

hyperMILL®



© The helmet was programmed and produced by PAISHIN

Software documentation



This software documentation applies to *hyperMILL* and *hyperMILL* SHOP Viewer. The content of this manual and the related software are the property of OPEN MIND Technologies AG. Any manner of reproduction shall require the prior consent of OPEN MIND Technologies AG.

All rights reserved.

Since we continuously work on further developments, we reserve the right to implement changes.
Last reviewed: September 2017.



OPEN MIND Technologies AG

Argelsrieder Feld 5
82234 Wessling
Germany

Phone: + 49 (0) 8153 933 -500
Fax: + 49 (0) 8153 933 -501
Email: Sales.Europe@openmind-tech.com

Web: <http://www.openmind-tech.com>

Symbols Used

In all OPEN MIND documentation, text sections are identified by symbols to facilitate the quick finding of information. The following symbols may be used:



Special notes and background information.



Indication of possible sources of error.



OPEN MIND tool database

Introduction	11-2
Starting the OPEN MIND tool database	11-3
User interface	11-3
Defining a tool	11-6
Defining an NC tool	11-44
Depots	11-52
Exchanging data with other databases	11-53
Additional functions	11-53
Interfaces	11-56
Icons in the database browser	11-57

hypeVIEW

User interface	12-3
Important workflows	12-3
Menu functions	12-7
Functions in <i>hypeVIEW</i> browser	12-15
Toolbars	12-35



11



OPEN MIND tool database

Introduction

Please note that the demo data in the tool database is provided for illustration, and not necessarily for production purposes. OPEN MIND disclaims any liability for damages resulting from the use of demo data in a production environment.

Database Formats

As the **MS Access (.mdb)** format does not support 64-bit technology, it has been changed to **SQLite (.db)** with Version 2011. Please note that **SQLite** tool databases cannot be protected by passwords at this time.

SQLite tool databases cannot be accessed by several users at the same time. This is only possible using **SQL Server technology**, which is supported as of Version 2014.1.



If you are using SQL Server databases, ensure that you are also using an up to date driver. The name of the driver must read:

SQL Server Native Client 10 or **SQL Server Native Client 11**.

Do not use drivers with the name **SQL Server**, even if these appear to be newer. This driver cannot transfer more than 400 kB of data to the server. As a result, you cannot use it to store 3D tools in your database.

Converting the tool database to the new database system

In order to use databases created under Version 2010 and older, you must convert them to the current format.

1. First, start your 2010 tool database from the Start menu under **Start > Programs > OPEN MIND > 2010 > Tool Database**.
2. Open the database which you want to convert to the new format, select it in the browser and select the **Export > *.xml** function from the shortcut menu (right mouse button). Your old database will be saved in a **neutral tool exchange format** which can then be imported using the tool database.
3. Next, start your current tool database from the Start menu under **Start > Programs > OPEN MIND > [Version number] > Tool Database**.
4. Create a new (empty) database (**File > New**) and select under **File type** the format **Sqlite database (.db)**.
5. In the browser, select the newly created database and then right-click the **Import** function in the shortcut menu.
6. Then select the previously created file in the **.xml** format that you wish to convert to the new format.

The **Import status** dialogue informs you about the current status of the data import.

Invalid lengths or duplicate designations are indicated in the **Constraint Violation** dialogue and must be edited. Constraint violations can be resolved by different methods:

Automatic: All conflicts occurring during the import will be resolved automatically, so that all elements can be imported from the old to the new database.

Ignore: Should only be used by experienced experts.



The import continues even if there is no conflict solution. Since the application of this option may result in faulty tools, it is only available if the options **Leave**, **Overwrite** and **OK** are disabled.

The **Leave/Leave All:** The current element/all elements of the old database is/are preserved.



Overwrite/Overwrite All: The current element/all elements of the old database is/are overwritten with the new element(s).

Deviation: The difference between the element of the old database and the element to be imported is shown.



Database files in the *.mdb format will not be deleted after import to the current database format, but saved to your system in the same location under the name [Database_saved.mdb].

During the import, the tool **heads** defined in older versions are converted to tool **spindles**. Consequently, all references are retained. However, the converted tool spindles are no longer displayed in the OPEN MIND tool database. You can nevertheless use them when assembling a tool in *hyperMILL*.



A database conversion is performed when importing tool data from an older version into the current format. This operation should not be stopped, as otherwise *hyperMILL* will not be able to load any data and joblists and jobs will no longer be available.

Starting the OPEN MIND tool database

Start as a separate application:



Via the Windows Start menu: All programs > OPEN MIND > hyperMILL > [Version] > Tool database [Version]

Start from *hyperMILL*:

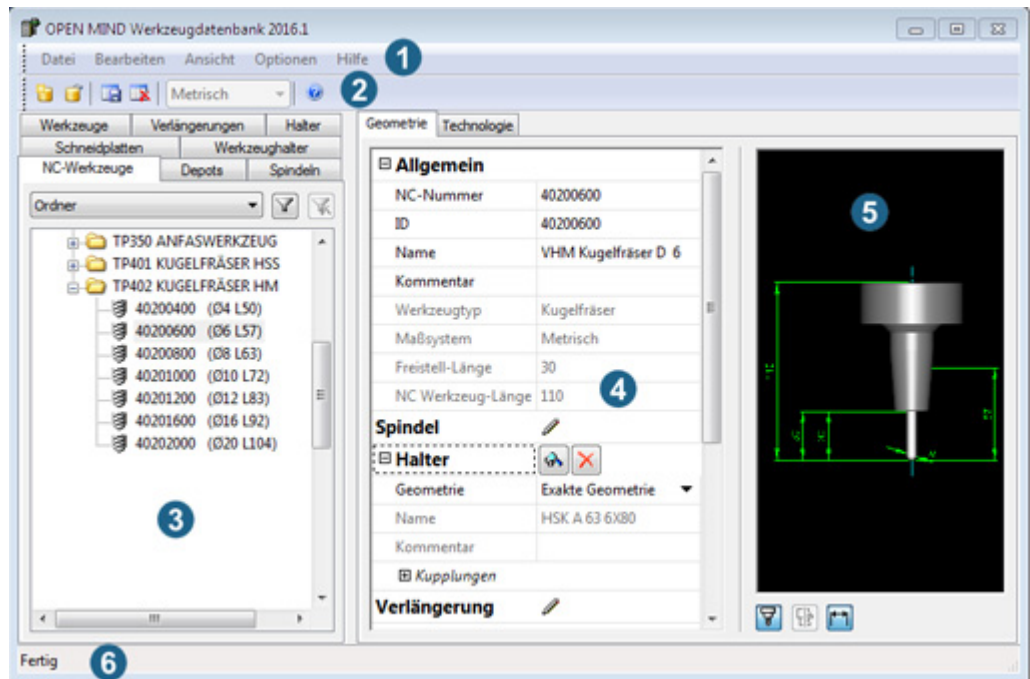


Click the icon in the *hyperMILL* browser, **Jobs** tab.

User interface

The user interface consists of the following elements:

(1) Menu bar, (2) Toolbar, (3) Browser, (4) Data view of the overview window, (5) Graphics area, (6) Status bar.








The menu bar

File	New...: Create a new database. Open...: Open an existing database.
Edit	Cut, Copy and Paste: Standard Windows functions that are valid for all entries in the data-base browser.
View	Show or hide the toolbar , status bar , browser window .
Options	Accuracy: Measurement system configuration.
Help	Tool Database Help: Start. Info...: Display version number and other version details.



The toolbar

	Save or Delete a data record.
	Switch measurement system (metric/inch).
	Start the online help.
	Geometry preview 1. Switches between 2D and 3D representation. 2. Switches between parametric and free tool geometry. 3. Show dimension.
	Create a new database or open an existing database .

The browser

In the browser, all elements of the database are created and displayed. You can use the dropdown combo box to change the structure of the browser display.

For information on the entry selected in the browser window, refer to the data view of the overview window and the graphical preview.

The display structure for the data view of the overview window (detailed view / list view) depends on the entry selected in the browser.

If you have selected a folder, the elements of this folder are displayed in a list view within the data view.

You can define the parameters displayed in the list view by right-clicking in the upper area of the data view and activating the required parameters.

Tool name	▲ Tool length	
Rad. D 5.0 HSS	60	<input checked="" type="checkbox"/> Tool name
Rad. D 4.0 HSS	80	<input checked="" type="checkbox"/> Tool length
Rad. D 3.0 HSS	80	<input checked="" type="checkbox"/> Tool diameter
		<input checked="" type="checkbox"/> Tip length

If you have selected one element within a folder, this element is displayed in a detailed view within the data view.

Create database

To create a new database:

1. Select **File > New** on the menu bar and specify a **name**.
2. Now click **Create** to create the database.



You can use a password to protect all database entries. Information in section **Changing the password**.



The newly created database appears as an entry in the database browser.

To close the database, right-click the database entry and select **Close**.

To clearly group your tools (and other database elements such as holders or extensions), you can create new folders in the database browser. If the folder names begin with a combination of numbers (such as 001, 002 and so on), the folders are listed in the browser in



numerical order. We also recommend using two or three (maximum) hierarchy levels for the folder structure of your database.

Creating a folder

To create a new folder in the browser window:

1. Select the **Folder** entry from the combo box for the browser window.
2. Right-click the database entry and select **New > Folder**.
3. Specify a **name** and, if necessary, enter a **comment**. Comments appear as a tooltip.

Defining a tool

After you have created a new database and the necessary folders, you can begin to define a new tool.

1. Switch to the **Tools** tab, right-click the previously created database entry and select **New**.
2. Select the required tool type from the list.
3. Specify a unique **name** for the tool type in the data view. A **comment** can later make it easier for you to select the correct tool. Enter any other required parameters.



If you define a non-permitted value for a parameter, the corresponding line in the data view is marked in red.

The newly created tool appears as an entry in the database browser. Geometry and technology data is displayed in the data view of the overview window. You can also save a tool as a default. All the geometry and technology parameters are saved and used if you define this tool type again.

To save a tool as default, select the tool in the database browser and select **Save As Default** in the shortcut menu.

The following sections list all tool types available in the tool database.

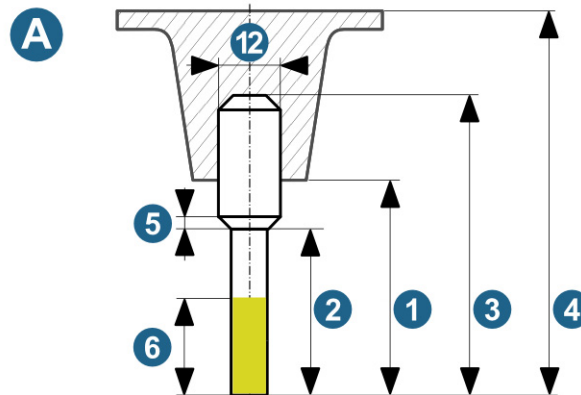


The **chamfered profile cutter** and **countersink** tool types are not currently supported by the *hyperMILL* cycles.

The **Reamer** tool type has been supported since Version 2014.1.

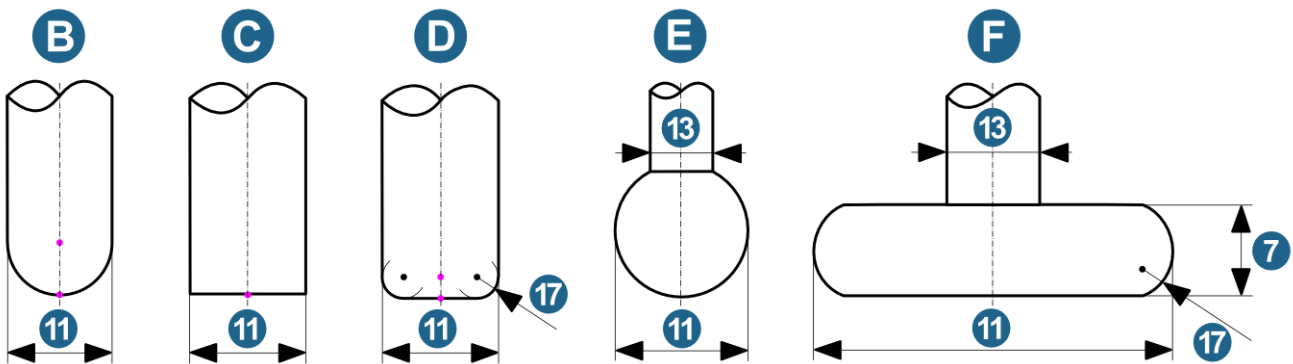
Milling Tool

Generally applicable (A) and type-specific geometry parameters.



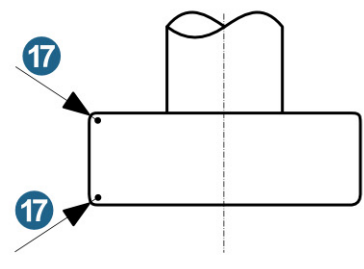
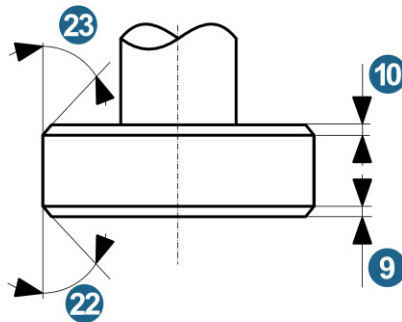
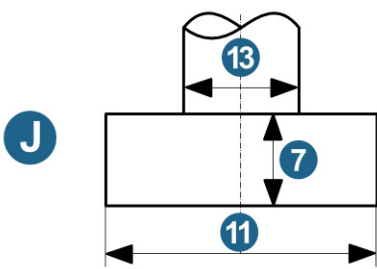
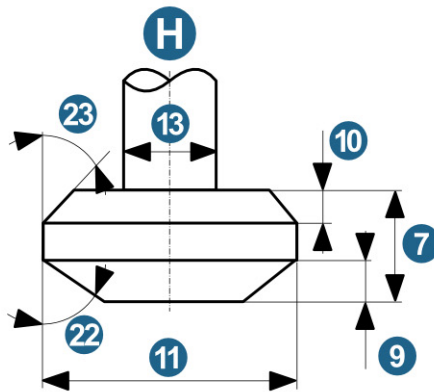
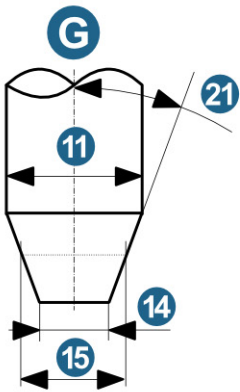
Lengths	(1) Tool reach, (2) Tip length, (3) Tool length, (4) Gage length, (5) Chamfer length, (6) Cutting length, (7) Disc height, (8) Barrel height, (9) Lower chamfer height, (10) Upper chamfer height,
Diameters	(11) Diameter, (12) Shank diameter, (13) Collar diameter, (14) Tip diameter, (15) Nominal diameter, (16) Base diameter
Radii	(17) Corner radius, (18) Lens radius, (19) Barrel radius, (20) Base corner radius
Angles	(21) Chamfer angle, (22) Lower chamfer angle, (23) Upper chamfer angle, (24) Barrel taper angle

Ball endmill (B), Endmill (C), Bullnose endmill (D), Lollipop, Touch probe tool (E), Woodruff (F):

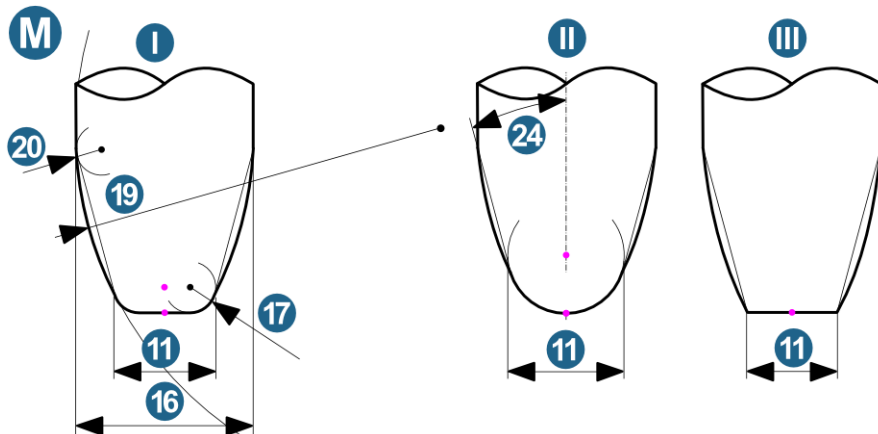
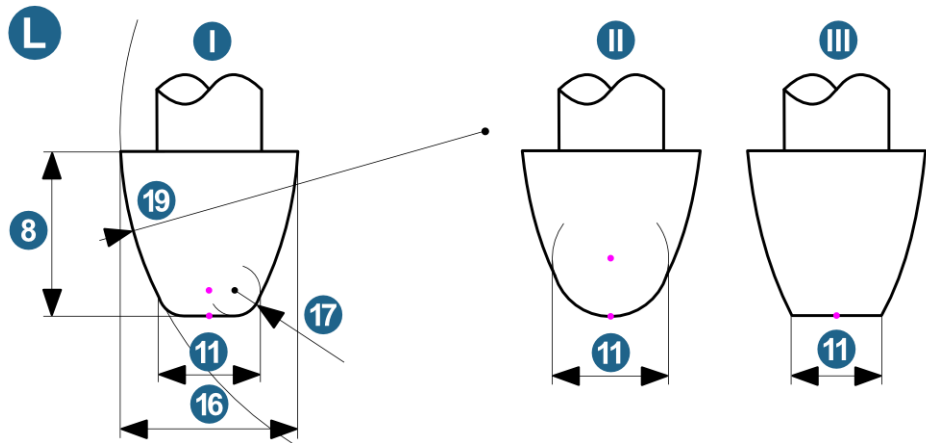


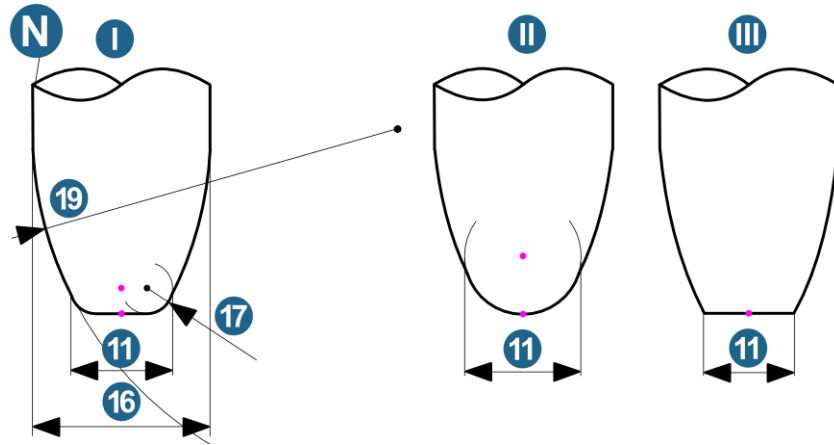
For the ball mill, the radius of the cutting edge always corresponds to half of the tool diameter.

Chamfer cutter (G), Chamfered profile cutter (H), T-slot cutter (J), Lens cutter (K):



General barrel tool (L), Conical barrel tool (M) and Tangent barrel tool (N) ball (I) and sharp (III) variant, with corner fillets (II).



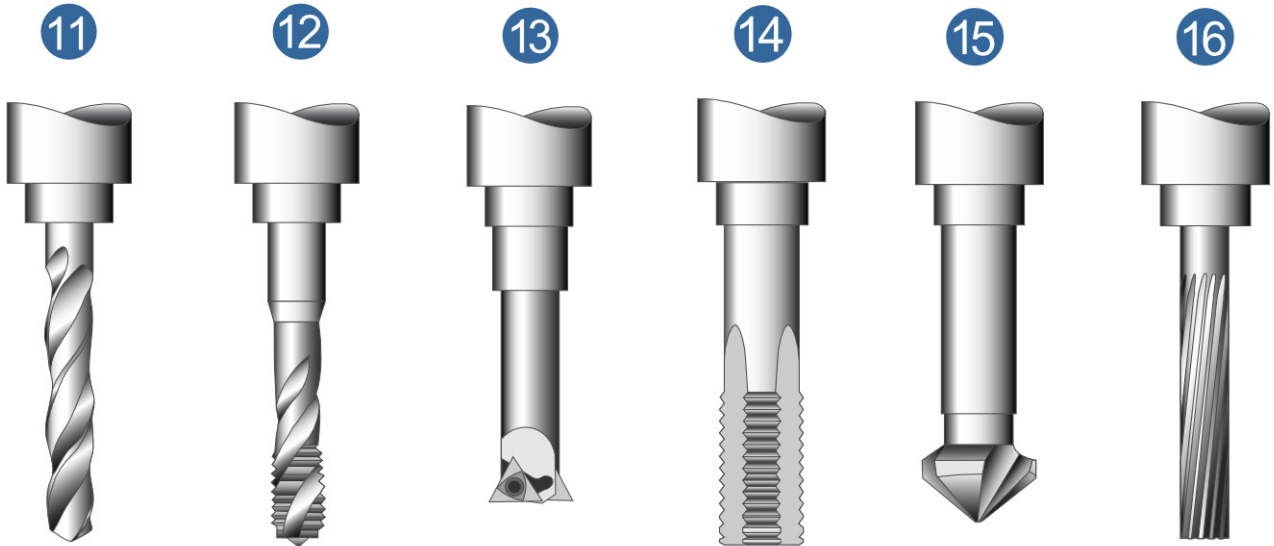




Drilling tools

The following drilling tools are available in the tool database:

(11) Drilling tool, (12) Tap drill, (13) Boring bar, (14) Thread mill, (15) Countersink, (16) Reamer.

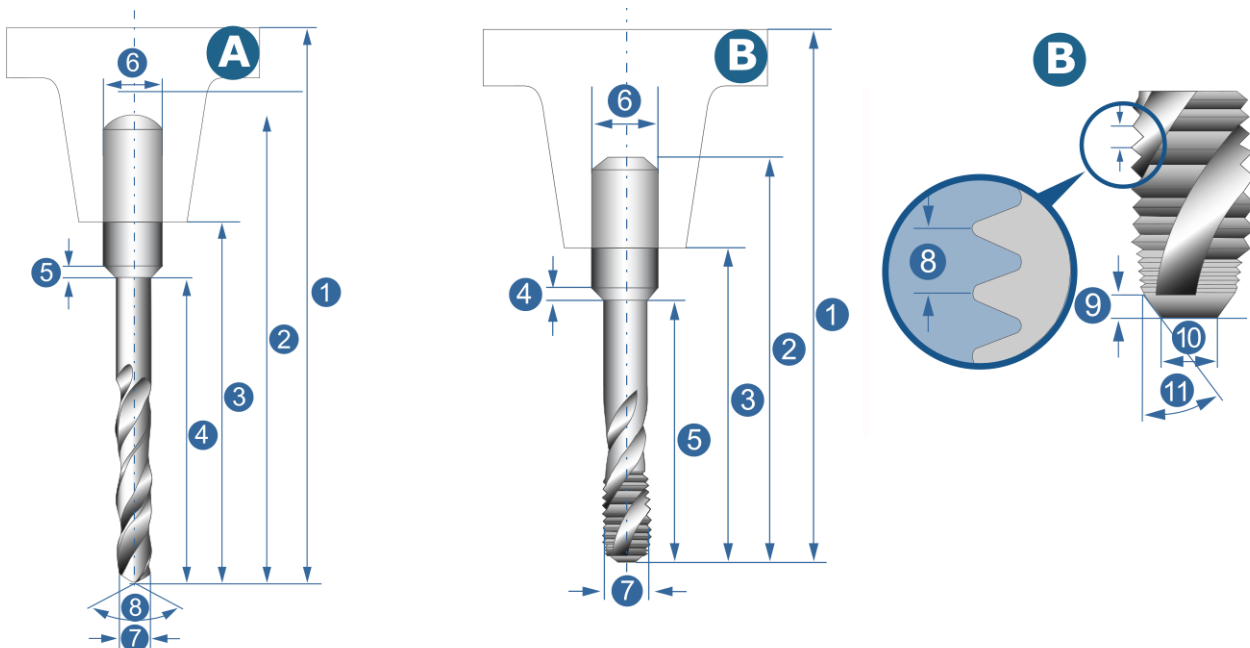


Geometry parameter tool type: **Drilling tool (A), Tap drill (B)**

(1) NC tool length, (2) Tool length, (3) Tool reach, (4) Tip length, (5) Chamfer length, (6) Shank diameter, (7) Diameter.

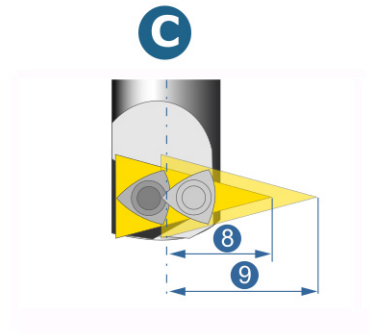
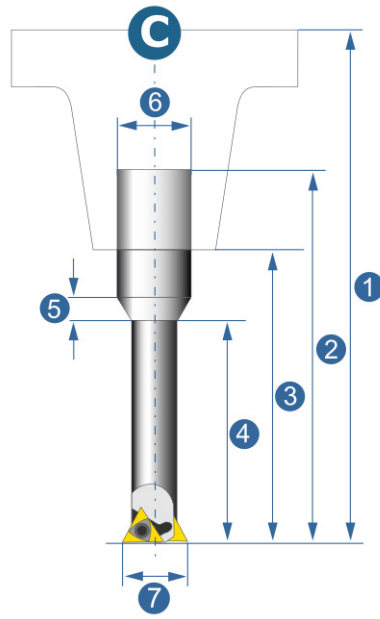
Drilling tool (A): (8) Tip angle.

Tap drill (B): (8) Pitch (p), (9) Lead in length, (10) Lead in diameter, (11) Back angle.



Tool type: **Boring bar (C)**

(1) NC tool length, (2) Tool length, (3) Tool reach, (4) Tip length, (5) Chamfer length, (6) Shank diameter, (7) Diameter, (8) Minimal diameter, (9) Maximal diameter.

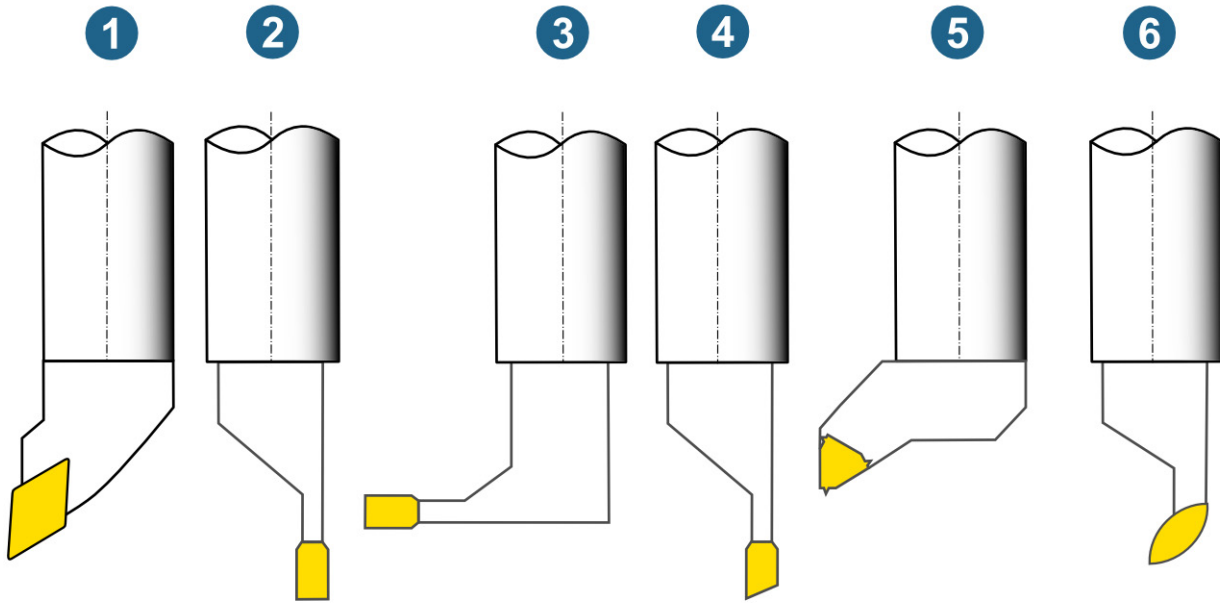




Turning Tool

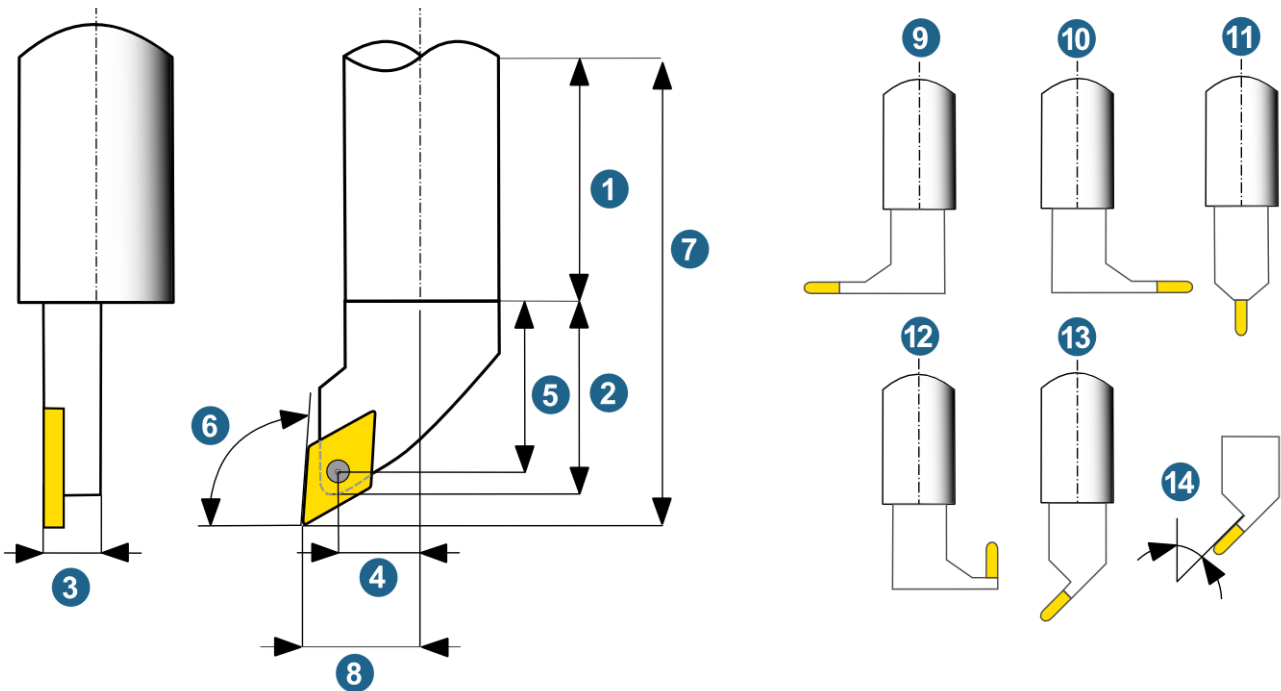
The following turning tools are available in the tool database:

(1) Turning tool, (2) Recessing tool, (3) Axial recessing tool, (4) Parting tool, (5) Threading tool, (6) Roll turn tool.



General geometry parameters

(1) Shank geometry, (2) Extruded geometry, (3) Extrusion thickness, (4) Axis distance, (5) Shank distance, (6) Approach angle, (7) Setting length X (8) Setting length Z (9) Mounting direction left, (10) Right, (11) Top, (12) Inverse, (13) Angle, (14) Mounting direction angle.



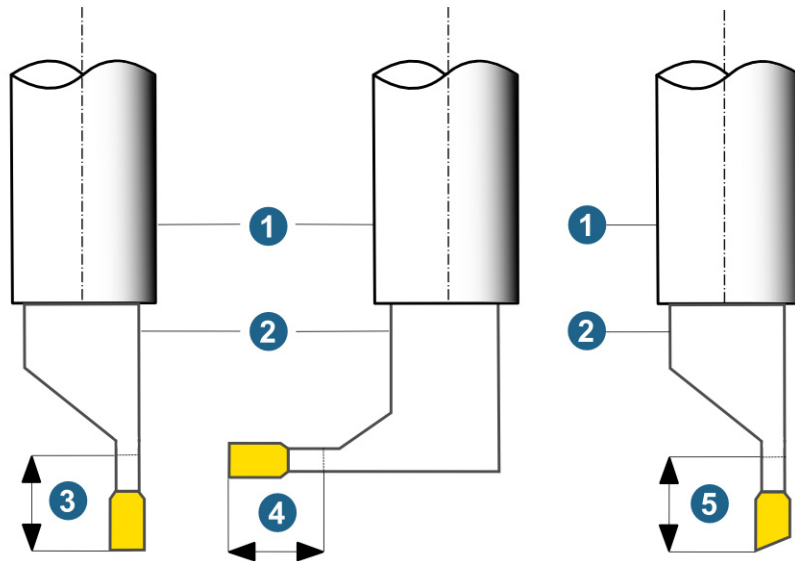


The Axis distance (4) and Shank distance (5) geometry parameters in the graphical preview are with reference to the insert lying flat in the Z-X plane and not to the actual tilted view.

Toolpath calculations are also with reference to a cutting point, which is calculated in this view.

Geometry data: recessing tool, axial recessing tool, parting tool

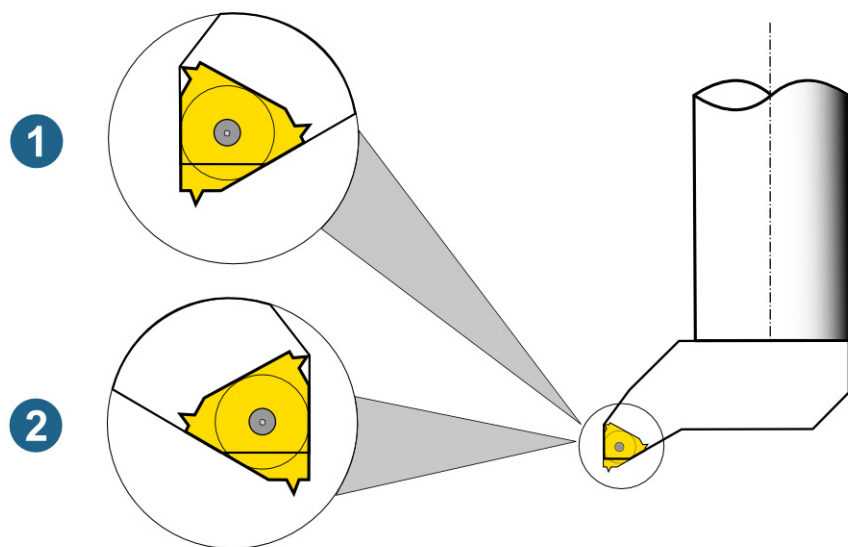
(1) shank geometry, (2) extruded geometry, (3) maximal radial depth, (4) maximum axial depth, (5) maximum cutting depth.



The largest radial / axial depth and the maximum depth of cut relate to the groove or recess to be machined. The value may be greater than the length of the insert.

Geometry parameters threading tool

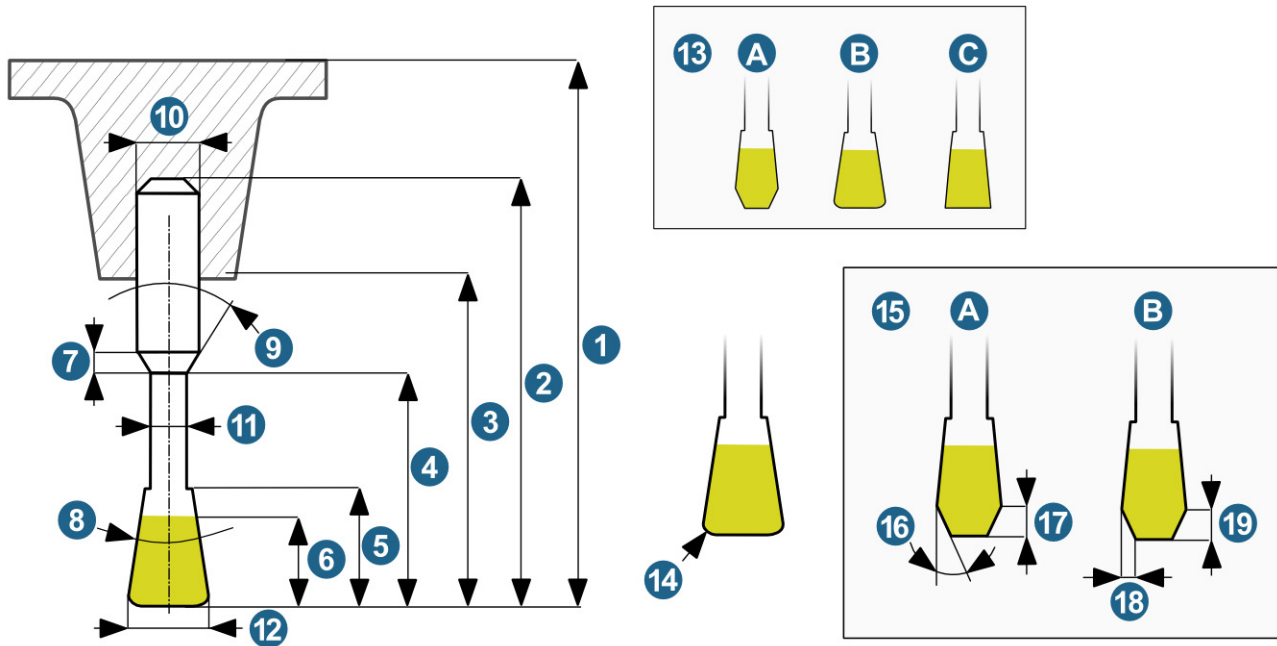
(1) reversed not activated, (2) reversed activated





Grinding bit

(1) NC tool length, (2) Tool length, (3) Tool reach, (4) Tip length, (5) Grinding height, (6) Cutting length, (7) Chamfer length, (8) Cone angle, (9) Chamfer angle, (10) Shank diameter, (11) Collar diameter, (12) Diameter, (13) Lower cutting edge: (A) Chamfer, (B) Radius, (C) Sharp, (14) Radius, (15) Lower chamfer mode: (A) Angle and height, (B) Width and height, (16) Angle, (17) Height, (18) Width, (19) Height.



Defining tool geometry

General

Define **name**, **comment** (optional) and **measurement system**.

Geometry

Define the **Diameter** and **Length** of the tool.

Shank

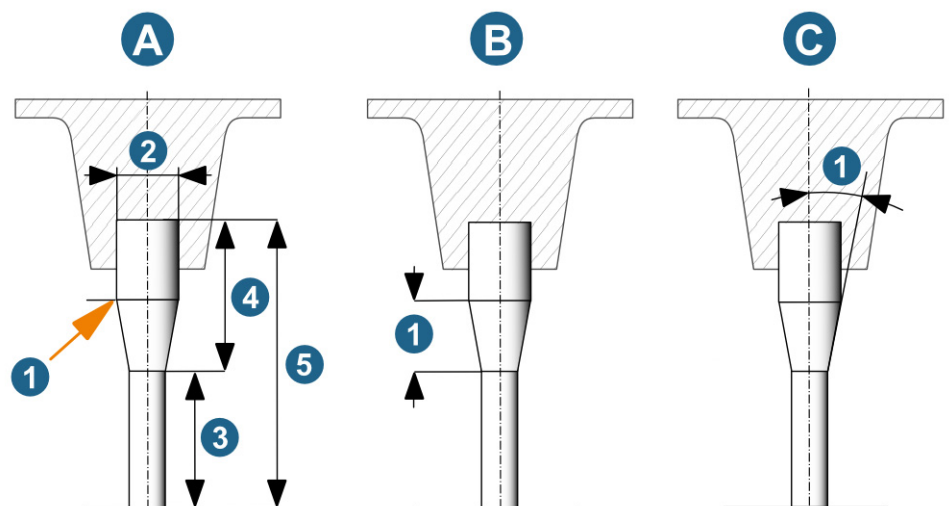
Shank mode: Select either the **Free** or **Parametric** option to define the shank. Select **None** if a shank is not necessary.

In the **Free** shank mode, use the geometry editor to define the shape of the tool shank. See chap. **The geometry editor** for information on the geometry editor.

	<p>Open the geometry editor in the Free shank mode. Information in section The geometry editor.</p>
	<p>Delete freely defined geometry.</p>

In the **Parametric** shank mode, specify the **shank diameter** and the method of **chamfer definition**. With the **Absolute** (A) chamfer definition, specify a point for the **chamfer end position** (1) as an absolute value. With the **Length** (B) chamfer definition, specify the **chamfer length** (1) and with the **Angle** (C) chamfer definition, specify the **chamfer angle** (1).

(2) Shank diameter, (3) Tip length, (4) Shank length, (5) Tool length.



Tip

Define the parameters for the tip of the tool. Information in chap. **Milling Tool** and chap. **Drilling tools**.

To define a conical tool, enable the **Conical tip** option and define the **Cone angle**.

	<p>Free tip geometry: Open the geometry editor and define the tip geometry. In the Cutting area column, enable line segments that define the cutting part of the tool tip.</p>
--	--



The non-cutting part of the geometry is used in the simulation to perform collision checks.



The length of the tool tip is calculated automatically if the following conditions are fulfilled: The Conical tip parameter is enabled, the shank mode is set to Parametric, Length is defined as the chamfer definition and the Chamfer length is 0.

The length of the tool tip is calculated automatically if the following conditions are fulfilled:

- The Conical tip parameter is enabled,
- the shank mode is set to Parametric,
- length is defined as the chamfer definition and
- the Chamfer length is 0.

The shank diameter is calculated automatically if the following conditions are fulfilled:

- The Conical tip parameter is enabled,
- the shank mode is set to Parametric,
- a free tip geometry is defined,
- the Chamfer length is 0.

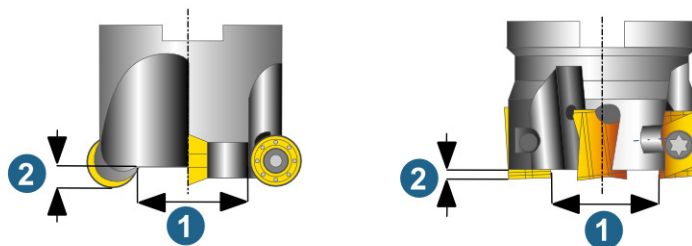
The following applies to the shank definition: Length of the chamfer ≥ 0 and length of the chamfer \leq shank length.

Defining a tapered tool with a shank

1. Select the required tool type on the **Tools** tab.
2. Specify the **Shank mode** on the **Geometry** tab in the data view in the **Tool** area and define the values for the **diameter** and **length** of the tool
3. Enable the **Conical tip** parameter in the **Tip** area and define the **Cone angle**.
4. Specify the required parameters in the **Shaft** area

Insert Parameters

It is possible to define the two **Core diameter** (1) and **Core height** (2) parameters for tools with inserts.



Top coupling

Define as standard for all tools. The **top coupling** connects the tool with another component (holder, extension) and is used when assembling an NC tool. Define the **Minimal length** and the **Maximal length**.

Defining a turning tool

1. On the **Tools** tab, right-click the corresponding database entry and select **New > [Tool type]**.
2. Select or redefine the **Insert** and **Tool Holder** components.



The Tool holder and Insert components are necessary to define a turning tool.

A Tool holder is made up of the Shank geometry (Part 1 and 2) and the Extruded geometry. It contains all placement information to support inserts.

By defining the Insert support you also set the Insert type to be supported by the tool holder, and via the Clamping method, the manner in which an insert is to be mounted on the tool holder.

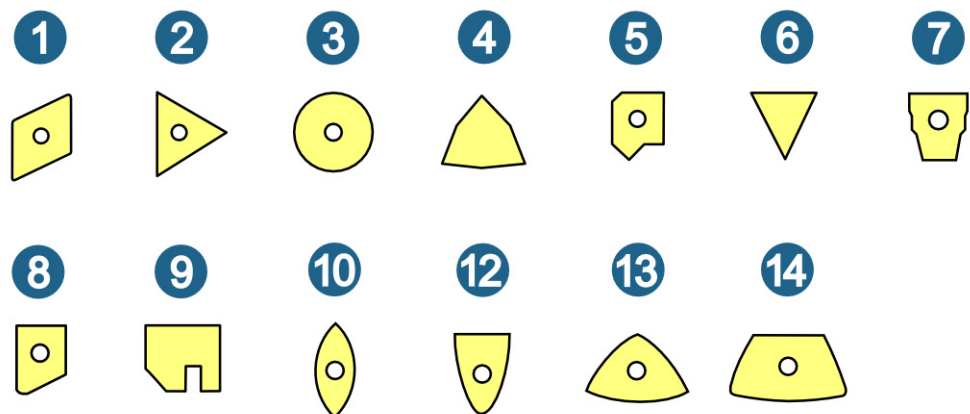
The Insert component is also characterised by geometric parameters as well as a Mounting type. In addition, it is possible to assign to an insert the technology parameters Material, Cutter material and Purpose.

Defining an insert

Select a shape of insert in the **Insert** tab via the **New > [Shape of insert]** shortcut menu.

Shape of insert:

(1) Rhombic, (2) Triangular, (3) Round, (4) W-shape, (5) Thread, (6) Simple thread, (7) Rectangle, (8) Parting, (9) Free form (10) X Full Roll Turn, (11) X Short Roll Turn, (13) T Full Roll Turn, (14) T Short Roll Turn.

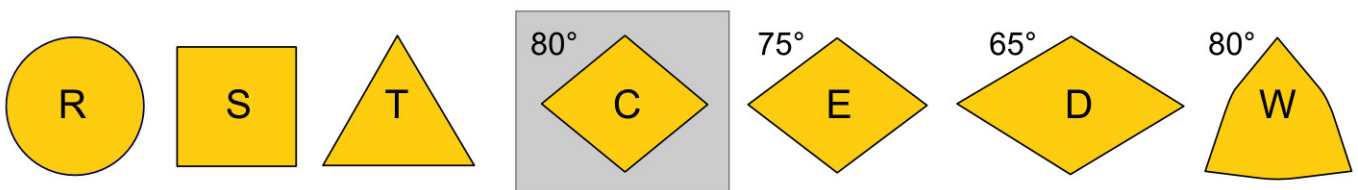


Alternatively, open the **Edit ISO/ANSI code...** dialogue via the **From ISO/ANSI Code...** short-cut menu. Each digit of the eight-digit ISO code stands for a parameter that can be defined using a selection field. Information on the definition of the inserts according to the ISO/ANSI standard can be found in the following example.

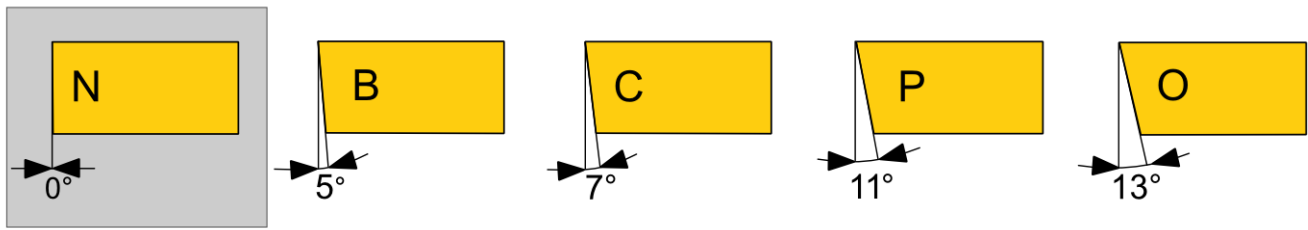
Code	C	N	M	G	12	04	08	EL-
Digit	1	2	3	4	5	6	7	8

Numbers 1 to 8 of the ISO/ANSI code:

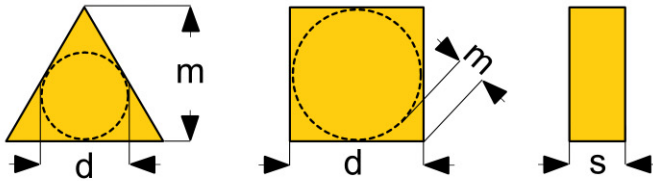
(1) Shape



(2) Free angle

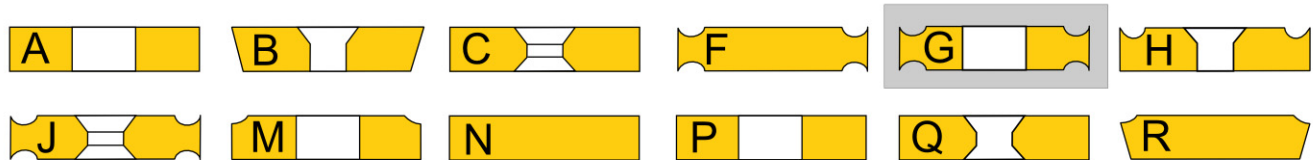


(3) Tolerance

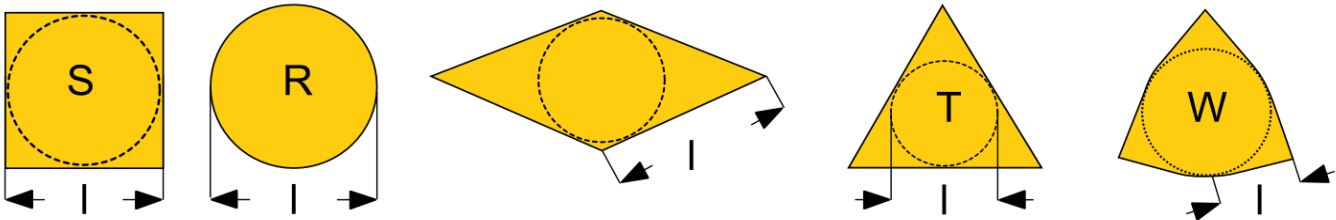


	m	s	d
E	±0.025	±0.025	±0.025
G	±0.025	±0.13	±0.025
M	±0.08 <> ± 0.18	±0.13	±0.05 <> ±0.13
U	±0.13 <> ± 0.18	±0.13	±0.13 <> ±0.25

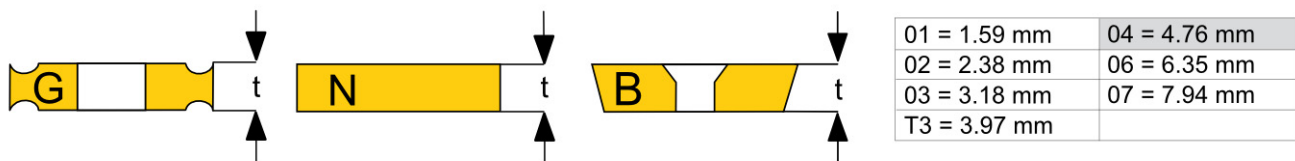
(4) Type of insert



(5) Length

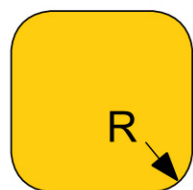


(6) Thickness



01 = 1.59 mm	04 = 4.76 mm
02 = 2.38 mm	06 = 6.35 mm
03 = 3.18 mm	07 = 7.94 mm
T3 = 3.97 mm	

(7) Corner radius

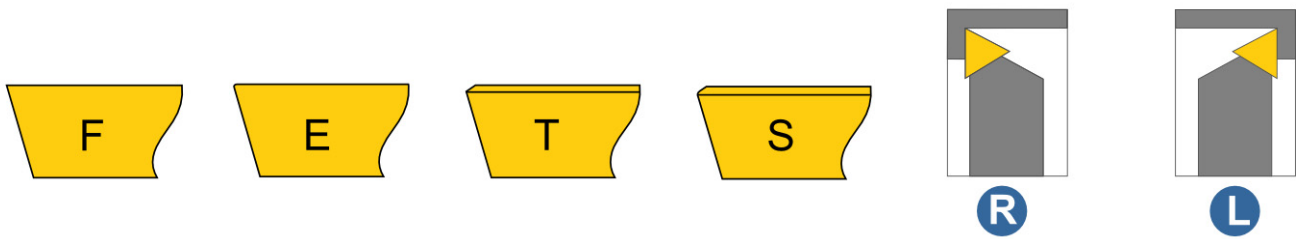


02 = 0.2 mm	16 = 1.6 mm
04 = 0.4 mm	20 = 2.0 mm
08 = 0.8 mm	24 = 2.4 mm
12 = 1.2 mm	

(8) = Forming of the cutting edge and chip former designation:

(F) Sharp, (E) Rounded, (T) Chamfered (negative chamfer), (S) Chamfered and rounded,

(R) Right model, (L) Left model.



Permitted combinations of tools and inserts:

Turning tool: Rhombic, triangular, round and W-shape insert

Recessing tool: Round, rectangular and free form insert

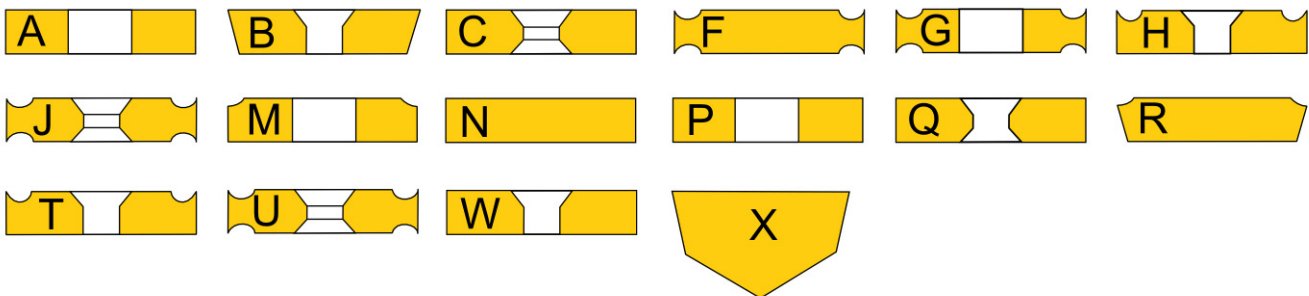
Axial recessing tool: Round, rectangular and free form insert

Threading tool: Threading insert

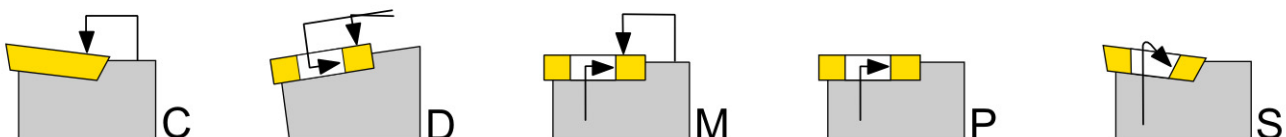
Specify a **name**, **comment**, **measurement system** and **type of insert**.

For free form inserts, the **Insert type** is not set, because this insert cannot be assigned to any specific **Clamping method** (on the part of the tool holder).

Insert types: A- Through hole - No chip breaker, B - Hole chamfered at one side - No chip breaker, C - Hole chamfered at both sides - No chip breaker, F - No hole - No chip breaker, G - Through hole - Chip breaker at both sides, H - Hole chamfered at one side - Chip breaker at one side, J - Hole chamfered at both sides - Chip breaker at both sides, M - Through hole - Chip breaker at one side, N - No hole - No chip breaker, P - Through hole - No chip breaker, Q - Hole chamfered at both sides - No chip breaker, R - No hole - Chip breaker at one side, T - Hole chamfered at one side - Chip breaker at one side, U - Hole chamfered at both sides - Chip breaker at both sides, W - Hole chamfered at one side - No chip breaker, X - Special shape



Available clamping methods (tool holder): C - Top clamping, D - Rigid clamp, M - Top and hole clamping, P - Hole clamping, S - With screw through hole.




Permitted combinations of insert type and clamping method (tool holder) are shown on the following table.



		Insert type (insert)													
		A	C	F	G	H	J	M	N	P	Q	R	T	U	W
Clamping method (tool holder)	C	X	X	X	X	X	X	X	X	X	X	X	X	X	X
	D	X	X		X	X	X	X		X	X		X	X	X
	M	X	X		X	X	X	X		X	X		X	X	X
	P	X	X		X	X	X	X		X	X		X	X	X
	S		X			X	X				X		X	X	X

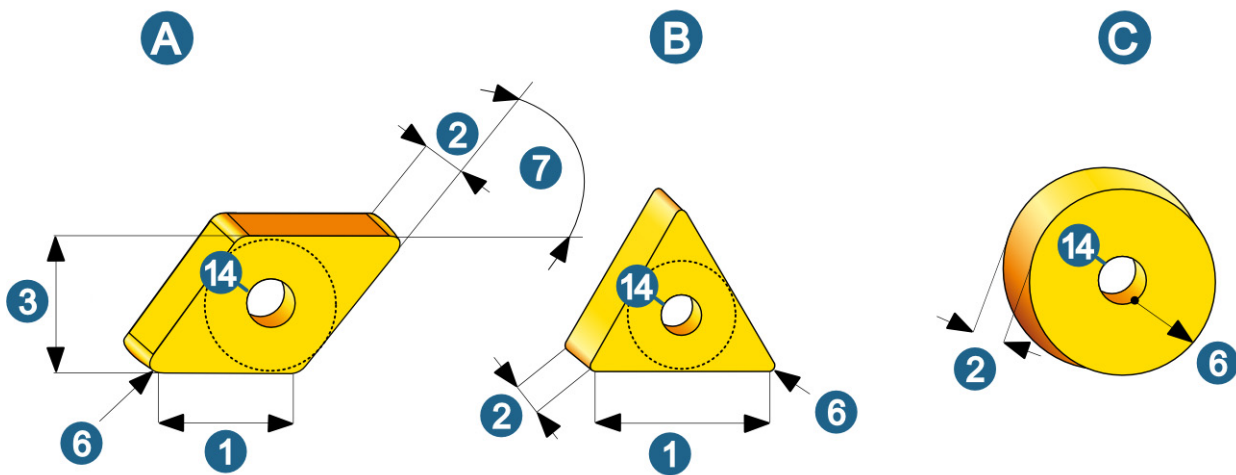
To define the insert, specify the required parameters in the **Geometry** area. To change the geometry of an insert that was defined with an ISO/ANSI code, remove the checkbox at **ISO/ANSI code active**.

	Click the icon to change the ISO/ANSI code.
---	---

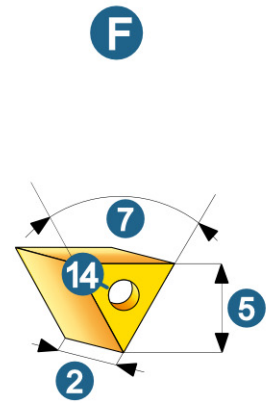
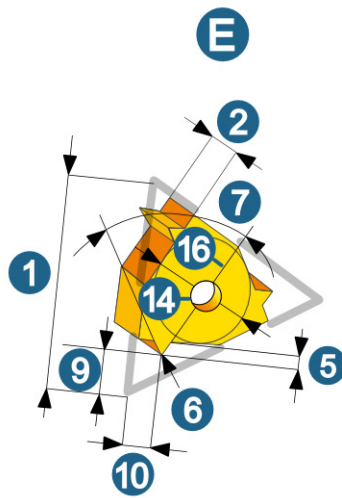
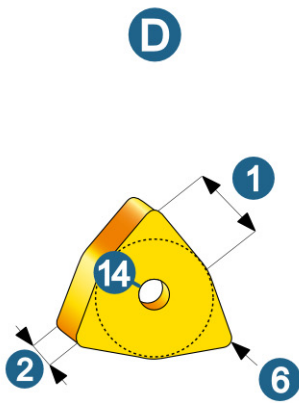
Insert geometry parameters

Lengths	(1) Length, (2) Thickness, (3) Height, (4) Cutting width, (5) Cutting height, (20) Width
Radii	(6) Corner radius, (17) Radius
Angles	(7) Angle, (8) front angle
Offset	(9) Tip X offset, (10) Tip Z offset (11) Reference offset X, (12) Reference offset Z,
Other	(13) Reference point, (14) Mounting point, (15) Cutting area, (16) Inscribed circle diameter, (18) Centre distance Z, (19) Centre distance X, (21) Mounting point, (22) Primary centre, (23) Secondary centre

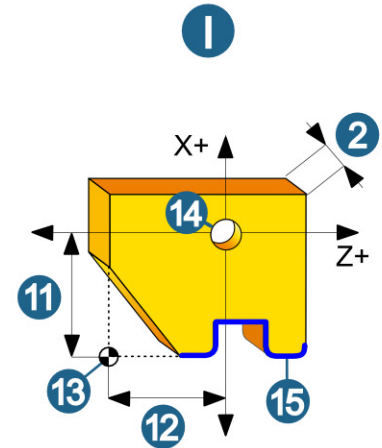
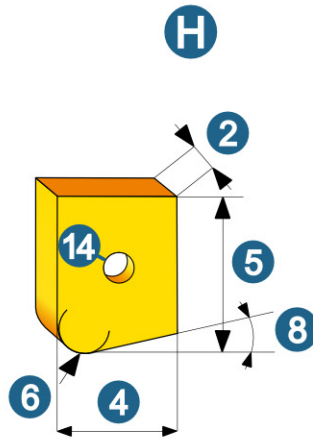
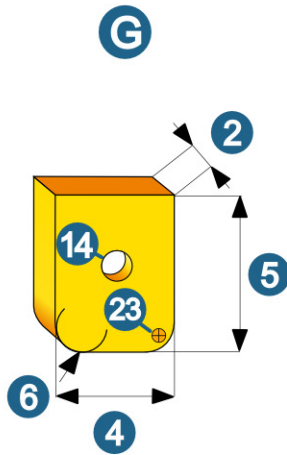
(A) Rhombic insert, (B) triangular insert, (C) round insert.



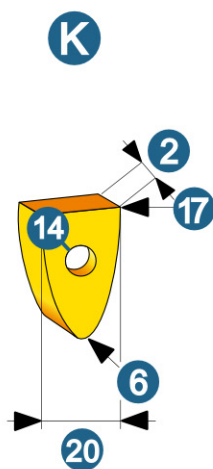
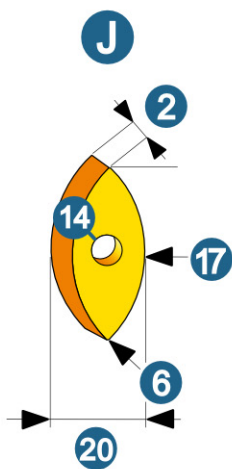
(D) W-Shape insert, (E) threading insert, (F) simple thread insert.



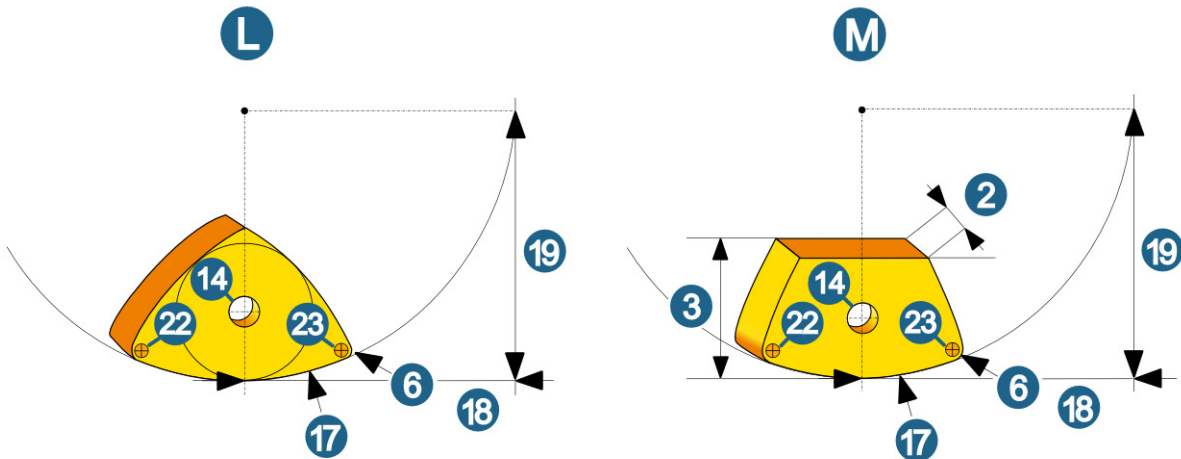
(G) Rectangular, (H) parting insert, (I) free form insert.



(J) X Full Roll Turn, (K) X Short Roll Turn.



(L) T Full Roll Turn, (M) T Short Roll Turn.



Defining the geometry of the free form insert

The geometry of the Free form insert is defined together with the tool definition in *hyperMILL* by means of contour and point selection in the CAD system.

1. On the **Geometry** tab in the tool definition, click **Insert** in the **Tool** area and select **Free form insert**.
2. Click **Geometry** and then click icon 1 to start contour selection in the CAD system.
3. Select contour, **Cutting area**, **Mounting point** and **Mounting direction point**.

☐ Geometry	
Reference offset x	0
Reference offset z	0
Thickness	3
Geometry	
1	

You can also use the Geometry editor from the tool database to define the geometry of the Free form insert. The geometry of the Free form insert must be defined as a closed contour. See chap. **The geometry editor** for information on the geometry editor.

In the **Cutting area** column of the Geometry editor, select the contour elements that represent the **Cutting area** of the free form insert and confirm your definition with **OK**.

The **Cutting area** must match the contour that is to be processed.

Reference point and **Cutting area** are displayed in the Preview window. The exact position of the **Reference point** results from the **Reference offsets X** and **Z**.

Inserts technology parameters

The Feedrate unit parameter (mm/min or mm/rev) is used to simultaneously switch the values for the Feedrate XY, Reduced feedrate and Feedrate retract parameters. These parameters are also displayed in the NC tool cutting profile, but cannot be edited there.

Defining a tool holder

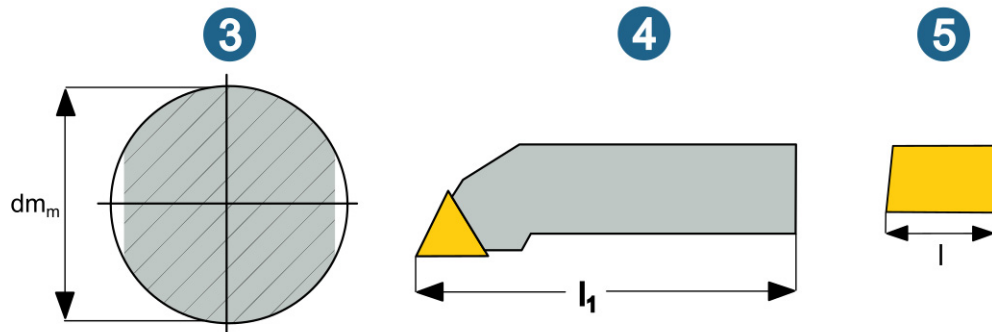
1. On the **Tool holder** tab, select the **New tool holder** function from the shortcut menu.

2. Under **Type**, select **Internal** for turning tools for inside machining, or **External** for turning tools for outside machining. If the later application of the tool holder has not yet been determined, select the **Unknown** option.
3. Specify a **Name** and optionally a **Comment** for the tool holder.
4. Define the **Measurement system** (metric / inch).
5. If the tool holder should be defined using an ISO/ANSI code, enable the **ISO/ANSI code active** checkbox.

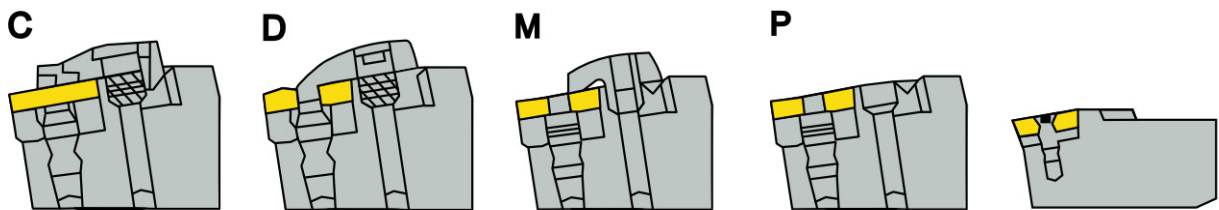
6.  Open the **Edit ISO/ANSI code** dialogue by clicking the icon.

Each digit of the ISO/ANSI code stands for a parameter that can be defined using a selection field. Depending on whether a tool holder is defined for inside or outside machining (internal/external), different parameters will be available for the ISO/ANSI code. Possible ISO/ANSI parameters are:

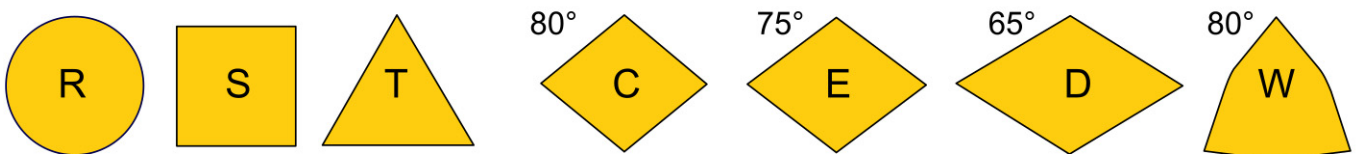
- (1) Type: Internal: Turning tool for inside machining, External: Turning tool for outside machining
- (2) Boring bar type
- (3) Shank diameter, (4) Length, (5) Insert length



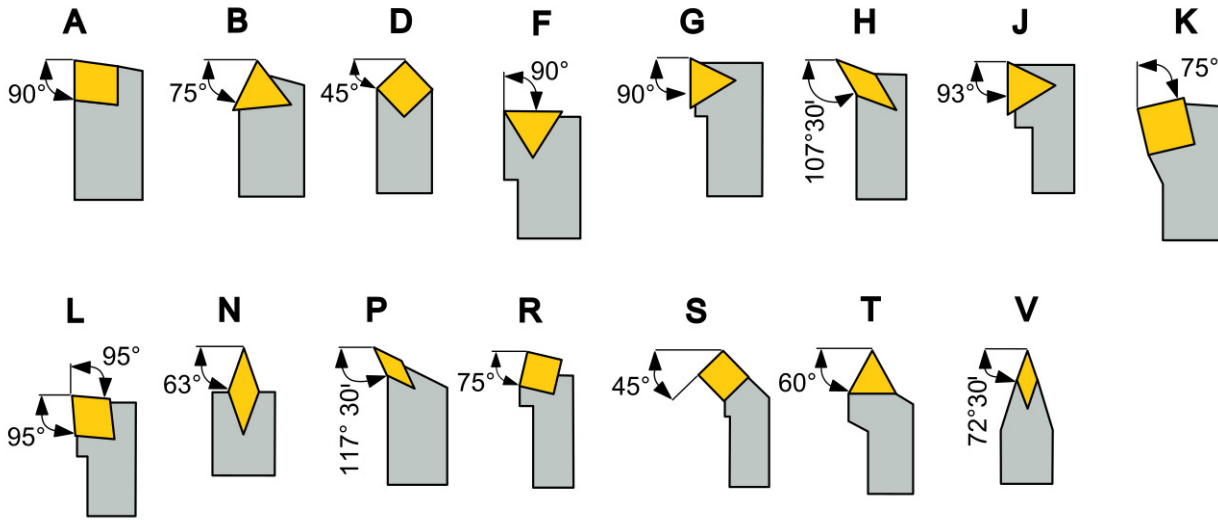
(5) Clamping method



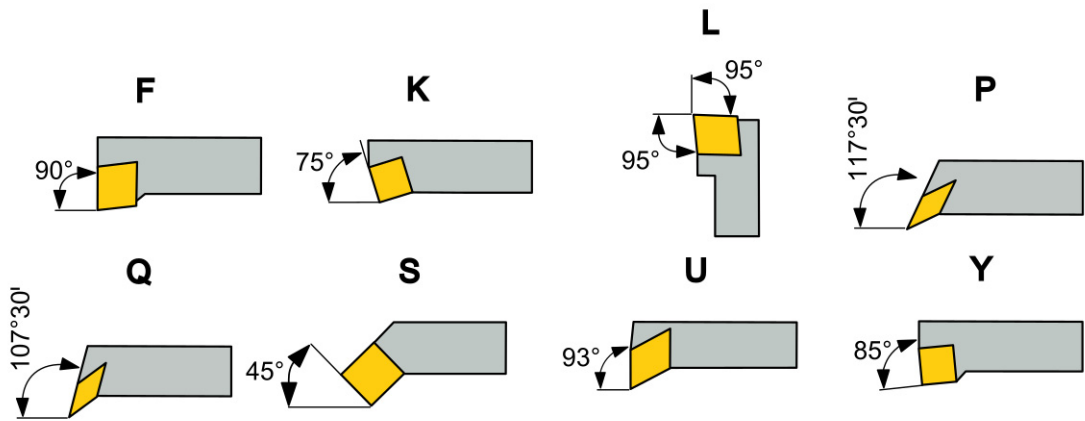
(6) Shape of insert



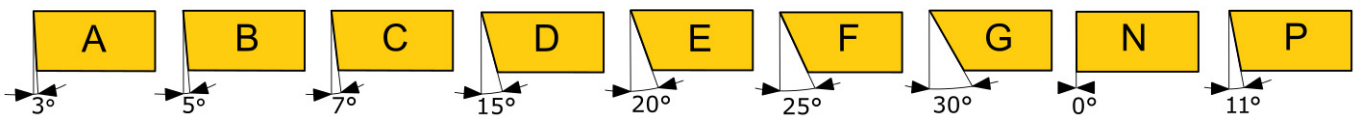
(7) Style, Type: external



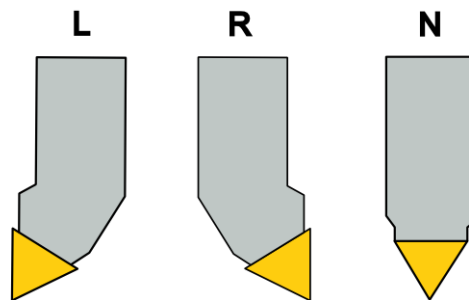
(7) Style, Type: internal



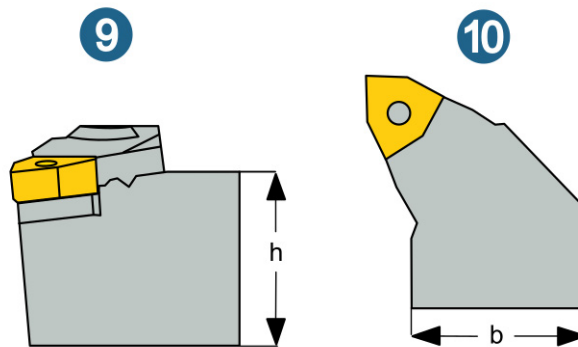
(8) Free angle



(9) Mounting direction (left, right, neutral)



(9) Shank height, (10) Shank width



Define all further necessary parameters in the **Insert support**, **Geometry** and, if necessary, **Advanced tool definition** areas.

Defining an insert support

1. Define which insert the tool holder is to support. For information on the available inserts, see chap. **Defining an insert**.
2. The following options to define the Insert support are available:
 - Use the selection field (1) to select the shape of the insert and define the geometric parameters or
 - select an existing insert directly by clicking the icon (2) or click the icon (3) to define an ISO/ANSI code using the **Edit ISO/ANSI code** dialogue.

[-] General	
Name	Insert 01
Comment	
Measurement System	Metric
[-] Insert support	<div style="display: flex; align-items: center;"> <div style="border: 1px solid gray; padding: 2px; margin-right: 5px;">Rhombic insert</div> <div style="margin-left: 5px;"> </div> </div>

If you define the insert support by means of the ISO/ANSI code, the geometry parameters are initially firmly linked to the ISO/ANSI code and cannot be edited. To edit the geometry parameters, click the icon (4) to break the link between the geometry parameters and ISO code.

To mount an insert to the tool holder, you must use an appropriate clamping method. Information on possible combinations of clamping methods and insert types can be found in chap. **Defining an insert**.

Defining a tool holder geometry

The following options are available to define the tool holder geometry:


- Import the geometry as a 3D model using the *hyperMILL* TOOL Builder
- In the tool database with the geometry editor
- In *hyperMILL /* in the CAD system

Importing as a 3D model

Import a tool holder from a 3D model (format: *.stl) using the *hyperMILL* TOOL Builder. To do so:

1. Enable the **3D data** option.






-  Start the *hyperMILL* TOOL Builder. For more information, see the *hyperMILL* TOOL Builder product documentation.



For 3D geometries of tool holders, it is no longer possible to edit the profile in the Geometry editor. For information on the *hyperMILL* TOOL Builder, see the *hyperMILL* TOOL Builder product documentation.

In the tool database with the geometry editor

To define the geometry in the tool database, click the icon (1) to open the Geometry editor. See chap. [The geometry editor](#) for information on the geometry editor.


<input type="checkbox"/> Geometry	
Shank geometry (Part 1)	 
Shank geometry (Part 2)	
Extruded geometry	
Extrusion thickness	



For tools with a free tip geometry, it is not possible to edit the cutting area in the geometry editor.

In *hyperMILL* / in the CAD system

To define the geometry by selecting contours in the CAD system:

- Create a new turning tool either using the job definition or in the tool browser.
-  Click the icon in the **Edit turning tool** dialogue (Shank geometry (Part 1) area) to select the required contour in the CAD system (warning: the contour should not be closed!). If necessary, define the **Shank geometry (Part 2)** and **Extruded geometry** in the same way.
- Then define the **Mounting point** and **Reference point** for the **Extruded geometry**.



The Mounting point defines the location where the extruded geometry is mount-flanged to shank geometry.

The Reference point defines the alignment of the extruded geometry.

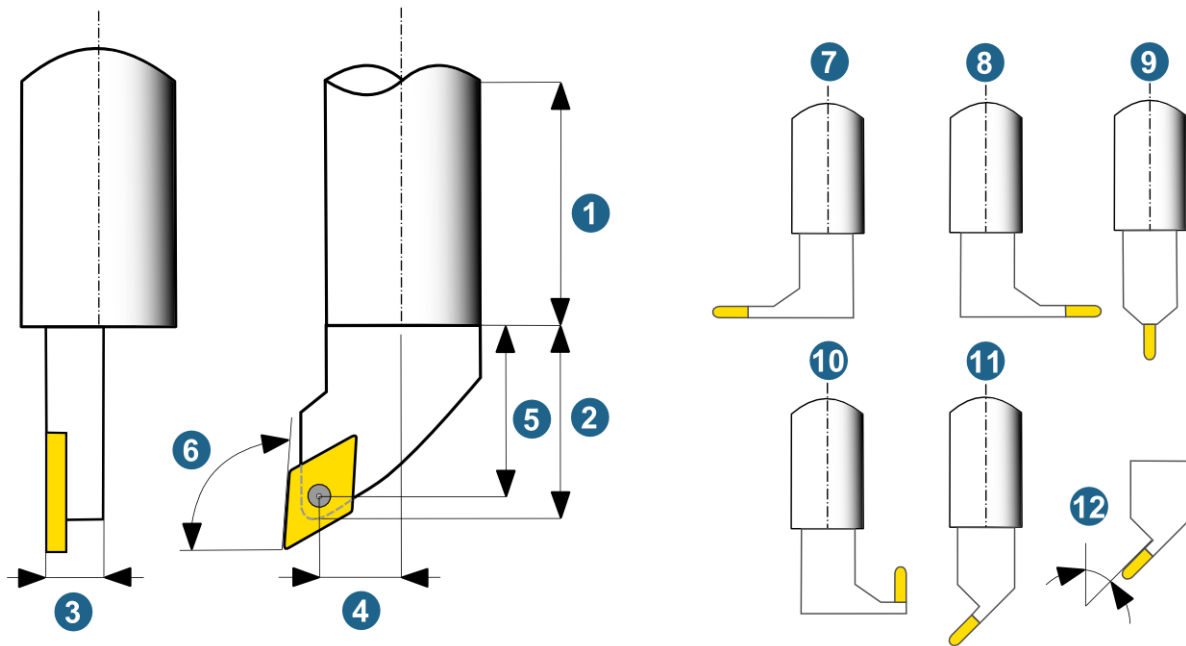
The following applies to the extruded geometry of the tool holder: The placement of the upper edge of the geometry (mounting point) must be on the coordinate $y=0$.

Tool holder geometry parameter

(1) Shank geometry, (2) extruded geometry, (3) extrusion thickness, (4) axis distance, (5) shank distance, (6) approach angle, (7) mounting direction: left, (8) mounting direction: right, (9) mounting direction: top, (10) mounting direction: inverse.

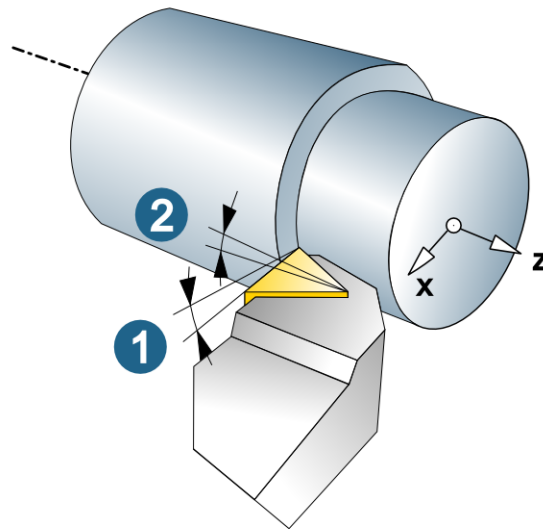
(11) mounting direction: angle and (12) mounting direction angle are also available for tool holders of the recessing tool, axial recessing tool and threading tool types.

The **Side of insert** parameter (left, right, top) is also available for inserts of the recessing tool, axial recessing tool and parting tool types. For the mounting directions left, right and top, the **side of insert** parameter is pre-set accordingly, and cannot be changed manually.

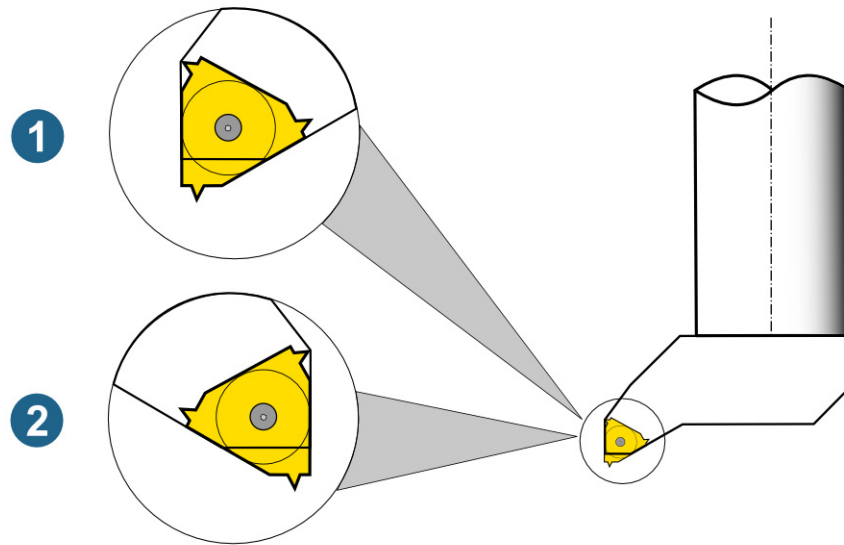


Tool holder geometry parameters: *hyperMILL* TOOL Builder

The *hyperMILL* TOOL Builder calculates the Inclination angle (1) and Rake angle (2), and shows it after import in the tool database graphical preview. These cannot be edited here.

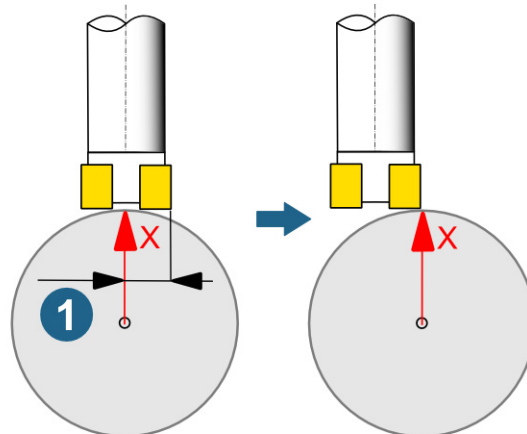


Reversed: enable when the insert is to be installed inverted (only with tool holders for threaded inserts). (1) reversed not activated, (2) reversed activated



Advanced tool definition

Y offset (1): for tools with multiple inserts, the Y offset results in the tool moving from the centre position.



Define couplings

1. In the database browser, select the required database entry and then select **Couplings...**
2. Select the Coupling type in the Couplings dialogue.
3. Specify a **class** to which you want the coupling to belong.
4. To do this, right-click in the lower window pane and select **New**.
5. Define the required geometric parameters and the measurement system for the respective coupling type and then click **OK** to close the window.


Coupling types

The following coupling types are available in the tool database:

	<p>Machining adapter</p> <p>The machining adapter for the tool holder is an important component when using machining simulation in <i>hyperVIEW</i>. Combinations are only possible when the connections fit.</p>
	<p>Morse taper</p> <p>Geometric parameters:</p> <ul style="list-style-type: none"> - Length (1). <p>Combination conditions:</p> <ul style="list-style-type: none"> - can be combined with: Morse taper. - names of couplings are the same.
	<p>Arbor</p> <p>Geometric parameters:</p> <ul style="list-style-type: none"> - Diameter, length. <p>Combination conditions:</p> <ul style="list-style-type: none"> - Components with an arbor - Identical coupling names - Fixed clamping area
	<p>Side lock with external thread</p> <p>Geometric parameters:</p> <ul style="list-style-type: none"> - Diameter, length. <p>Combination conditions:</p> <ul style="list-style-type: none"> - Components with a side lock (internal thread), - identical coupling names. - External thread completely fits into internal thread, - fixed clamping area.
	<p>Side lock with internal thread</p> <p>Geometric parameters:</p> <ul style="list-style-type: none"> - Diameter, maximum length. <p>Combination conditions:</p> <ul style="list-style-type: none"> - Components with a side lock (external thread), - identical coupling names, - external thread completely fits into internal thread, - fixed clamping area.



	<p>Shrink fit</p> <p>Geometric parameters:</p> <ul style="list-style-type: none"> - Diameter, minimum/maximum length. <p>Combination conditions:</p> <ul style="list-style-type: none"> - Components with a shank or shrink fit - Identical coupling names - Variable clamping area
	<p>Collet</p> <p>Geometric parameters:</p> <ul style="list-style-type: none"> - Min./Max. Diameter, Min./Max. Length. <p>Combination conditions:</p> <ul style="list-style-type: none"> - Components with a shank or Weldon shank - Identical coupling names - Drill chuck can be represented - Variable clamping area.
	<p>Collet with square drive</p> <p>Geometric parameters:</p> <ul style="list-style-type: none"> - Diameter, Min./Max. Length, Min./Max. Square size <p>Combination conditions:</p> <ul style="list-style-type: none"> - Can be combined with: straight shank with square drive. - names of couplings are the same.
	<p>Square drive shank</p> <p>Geometric parameters:</p> <ul style="list-style-type: none"> - Diameter, Min./Max. length, square size. <p>Combination conditions:</p> <ul style="list-style-type: none"> - Can be combined with: Collet with square drive. - Names of couplings are the same.
	<p>Weldon</p> <p>Geometric parameters:</p> <p>Diameter, length.</p> <p>Combination conditions:</p> <ul style="list-style-type: none"> - Components with a Weldon shank - Identical coupling names - Suitable for Whistle Notch - Fixed clamping area.

	<p>Weldon shank</p> <p>Geometric parameters: Diameter, Min./Max. Length.</p> <p>Combination conditions:</p> <ul style="list-style-type: none"> - Components with a Weldon coupling, shrink fit (not recommended, because imbalances may occur), collet, - Identical coupling names, - Fixed clamping area (combination with a Weldon coupling), - Variable clamping area (combination with a collet).
---	---

Possible coupling type combinations

Coupling types and possible combinations	Adaptor	Arbor	Morse taper	Collet	Shrink fit	Side lock with external thread	Side lock with internal thread	Collet with square drive	Straight shank with square drive	Weldon	Weldon shank	Shank [cylindrical]
Adaptor	+											
Arbor	-	+										
Morse taper	-		+									
Collet					+						+	+
Shrink Fit					+							+
Side lock with external thread												
Side lock with internal thread							+					
Collet with square drive									+			
Straight shank with square drive									+			
Weldon	-											
Weldon shank	-			+	+					+		
Shank (cylindrical), can be defined for each component				+	+							

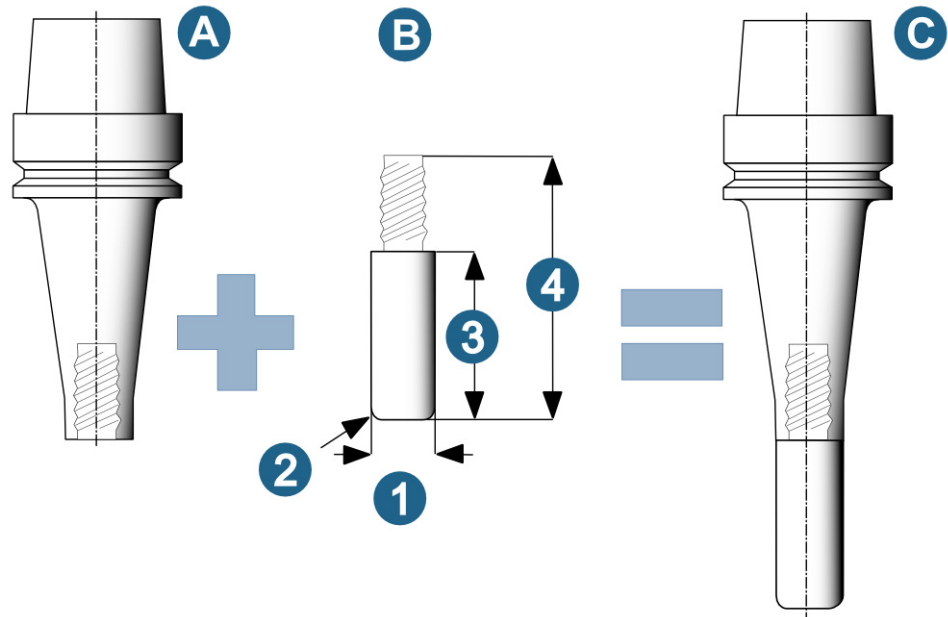
Combine the couplings with the **Holder** and **Tool** components to form an **NC tool**.

Example:

(A) Holder = holder for a screw-in cutter (e.g., HSK 63 A), side lock with internal thread (M12).

(B) Tool = bullnose end mill: diameter (1), radius (2), length (3), tip length (4), side lock with external thread (M12).

(C) NC tool



Possible coupling type applications

	Tool		Extension		Holder	
	Top	Bottom	Top	Bottom	Top	Bottom
Machining adapter	-				+	
Arbor	+			+		+
Morse taper	+		+	+		+
Collet				+		+
Shrink fit	-	+	+	+		+
Side lock with external thread	+		+	+		+
Side lock with internal thread	+		+	+		+
Collet with square drive				+		+
Straight shank with square drive	+		+			
Weldon	-			+		+
Weldon shank	+		+			

Technology data



Technology data and **cutting profiles** allow you to create tools for special use areas. If a tool is to be used for **finishing** and **roughing**, define a corresponding Usage in the cutting profile and assign this to the tool according to its use.

Technology data can be assigned either to **Tools** or **Inserts**. In doing so, the system differentiates between milling and turning jobs. Thus, individual technology data can take on different values for milling jobs and turning jobs.

Define technology data

1. In the data view, click the **Technology** tab and set the **Cutter material**, **Cutting edges** and **Spindle orientation**.
2. Then, in the lower area of the dialogue box select the required **Material** and **Usage**.



The technology data Cutting edges and Spindle orientation are available only for tools, but not for inserts.

If an insert with defined technology data is assigned to a tool with technology parameters, the technology data of the tool is overwritten with that of the insert.

Information on:

Material and cutter material	see section Define material
Technology usages	see section Defining a technology usage



In the OPEN MIND tool database, the following technology data can be defined:

Parameters	Explanatory note
Material	Define material
Cutter material	Define cutter material
Usage	Defining a technology usage
Spindle RPM (n)	Machining speed
Feedrate XY	Feedrate in the work plane
Axial feedrate	Feedrate in stepover direction
Reduced feedrate	Feedrate for plunge movements into material.
Retract feedrate,	Feedrate for retract movements (retract macro).
Feedrate approach	Feedrate for approach movements (approach macro).
Feedrate retract	Feedrate for retract movements (retract macro).
Feedrate stroke	Feedrate for stroke movements (Grinding bit tool type).
Overlap top	Length of the distance that the tool travels beyond the machining area in the axial positive direction.
Overlap bottom	Length of the distance that the tool travels beyond the machining area in the axial negative direction.
F/edge (fz)	Feed on the edge.
Cutting speed (Vc)	Speed of the tool geometry in the cut direction through the material to be machined.
Fz drill (f)	Feedrate in z direction/rotation.
Max. angle for reduced feedrate	Maximum permitted angle of the tool for machining at reduced feedrate.
Plunge angle	Plunge angle of the tool.
Infeed width (ae)	Area of the tool tip used for cutting. Applies only to milling tools that do not centre cut.
Infeed length (ap)	Length of the cutting edge(s) that is/are used for machining the material.
Cutting direction	Cutting mode of the tool (climb milling, conventional milling).
Coolants	Defining a coolant

The values for **spindle RPM** and **feedrate** can be defined using pre-defined or user-defined formulas.

Factor Spindle RPM/Factor F: Factor values that are applied to the spindle RPM and feedrate values of the tool.

Infeed width factor (ae) / infeed length factor (ap): define a factor to modify the infeed width (ae) and infeed length (ap) of holders, extensions and NC tools.

Maximum spindle speed/Maximal feedrate: Limitation of feedrate and spindle RPM.

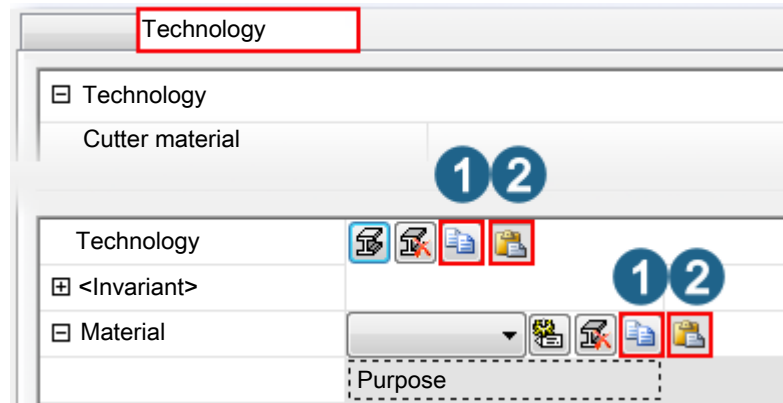
5X comp. length: Value for calculating the linear offset as a result of rotating the tool.

Copy and delete technology data

Copy technology data for tools and inserts by clicking the icon (1). Insert copied technology data by clicking the icon (2). Pasting the copied data in the browser is possible for tools, inserts and folders in the tools and inserts tab. If technology data that is being inserted that

already exists, a prompt is shown asking if the data should be overwritten. Technology data sets shall be deemed to exist when material and usage correspond to each other.

If materials that are in use are deleted, the technology data sets used by the material are also deleted.

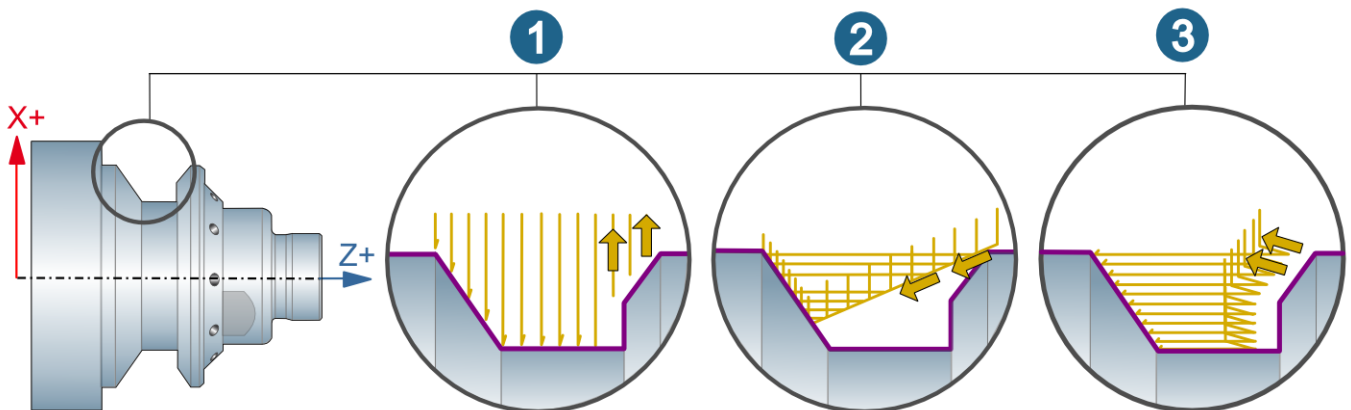


Technology data for turning tools

Retract feedrate: Feedrate used for retract movements. The Retract feedrate is used with the Grooving cycle if the tool cannot move freely via the retract macro because of the model geometry, but instead retracts along the material (1).

Feedrate reduced: Feedrate for plunges into the material at a specific angle. Application cases:

1. Cycle Turning, strategy: Axial, Radial and Contour parallel roughing. Options: Falling contours allowed is enabled. Feedrate reduced is always used when a combined X-Z movement occurs (2).
2. Cycle Grooving, Strategy: Axial roughing with ramp. The ramping-in movement to the next infeed plane always occurs in the Feedrate reduced mode (3).



Technology data for drilling tools

The following applies to the drilling tools, tap drills and boring bars tool types:

Drilling tool, boring bar	
Cutting speed (Vc)	m/min
Fz drill (f)	mm/rev
Spindle RPM (n)	$\frac{\text{Cutting speed (Vc)} \cdot 1000}{\pi \cdot d}$



Axial feedrate	Spindle RPM (n) * Fz drill (f)
Tap drill	
Pitch	p
Spindle RPM (n)	$\frac{\text{Cutting speed (Vc)} * 1000}{\pi * d}$
Axial feedrate	Speed (n) * Pitch (p)

Updating technology data

The Cutting speed (Vc), F/edge (fz) and Fz drill (f) technology data can be defined under material or in the data view of the tool (Technology tab).

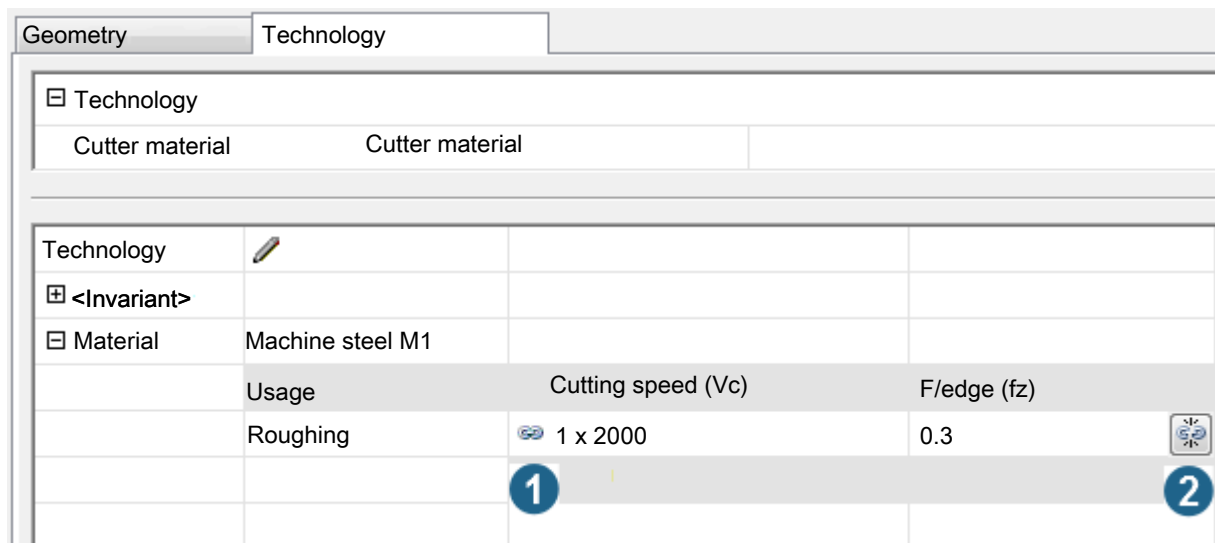
In the material dialogue, if you set values for this technology data, define a new tool and assign a material to it, the previously defined material technology data is linked to the tool technology data.

This means that any change in the material technology data will be adopted automatically from the linked tool technology data, including any necessary re-calculation of dependent values.

A linked value, then, can be modified on the tool technology data record only by one factor (see 1 in the screenshot and example A).

The link can be removed for each value individually. This way, the tool technology data remains unchanged in case of changes to the material technology data (see 2 in the screenshot and example B).

Figure: Data view (tools)



Example A:

Standard values in the material dialogue:

Cutting speed (Vc) = 200

F/edge (fz) = 0.5

Tool 0001 defined with standard values from the material dialogue:

Cutting speed (Vc) = 200

F/edge (fz) = 0.5

Standard values in the material dialogue changed to:

Cutting speed (Vc) = 220

F/edge (fz) = 0.6

For tool 0001, the cutting speed (Vc) changes to 220 and the F/edge (fz) to 0.6. At the same time, all dependent data that is defined using formulas is updated (feedrates and spindle speeds).

Example B:

Standard values in the material dialogue:

Cutting speed (Vc) = 200

F/edge (fz) = 0.5

Tool 0001 defined and standard values for material and cutter material overwritten by

Cutting speed (Vc) = 80

F/edge (fz) = 0.3

Standard values in the material dialogue changed to:

Cutting speed (Vc) = 220

F/edge (fz) = 0.6

For tool 0001, the cutting speed (Vc) remains at 80 and the F/edge (fz) at 0.4.

Using the Limiting diameter parameter, the material technology data Cutting speed (Vc), F/edge (fz) and Fz drill (f) is assigned to a specific tool diameter (upper limit). This also means that the above-mentioned material technology data is updated if the tool diameter changes (see data view of the tool, Geometry tab).

Using standard formulas

1. In the **Usage** area, select the parameter.
2. Use the combination field to select the required **formula**.

Technology			
<input type="checkbox"/> <Invariant>			
<input type="checkbox"/> Material			
	Purpose	Feedrate XY	Axial feedrate
		f_x 53	f_x 1
			fF fFinfeed



The f_x formula icon shows that the corresponding value has been defined using a formula. This also applies to the data view of the NC tool.

For the **Spindle RPM** parameter, there is a formula for the **Metric** measurement system and one for the **Inch** measurement system.

Defining a new formula

1. Right-click the database entry and select **Formulas...**
2. In the **Formula** combo box, select the area for which you want to define a new formula.
3. Right-click in the **Formula** dialogue box and select **New**.
4. Enter a **name** for the new formula (1), double-click the **Formula** column (2), click the icon for displaying the formula elements (3), define the formula you require (4) and select **OK** to close the dialogue.



Name	Formula	
fFinfeed	fz*z*n/3	

fz - feedrate per tooth

z - number of cutting edges

Vc - cutting speed

Diagram illustrating the formula definition interface. A table shows the 'Name' field containing 'fFinfeed' and the 'Formula' field containing 'fz*z*n/3'. A dropdown menu is open, showing variables: 'fz - feedrate per tooth', 'z - number of cutting edges', and 'Vc - cutting speed'. Numbered callouts 1-4 indicate: 1. Name field, 2. Formula field, 3. Formula field dropdown arrow, 4. Variable list.

Formulas can be used in all numeric input fields.

Example: Diameter: $(5+2)*3 = 21$.



Only points, but not commas, are allowed, as decimal separators in formulas. Only certain variables are allowed for formula definition, depending on the formula type. The following limitations shall apply:

Speed: spindle RPM (n) is not acceptable.

Length: ae, ap, fz, z, Vc, n and f are not acceptable.

Feedrate: all variants are allowed.

Define material

1. Select the database entry in the database browser and then select the **Materials...** menu item.
2. Right-click in the **Materials** window and select **New**.
3. Enter the **name** of the material you require and, if necessary, enter a suitable **comment**.



The material name must be unique. Otherwise, you will get an error message when closing the window.

Because the cutting conditions for the tool are largely affected by the **cutter material**, you can assign a defined cutter material to each material in the **cutter material** dialogue.

For information on assigning cutter materials to materials, see section **Assigning a cutter material to a material**.

Define cutter material

1. Select the database entry in the database browser and then select the **Cutter Materials...** menu item.
2. Right-click in the **Cutter Materials** window and select **New**.
3. Enter the **name** of the cutter material you require and, if necessary, enter a suitable **comment**.

Assigning a cutter material to a material

In the lower part of the **Materials** dialogue box, you can assign each **material** one or more **cutter materials**.



In such assignment, a distinction is made between the two measurement systems **Metric** and **Inch** as well as between **milling** and **turning technologies**.

Assigning material to cutter material



1. Select the material and then define whether the material-cutter material combination is to be used as **milling technology** or **turning technology** with a **metric** or **inch**-based measurement system.
2. Click icon 1 and select the required Cutter material from the list.

Name	Comment

Default milling technologies

<input type="checkbox"/> Metric	 1		
<input type="checkbox"/> Inch			

Default turning technologies

<input type="checkbox"/> Metric			
<input type="checkbox"/> Inch			

For each cutter material, you can define the Limiting diameter, Cutting speed (Vc), F/edge (Fz) and Fz drill (f) parameters as well as a measurement system.

Depending on the tool diameter, the F/edge (Fz) feedrate defined for the (upper) limiting diameter of the cutter material and the cutting speed (Vc) are incorporated into the definition of the tool.

Example:

Material	Cutter material	Limiting diameter	fz	Vc
1.30255 (Al 99.5)	SC (Solid Carbide)	10	0.3	250
1.30255 (Al 99.5)	SC (Solid Carbide)	20	0.4	265
1.30255 (Al 99.5)	SC (Solid Carbide)	30	0.6	280

Taking technology parameters from the cutter material for an NC tool

To apply the F/edge (fz) and cutting speed (Vc) technology parameters from the cutter material definition for an NC tool, follow the steps below:

1. Define a **material** and **cutter material** (including a limiting diameter, fz and Vc).
2. On the **Tools** tab, select the tool you require or define a new tool.
3. In the **Technology** area, select the relevant **material** and the associated **cutter material** and select **Standard** as the **purpose**.
4. Based on the sample data contained in the above table, a defined tool diameter of 15 results in the value 0.4 for the **F/edge** (fz) parameter and the value 265 for the **Cutting speed** (Vc) parameter (see the table).

Defining a technology usage

Assign different technology parameters to a cutter material, depending on the machining situation (e.g., roughing or finishing). This assignment is made in the data view of the tool definition in the technology parameters area (Material > cutter material).

1. Select the database entry in the database browser and then select **Technology Usages**.
2. Right-click in the **Technology Usages** window and select **New**.
3. Specify the **Usage** and, if necessary, enter a **comment**.

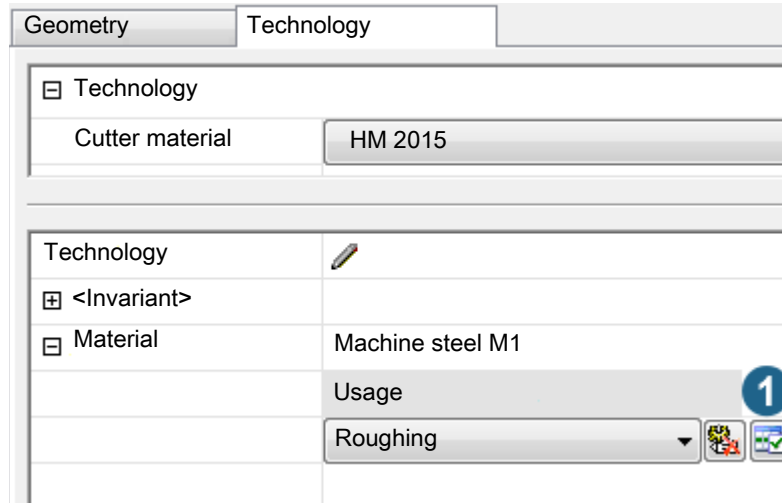
Save as default

1. Click the **Technology** tab of the tool's data view.



2. In the lower area of the dialogue box, select the required **Material** and click the **Usage** entry.
3. Select the **usage** and then click the icon (1).

All of the technology parameters defined for this purpose (including all of the **formulas** used) are saved in the default purpose.



If you then insert a new **material** in the data view of the tool, the parameters previously saved in the default usage will always be used.

Defining a coolant

To define a new coolant:

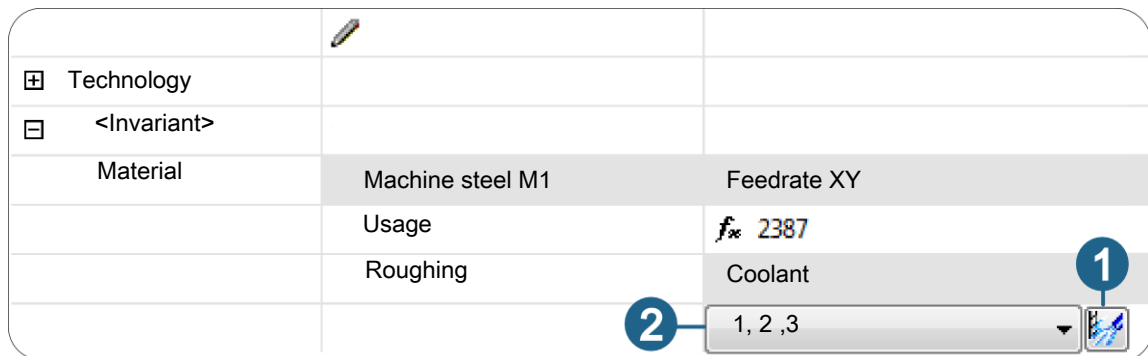
1. In the database browser, select the required database.
2. Right-click and select the **Coolant** function on the shortcut menu.
3. The two coolants with the ID 1 and ID 2 are already predefined by default. Right-click and then select **New**. Newly added coolants are numbered consecutively (ID 3, ID 4...).
4. If necessary, enter a comment for the coolant properties or coolant type.

You can define up to nine different coolants.

Assigning coolant to a usage

1. In the data view of the tool, switch to the **Technology** tab.
2. Under **Material > Usage > Coolant**, click icon 1 and open the Coolant dialogue.
3. Activate the coolant intended for the usage.

The IDs of the activated coolants are displayed (2).



Define spindle

1. In the database browser, switch to the **Spindles** tab.
2. Right-click the database entry.
3. Select **New spindle** and specify a **Name**, if necessary, a **Comment** (tooltip) and the **Measurement system** for the spindle.
4. To link the tool spindle with other components, specify the **Type** and **Class** of the lower **Coupling**. Start the geometry editor in order to specify the spindle **geometry**.

Define holder/extension

1. In the database browser, switch to the **Holder / Extension** tab.
2. Select the database entry you require and then select **New Holder/New Extension**.
3. Give the **Holder / Extension** in the data view a unique **Name** and select a **Measurement system**.

You can also save the components **Spindle**, **Holder** and **Extension** as default. All the geometry and technology parameters are saved and used if you define this component again.

To save a component as default, select the component in the database browser and select **Save As Default** in the shortcut menu.

Information on

Technology parameters	see section Technology parameters
Couplings	see section Define couplings
Geometry editor	see section The geometry editor

Importing a holder/extension as a 3D model

Use the *hyperMILL* TOOL Builder to import a holder or an extension from an existing 3D model (format: *.stl).

	In the Geometry area, open the <i>hyperMILL</i> TOOL Builder.
--	--

For information on the *hyperMILL* TOOL Builder, see the *hyperMILL* TOOL Builder product documentation.

	For 3D geometries of holders and extensions, it is no longer possible to edit the profile in the Geometry editor.
--	---

The geometry editor

Use the geometry editor to create freely definable **geometries** for tool components.

Click the Geometry line in the data view of the **component** and then click the icon (1). Specify a **name** for the geometry and define whether you want the geometry to be **active** and whether a **coupling** exists. To edit an existing geometry, click the icon (2) and open the geometry editor.

[-] Geometry	
[-] Variant 1	
Name	
Active	<input type="checkbox"/>



Only one geometry can be active for each component. You cannot delete an active geometry. To delete a geometry, disable this and use the icon (3).

The geometry editor comprises an input area for defining line segments and a graphical preview (2D). To add a geometry, right-click in the input area and specify the **Height** for the **Origin** and the **Diameter** and **Height** for the line segment. Add a line segment using the Paste shortcut menu.

Type	Diameter	Height	Report
Origin		0	<input type="checkbox"/>
Line		-0	<input type="checkbox"/>
Line		0.2	<input type="checkbox"/>
Line		12.5	<input checked="" type="checkbox"/>



Since the component geometries are always rotationally symmetrical models, you can uniquely define a line segment by defining a Diameter and Height. The radius is displayed in the graphical preview.



Component geometries must not contain any undercuts in the X direction.

Enable the checkbox in the **Report** column to copy the line segment to a report. The first and last line segment are always selected by default, so that they are output in a report.

If the check box in the Report column is enabled, its dimensions are also displayed in the geometry editor. The same also applies to the components Spindle, Holder and Extension.

To edit or delete a line segment, right-click in the input area and select **Edit** or **Delete**.



Load: Load a contour from a dxf file.

The following requirements must be met in order to be able to correctly load the dxf file:

- The contour must be on layer 1,
- the contour must be closed,
- both halves must be present for rotating bodies.
- The axis must be horizontally aligned.



Insert: Insert a contour in the *.omx format and import freely defined tool geometries directly from *hyperMILL*, for example.



Copy: Copy a contour from the database to the clipboard.



Mirror: Mirror a contour.



Delete: Delete a contour.



Complete your entries by clicking **OK**. A contour is now displayed in the graphical preview as a 3D bounding geometry.



Defining an NC tool

1. In the database browser, switch to the **NC tool** tab.
2. Select the database entry you require and select **New > NC tool**.
3. The **Select Tool** dialogue box opens. Here, select the tool you require and then click **OK**.
4. The newly created NC tool appears in the data view. Here, specify the required components and parameters.

Or:

1. Switch to the **Tool** tab in the database browser.
2. Select the **tool** that you want to use to define the **NC tool** and select **Create NC-Tool**.
3. In the data view of the NC tool, define the components and parameters you require.

The data view of the NC tool is divided into the **Geometry** and **Technology** tabs. On the **Geometry** tab, specify the general characteristics (NC number, Name) as well as the components that make up the NC tool. On the **Technology** tab, define the material used and the cutter material, the usage as well as the spindle speed and feedrates. If these have already been defined for the tool, they can be overwritten for the NC tool.

Save NC tool as default

To save a NC tool as the default, select the desired NC tool and select **Save as default** from the shortcut menu. This will also save reference points.

Basic parameters

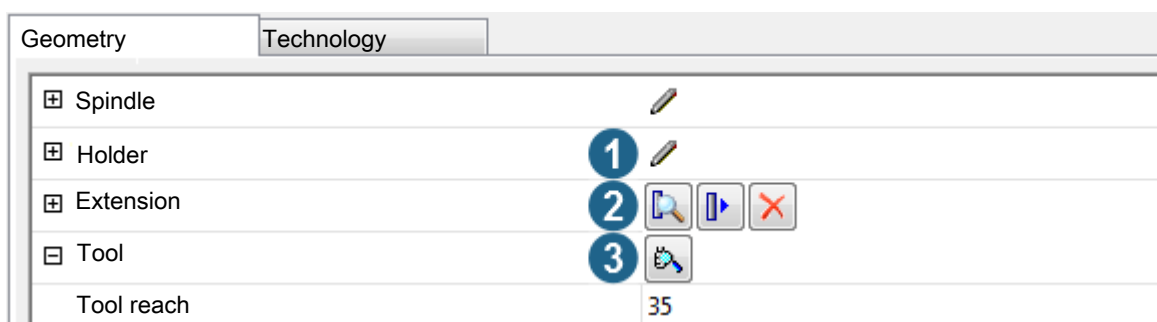
The NC number (maximum: 10 digits) is transferred to the NC file and is used to identify the tool on the machine that is used. For machines that can also evaluate non-numeric characters, *hyperMILL* also allows you to use an ID to define any combination of digits/characters (maximum: 32 characters) in order to uniquely identify the tool.

The tool type and measurement system are taken from the selected tool.

Select tool, spindle, holder and extension

NC tools are created in the data view of the overview window. Here, you can select the **spindle**, **holder**, **extension** and **tool** components for a newly created NC tool.

1. In the data view of the NC tool, click the **Holder, extension, tool** entry or on the icon (1),
2. click on the icon (2) or (3) to select the desired component,
3. confirm the selection with **OK**.



When assembling an NC tool, the following conditions apply:

1. The measurement system of the selected tool determines the measurement system of the NC tool.
2. All installed components must have the same measurement system.



If you assign a **holder**, **extension** or **tool** to an **NC tool**, only those components that can actually be installed at the appropriate place are displayed in the relevant selection dialogue boxes.

To connect components with each other, you use **couplings**, which can be categorised in the tool database according to different basic types and can be installed according to classes.



The coupling types used and the size of each coupling (length, diameter) determine which components are assembled together to form an **NC tool**.

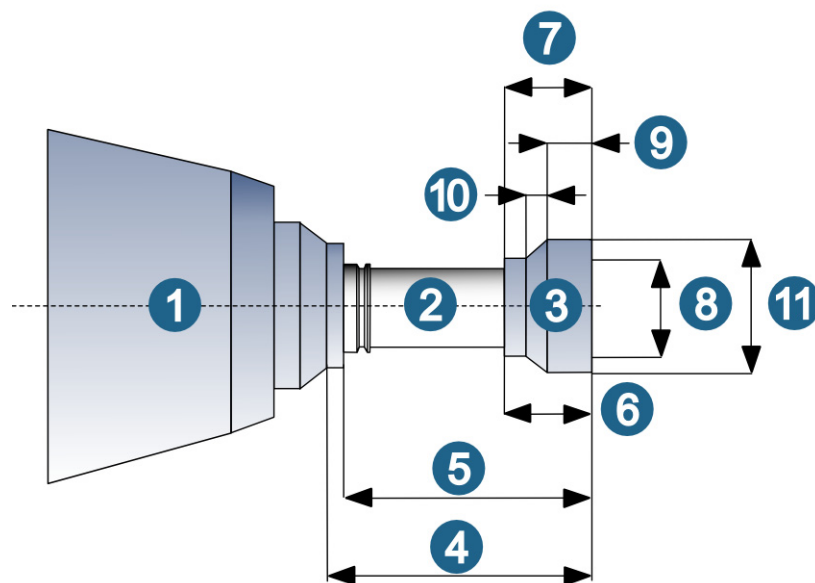
If you use the **holder** or **extension** component when assembling an **NC tool**, *hyperMILL* also checks the **maximum spindle RPM** of the components and takes the lowest value for the NC tool.

Example:

Holder	10,000
Extension	7,500
Tool	15,000
NC-Tool	7,500

Geometry parameters

(1) Spindle, (2) holder, (3) tool, (4) clearance length, (5) NC tool length, (6) tool reach, (7) tool length, (8) shank diameter, (9) tip length, (10) chamfer length, (11) tool diameter.



Gage point offset: Represent elements that extend beyond the standard geometry used in the vertical direction when assembling the NC tool and milling spindle. These can be:

- Elements of the milling spindle such as coolant nozzles.
- Elements of the holder such as a fixture cone.

The following figures show the parameters for **Milling spindle**, **Holder** and **Tool (NC tool)**:

(1) Milling spindle



2 = Gage point: Used when connecting the **tool** to the machine to determine the **Gage point offset** (milling spindle/holder) and to calculate the **NC tool length**.

(3) = **Gage point offset** (milling spindle): As a parameter of the milling spindle, this describes the offset from the gage point to the lower edge of the spindle elements which extend beyond the gage point (e.g., cooling nozzles).

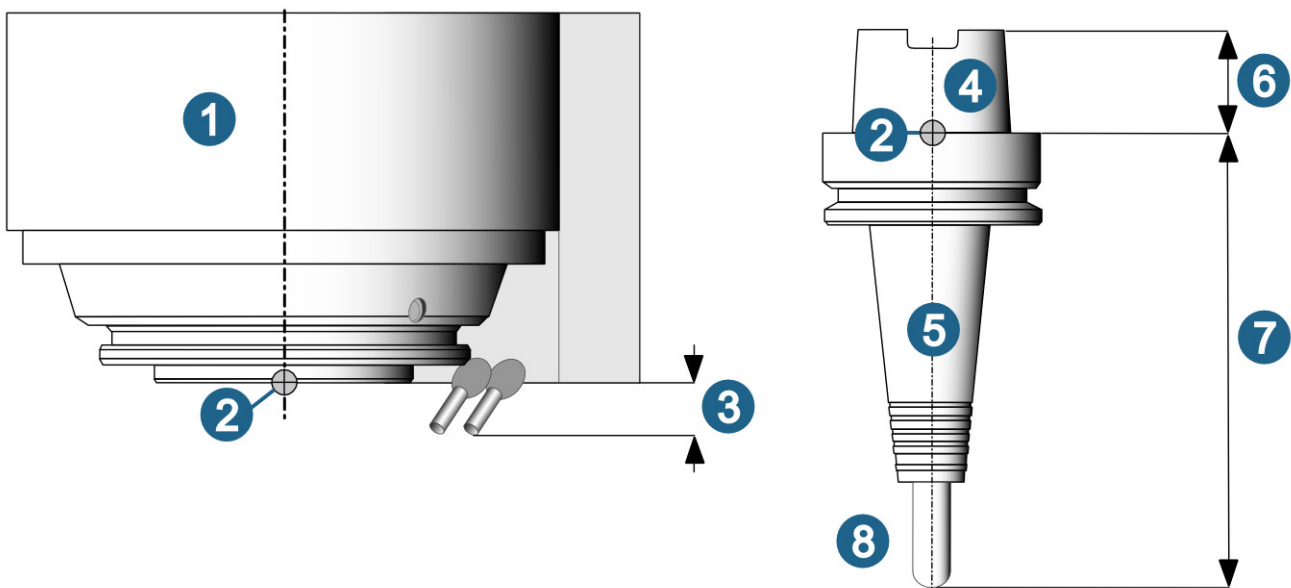
4 = **Holder** fixture cone

(5) **Holder**

(6) **Gage point offset** (holder): As a parameter of the coupling, this describes the gage point offset to the geometric upper edge of the holder (fixture cone). The **Gage point offset** is only available for the **Adaptor** coupling type.

(7) = **Gage length**: Length from the **tool tip** to the gage point.

(8) **Tool**



Technology parameters

To define technology parameters for an NC tool, switch to the **Technology** tab of the data view.

If **Spindle RPM (n)** and **Feedrates** have been defined for a material, they can be overwritten in the data view of the NC tool and, if necessary, modified through the use of factors.

For information on defining cutter materials, see section **Define material**.

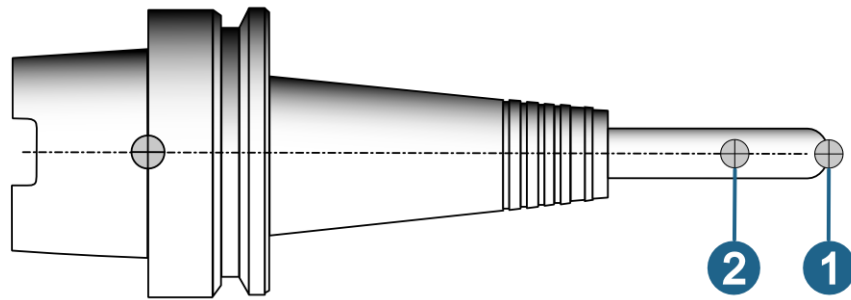
Tool reference

For the following tool types, either a **tip** or **centre** can be set as the tool reference:

Ball mill, bullnose end mill, lollipop, Woodruff cutter and lens cutter.

Tip (1): Z positions in the NC program are relative to the tool tip.

Centre (2): Z positions in the NC program are relative to the tool's centre point.



You can define several reference points for the following tool types:

T-Slot Cutter Drilling tool, Tap Tool and Boring bar.

For the **Barrel cutter** tool type, only the **tip** reference point is supported.

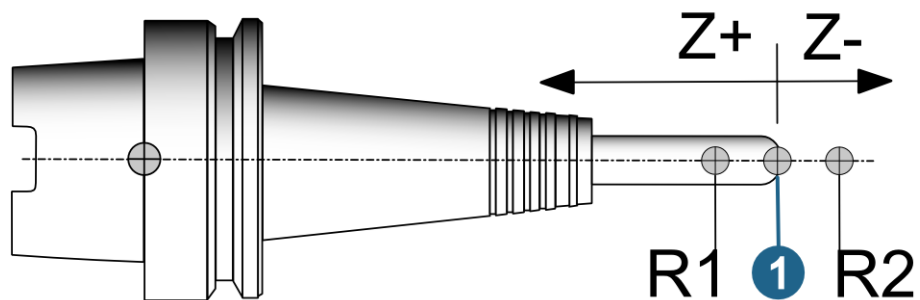
Reference points

Specify the **name** and **position** of the reference point. You can define as many reference points as needed.

Example

The value of the reference point position always refers to the tool reference (see the image).

R1: Position = 2, R2: Position = -2, Tool reference = 1



A positive value for the reference point means that this point is moved two units in a positive direction on the Z axis.

A negative value for the reference point means that this point is moved two units in a negative direction on the Z axis. At the same time, the clearance plane is aligned accordingly.

An offset of 0 corresponds to the tool reference.



The tool reference must match the tool's measuring system and applies to all 5X and 3D jobs in a joblist.

Factor spindle RPM (n) / Factor F: can be defined irrespective of the material and they are multiplied by the Spindle speed / Feedrate values. The **Maximum spindle RPM** and the **Maximum feedrate** limit the spindle RPM and feedrate values.



Both the **factor** values and the **maximum spindle RPM** or **maximum feedrate** also apply to the **holder** and **extension** components.

Factor values are also applied to **Feedrates** in the cutting profile if the feedrate unit is set to **mm/rev** or **inch/rev**.

5X comp. length: Value for calculating the linear offset as a result of rotating the tool. The distance from the tool reference point to the pivot point is normally composed of a tool-independent and a tool-specific part (correction length).

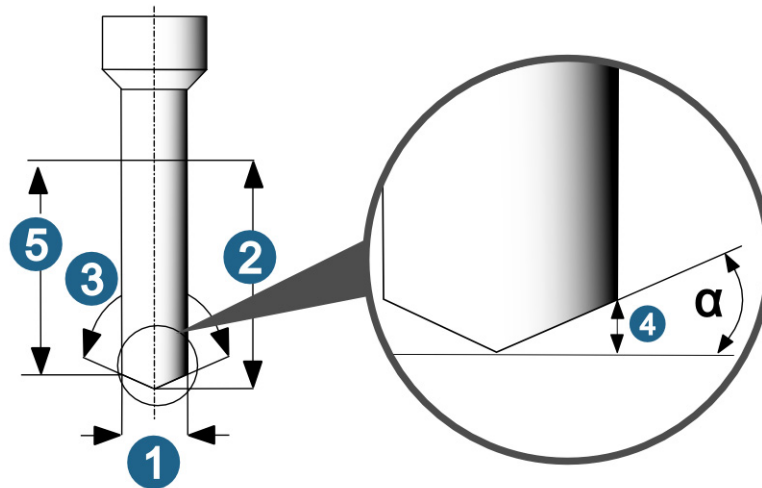


Usable length: For drilling tools and tap drills, the value is determined as follows: cutting length - tip length (drilling tool), cutting length - lead in length (tap drill). The value can be modified later on.

The following applies for all other tool types: Usable length (5): Determined based on the Tool diameter (1), Tip length (2), Tip angle (3) and Lead in (4) parameters. Tip angle (3) and Lead in (4) are calculated based on factor values.

Tip angle (3)	Tip length / Lead in length (4)
80°	0.6 * D (Tool diameter)
118°	0.3 * D (Tool diameter)
130°	0.23 * D (Tool diameter)
140°	0.18 * D (Tool diameter)

Tool diameter (1), Cutting length (2), Tip angle (3), Tip length, Lead in length (4), Usable length (5)



For milling tools the following also applies: Usable length = Tip length. If the tool reach is lower than the tip length, the usable length corresponds to the cutting length.

NC tool length: tool length + holder length.

The technology values for the **cutting edges** and **spindle orientation** are taken from the selected tool.

Material

A material and usage are always displayed if they are linked with the selected cutter material (see above). The material and usage are then taken from the selected tool.



The material-dependent spindle speed and feedrate values such as Spindle RPM (n), Feedrate XY, Axial feedrate, Reduced feedrate as well as the Infeed width (ae), Infeed length (ap) and Plunge angle parameters can be directly overwritten for the NC tool.

To directly overwrite these material-dependent parameters, click the corresponding row (1). If the value is linked, first click the icon (2) to clear the link. Then enter the desired value.

☐ Material	Machine steel M1		
	Purpose	Spindle RPM (n)	Feedrate XY
	Roughing	1 2000	2 1 x 200
		Coolant	Additional coolant



Meaning of the icons

	A formula is used to define a value for the tool .
	The value has been overwritten for the NC tool .



Cutting point setup (turning tools)

Specify the clamp of the tool and the position of the cutting edge using the **Tool angle**, **Tool reference**, **Cutting edge position** and **Reference angle** parameters.

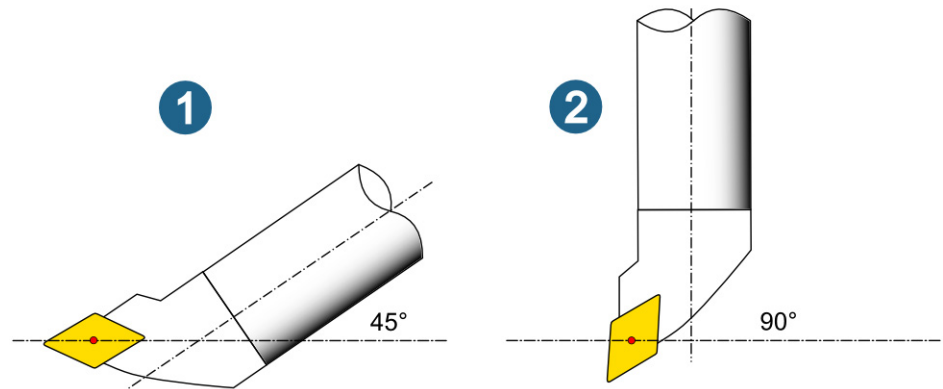
The parameters are saved in one parameter set. It is possible to name, copy and paste a parameter set. You can create as many parameter sets as desired for a tool.

Parameter sets

	Parameter sets: Define a new turning parameter set.
	Location: Delete location / copy location to clipboard.

Name location.

Tool angle: Specify depending on the type of machining. Inside (1), outside (2). Default setting = 90°.

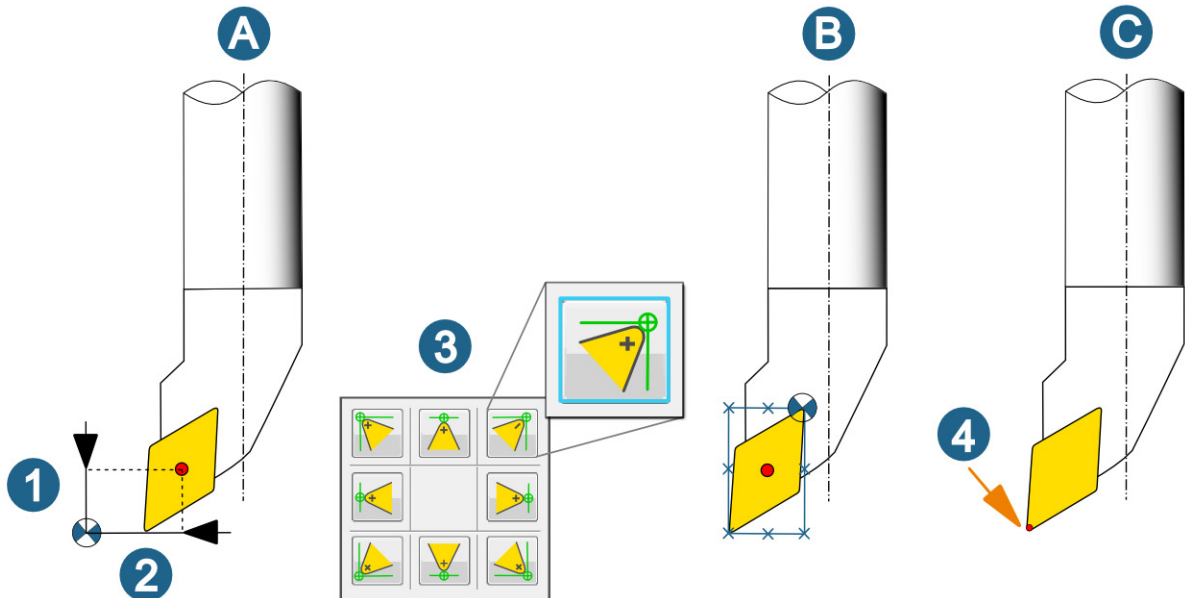


Tool reference: Specify the tool reference for calculation of the toolpath using the **Free point definition**, **Contact point**, **Orthogonal gauge point** or **Primary centre** options.

Free point definition (A): Define any reference point. Use the **Mounting point x offset (1)** and **Mounting point z offset (2)** parameters.

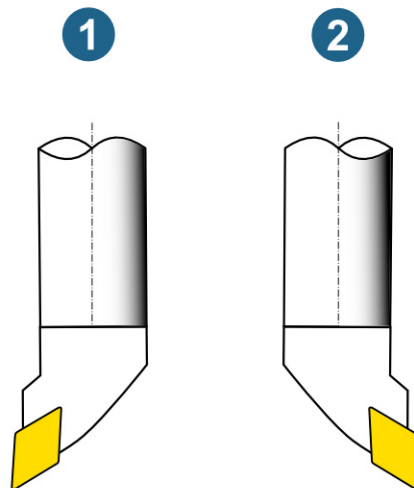
Orthogonal gauge point (B): Select the **Cutting edge position** parameter using the buttons (3) for machining operations with path compensation (G41/G42). Defines the location of the reference point relative to the **Mounting point** (centre point) of the insert.

Primary centre (C): The radius of the insert is used as a reference point (4).



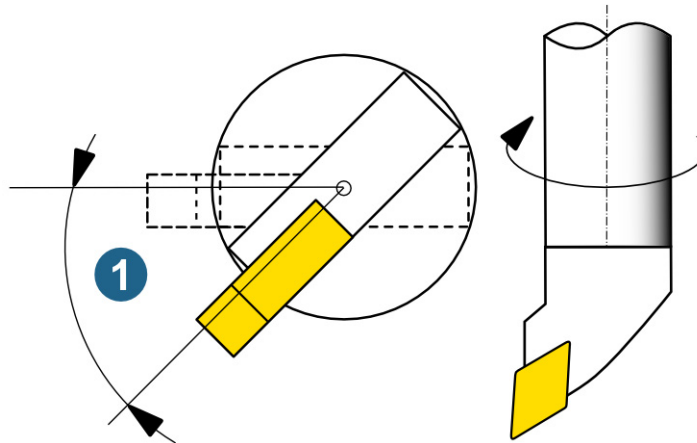
Correction list: Correction space in the machine.

Invert tool: Function not enabled (1), function enabled (2).



Advanced

Reference angle (1): Turn the tool holder and insert around the own axis in the tool angle (overhead setup). No graphical preview available.



Depots

Use depots to group and administer tools according to your own criteria.

Defining a depot

1. Go to the **Depots** tab in the database browser.
2. Right-click the database entry, select **New > Depot** and specify a **Name** and, if necessary, a **Comment** (tooltip) for the depot.
3. Enable the **Contains NC tools** option if you want the depot to contain the **NC tools**.

Saving NC tools in a depot

1. Select the relevant **depot** and right-click the entry.
 2. Select **New > Depot Item...**
 3. In the **Select NC Tool** dialogue box, select the tool you require and confirm by selecting **OK**.
- or:
1. Select the required tools on the **NC tool** tab,
 2. select the **Create Depot Items...** function from the shortcut menu.
 3. In the **Select Depot** dialogue box, select the depot you require and confirm by selecting **OK**.

The NC tools are now available in the **depot** and the parameters are displayed in the data view. If some of the NC tools already exist in the respective depot, these are not added and a corresponding message is displayed.



The **Create Depot Item...** function is also available for entire folders. It enables you to transfer the content of an entire directory to a depot.

NC tools are indicated by an **alternative NC number**, an **alternative NC number (with text)** and an **alternative NC name** when they are added to a depot. These initially correspond to the **NC number**, **ID** and **Name**.

You can, however, change these manually: if you create a depot as a magazine, for example, the **alternative NC number** can be used as the place number of the tool. This number therefore always needs to be unique.

If you transfer tools from a depot directly to *hyperMILL*, the **Alternative NC-Number** is transferred as the **Number** and the **Alternative NC-Name** as the **Name** in the tool browser.

To transfer an entire depot or individual tools from a depot to *hyperMILL*:

1. Select the required entries, right-click and
2. select **Transfer to *hyperMILL***.



Transferring a function to *hyperMILL* is only available if you start the tool database from the *hyperMILL* browser.

How to create a new **depot** within an existing depot:

1. Select the relevant **depot**, right-click and
2. select **New > Depot**.

Exchanging data with other databases

To exchange data with other tool databases, a neutral format (*.xml) is available.

You can import or export both an entire database and individual folders or elements.

For more information on the **XML** data exchange format, please contact your OPEN MIND partner.

Importing/exporting data

1. Select the required database entry in the database browser.
2. Select **Import** or **Export** on the shortcut menu.
3. Then choose **Neutral Tool Exchange Format (*.xml)** for file type.

Please note: the **Import** shortcut menu function is only available for database nodes.

Exporting a structure

To export just the folder structure of your OPEN MIND tool database:

1. Select the element you require (database or individual folder) in the database browser and
2. select the **Export structure** option on the shortcut menu.

Synchronization

The **Synchronization** function allows data to be compared automatically between another database system (such as Walter) and the OPEN MIND tool database.

1. In the database browser, select the database into which you want to import the data of another system.
2. Select **Synchronization > Input** from the shortcut menu.
3. Click the **Enabled** checkbox in the **Input Synchronization** dialogue.
4. For **Data exchange folder**, select the folder that contains the database whose data is to be synchronized with the OPEN MIND database.
5. Enable the **Slave database** option (OPEN MIND database). Please note: if this option is enabled, you cannot edit the contents of your OPEN MIND database. Enable the **Pre-serve folder structure** option to preserve the folder structure of the OPEN MIND database when synchronizing the data.
6. For **Workstation ID**, click **Update** , to read the ID of the system on which your OPEN MIND database is located.

Additional functions

Generating reports

Generate report files with the parameters of the defined tools and/or tool components. You can generate reports for **tools**, **NC tools**, **extensions**, **depots** and **holders**.



1. Select the element you require in the browser (you can also make multiple selections).
2. Right-click and
3. select **Report...** from the shortcut menu .

The data is initially temporarily saved in the ***.html** format and displayed in your default browser. You can now back this data up anywhere on your system.

The creation of reports can be controlled from the **Options > General** menu.

Show report after generation

The generated report file is opened automatically in the default program.

Overwrite report output folder

When creating a new report, an existing report directory is deleted and a new directory is created. By default, reports are saved in the [tooldbReport] directory of the **Global working space**. If you use project paths, they are saved to the [tooldbReport] directory of the project path defined.

For information on creating reports, see section **ReportDesigner**.

Specifying the accuracy

You can specify the accuracy for both the **metric** and **inch** measurement systems, as well as separately for **angles** and **factors**.

1. Right-click the required database in the database browser and
2. select **Accuracy** from the shortcut menu.
3. Specify the corresponding accuracies for the measurement system used and for the **angle** and **factor**.
4. Alternatively, you can specify the accuracy via the menu by choosing **Options > Accuracy**.

Specifying a measurement system

1. Right-click the required database in the database browser and
2. select **Measurement system > Metric / Inch** from the shortcut menu.

Changing the password

1. Right-click the database entry in the database browser and
2. select **Change password** from the shortcut menu.
3. Enter the old password, then specify the new password.
4. Confirm the new password.

Showing properties of the tool database

1. Right-click the required database in the database browser and
2. select **Properties...** from the shortcut menu.

The following properties are displayed: File name, User, Version and URL.

Defining a filter

Filter the display of elements in the browser window and in the data view according to certain criteria.





1. Click the icon (1) to open the Filter dialogue.
2. In the Filter dialogue, select and activate the filter. To do this, enter the required values or select one of the options offered.

The column width in the list display depends on the tool type and the activated (displayed) options. For information on configuring the list display, see section **The browser**.

Deleting a filter

Click the icon (2) to delete the defined filter.

If you start the OPEN MIND tool database from the Tool dialogue page of the job definition, previously defined tool types are also considered as filters. Please note that when you delete a filter, this default filter in *hyperMILL* is not taken into account.

If you start the Select NC-tool dialogue from *hyperMILL*, the filter in the Technology area contains a complete list of all materials already defined for the material selection.

If you clear the check box in the middle column of the Filter dialogue box, the respective filter is also disabled; however, the filter definition remains the same.

Multiple selections

Select multiple elements in the browser at the same time. To do this, click to select the required elements and hold down the **CTRL** key at the same time. Depending on the type of elements selected, different functions are available in the shortcut menu (right-click) (including **Copy/Export**).

You can also use the multiple selections function to transfer several tools to *hyperMILL*. For further information, see section **Import tools from external database**.

Copy & Paste

In the browser of the OPEN MIND tool database, move or copy elements to a different place in your database using drag-and-drop operations.


To move an element, select the required element with the left mouse button and move it to the required location while holding down the mouse button.

To copy an element, hold down the CTRL key at the same time. This function is available both for elements and for folders.




Interfaces

TDM Systems integration


	<p>Manage the exchange of data with the TDM Systems tool management system using the TDM Systems integration.</p>
---	---

Select the TDM Systems database from the *hyperMILL* > Setup > *hyperMILL* Settings > Database > Settings Wizard dialogue.

Importing tools from TDM Systems to *hyperMILL*

- 

Click the icon to start the **TDM Tool Manager**. This can be done either directly in the job definition tool dialogue, from the *hyperMILL* browser or from the **hyperMILL** > **Tools** menu.

- 

The tool import dialogue is already enabled by default.

- 

Click on the tool icon to first select the desired tool type, and then the tools to be imported. If the **TDM Tool Manager** was opened from the job dialogue, then only the tool type already selected in the job dialogue is selectable.


It is not yet possible to import turning tools with *hyperMILL* Version 2017.1.



Tools which were imported using the TDM Tool Manager can be used in macro programming. A prerequisite is that the tools are saved in the database as an NC tool with a unique ID.

In the Apply macro dialogue, NC tools used in macro jobs which come from the tool database and have a unique ID are marked as DB.

Exporting a tool list from *hyperMILL* to TDM Systems

- Open the TDM Tool Manager from the **hyperMILL** > **Tools** > **TDM Systems Tools** menu.
- 

Click the icon to open the Tool list export dialogue. All local tools defined in the model file (*.hmc) will be exported.
- Define the name of the tool list to be exported using **List ID**.

WinTool integration

The WinTool integration enables data exchange with the WinTool tool management system and is available via the **hyperMILL > Tools** menu.

It is possible to access complete tools and tool lists directly from *hyperMILL*. The tools exported from WinTool are not written to the OPENMIND tool database, but directly to the *hyperMILL* document. This way, a redundant tool management system is avoided.

To enable the WinTool integration, right-click in the CAD toolbar and select the **WinTool hyperMILL interface**.

	<p>To enable the WinTool integration, right-click the CAD program's toolbar and select the WinTool hyperMILL interface option.</p>
--	---

WinTool *hyperMILL* data exchange

During the exchange of data between WinTool and *hyperMILL*, the existing tool IDs are checked.

When importing tools from WinTool, *hyperMILL* assigns NC numbers automatically. The next NC number is the highest existing number + 1.

Using the **WT-MakeList** function, all tools used in a *hyperMILL* document are adopted in a list and exported to WinTool. The list consists of a set of IDs and NC numbers for each tool used.

For more information on the functions of WinTool integration, see your WinTool documentation.

Zoller integration

The Zoller integration enables data exchange with the Zoller tool management system and is available via the **hyperMILL > Tools** menu.

	<p>Import Zoller tools: Opens a window of the Zoller database to choose the tools that are to be imported to <i>hyperMILL</i>. The imported tools are automatically entered in the tool list of the current document.</p>
	<p>Export Zoller tools: All tools used in the document are exported to the Zoller database. If the document contains multiple joblists, a usage list is created for each joblist of the document</p>

















Import and export are available only if the Zoller TMS has been started.























	<p>Only <i>hyperCAD</i> 2009.3: Optionally, display the buttons for the interface. To do this, right-click the toolbar, use the Adjust function to open the Toolbars dialogue and enable the Zoller TMS hyperMILL option.</p>
--	--

Icons in the database browser

Milling Tools	Drill Tool	Turning tools
Ball endmill	Drilling tool	Turning tool
Endmill	Tap drill	Recessing tool
Bullnose endmill	Threadmill	Axial recessing tool



Milling Tools	Drill Tool	Turning tools
 Lollipop	 Boring bar	 Parting tool
 Woodruff cutter	 Countersink	 Threading tool
 Barrel tool	 Reamer	 Touch probe tool
 Chamfered cutter		 Grinding bit
 Chamfered profile cutter		
 T-slot cutter		
 Lens cutter		
 General barrel tool		
 Conical barrel tool		
 Tangential barrel tool		

Production resource	Other	Inserts
 Cutter material	 Depot	 Rhombic insert
 Material	 Holder	 Triangular insert
 Couplings	 Extension	 Round insert
 Technology usage	 Depot element	 W-shape insert
 Coolant	 Tool holder	 Threading insert
 Formula	 Spindle	 Simple thread insert
	 NC tool	 Rectangular insert
		 Parting insert
		 Free form insert

12



hyperVIEW

Structure of documentation

This documentation provides you with:

- A brief overview of the basic functions in the following section.
- Information on the screen layout in section **User interface**.
- A brief summary of the most important workflows in section **Important workflows**.
- Detailed descriptions of the individual functions on the menu in section **Menu functions**.
- Descriptions of the *hyperVIEW* browser functions in section **Functions in hyperVIEW browser** and
- information on the toolbars in section **Toolbars**.

Basic functions

hyperVIEW's basic functions include:

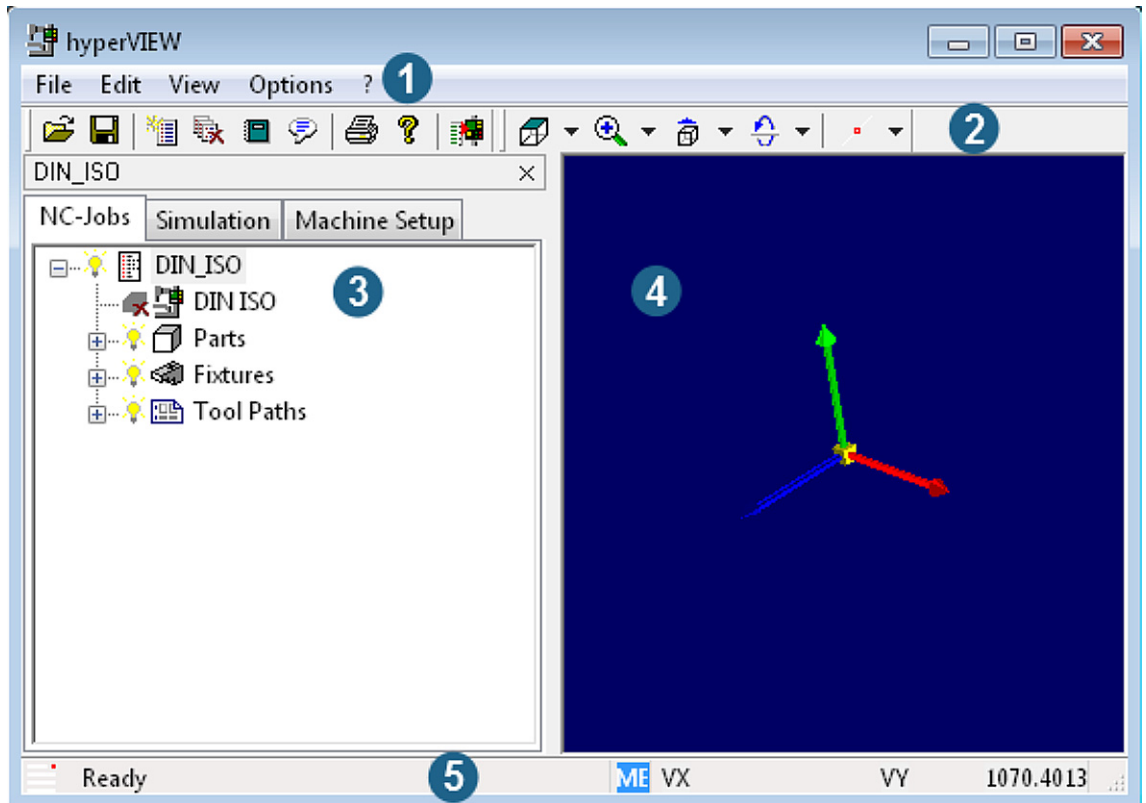
- Display of the toolpaths and control of the tool's movements (**Toolpaths**).
- Machine simulation based on toolpaths (**Setting up the machine simulation**).
- The generation of NC files (**Generate NC program**).
- The generation of reports (**Generating reports**) and
- print functions (**Printing information**).

To unlock some of the *hyperVIEW* functions described here, you will need to purchase a special licence. Please contact your OPEN MIND dealer. With the relevant licence, you can also simulate multi-axis and 5-axis machining.

User interface

The user interface consists of the following elements:

(1) Menu bar, (2) Toolbars, (3) Browser, (4) Graphics area, (5) Status bar




Important workflows

Important workflows are outlined below.

Defining a machine

For much of the work carried out, you need to specify a machine to use for simulations. When you run *hyperVIEW* for the first time, you will be prompted to define a machine.


Proceed as follows to set up a machine:

1.  Open the **Machine administration** dialogue.
2. Alternatively, open the Machine administration dialogue via the joblist in *hyperMILL* (Postprocessor > Machine).
3. Specify the machine's name and postprocessor in the **Common** category of the **Machine properties** dialogue.



The postprocessor's parameters need to be suitably configured before it is used for the first time. Please contact your OPEN MIND dealer to get information regarding this.

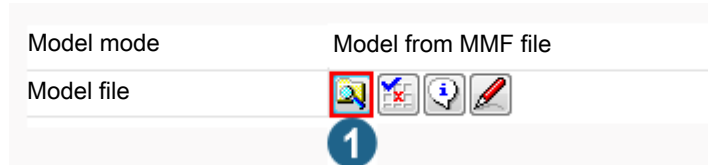
Proceed as follows to modify a postprocessor's configuration:

1.  Open the **Postprocessor configuration** dialogue and, if necessary, change the name, type and value of individual parameters.
2. Make sure to document all changes made in the configuration dialogue!



Model mode: Specify whether a machine model is to be used for the simulation or whether only the part and tool are to be displayed. If a machine is used, define the format of the machine model file.

Model from MMF file / VMM file: Select if a machine is to be displayed in *hyperVIEW*. Click the icon 1 to select the machine model file. The settings for Model from VMM file correspond to the settings for the Fixed part option.



Model from postprocessor: Select if the postprocessor (*.oma) contains a machine model. Only licensed tool machine models can be used. For more information on this, please contact your OPEN MIND partner.



Please note that these machine models are not assured to match any particular machine in your machine shop. Functions like collision control and workspace monitor may vary from our typical model to your actual machine.
OPEN MIND or your OPEN MIND partner can adjust the machine to your particular requirements. This service, which does not form part of the standard delivery, is charged separately.

Fixed part / Fixed tool: Select if there is no machine model or if the machine model is irrelevant to the simulation. Only the part and the tool are displayed in the simulation; one of the two needs to be fixed.

For further information on the Machine definition dialogue and on how to configure a postprocessor, see section **Machine Properties / NC-File properties**.

Generate NC program

Proceed as follows to write an NC file:

1. Define a machine, see **Defining a machine**.
2. Create an NC job (right-click > **Create NC job**). *hyperVIEW* creates an entry in the browser consisting of the following elements: machine, several parts (stock, model, result) and toolpaths.
3. Insert a toolpath (select the **Toolpath** entry in the browser, right-click and select > **Insert toolpath**).
4. Start the postprocessor run (select the **NC job** entry, right-click and select > **Write NC-File**).

The postprocessor run terminates with an error message if no licence is found.



OPEN MIND has enhanced the postprocessor run to include an analysis of the postprocessor. The following message appears if a security-related issue is detected:

Important note on your postprocessor. OPEN MIND detected a security issue. Please contact your OPEN MIND business partner.

Setting up the machine simulation

Proceed as follows to set up the machine simulation:

1. Define a machine, see **Defining a machine**.
2. Create an NC job and insert toolpath (see section **Generate NC program**).

- Define the machine simulation conditions. To do this, switch to the **Simulation** tab in the browser and define the machine origin, the movement display of your choice and the speed (animation control).



Machine origin: If you select the **Automatic** option, the parts used and the toolpaths are positioned automatically. Use Machine Setup to define other origins.

- Define the display options for the tool, toolpath, machine and model.

Animation control

C: All the tool movements of the respective job are simulated.

G1: Only the tool movements on the G1 plane are simulated.

G0: Only the tool movements on the G0 plane are simulated.
















If you set the (Program) halt option to On, the simulation will be stopped at the point specified in the NC file.

Mode

If you want to perform a material removal simulation, define a stockmodel.

	Material removal simulation disabled. Open the Create stock dialogue. Define the shape and display for the stock.
	Material removal simulation enabled (only if a stockmodel has been defined).
	Material removal shaded
	Material removal transparent
	Material removal wireframe
	Save the result of the material removal simulation in the *.stl, *.vis or *.omg format.
	Model shaded
	Model transparent
	Model wireframe
	Model invisible
	Automatic collision checking disabled.
	Automatic collision checking enabled.
	Enable collision checking to include the activated checks. A reverse simulation is not possible.



	Open the Machine structure dialogue.
	Machine component visible.
	Machine component invisible.
	Show all machine components.
	Hide all machine components.
	Tool shaded
	Tool transparent
	Tool as wireframe
	Tool invisible
	Display all toolpaths.
	Only display the current toolpath.
	Only display the current tool movement.
	Do not display any toolpaths.



An activated machine model is required to display the machine.

For information on the machine origin, animation control and the display mode for the simulation, see section **Simulation tab**.

Generating reports

Generate a report for tool lists, joblists or statistical information (tool movements, dwell times, coolants and so on) in the formats *.htm , *.rtf , *.txt or *. xml (for Excel 2002).

Proceed as follows to create a report:

1. On the **File** menu, select the **Report** entry or select the **NC job** entry in the browser, right-click and select the **Report** option.
2. In the Report dialogue, select the required **format** and a template file.
3. Specify a path and file name and save the file.

For the generated report file to open automatically, you need to select the **Show report after generation** option on the Options > General menu.



Reports including statistical evaluations can only be created if in *hyperMILL* (job definition > **General** dialogue page) the **Generate NC file** option is enabled.

Use a text editor to customise script files (*.JLR) for modifying the content of report files according to your individual requirements.

Printing information

Print information on the NC file, machine, tool parameters and machining cycles for an NC job.

Defining the print layout

To define the print and CSV layout:

1. On the **Options** menu, select the **Print and CSV Layout** entry.
2. Select the **<new layout>** option and specify a name.
3. Define the printing values (double-click the **Available values** column).
4. Use the arrow keys to specify the order of the values.
5. Click **OK** to confirm the print and CSV layout.

Print

To print the layout:

1. Select **Print** from the **File** menu.
2. In the **Print** dialogue, define the printer, print area and layout.
3. Click **OK** to close the **Print** dialogue.



To print out NC jobs, the browser window needs to be active, as otherwise the current contents of the graphics window will be printed instead.

CSV format: The defined parameters are saved in a text file with comma separators between the values.



This format enables further editing in a spreadsheet program such as Excel.

To generate a file in *.CSV format:

1. Select **CSV** from the **File** menu.
2. Select the storage location, file name, file type and layout.
3. Click **Save** to close the dialogue.

Menu functions

File

Open: Use this command to open files (file types *.hv or *.hvx). When opening *.hvx files, the files contained in the archive are unpacked to a temporary directory (tmp).



This automatically deletes all browser entries.

Append: Use this command to append the selected file (*.hv / *.hvx) below the last browser entry. Existing entries are not deleted.

Save as: Save a file in the format:

- ***.hv:** With this file type, information about the machine, stock, model and toolpaths is linked to an NC job. All of the NC jobs in the browser are saved.



This gives you the option of saving a project (NC file) even if you haven't yet defined all the areas. Missing data can still be added at a later point.

- ***.hvz:** All of the files associated with a project file (*.hv) are saved together in a ZIP file with the following file structure:

NC-Job X

- 3DF/
- POF/
- STOCK/
- NC/

NC-Job Y








With postprocessor: Include the postprocessor in the .hvz file. This includes all *.def files that are located in the same directory as the postprocessor (.oma file).

With machine model: Include the machine model in the .hvz file.

The options are important for the OPEN MIND support team and are enabled by default.

Unpack Archive: Unpack all the files associated with the project file (that is also contained in the ZIP archive) to the selected directory.

Report: Open the **Report** dialogue. It is possible to output tool lists, statistical evaluations and job list reports; to some extent also with a graphical representation of the toolpaths. The following file formats are available: HTML, RTF, TXT and XML (Excel 2002). Select the file type and report required, confirm your selection by clicking **OK** and then enter the path and file name to save the report file. A report file is generated for every NC job in the browser.

	Move the report to the top of the list.
	Move the report one line up.
	Move the report one line down.
	Move the report to the bottom of the list.
	Create a new report.
	Edit the report.
	Delete the report.

Save to project folder: Save a report directly in the **Joblistreport** folder inside the project folder.

CSV: Save to a file in the *.CSV format using parameters which were selected in the **Print and CSV Layout** dialogue (Options menu).

Print: Open the Printer setup selection dialogue to print data for the NC job selected in the browser.

Page Setup: Specify the page options for printing (default Windows dialogue).

Write NC-File: Create NC files for all of the NC jobs selected in the browser. Before the NC file is generated, you can still edit and change the relevant tool data and the stock. For further information, please refer to **NC file**.

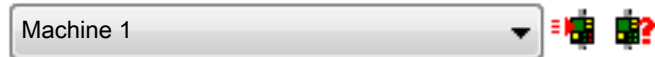
Edit

New NC-Job: An empty NC job (data format: *.hv) is created in the *hyperVIEW* browser and assigned the following (tree structure): machine, stock, model, result, rest material, clamp and toolpaths.

The default machine is the one specified in the Machine Administration dialogue box. The Machine Definition dialogue box also specifies the name of the NC job.

Delete all NC-Jobs: Delete all the NC jobs created in the *hyperVIEW* browser.

Machine administration: Opens the dialogue for creating and editing machines.



The default machine specified in Machine Administration is also displayed in the Machines toolbar.



By default, the Machines toolbar is hidden when the program is first started.

View

Display: Option to display frames, NC systems and world coordinate system in the graphics window.

Toolpath: Show and hide toolpath and NC paths.

Orientation: Viewing perspective for the data currently displayed in the graphics window.

Zoom: Enlarge or reduce the current view in the graphics window.

Move: Pans the current view in the graphics window.

Rotate: Rotates the current view in the graphics window.

You can assign these functions to key combinations so that they are more directly accessible from the keyboard. For instructions on how to assign key combinations, see section **Keyboard**.

For information on the increments of the Pan and Rotate functions, see section **General**.

Browser: Show or hide the hyperVIEW browser.

Toolbars: Show and hide the following hyperVIEW toolbars: Standard, Machines, View & Measure, Status bar and Video.

Measure: Information in section **View and Measure**.

Capture: Create images of the graphics window, browser or the entire application. Available file formats: *.png, *.gif and *.bmp.

Create *.avi-formatted videos (standard Windows file format for video content). Specify the video capture settings on the menu under Options > Video . Define the file name and click **Save** to start the video capturing.

	Start capturing.
	Stop capturing.

Information on the formats and how to configure them is provided in the documentation of the respective manufacturer.

Simulation

All the functions of this menu can also be controlled with keyboard shortcuts.



Machine Setup

The menu item **Machine Setup** is only available after you select the Machine Setup tab in the *hyperVIEW* browser. All the functions of this menu can also be controlled with keyboard shortcuts.

Options

General

Toolpath

Dim inactive toolpaths in highlight mode

Use this command to control the toolpath display when in highlight mode (shortcut menu > Highlight mode). If this function is enabled, all unselected toolpaths are dimmed.

You can define the colour of the dimmed toolpaths in a profile (menu: Options > Profiles > Colours). If this function is disabled, all unselected toolpaths are hidden.

Frame axes length: Use this command to control the length of the axes of the frame's coordinate system.

Tool vector length: Use this command to control the length of the tool vectors in 5-axis cycles. This is only available if the Show 5-axis tool vectors option has been enabled (see also the Toolpath Properties dialogue box).

Graphic view

Pixels per pan step: Use this command to control the increment when panning across objects in the graphics window.

Degrees per rotation step: Use this command to control the increment when rotating objects in the graphics window.

Temporary files

Delete temporary files after use: Use this command to delete temporary files from the hyperVIEW/tmp folder after they have been used.



Delete all temporary files now: Deletes all files located in the temporary directory. The function is only available after *hyperVIEW* has been started, provided no files have been loaded.

Report

General

Show report after generation: Controls whether the file created with the Report command is automatically opened in the standard program after the end of the command. The option has no effect on reports that are generated within a postprocessor run.

Execute report 2017 after NC generation: Controls whether, after generating an NC file (only on the job list level), reports should also be generated.

	Add *.xsl report as template.
	Remove report

If no report template is entered in the list, each time after generating the NC file the user will be prompted to select the required report. But only if the template directory "...reportScript\[Language]joblist\template\..." contains more than one report template.

Overwrite Report 2017 output folder: If the option is enabled, an existing report directory is deleted and a new directory is created.

Storage location

Specify where a created report is to be stored.

Default: The report is stored in the Global Working Space or in the project folder.

Ask: It is possible to define the storage location before saving the report.

Other: Specify another storage location for the report.

	Select report storage location
---	--------------------------------

Logo

Displays the logo used for the report.

	Select another logo.
---	----------------------

Expert

Machine change

Enable query when the machine of a NC-Job changed: When this option is enabled, the program queries upon each machine change whether the machine is to be used as the default machine for all NC jobs.

When the option is disabled, use one of the following two options to specify whether the selected machine is to be used for all NC jobs and/or as default machine

Enable query after changing the default machine: When this option is enabled, then each time the default machine is changed the program queries whether all the NC jobs are to be sent to the default machine.

When this option is disabled, use the following option to specify whether the selected machine is to be the default machine for all NC jobs.

Simulation

Apply tool colours from tool colour table: As a default, you can disable the tool colours list or assign it to the used tools or paths.

Enable the **Off** function to configure the tool colours manually (NC Jobs tab in the *hyperVIEW* browser).

Movements to show when partial tool path display is enabled: Specifies the length of the partial toolpath displayed in the simulation.

Consider spindle: Enable this option to display the tool head in the simulation.

Consider additional information on tool length: Value that is automatically assigned to the tool length (if defined, see the Tool dialogue). The simulation runs without the need for any manual intervention. All report files calculate this additional value and specify the tool length actually required.

Use hyperMILL colours: Enable to take the colours defined in hyperMILL for G1 and G0 movements. If this function is disabled, you can define the colours on the *hyperVIEW* menu under Options> Profile > Colours.

Profiles

Profile: Select the required profile or select the <New profile> entry. Rename the new profile as needed. Use the profiles to define the colours and options of your mouse button functions and key assignments.

Colours: Select the colours required for the available options: toolpaths, surfaces and solids, background and simulation.




Tool Colours: Define the colours for tools.



Colours

Here, you can specify colours for the individual tools. The specified colours are used in the simulation to visualise the tool, the path taken by the tool the material removed by the tool (cutting edge).

The individual colours are ordered numerically.

	Define a new colour.
	Edit the selected colour.
	Delete the selected colour.

View

Left mouse button / Middle mouse button / Right mouse button / Mouse wheel

Here you can define the following functions for the graphics window: rotate, pan, zoom. You can assign functions to the left, centre and right mouse buttons as well as to the scroll wheel. You can also assign them functions that use combinations of the SHIFT or CTRL keys.

Keyboard

Command

For all of the commands contained in the *hyperVIEW* menus, you can assign key combinations consisting of up to four keys.

Category: Select the category in question from the list (the categories correspond to the commands in the *hyperVIEW* menus).



Name: Select the command in question and position the mouse pointer inside the entry field.

Specify key

New key: Press the key combination of your choice.

You can combine any of the following: Shift key, Ctrl key and Alt key with numeric or alpha-numeric keys.

The following keys cannot be used for key combinations: Backspace, Return/Enter, Tab and Escape.

	Assign: Link a key combination with a command.
	Delete: Delete an assignment.

NC file

Common

Use toolpath name of first loaded tool path as default for NC file.

See the information in section **Machine Properties / NC-File properties**.

NC file generation

Ask for tool data modification

Before the NC file is generated, the **Tool data / Stock** window opens for you to modify the entries.

Start NC-Editor after file generation

After the NC file is generated, the editor assigned to the file type in question opens automatically. If several NC files are generated, the NC file selection dialogue is called up.

The function is not activated by default.

Machine model

Rotary axes / Tool axis

You can define the display, colour and length for the machine elements rotary axes, tool axis and reference point for the machines that are used in the simulation.

Increments

Length and angle by which the machine axes can be moved.

Origin name scheme

In addition to the prefix and suffix, you can also include axis descriptions and values in the name scheme.

Print and CSV Layout

Create print layout

Select the **<New layout>** option and enter a name for the layout. Confirm the dialogue box with **Yes** and close it.

Define printing information

Double-click an entry in the **Available values** column to move the selected value to the **Selected values** column and include it in printing.

	Include the selected value in the print and CSV layout.
	Move the selected value to the top of the list.
	Move the selected value one line up.
	Move the selected value one line down.
	Move the selected value to the bottom of the list.
	Delete the selected value.
	Delete all the values in the list.

Accuracy

Metric / Inch

Here, you can define the display accuracy for the following parameters: Position (of tool), Feed rate (of tool), Direction, Spindle speed and Simulation.



The default value defined for simulation can be overwritten in the Tool dialogue (shortcut menu : Toolpath).



Video

Video Options

Video codec: You can specify the video format here. The following formats are available by default under Windows: Cinepak codec (Radius), Uncompress, Indeo Video 5.1.



Configuration

For further information on how to configure the individual video formats, consult the documentation supplied with the formats.

Quality: If the selected format supports this option, this is where you can set the display quality.

Frame rate: Use this parameter to specify the number of frames (images) per second. High frame rates increase the file size and slow down simulation. Our recommended frame rate is 5 fps; this still yields a reasonably high display quality.

Source: Specify the video source. You can choose between the following options: Application: the content of the entire application window is captured. Graphics window: the content of the graphics window is captured. Screen: the content of the screen is captured.

With mouse cursor: the mouse cursor is included in the capture.

Functions in *hyperVIEW* browser

All of these functions can be accessed from the **NC-Jobs**, **Simulation** and **Machine Setup** tabs of the browsers.

NC-Jobs

Manage NC jobs. An NC job (*.hv) consists of the following elements: machine, stock, model, result, rest material, clamp and toolpaths.

Simulation

Simulate NC jobs. To control the simulation, set the machine origin, configure the display of the machine, tool and model and configure the collision check.

Machine Setup

Specify the settings for the machine model, including the active origin, machine position, the available workspace and the machine display. These settings are only available if the machine definition includes a machine model.

NC-Jobs tab

To create a new NC job, select **Edit > New NC-Job** on the menu bar or right-click in the browser window and select **New NC-Job**. A new NC job entry appears in the browser alongside a list of all associated elements (tree structure).





The name defined in the Machine Definition or NC-File Properties dialogue is used as the file name of the new NC job.

Icons in the *hyperVIEW* browser

	NC Job (postprocessor run has not yet been carried out)
	NC Job (postprocessor run has been carried out and an NC file has been generated)
	Machine
	Part (model, result, residual stock) is defined
	Part (model, result, residual stock) is not defined
	Stock is defined
	Stock is not defined
	Clamping devices
	Toolpaths
	Frame system
	Element is visible



	Toolpaths in highlight mode
	Element is not visible

Drag-and-drop functions

You can copy and move the stock, model, result and toolpath entries of an NC job using drag-and-drop operations.



Exception:

The result of one NC job cannot be used as the result of another NC job.

To move an entry, select the entry and drag it the required position in the *hyperVIEW* browser while holding down the mouse button.

To copy an entry, select the entry and drag it the required position in the *hyperVIEW* browser while holding down the **CTRL key**.

The standard **Copy** and **Paste** commands apply to all of the entries.

To edit the NC job that has been created, select the NC job and choose the **NC-File Properties** function on the shortcut menu.

To edit a machine assigned to the NC job, select the machine and choose the **Machine properties** function on the shortcut menu.

hyperVIEW usually creates one NC job for each joblist received from *hyperMILL*. To combine toolpaths from different job lists into one NC file, use the **Combine all** function.

Machine Properties / NC-File properties

Use the two dialogues to specify which parameters are included in the generation of NC files. Note that the data entered in the **Machine properties** dialogue is saved as part of the machine definition. In the **Machine properties** dialogue, you can define additional machine-specific information such as name, postprocessor or origins. When you create a new NC job, all of the properties of the current machine are imported. You can use the **NC file properties** dialogue to overwrite specific machine definition properties.

General

Name/Type: Enter machine designation and type.

Adaptor: Define adaptor.

Multiple solution support: Available if several solutions are possible for the rotation axes of the defined machine.

Postprocessor: Click this entry to show the icons for defining and configuring the postprocessor.



Select the postprocessor required for the machine.

OPEN MIND recommends that all postprocessor files be managed in the **Global Working Space** in the *hyperVIEW* directory. Due to compatibility with Windows UAC, you can no longer manage files in the **C:\ProgramData\OPEN MIND\hyperVIEW** folder.



You can also use a postprocessor in a shared network. In this case, you should always save the *.def file as a write-protected file in order to protect it from unwanted changes.

	Configure the postprocessor.
	Show information on the machine model.
	Copy information on the postprocessor to the Windows clipboard and, if necessary, make it available to your OPEN MIND partner. To do this, paste the information stored in the clipboard into an e-mail.
	Select a machine model file (*.mmf/*.mmb). Machine definitions (*.mdf) are managed both for <i>hyperVIEW</i> 32-bit and <i>hyperVIEW</i> 64-bit in the Global Working Space of the MDF directory.
	Configure the machine model.
	Show information on the machine model.
	Edit the machine model file (currently intended for internal OPEN MIND use only).

External simulation: VERICUT is available as an external simulation program.

Origins

Select the **Enabled** checkbox and select an origin table file (.odf) via the **Origin table** entry.

	Select an *.odf file. If the file is not located in the C:\Users\Public\Documents\OPEN MIND\mdf [Global Working Space] directory, the origin table file will be copied into this directory.
	Edit the origin table file. The origin definition in the machine administration is only available with the proper licence and postprocessor. 5

NC file

	Select a target folder for new NC file.
--	---

Specify the **file name** and **file extension** for the NC file created in the postprocessor run. The name of the NC file is also used when a new NC job is created.

Write comments to NC-File: Select this option to adopt comments from toolpaths in the NC file.

If the **Use POF-file name of first loaded tool path as default for NC-File** option is selected on the **Options > NC-file** menu, the name specified for an NC file in the **Machine properties** dialogue is replaced with the name of the first loaded toolpath.

Prologue, Epilogue: Text issued at the beginning and end of the NC file which may contain the following variables:

Version > Version

<VERSION>

Version > Postprocessor version

<OMA_VERSION>



Version > Postprocessor program version	<PRUN_VERSION>
Date/Time > Creation (day)	<DAY>
Date/Time > Creation (month)	<MONTH>
Date/Time > Creation (year)	<YEAR>
Date/Time > Creation (hour)	<HOUR>
Date/Time > Creation (minute)	<MINUTE>
Date/Time > Creation (second)	<SECOND>
Job list comments > 1, 2, 3...	<JOBLIST_COMMENT_1, 2, 3>
Machine > Name	<MACHINE_NAME>
Machine > Type	<MACHINE_TYPE>
Machine > Adaptor	<MACHINE_ADAPTOR>
Machine > Model name	<MACHINE_MODEL_NAM
Machine > Model path	<MACHINE_MODEL_PATI
Statistics > Total machining time - hours	<TOTAL_TIME_H>
Statistics > Total machining time - minutes	<TOTAL_TIME_M>
Statistics > Total machining time - seconds	<TOTAL_TIME_S>
NC > Path	<NCPATH>
NC > File	<NCFILE>
NC > File name	<NCFILE_NAME>
CAD > Path	<CAD_PATH>
CAD > File	<CAD_FILE>
CAD > File name	<CAD_FILENAME>
CAD > File path	<CAD_FILEPATH>
Other > NCS name	<NCS_NAME>
Other > NCS comment	<NCS_COMMENT>
Other > Material	<MATERIAL>
Other > Fixture name	<FICTURE_NAME>
Other > User	<USER>

DNC start/DNC end: information that is not processed by the postprocessor itself, but by programs that are invoked after the postprocessor run (for example, management tools for NC files). You can use the same variables as in the fields **prologue** and **epilogue**.

Job mode

One NC-file per job: Select this option if a separate NC file is to be generated for each tool-path. Otherwise, all toolpath files will be written to a single NC file during the postprocessor run. The job name or joblist name is used here. If a separate NC file is written for each tool-path, the rule for file name generation is applied when the respective NC file is generated.

Compound job as single job: A separate NC file is generated for a compound job.

Rule for file name generation: You can enter any combination of numbers and characters. The pre-defined parameters for the job list name, job name, job ID and counter are also available.

Start of counter: The generated files are numbered consecutively, beginning with the first number specified (n; n+1; n+2).

Increment to job ID: Amount of incremental increase in job ID.

Numbers

First program number / First block number: For program and block numbering, you can specify the first number as well as the required increment (depending on which postprocessor is selected).

Tool

Type of tool change: Tools can be changed manually or automatically. Tool change position: If the tool is changed manually, you need to enter the corresponding position here.

Go to position before first tool change: The position of the manual tool change will also be considered/approached for the first tool.

Time for automatic tool changing: Enter the time available for automatic tool changing.

Positions

Start of machining X/Y/Z and End of machining X/Y/Z: If required for control, enter the tool position at the start and end of machining.

4/5 axes

Rotation axis X, Y, Z: Centre position of rotation axis in NCS.

Clearance radius / Clearance distance: Define the clearance radius for machines with more than three axes. Define the clearance between the tool and the rotation axis or workpiece (when indexing).

Clearance height: Z position of the tool before tilting.

Axis interpolation: Specify the angle or vector (great circle).

Max. deviation (degree): Enter value.



The evaluation of this parameter is PP-specific.

Limits

Maximum rapid feedrate / Maximum machining feedrate / Maximum spindle speed / Maximum file size / Maximum number of NC lines: If the toolpath file (*.pof) contains large values for the above parameters, they are automatically reduced to the maximum values specified here if the corresponding option is selected.



Withdraw and retract movements and machine stop are entered prior to separation.



Cylinder Barrel

Settings in this category are only required in the specific case of 2.5D cylinder barrel machining. This requires a special postprocessor.

Program halt

Enter commands in the NC file that trigger a program halt at critical points.

Under **Mode**, specify whether these commands should be inserted before or after a tool change.

Commands: Enter the commands you want to insert in the NC file.

NCS Transformation

Define a predetermined offset for the NCS. First, the rotation is executed, followed by the displacement. Thus, a displacement refers to the rotated NCS. A warning is issued in this context when loading data in *hyperVIEW* and during the postprocessor run.

Select the **Enabled** checkbox to turn the NCS transformation on. **X offset** moves the NCS in the direction of the X axis, **Y offset** in the direction of the Y axis, and Z offset in the direction of the Z axis. A value must be entered for the required distance. **Rotary axis** determines the axis of the NCS around which the NCS is to be turned. **Rotation angle** defines the value for the rotation. It is applied only if the **rotation axis** has the value **x**, **y** or **z**.








Collision check

Only available in the **Machine properties** dialogue.

Collision check options for the simulation. Select the **Enabled** checkbox below the required option in order to enable it. **Tolerance** and **colour** can be specified for each individual option. All other information, such as machine, stock and so on, are taken from the first NC job in the *hyperVIEW* browser.

Machine Administration

Use Machine Administration to create new machines or edit existing machines.

	Open the Machine administration dialogue.
	New: Create a new machine.
	Edit: Edit a machine. Specify the parameters of your choice in the Machine Definition dialogue.
	Delete: Delete the selected machine.
	Copy: Copy the selected machine.
	Import: Import a defined machine (load a machine definitions file in *.mdf format).
	Default: Define the selected machine as a default for when a new NC job is created.

Select the machine by entering characters

When entering letters from the keyboard, a machine below the currently selected machine is selected that matches the letter(s) entered. In case of 'rapid' input, the character string is evaluated in context.

Stock

Use the shortcut menu of the Stock entry to assign a stockmodel to the NC job by defining it as a cube/cylinder or importing it as a mesh in *.STL or *.VIS format. If you open the program from *hyperMILL*, a stockmodel can also be transferred. The stock is used for the following purposes displaying the simulation, and calculating the results.

Machining sequence

To use the calculated stockmodel as a basis for the stockmodel of the subsequent NC job, copy the stockmodel to the stockmodel of the NC job using a drag-and-drop operation. Insert a stockmodel using the shortcut menu **Insert cube / Insert cylinder / Insert mesh**. Specify the required stock parameters in the **Cubical stock model** or **Cylindrical stock model** dialogue.

Dimensions

X, Y, Zmin /Zmax: The minimum and maximum dimensions of the cube are proposed by the program and can be changed as required.

Appearance

Select the view options of your choice.

Centre of bottom

X, Y, Z: A centre value is proposed by the program. You can change this as required.

Boundary Offset

Diameter, Height: Use the values or enter your own.

Direction: Select the orientation of the stockmodel along the X, Y or Z axis.

Appearance

Select the view options of your choice.



If one or more toolpaths have already been assigned to the NC job, the measurements of the stock are automatically adapted to the existing toolpaths.

If a stock (cuboid/cylinder) has already been defined in the CAM application, this is proposed for the simulation.

Modify the dimensions so that they match the toolpath to be simulated.

The stockmodel can be made visible or invisible, or displayed in shaded, transparent or wire-frame mode. The **Properties** shortcut menu of a defined stockmodel allows you to display additional properties (Number of elements) and edit others (Visibility, Colour, Mode).

Model

Use the **Insert part** shortcut menu to assign a model in *.3DF, *.STL or *.VIS format to the NC job. If you open the program from *hyperMILL*, a model can also be transferred.

Result

Use the **Calculate** shortcut menu to calculate the resulting stock. Prerequisite: Stock and toolpaths have been assigned. The calculated stock is displayed in the graphics window under **Result**.







Toolpaths

Using the shortcut menu of the **Toolpaths** entry, you can assign toolpaths to an NC-Job and check and change the properties of toolpaths. To insert a toolpath, use the **Insert toolpaths** shortcut menu and open the **Insert toolpath file** dialogue:

File selection

set filter (*.pof/*, etc.).



	Set the drive and directory or enter the path manually and click the arrow to apply.
	Select the path from your favourites.
	Click the Insert button to copy the set path into your favourites list. Accept files. Select the required files in the top list displayed.
	Files are copied to the lower selection list by clicking the arrow with the left mouse button or by double-clicking the file name. Import files with OK .
	Remove selected entries / all entries from selection list.
	Move selected entry upwards / downwards.

To improve clarity when several toolpaths are displayed in the graphics window, the toolpaths can be displayed in highlight mode and/or in different colours.

The **Tools...** shortcut menu of the **Toolpaths** entry provides a list of the tools used in the NC job.



If the tool does not have a unique NC number, for example, if several tools with different geometries use the same NC number, the respective tools are highlighted in red.

Various tool parameters are displayed in the list display in the dialogue. Some of these parameters (Accuracy, 5X comp. length) can also be edited directly. Select the desired entry in the list, right-click, and select **Edit**.

Double-clicking an entry in the list opens a Tool dialogue box. This dialogue box displays an overview of all the tool's geometry and technology parameters. The most important technology parameters can also be edited.

When you are done, click **Close** or **Exit**.

As well as tool parameters, the list also displays the tool/toolpath colours defined in **Options > Profiles**. If no default behaviours are defined in the **Options** menu, you can assign the defined colours to the tools or toolpaths here.

Click the **Reset** button to use the default colour for all the tools. In addition to the colours already defined in **Options > Profiles**, you can define new colours for the individual tools/toolpaths here.

The **Coordinate systems** shortcut menu lists all the coordinate systems of the current NC job including **origin** and **reference system**.

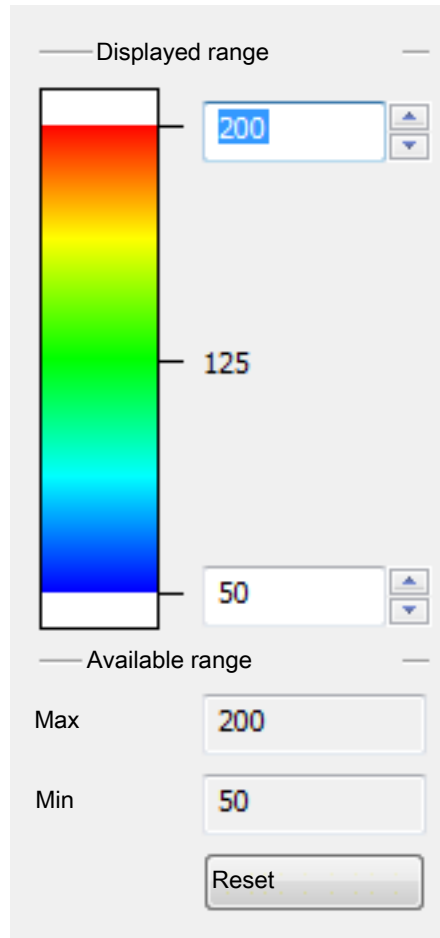


The report file that is generated only contains information on the selected NC job or toolpath. The printout lists information on all of the NC jobs contained in the browser.

In addition, you can list individual statistics for each toolpath, such as: length of movement in G0, G1, G2/G3, dwell times and the number of tool changes.

Feedrate Analysis

The feedrate values of the calculated toolpaths are displayed using different colours. This allows for an easier and faster check of the toolpaths and the feedrates selected or calculated. Particularly during **High Performance** roughing, the realised feedrate adjustment can be checked and reproduced easily.



Displayed range

Change the largest or smallest feedrate displayed manually or with the arrow keys. The colours used for each feedrate area are displayed in the colour scale.

Toolpaths with only one feedrate value are shown in red,

Feedrates outside the displayed range are shown in white.

Select **Reset** to restore the original values.

Available range

Max / Min: Largest and smallest feedrate in the toolpath.



In 2D Contour Milling, only the centre path is displayed.

The **Properties** shortcut menu displays the following parameters for editing: the most important technology parameters: Feedrate, Infeed, Spindle speed.

The toolpaths can be displayed in an external editor as follows: In the Properties dialogue, right-click the respective entry next to **Source file** and choose **Show toolpath**.



Write an NC file. Select storage location.

The following parameters are displayed on the **Properties** shortcut menu but they cannot be edited: dimensions, internal number (relating to the current NC job), cycle description (including creation date) and display settings for the graphics window (visibility...).

Appearance

Visible: Select this option to display the toolpath that has been inserted in the graphics window.




Show 5 axis tool vectors: Select to display the 5-axis vectors for 5-axis toolpaths.

Display truncated toolpath: Toolpaths that have been cut off and that led to a holder collision (stored in the toolpaths) are displayed in the simulation.



Show NC path: This option is not available unless the toolpath has been created with a cycle that contains the **Path compensation** option (2D Contour Milling, 2D Contour Milling on 3D Model and turning cycles). The difference between the corrected and the simulated toolpaths is shown.

Double-clicking the colour fields also allows the respective colours to be defined for G0 and G1 as well as for NC paths (G0 / G1) and local points.

	Use the Point properties function on the Measure toolbar to display local points in the graphics window.
	Generate a list of the cycle parameters set.
	Open the toolpath of a job in an external editor.

NC text: Enter user-defined text blocks such as control instructions or other similar parameters in the NC text dialogue.

Insertion position / NC text

Select the corresponding position where the NC text is to be inserted. There are no limitations to the text length or the characters used (including symbols).

Copy: Copy the NC text defined to the clipboard.

Load: Open any text file. The content of the file is displayed in the NC text area.



As *hyperVIEW* does not perform any syntax or collision checks on the entered data, you should only use this function if you are certain the used machine can interpret the data correctly.

Simulation of compound jobs

A **compound job** is shown as an entry in the *hyperVIEW* browser (1). The following functions are available on the shortcut menu of the **compound job**:

Insert toolpaths: Load a toolpath file and add the job to the compound job.

Edit NC-File: Open and edit the generated NC file in the editor. Requirement for generating an NC file for a compound job: In the **NC-file properties** dialogue, the **Job mode > Compound job as single job** option must be selected.

Simulation: Simulate all the jobs of the compound job.

Tools: Show all the tools of the joblist.

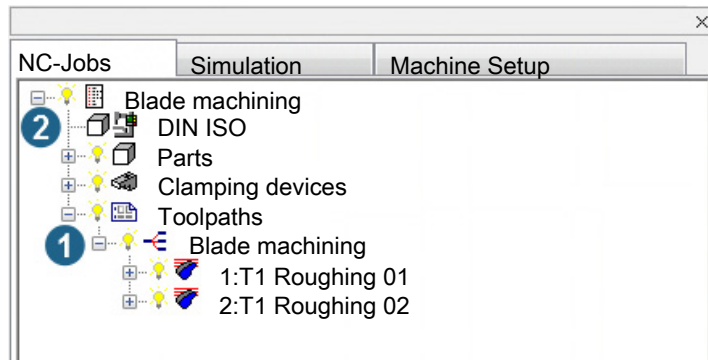
Statistics: Show the statistical data of all the jobs in the compound job.

All other functions in the shortcut menu correspond to the shortcut menu for an individual job. For more information on this, please see the previous section.

The following functions are available on the shortcut menu of the **Machine properties** entry (2):

Job mode > One NC-file per job: Select this option if a separate NC file is to be generated for each job.

Compound job as single job: Select this option to write all the jobs in a compound job to a single NC file. This function can only be enabled if the **One NC-file per job** function is also enabled. The name of the compound job is used for the <JOB> placeholder in the **NC file**.





Simulation tab

Simulate NC jobs. To control the simulation, set the machine origin, configure the display of the machine, tool and model and configure the collision check.

Machine origin

Select the machine origin from the list. The toolpaths and model are aligned to the origin.

State

List of all simulated jobs. Information no the tool number, tool geometry and coolant.

	Open the Solutions dialogue
---	------------------------------------

Select solutions for rotary axes


Depending on the machine kinematics, there may be several approach and retract paths for different tools. Select the most suitable solution in the dialogue.

The availability of a machine model is a prerequisite to calculate possible rotary axis solutions. If these solutions will also be used by a postprocessor, the machine model and the postprocessor must be synchronized. Please contact your OPEN MIND dealer to arrange this. Implementation may involve adaptating your postprocessor and this might incur a fee.

Toolpath: Shows the job.

Solution: All possible solutions for the selected job as calculated by *hyperVIEW*. If you have machines with serration, possibly more than two solutions are found. In this case, the number of offered solutions depends on the defined Maximum deviation (degree) for tilting, which can be defined in the Machine Properties dialogue, 4/5-Axes category.



Deviation: Real value for the tilt deviation.


	In some types of machines (such as those with parallel rotary axes), one of the rotary axes may need to have a fixed position.
---	--

Enable solution

Double-click the **Solutions** column to enable and use a solution. The solution will be shown in a bold font. The selected solution is then used in the simulation, but not for NC output. To also have the selected solution used for NC output, you must save the selected solution.

Save active solutions.

	Save the active solutions for all jobs. These will be stored after closing the program and remain available the next time the program is started. An exception to this is if the job was recalculated in <i>hyperMILL</i> .
	Delete the active solutions.

	After switching to the Simulation tab at least once, the stored solutions are shown as a tooltip for the corresponding toolpath in the <i>hyperVIEW</i> browser.
---	--



Application of active solutions

In the *hyperVIEW* menu, click **Options > NC file**. Select the desired option under **Stored solutions** and close the dialogue with **OK**.

Delete stored solution

To delete the stored solution, select a toolpath in the *hyperVIEW* job browser, and select the **Delete stored solution** function from the shortcut menu. The stored solution for the marked toolpath is deleted.

There is currently no selectable solution available for jobs containing transformations. OPEN MIND recommends combining jobs containing transformations into a linking job. Please note that to receive a solution selection, the transformations must be defined in the sub-jobs.



	<p>Display the fixed rotary axis. You can specify the corresponding angle directly in the box (double-click). After the angle has been entered, the available solutions are recalculated.</p>
	<p>Compare the available and used workspace.</p>

Workspace

Limit switch: Available workspace of the machine model.

Min / Max: Workspace used in the simulation. All displayed values refer to the defined reference point.







Additionally, the difference values between the used and available workspace can be shown.

	<p>Display the active reference point.</p>
	<p>Violations of the available workspace. You will need a special licence to control the defined workspace. Please contact your OPEN MIND dealer to arrange this.</p>

Machine origin

Find best fit: Looks for the machine origin that produces no workspace violation or the smallest possible workspace violation.

Define best fit: This function opens the Machine Origins and Reference Points dialogue and displays the automatically defined Best Fit location in a blue colour.

	<p>Insert a new origin/reference point</p>
	<p>Edit an origin/reference point</p>
	<p>Delete an origin/reference point</p>
	<p>Activate an origin/reference point</p>
	<p>Insert the origin from current machine position. Set reference point from current machine position.</p>
	<p>Move the machine to the selected origin/reference point</p>



Legal Reminder for the Usage of the Machine Tool Models and Machine Tool Simulation

The terms of the End-User Licence Agreement are also valid here. OPEN MIND does not warrant in any way situations where incorrect data and incorrect information delivered by the licensee have lead to the creation of an incorrect machine tool model. Licensee shall make an incoming inspection immediately after the delivery of each machine tool model.

Use selected: The currently set machine origin is used.

	Copy the maximum and minimum values displayed to the clipboard.
	Open the Collision check options dialogue.

Enable: List of available checking options. Enable the desired option. Only those options that are useful for the NC job to be simulated are available. However, all options are available within the Machine properties in the Collision check dialogue.



Tool against model: If this option is selected, only the non-cutting part of the tool will be considered in *hyperVIEW*. This is defined using the Cutting length parameter of the tool. If the Cutting length has the value 0, it will be assumed that the entire tool cuts.

When the collision check is enabled, all G0 traversing movements are checked against the model, whereby the tool and holder can be checked separately. G1 traversing movements are not checked against the model.

Double click an entry in the **Max. increment** list to specify the accuracy of the collision check. Note that higher levels of accuracy will slow down the simulation.

Double-click an entry in the **Colour** list to define the colour used to display detected collisions during simulation.



Both the elements enabled in **Collision Check Options** and the toolpaths will be highlighted in the specified colour.

All of the options provided in this dialogue box can also be accessed via the corresponding right-click shortcut menu. This way, you can modify the overall definitions for the list entries. You can also enable/disable collision checking for all of the list entries.

Shaft allowance: During simulation, the tool shaft is increased/decreased (positive/negative values) by the specified shaft allowance. This helps to reveal collisions in complex situations where the shaft is required to have a minimum distance to the part.

	Show collision status: Display collisions that were detected during the simulation in a list.
	Show properties of the active toolpath's tool: Display an overview of all the tool parameters. This dialogue is the same as the dialogue available from the Toolpaths shortcut menu in the <i>hyperVIEW</i> browser.
	Show properties of the active toolpath: Display the parameters of the currently simulated toolpath. This dialogue is the same as the dialogue available from the Toolpaths shortcut menu in the <i>hyperVIEW</i> browser.



In contrast to the dialogues available on the shortcut menu (NC-Jobs tab), the parameters in the two dialogues on the Simulation tab serve only to provide a better overview. They cannot be edited.

Controlling the simulation

Setup

Stop simulation: In some situations, the simulation can be stopped automatically. Set the corresponding switches to **On** if the simulation is to be stopped.

M0: Stop at the next halt marker.

TC: Stop at the next tool change.

Coll: Stop at the next detected collision.

LS: Stop at the limit switch.

Use **Continuous** to simulate the tool movements continuously; this is done in **Step Mode** in single steps corresponding to the toolpath.











Override: Please use the scroll bar to control the simulation speed. In continuous mode, you can freely control the speed. In step mode, you cannot select a value below 1x.

Override

Use the slide control to set the speed of the simulation.

Animation control























Control the simulation.





	Go back to first toolpath
	Previous toolpath
	Fast backward
	Backward
	Stop
	Forward
	Fast forward
	Forward to movement no.
	Next toolpath
	Go to end of last toolpath



Mode

Configure the display for material (removal), model, machine, tool and toolpath.

	Material removal simulation disabled. Open the Create stock dialogue. Define the shape and display for the stock.
	Material removal simulation enabled (only if a stockmodel has been defined).
	Material removal shaded
	Material removal transparent
	Material removal wireframe
	Save the result of the material removal simulation in the *.stl, *.vis or *.omg format.
	Model shaded.
	Model transparent.
	Model wireframe.
	Model invisible.
	Automatic collision checking disabled.
	Automatic collision checking enabled.
	Enable collision checking to include the activated checks. A reverse simulation is not possible.
	Open the Machine structure dialogue.
	Machine component visible.
	Machine component invisible.
	Show all machine components.
	Hide all machine components.
	Tool shaded
	Tool transparent
	Tool as wireframe
	Tool invisible

	Display all toolpaths.
	Only display the current toolpath.
	Only display the current tool movement.
	Do not display any toolpaths.

Clears all conflicts: All of the conflicts displayed in the simulation (collisions and workspace violations) are deleted. This applies to the toolpaths as well as to all other visible elements in the simulation (model, machine, etc.).

Collisions Visible: Select this option to display the affected parts (machine, model, and so on) in the defined colour.

Clears collisions: All of the collisions displayed in the simulation are deleted.


Linear Axes Conflicts Visible: Violations of the maximum permissible workspace are displayed in the defined colour.

Clears linear axes conflicts: All of the displayed conflicts (see above) are deleted from the simulation.

Rotary Axes Conflicts Visible: Positions where the values of the machine's rotary axes are not permitted are displayed in the defined colour.

Example: a rotary axis may only move between +90° and -90°.


Clears rotary axes conflicts: All of the displayed conflicts are deleted from the simulation.

	Open the Machine structure dialogue.
---	---

Part: Display components available for the machine model.

Colour: Define the colour for the machine component. Double-click the component line, then click the triangle icon to open the colour selection dialogue and select the required colour.

Display: Define the display option for the machine component. Double-click the component line, then click the line to show the available options (Shaded, Transparent, Wireframe) and select one. Use the machine component shortcut menu (right-click) to unify the visibility, display and colour for all components. Alternatively, control the visibility using the **Show all** and **Hide all** buttons.

	Open the Simulation info dialogue.
---	---

The following parameters are displayed:

General

Movement no. / all, movement type, machining time.

You can also obtain this information and the tool change times from the Statistics shortcut menu of the Toolpaths entry on the *hyperVIEW* browser > NC-Jobs tab.

Position

The current position of the tool (X, Y, Z), the workspaces defined for each direction as well as workspace violations.

Rotary axes

The movements of the Rotation axes (if available on the respective machine model) and their workspaces.

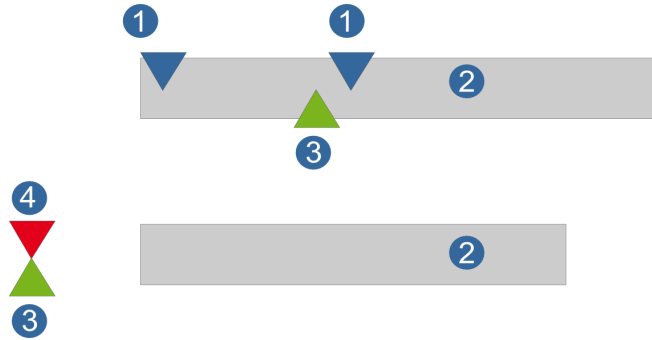


Tool vector

For turning cycles, display is extended by adding the turning-specific tool vector parameters (U, V, W), as well as the feedrate and spindle RPM.

Technology

Displaying the tool position and workspace in the Simulation Info dialogue box



- (1) Maximum start and end points that were reached in the current simulation.
- (2) Defined workspace.
- (3) Current position of the tool.
- (4) Workspace violation.

Machine Setup tab

Specify the settings for the machine model, including the active origin, machine position, the available workspace and the machine display.

Active origin

The active machine origin is the machine's NC zero point. Select the **<Automatic>** option to use the origin that is suitable for the machine model. If the automatic machine origin fails to function correctly, there is a fault in the machine model. In this case, please contact your OPEN MIND dealer.


Select the coordinate values for the machine origin from the list or

	open the Machine origins and reference points dialogue.
---	--

The machine origin is based on one of the following: the program coordinate system or the machine coordinate system (and a corresponding reference point).





Machine Position

Individual **machine positions** can be approached with full accuracy. To do this, right-click in the **Position** column for the relevant axis and select **Edit** or double-click in the relevant line.

	Move the respective axis of the machine model in the graphics window.
--	---



Use the shortcut menu to enter additional appearance options. To do this, right-click in the **Position** column and select **Options** or double-click in the relevant line.

The **Machine model options** dialogue opens.

	Available workspace: If the workspace is not defined in the machine model, define the minimum and maximum values for the machine's linear axes directly in the input line.
	Take the minimum and maximum values for the machine's linear axes from the current machine position.
	Move the machine model to the defined position. The values defined here for the linear axes (X, Y, Z) are transferred to the Available and Used Workspace dialogue and are displayed as values for the limit switch.
	Open the Rotary Position dialogue.

Tool vector




Enter the tool vector (U, V, W).

	Calculate the position for the rotary axes (based on the tool vector).
	Move the machine model to the defined position.






Polar angles

Define the tilt angle (B axis) and rotation angle (C axis). These options are only available for machines that support serration. Special codes are required for this functionality. Please contact your OPEN MIND partner.












	Graphical preview of the defined angle.
	Define the slope and rotation angle directly from the displayed machine model.
	Define the tool vectors and calculate the rotary axes.

Mode




	Display the stock, model and result in shaded mode
	Display the stock, model and result in transparent mode
	Display the stock, model and result in wireframe mode
	Stock, model and result invisible
	Open the Machine structure dialogue.

Toolbars

Standard

	Open <i>hyperVIEW</i> file or <i>hyperVIEW</i> archive file.
	Save file in *.hv or *.hvz format
	Create new NC job
	Delete all NC jobs
	Generate report file
	Open message window (postprocessor)
	Open the Print dialogue box
	Start the online help
	Write NC file

Machines






















	Choose a machine to use for the simulation.
	Open the machine definition dialogue
	Open the machine administration dialogue










The Machines toolbar is hidden by default. To show the toolbar, select View > Toolbar > Machines.



View and Measure

	Top view
	Front view
	View from left
	View from right
	Back view
	Bottom view
	Back view from right
	Back view from left
	Front view from left
	Front view from right
	Zoom into graphics window
	Zoom out of graphics window
	Match graphics window magnification to the model
	One step up
	One step down
	One step to the left
	One step to the right
	X axis anti-clockwise
	X axis clockwise
	Y axis counterclockwise
	Y axis clockwise



	Z-axis counterclockwise
	Z-axis clockwise
	Analyse the properties of a point, display point coordinates.
	Analyse the properties of a movement. Show type of movement (feedrate G1, rapid G0), point coordinates of start and end point, length, angle in Z and XY.
	Measure the distance between two points. Display the point coordinates and the distance.
	Measure the distance between a point and a movement. Display the coordinates of the selected point, the nearest start or end point and its distance.
	Measure the distance between two movements. Display the coordinates of the start and end points and their distance.

Status bar

(Me) metric, (In) inch: Indicates the measurement system used. To change, click the entry in the status bar and select the required measurement system.

Video

(1) Start video capturing, (2) Pause video capturing, (3) Stop video capturing (4) Open the Video options dialogue to configure video capturing. Information in section **Video**.



Index



- A**
Accuracy
 hyperVIEW 12-13
Active machine origin
 hyperVIEW 12-33
Animation control
 hyperVIEW 12-29
- B**
Browser icons
 OPEN MIND tool database 11-57
Bullnose end mill
 insert parameters 11-16
- C**
Calculating a result
 hyperVIEW 12-21
Clamping method 11-19
Controlling the simulation
 hyperVIEW 12-29
Coupling types
 combination options 11-31
Creating an NC job
 hyperVIEW 12-15
Creating depot elements
 OPEN MIND tool database 11-52
Cutting point setup
 Turning tools 11-50
Cutting point setup (turning tools) 11-50
- D**
Data exchange
 OPEN MIND tool database 11-53
Database formats 11-2
Define insert
 From ISO code 11-17
Define tool holder
 by ISO/ANSI code 11-23
Defining a coolant 11-40
Defining a depot
 OPEN MIND tool database 11-52
Defining a formula 11-37
Defining a free geometry for a tool 11-15
Defining a machine
 hyperVIEW 12-3
Defining a material/cutter material 11-38
Defining a tool holder 11-22
Defining an insert support 11-25
Defining insert 11-17
Defining turning tool 11-16
Drag-and-drop functions
 hyperVIEW 12-16
- E**
Enable solution
 hyperVIEW 12-26
Expert mode
 hyperVIEW 12-11
- F**
Free point definition
 Turning tools 11-50
- G**
General geometry parameters
 turning tools 11-12
Generating a report
 hyperVIEW 12-6
Generating reports
 OPEN MIND tool database 11-53
Geometric parameters
 Tool holder 11-26
Geometry data
 ball mill 11-7
 boring bar 11-10
 bullnose endmill 11-7
 chamfered cutter 11-7
 chamfered profile cutter 11-7
 drilling tool 11-10
 endmill 11-7
 lens cutter 11-7
 lollipop 11-7
 NC-Tool 11-45
 recessing tool, axial recessing tool,
 parting tool 11-13
 T-slot cutter 11-7
 woodruff 11-7
Geometry editor
 OPEN MIND tool database 11-41
Geometry parameter
 Grinding bit 11-14
Geometry parameters
 Shape of insert 11-20
 threading tool 11-13
 tool holder 11-26
Grinding bit
 Geometry parameter 11-14
- H**
hyperVIEW
 Enable solution 12-26
hyperVIEW browser
 tab
 machine setup 12-33
 NC jobs 12-15
 simulation 12-26
- I**
Inclination angle
 Tool holder 11-27
Insert stock
 hyperVIEW 12-21
Inserting a model
 hyperVIEW 12-21
Inserting toolpaths
 hyperVIEW 12-21
- L**
Linear axes conflicts
 hyperVIEW 12-31
- M**
Machine administration
 hyperVIEW 12-20
Machine simulation
 hyperVIEW 12-4
Menu functions
 hyperVIEW 12-7
- N**
NC file properties
 hyperVIEW 12-16
- O**
OPEN MIND Tool database
 selecting components 11-44
OPEN MIND tool database
 Browser icons 11-57
 Creating depot items 11-52
 Data exchange 11-53
 defining a coolant 11-40
 defining a depot 11-52
 defining a holder and extensions 11-41
 defining a new NC tool 11-44
 defining a tool 11-6
 defining couplings 11-28
 Generating reports 11-53
 Geometry editor 11-41
 opening from the CAM application 11-3
 technology parameters 11-33
 updating technology data 11-36
 user interface 11-3
Orthogonal gauge point
 Turning tools 11-50
- P**
Performing a postprocessor run
 hyperVIEW 12-4
Primary centre
 Turning tools 11-50
Printing information
 hyperVIEW 12-7
- R**
Rake angle
 Tool holder 11-27
Reference angle
 Turning tools 11-51
Rotary axes conflicts
 hyperVIEW 12-31
- S**
Save Spindle, Holder, Extension As De-
fault 11-41
Save Tool As Default 11-6
Screen layout
 hyperVIEW 12-3
Solutions for rotary axes 12-26
- T**
Tab
 NC job 12-15
Tapered tool with shank 11-16
TDM Systems integration 11-56
Tool change position
 hyperVIEW 12-19
Tool colours
 hyperVIEW 12-11
Tool geometry
 Free shank geometry 11-15
Tool holder
 Inclination angle 11-27
 Rake angle 11-27
Tool reference
 centre 11-46
 tip 11-46
Touch probe tool 11-7



Turning tool
Y offset *11-28*

U

Updating technology data
OPEN MIND tool database *11-36*
Usable length *11-48*
Using standard formulas *11-37*

V

Video capturing
hyperVIEW *12-14*
Video format
hyperVIEW *12-14*

W

WinTool hyperMILL interface *11-57*





We push machining to the limit



We push machining to the limit