. **Example:**

A completed call process including installation / start / path and naming the hyperMILL database for synchronization of TDM data can look like this:

The synchronization service can be controlled via the Windows Control Panel. To do this, enter Control Panel in the search box of the taskbar and call the service under Administration → services → OPEN MIND TDM Synchronization Service. In general, however, there is no need for this, because the service is started automatically each time the computer starts up.

**Messages of the synchronization service (*.log file)**

Below the path for the synchronization folder, the folder `log` is created. After starting the service, the file `hmTDMSyncService.log` is located here. The synchronization service writes messages about activities and any import problems that may occur into this file. Each time the service is started, a new `*.log` file is created.

The current file is then renamed `hmTDMSyncService_1.log`. Older files receive correspondingly higher indexes. A maximum of 100 files is stored. If the service is running continuously, new files will be created as soon as they exceed 1MB in size. If there are errors during the import, the causative files are moved in the subfolder `invalid`. Successfully imported files are automatically deleted.
**NOTE**

Rebuild the synchronization service: The synchronization service must be rebuilt, if an update of hyperMILL is performed, hyperMILL is uninstalled or installed on a new machine. In these cases, the service needs to be rebuilt, because the required parameters are no longer known. In order to simplify the rebuild of the service, the settings made during the first setting up should be documented, since they can not be automatically transferred from a pre-installation.

---

**Prerequisites for installing the synchronization service**

To set up the synchronization service, the following software requirements must be met:

- **TDM Systems**: TDM 2018 HF5 + TDM Global Line hyperMILL interface (= Smart Interface Client). The installation of the Smart Interface Client must be installed on the same computer on which the synchronization service (hmTDMSyncService.exe) is installed. During installation of the Smart Interface Client specify the exchange directory; this corresponds to the directory already defined in the hmTDMSyncService.exe.

- **OPEN MIND**: hyperMILL 2019.2.

---

**How the synchronization service works**

The TDM user creates a new tool in TDM and exports it for hyperMILL (1).

The TDM Server informs all installed Smart Interface Clients (2) and stores corresponding files in the shared exchange directory.

The hyperMILL Sync Service uses these files and then creates the tool in hyperMILL Tool Manager.
**IMPORTANT**

Using the synchronization service in a local (document) database (*.hmc)

If you use TDM-Systems tools in a document database, make sure that the synchronized database is set up as default database in the hyperMILL Settings → Database. Avoid importing tools at the same time via the TDM Systems Plugin (see hyperCAD-S menu) hyperMILL → Tools → Import TDM Systems tools. This is the only way to ensure that all tools imported into the document database have a unique ID.
6. Solved problems and changes 2020.2 ServicePack1

**5X Multiblade Roughing**

- Improved collision detection/prevention with small tool diameter and large taper angle
- Collision protection introduced when machining in pocket mode on the side of the leading edge and path extension towards the leading edge

**3D Automatic Rest Machining**

Problem with incorrect toolpaths has been fixed.

**3D Z Level Shape Finishing**

Problem with a wrong calculation of tool paths is fixed.

*hyperMILL VIRTUAL Machining Center*

Error when creating an NC file containing a Toolpath Edit job is fixed.

**Mirror - 3D Optimized Roughing**

Pocket machining of mirrored jobs in cycle 3D Optimized Roughing improved.

*hyperMILL in Inventor*

Performance problems with the geometry check are solved.
7. Bug fixes - Hotfix 1

- **Transformation → Mirror**: A problem is solved with job **Mirror**, that resulted in an incorrect tool orientation for a specific post-processor configuration. OPEN MIND recommends all **hyperMILL** users who use the **Transformation → Mirror** function to install this HOTFIX.

- **Drilling → Optimized Deep Hole Drilling**: A problem with an incorrect feedrate at F1 has been solved.

- **Tire machining**: A problem with a wrong **Segment extension angle** is solved.

- **hyperMILL VIRTUAL Machining Optimizer**: A problem with a default value for the function **Angular increment solution** is solved (dialog page **NC → Use NC optimizer → Pole → Solution inc. angle**). The standard value was set to 180 degrees.

- **Virtual tool**: A problem with a missing variable when editing several features and using the **Set connection** function is solved.

- **Simulation - hyperMILL VIRTUAL Machining Center**: A problem with a part violation when simulating circle segments in the cycle **2D contour milling on a 3D model** is solved.

- **2D Cycles → 2D Contour Milling on 3D Model**: A problem with the **Smooth overlap** function and an incorrect collision message is solved (dialog page **Contours → Options → Smooth overlap**).

- **Simulation - hyperMILL VIRTUAL Machining Center**: A problem with a collision in connection with an interpolation movement during 5X drilling has been solved (**Drilling → Contours → Drilling mode → 5X drilling**).

- **5X Surface Milling → 5X Contour Machining**: A problem with the functions **Synchronize toolpaths → Interpolate surface normals** and **Synchronize toolpaths → Interpolate lines** in connection with a mirrored job list is fixed.