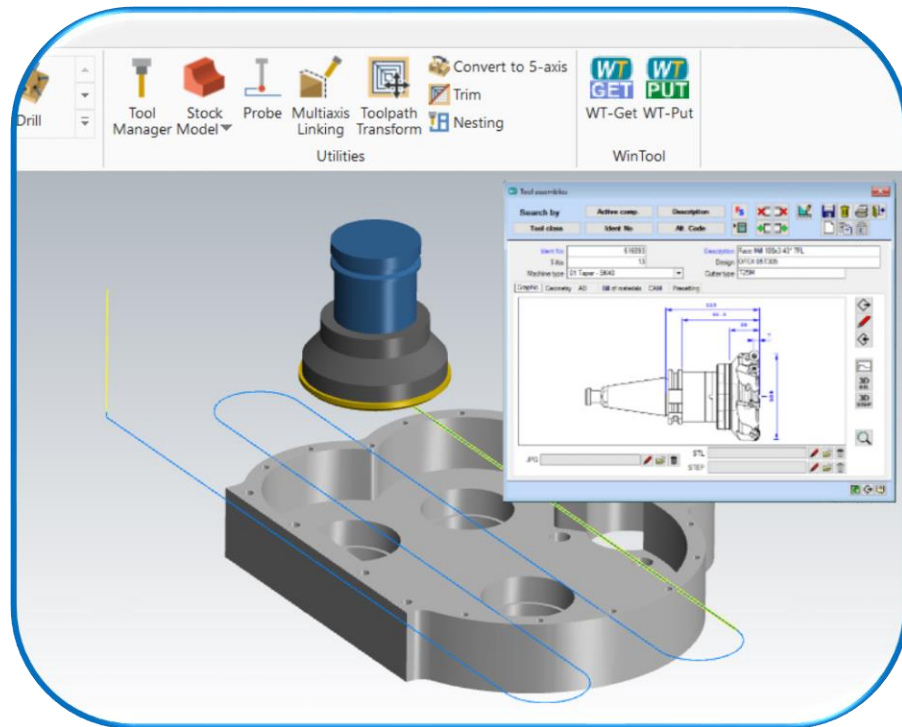


WT-Mastercam Interface



Manual

WinTool Interface 4.6.0 for Mastercam 2026

The WT-Mastercam-Interface enables the user to select and transfer tool assemblies from the *WinTool* database to Mastercam. After creating a NC program, the list of the tool assemblies used in the Mastercam operations manager will be stored back to *WinTool* for further processing in production.

Requirements

- *WinTool* 2018 Professional or later
- Mastercam 2026

WinTool AG
Flüelastrasse 7
CH-8048 Zurich
Phone: +41 (0)44 401 00 55
<https://www.wintool.com>
info@wintool.com

Contents

Summary	4
Job.....	4
Requirements	4
Supported Tool Types	4
Copyright	4
Installation	5
New licensing structure	5
Installing for WinTool prior 2019.1	5
New Directory Structure	5
Update Installation.....	5
Update from version 3.1 and newer	5
Update from version 3.0.1 or older	5
New Installation	5
Mastercam 2017 and up	7
Configuration	9
Mastercam Settings.....	9
WinTool Settings	9
Interface Settings	11
Configuration Window	11
Output Path	11
UserModels Path.....	11
Selection of cutting conditions	12
Tool List Exchange Path	12
Interface Settings for Mastercam 2017 and newer	13
Coolant0-Coolant9	13
Importing Rough Step Percentage Values.....	13
Interface Settings for Mastercam 2018 and newer	13
Tool Name/ Assembly Name	13
Getting Started	16
Sample Database.....	16
Import Tools	17
Tool Numbers.....	21
T-Number Assignment	21
WinTool Tool Assembly T-No	21
WinTool Tool List T-No	21
Duplicate T-Numbers	22
Coolant Import.....	22
Using Cutting Conditions.....	23

Export Tool List to <i>WinTool</i>	24
Step-by-Step	24
Mastercam data fields transfer	25
Preparing Tool Data in WinTool	26
User Classification	26
Machine Configuration	26
Tool ID and Name	27
Regular Tools	28
Special Tool Assemblies	29
Managing Special Tool Assemblies	29
Create a Special Tool Assemblies Contour DXF	29
Special Components	30
Managing Special Component Contours	30
Special Cutters	31
Managing Special Cutters	31
Create a Custom Cutter Contour	31
Cycle Type / Usage (C7)	31
Known Issues	32
Missing license error message	32
Uninstall Error-Message in Mastercam	32
No PUT and GET Buttons Available	32
Incorrect Diameter and Length Correction Numbers	33
Annex	34
Configuration File Parameters	34
General Information	34
WT-Mastercam-Interface.cfg	34
Windows Registry values	35
Local Machine	35
Current user	35
Supported Mastercam Tool Types	36
Milling Tools	36
Not Supported Mastercam Tool Types	43
List of Figures	44

Summary

Job

The WT-Mastercam-Interface enables the user to select and transfer tool assemblies from the *Windows* database to Mastercam. 3D Graphic representation for tools is supported as well as cutting conditions for work materials. After creating a NC program the list of the tools used in the Mastercam operations manager will be stored back to *WinTool* for further processing in production.

Requirements

This WT-Mastercam-Interface requires *WinTool* Professional 2018 or later and Mastercam 2026 or later.

Supported Tool Types

All rotating tool components such as holders, extensions, drills, taps, and mills are supported.

The WT-Shape module is a software component of the WT-Mastercam-Interface and creates assembly contours which are used in Mastercam.

Copyright

This documentation as well as the software is copyright of

WinTool AG

Flüelastrasse 7
CH-8048 Zurich, Switzerland

Phone: +41 (0)44 401 00 55

<https://www.wintool.com>

E-Mail: info@wintool.com

Installation

New licensing structure

WT-Mastercam-Interface 3.8.1 and higher uses the new licensing system WinTool is using since version 2019.1; CodeMeter. If Mastercam is installed alongside WinTool 2019.1 or newer no additional installation is required.

For more details on installation/usage of the new licensing system consider the WinTool manuals, or directly WiBu/CodeMeter manuals.

Installing for WinTool prior 2019.1

Using Mastercam 3.8.1 with WinTool prior 2019.1 requires to also install "CodeMeter Runtime" which is delivered with the installer (CodeMeterRuntime.exe).

New Directory Structure

WT-Mastercam-Interface 3.1 introduces a clear separation of program files and user data.

All user data is centrally placed the [\[Public Documents\]\WT-Mastercam-Interface-2025](#) folder:

User data	New location
Default location of UserModels folder	[Public Documents]\WT-Mastercam-Interface-2025\UserModels
Default location of Exchange folder	[Public Documents]\WT-Mastercam-Interface-2025\Exchange
Configuration files: WT-Mastercam-Interface.cfg WT-MakeList.cfg WT-ToolExport.cfg	[Public Documents]\WT-Mastercam-Interface-2025

Update Installation

A previously installed WT-Mastercam-Interface version will be uninstalled automatically before the new version is installed. The update instructions depend on the currently installed version.

Update from version 3.1 and newer

Follow the instructions in the paragraph "New Installation". After the installation, check the interface configuration using the configuration window (see page 11) and the configuration files "WT-MakeList.cfg" and "WT-ToolExport.cfg".

Update from version 3.0.1 or older

Follow the instructions in the paragraph "New Installation". After the installation, follow one of the two steps to recover the configuration:

- If the interface was installed in the **same directory** as the previously installed version, the configuration files are automatically moved to [\[Public Documents\]\WT-Mastercam-Interface-2025](#). Check the interface configuration using the configuration window (see page 11) and the configuration files "WT-MakeList.cfg" and "WT-ToolExport.cfg".
- If you chose a **different directory** for the interface, you must copy the configuration files manually from the previous installation directory to the directory [\[Public Documents\]\WT-Mastercam-Interface-2025](#)

The default location of the setting [UserModelPath](#) has changed. If you haven't set a [UserModelPath](#) in the interface configuration, in which case the UserModels folder is in the interface installation directory, you must move the contents of the folder to the new default location [\[Public Documents\]\WT-Mastercam-Interface-2025\UserModels](#).

New Installation

Make sure you are logged in with administrator rights to install the software on your PC.

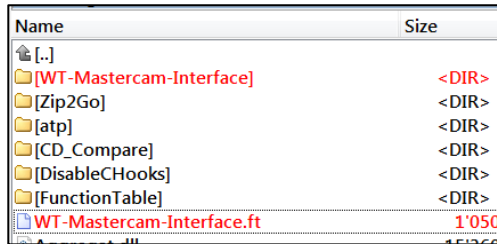
Install *WinTool Professional* first before installing the WT-Mastercam-Interface.

Close Mastercam.

Execute setup.exe from your WT-Mastercam-Interface CD or the download package from the *WinTool* homepage. The default installation directory is:

C:\Program Files (x86)\WinTool\WT-Mastercam-Interface-2025

Note: During the installation some WT-Mastercam-Interface software components will be stored in your Mastercam **chooks** directory:



Name	Size
[..]	
[WT-Mastercam-Interface]	<DIR>
[Zip2Go]	<DIR>
[atp]	<DIR>
[CD_Compare]	<DIR>
[DisableCHooks]	<DIR>
[FunctionTable]	<DIR>
WT-Mastercam-Interface.ft	1'050

Figure 1 Mastercam chooks directory

Setup the new WT-Mastercam Interface toolbar as follows:

Mastercam 2017 and up

Startup Mastercam. Right-click on the tab "HOME" and select "Customize the Ribbon..."

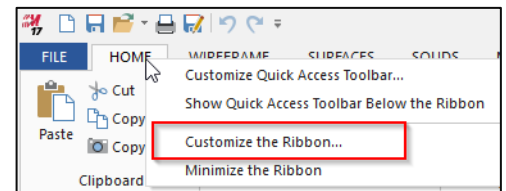


Figure 2 Customize the Ribbon

On the right side of the window select the "Tool Tabs" from the list.

Add a "WinTool" group for the interface buttons:

- For "Mill", right-click on "Toolpaths" and select "New Group"
- For "Lathe", right-click on "Milling" and select "New Group"
- For "Mill-Turn", right-click on "Milling" and select "New Group"

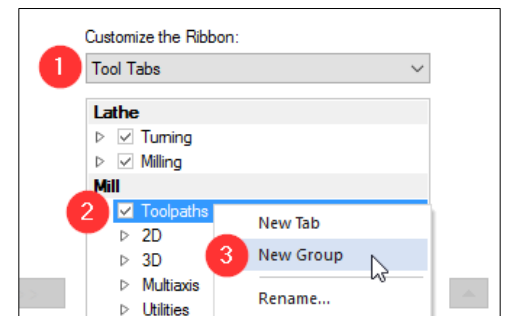


Figure 3 Customize the Ribbon with New Group

Right-click on the "New Group" and select "Rename...". Enter "WinTool" and select OK.

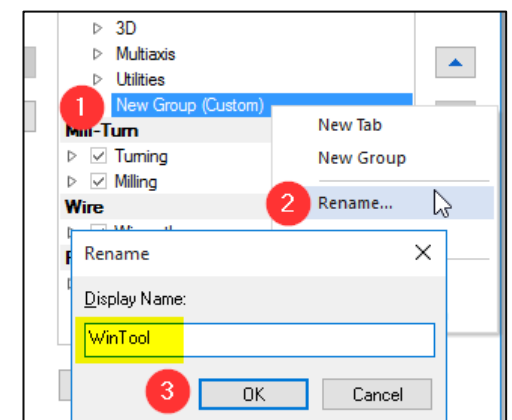


Figure 4 Steps to the WinTool Mastercam Installation

On the left side of the window select "Commands Not in the Ribbon" in the commands list. Select "WT-Get" and click on "Add >>" to insert the button in the *WinTool* group. Now select "WT-Put" and click on "Add >>":

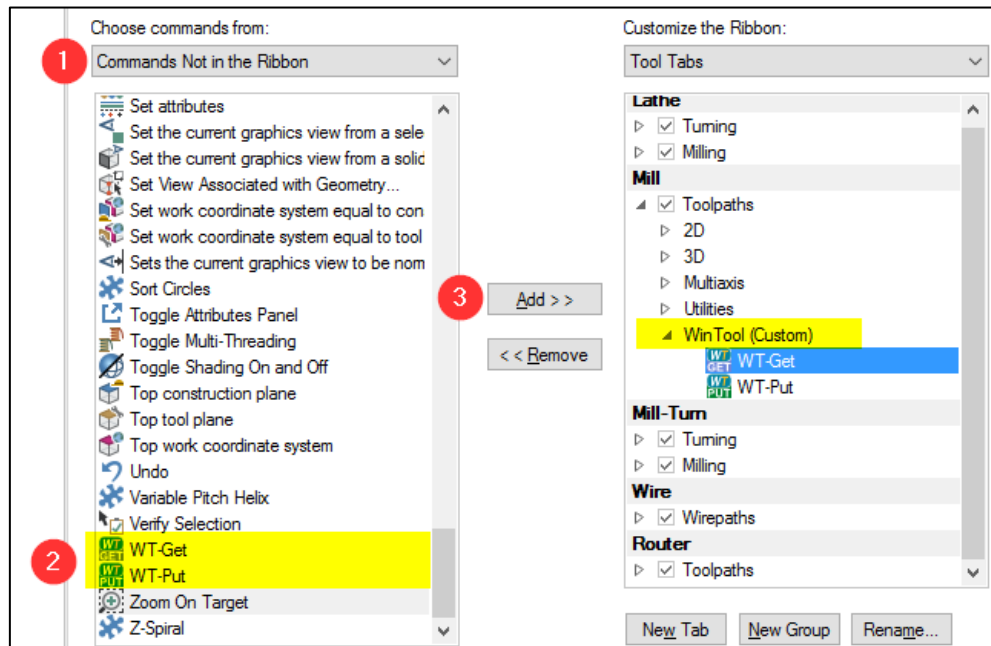


Figure 5 Choose commands and add to Tool Tabs

Configuration

Mastercam Settings

The flag "Assign tool numbers sequentially" in the Mastercam Machine Group Properties must NOT be set.

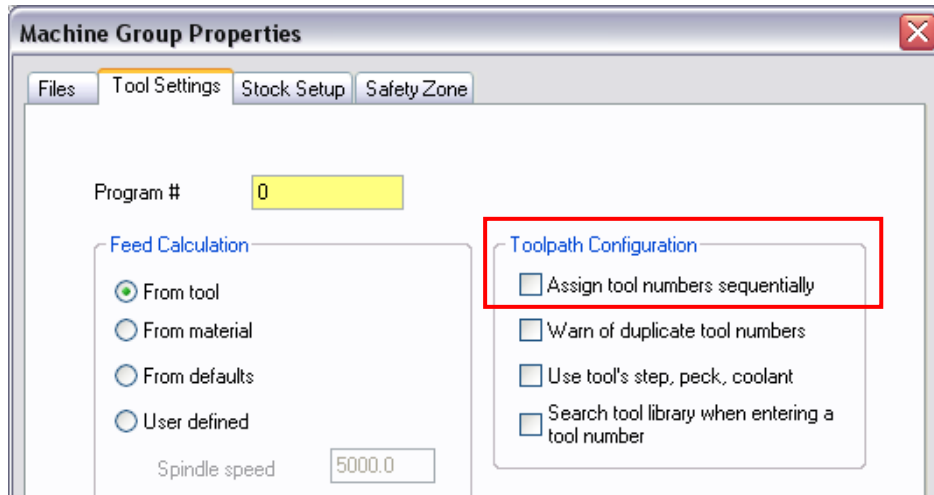


Figure 6 Mastercam Tool Settings

WinTool Settings

After Installation of the WT-Mastercam-Interface start up *WinTool* and set the flag for the Mastercam interface in Tools>Settings>Cam settings:

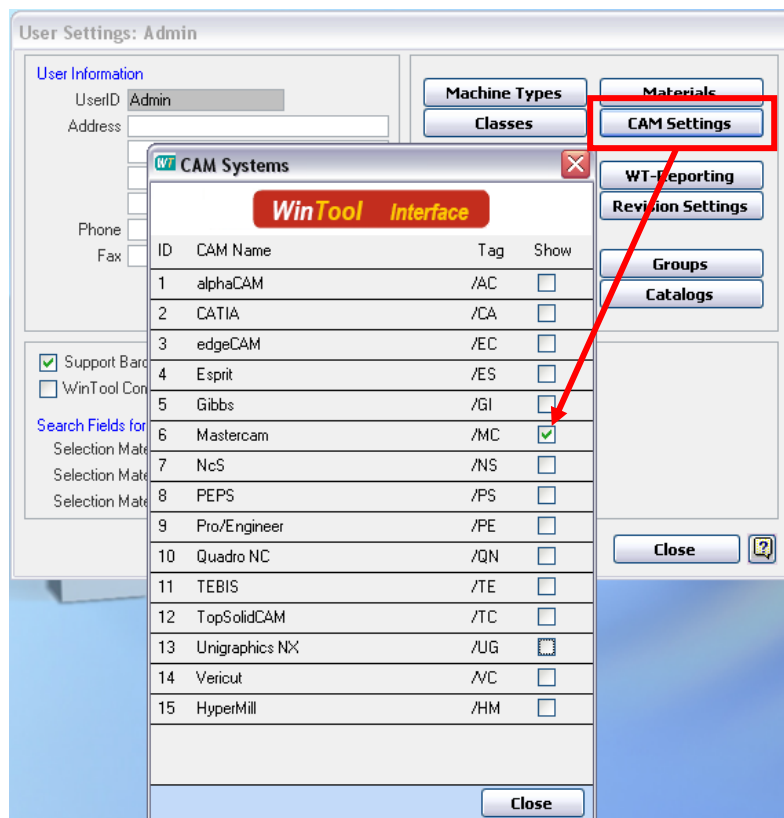


Figure 7 Choose Mastercam in User Settings

This enables in *WinTool* tool assemblies the *custom tools manager* in folder tab CAM:

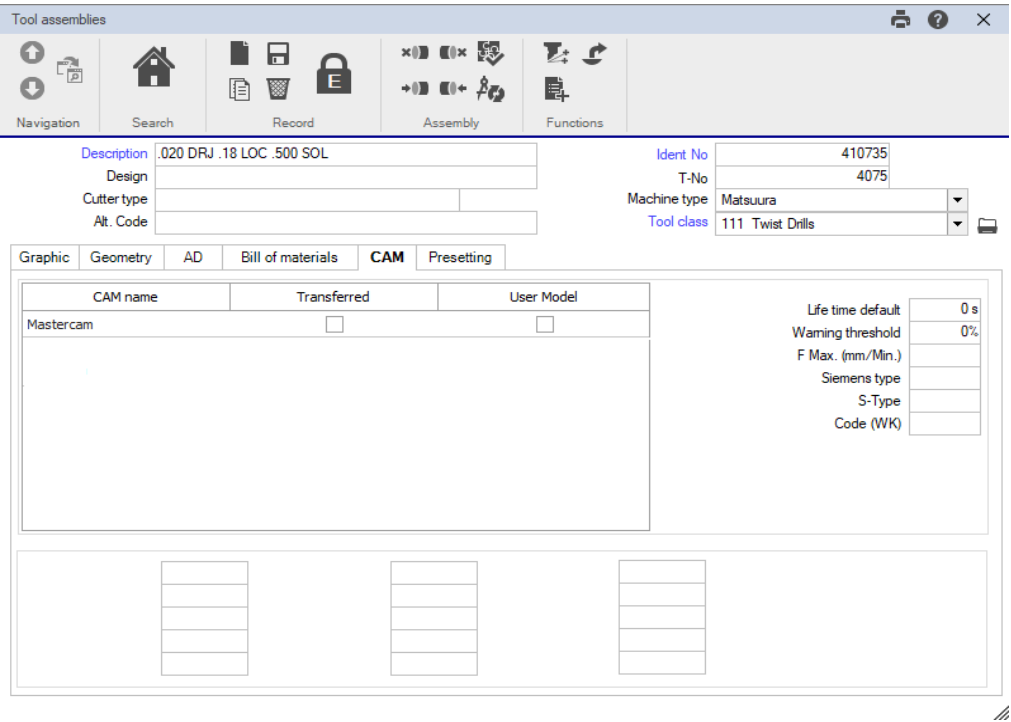


Figure 8 Tool Assemblies in CAM

Note: The settings of the activated CAM interfaces will be stored in the *WinTool* database (WTData). If you switch your *WinTool* Professional installation to another database you must activate the Mastercam interface in the new database as well (see chapter "Sample Database" on page 16).

Interface Settings

The following information is relevant for your understanding of the WT-Mastercam Interface data transfer and the configuration.

If you have installed the interface with default path settings no configuration changes are required to operate the interface locally.

Configuration Window

The configuration window allows you to check and change the settings of the WT-Mastercam-Interface. Open the configuration window in **START > All Programs > WinTool > WT-Mastercam-Interface > WT-Mastercam-Configuration**:

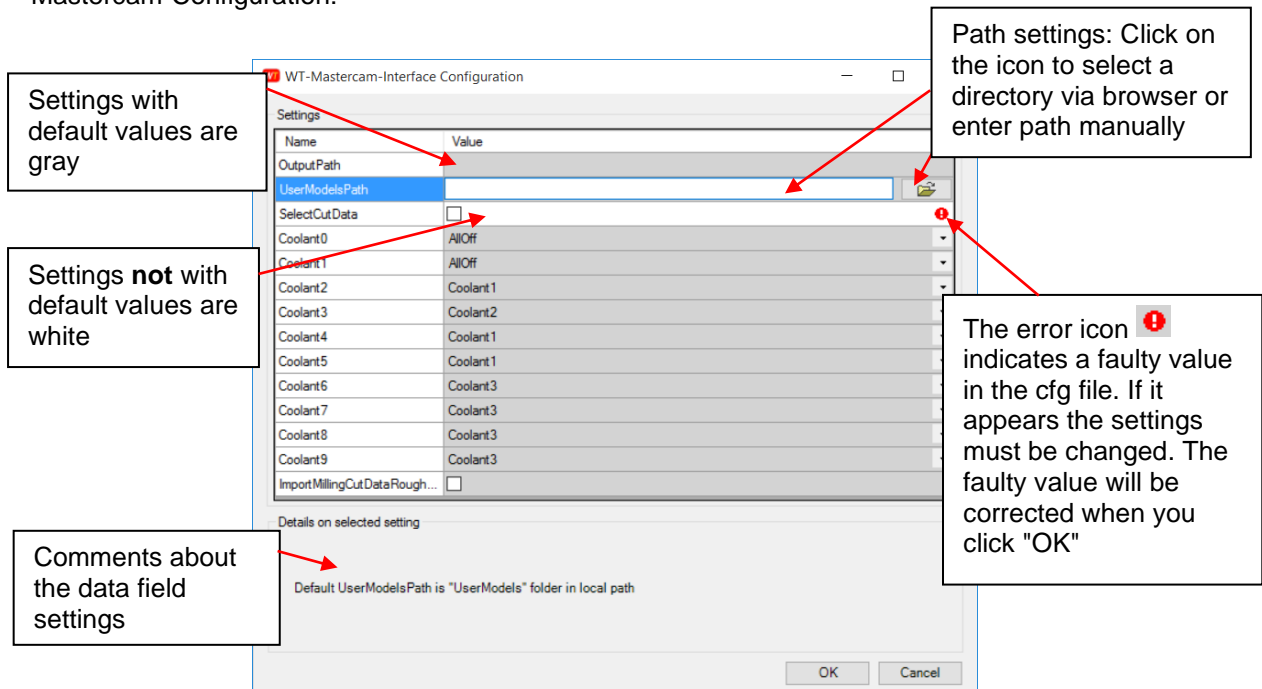


Figure 9 WT-MasterCAM-Interface configuration

<OK> stores all settings. <Cancel> exits the configuration window without saving.

The configuration window reads and stores settings in the file "WT-Mastercam-Interface.cfg" in the [\[Public Documents\]\WT-Mastercam-Interface-2025](#) directory.

The chapter "Configuration File Parameters" in the Annex describes details about the file "WT-Mastercam-Interface.cfg".

Output Path

All transferred tool assemblies are stored in a Mastercam tool database file which is stored in the file "WT-Mastercam.tooldb". It is stored in a dedicated Output Path that has been registered in cfg-file for each user. (For details on how to change the default setting see chapter "Configuration Window" on page 11).

UserModels Path

The WT-Mastercam-Interface processes user models (tool contour DXF files to produce 3D tool representations in Mastercam). The exchange path for DXF files is stored in a cfg-file for each user. (For details on how to change the default setting see chapter "Configuration Window" on page 11).

Note: User models must be stored in a network directory on a server so that all Mastercam users can access them. The directory with the user models must be included in the backups!

Selection of cutting conditions

Note: Starting with Mastercam X7, **all** cutting conditions for work materials of the imported tool assemblies can be accessed within Mastercam (see chapter "Using Cutting Conditions" on page 23), but it still allows importing one cutting condition which is stored in the tool assembly itself like in the previous Mastercam versions.

The interface imports cutting conditions for work materials for a tool assembly if this function is activated (`SelectCutData=True`). By default, it is activated.

For tool assemblies and tool lists, the import uses a different cutting condition selection procedure:

Import	Selection procedure
tool assembly data	The cutting condition window opens and all available cutting conditions can be selected.
tool list data	<p>For a tool list the interface imports all cutting conditions available for <i>one material only</i>.</p> <p>If the work material has been assigned to a tool list in <i>WinTool</i> (see folder tab "General Data") then the interface imports the tools with the cutting condition for this material automatically.</p> <p>If the work material has <i>not</i> been assigned to a tool list in <i>WinTool</i>, then the cutting conditions selection window appears for the first tool of the list and a value must be selected manually. The interface memorizes the work material of the previous tool and will suggest the same material for the next one.</p> <p>If a tool has more than one cutting condition for the same material or if no cutting condition exists for the material, the interface requests to select one manually.</p>

Note: The cutting condition window appears only if there is at least one cutting condition.

If this function is deactivated (`SelectCutData=False`) the cutting condition at the top of the list of the cutting condition table for the tool assembly is selected and imported automatically.

For details on how to change the default setting see chapter "Configuration Window" on page 11.

Tool List Exchange Path

A list of tools used in a Mastercam toolpath group can automatically be transferred back to *WinTool* tool lists with the interface function PUT which activates the software module "WT-MakeList". This software is installed in the interface installation directory. (See Annex of this manual for details on how to change the WT-MakeList default settings).

The path for the exchange file is the same as the "Output Path" (see above).

Note: Use a different WTMakeListPath for each user (e.g., use the local exchange directory path which is configured in the default settings).

Interface Settings for Mastercam 2017 and newer

Coolant0-Coolant9

The interface will assign the 10 *WinTool* coolants to the 10 Mastercam coolant types, based on the settings. If a value between "Coolant1" and "Coolant10" is assigned to a WinTool coolant setting, the corresponding coolant in Mastercam is set to "On", the rest to "Ignore".

The setting values "AllOff" and "AllIgnore" set all coolant values to "Off" and "Ignore" respectively.

Example:

If the setting "Coolant6" is set to 'Coolant2', and a cutting condition with the coolant type set to '6 on internal' sets the coolant nr 6 in Mastercam to "On" and all other coolants to "Ignore".

WinTool Coolant Types			
No	Description	No.	Description
0	-	5	5 Flood 2
1	1 Air	6	6 On internal
2	2 On	7	7 Mist internal
3	3 Mist	8	8 Flood 1 internal
4	4 Flood 1	9	9 Flood 2 internal

Importing Rough Step Percentage Values

When this setting is enabled, the values "Rough XY Step%" and "Rough Z Step%" in milling tools are imported using the formulas

- Rough XY Step % = 100 * (Cutdata.Ae / Cutdata.DM)
- Rough Z Step % = 100 * (Cutdata.Ap / Cutdata.DM)

Interface Settings for Mastercam 2018 and newer

Tool Name/ Assembly Name

As in Mastercam tool names are used to identify tools, the interface allows the user to configure how the names are generated while importing Tool assemblies from WinTool, modifying the settings "ToolName" and "AssemblyName":

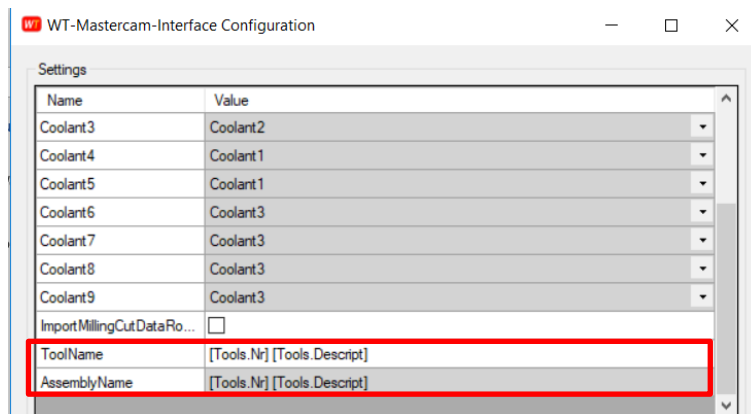


Figure 10 WT-MasterCAM-Interface configuration for naming conventions

Placeholders (put in square brackets) can be used to modify the Tool name/Assembly name. Most of the Tool Assembly fields in WinTool are supported. A short list of supported placeholders:

Tools.Nr, Tools.TNumber, Tools.Comment, Tools.Name, Tools.MachineNr, Tools.Descript, Tools.Design, Tools.MaskNr, Tools.ToolWidth, Tools.ToolLength, Tools.OldName, Tools.MDate, Tools.StockState

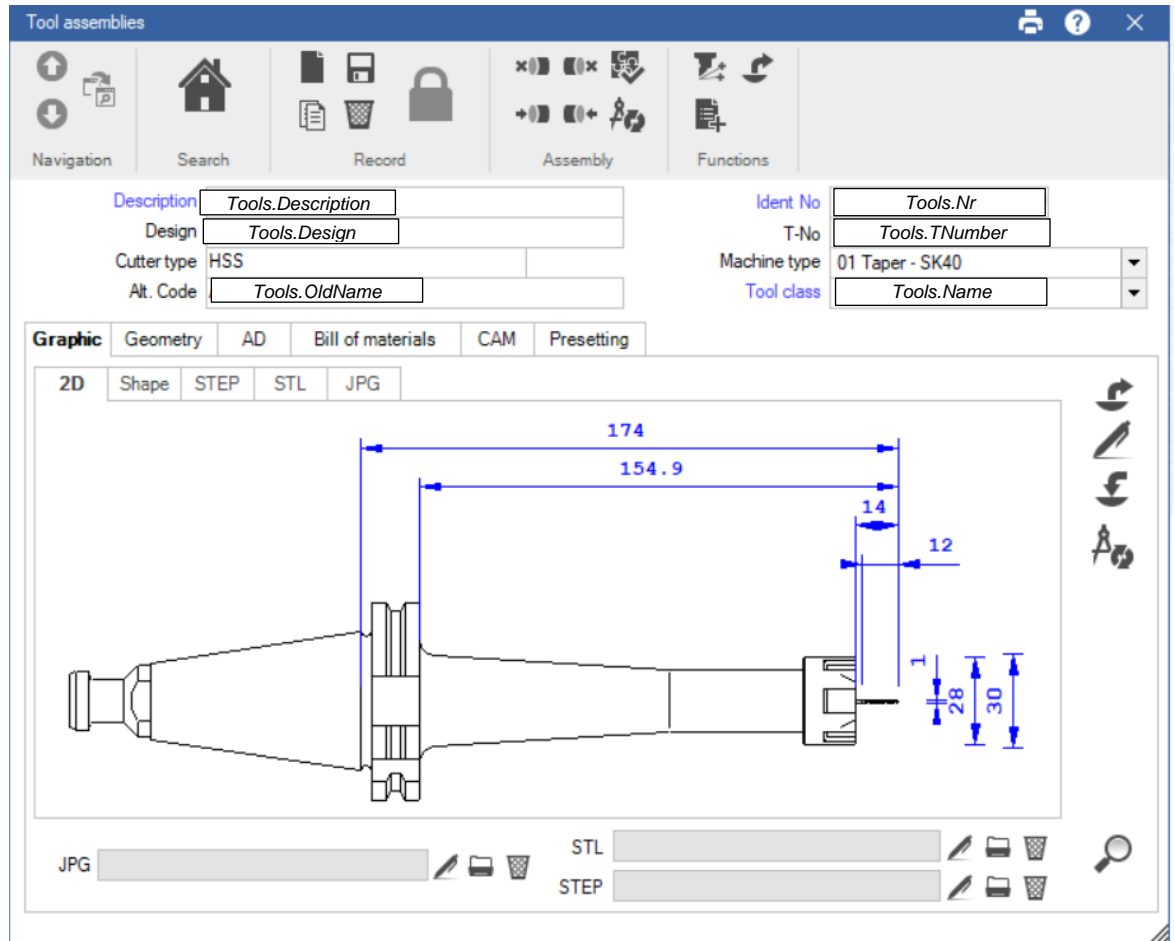


Figure 11 Main Placeholders

Special placeholders with dependent meanings:

- *TNumber* – (without Tools prefix) will become T from Lists if a list is imported, otherwise T from Tools if a tool is imported.

Important Notes:

- Placeholders must be put in square brackets.
- ToolName and AssemblyName text is limited by Mastercam to 100 characters each.
- For Identification, ToolName must contain the tool number ([Tools.Nr]) in either the following ways:
 - At the beginning
 - At the end
 - Somewhere else, marked with `#![Tools.Nr]!`
 - In case none of the above is used one will be appended automatically

Example:

A setting like

`[Tools.Nr] - [TNumber] - [Tools.Descript]`

Will be translated to

616021 - 0 - End Mill HSS 4x19 4FL

if imported via Tool Assembly, or to

616021 - 123 - End Mill HSS 4x19 4FL

when imported via Tool List.

EnableLathes

If setting it to "True" the import of Turning Tools will be possible. If setting it to "False" (default) the Import of Turning Tools will not work.

ImportMillingCutDataRoughStep

If enabled, the following calculations are used for milling tools:

Rough XY step (%) = $100 * (\text{CuttingCondition.Ae} / \text{CuttingCondition.Diameter})$

Rough Z step (%) = $100 * (\text{CuttingCondition.Ap} / \text{CuttingCondition.Diameter})$

ImportPitchValueFromComponent

If enabled (default) the pitch value will be taken from the Component (DMMin).

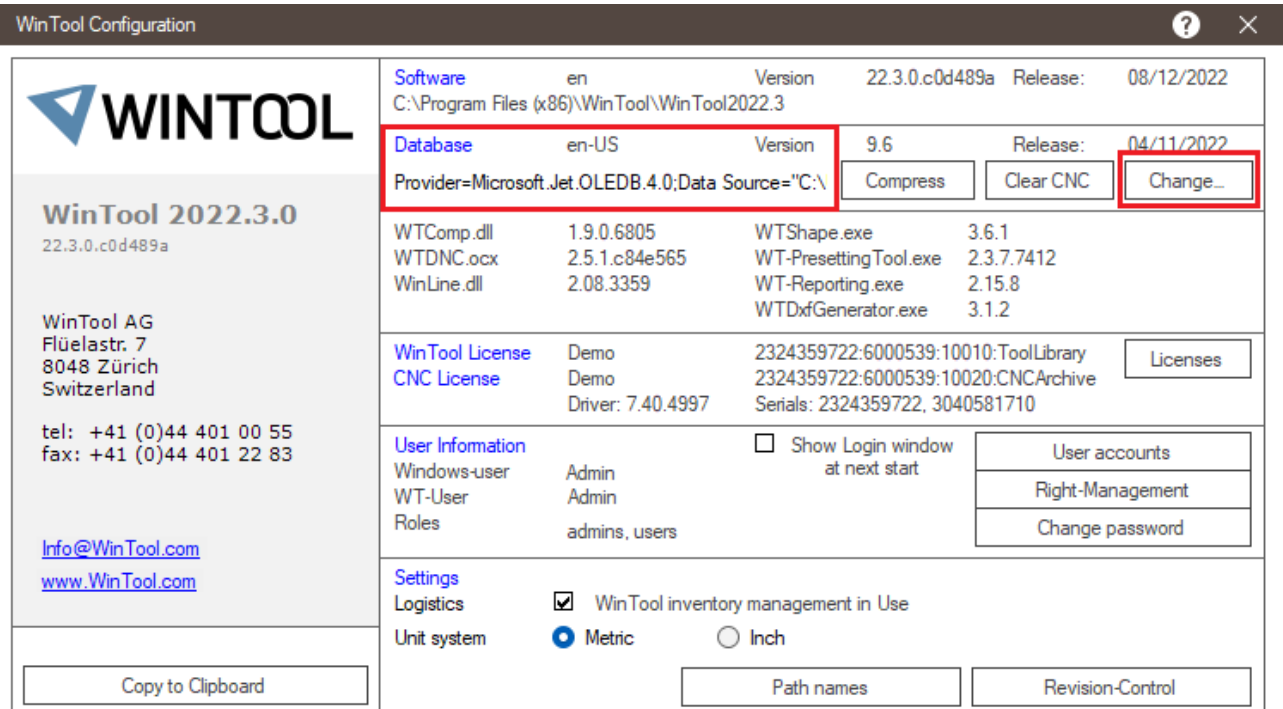
If disabled, the pitch value will be taken from the Tool Assembly (p).

Getting Started

Sample Database

The WT-Mastercam-Interface interfaces with the *WinTool* database that is currently linked to your *WinTool* Professional installation.

To test the interface installation and get yourself familiar with the functionality of the WT-Mastercam-Interface, please relink your *WinTool* Professional with the database supplied with the *WinTool* installer.




WinTool Configuration	
 <p>WinTool 2022.3.0 22.3.0.c0d489a</p> <p>WinTool AG Flüelastr. 7 8048 Zürich Switzerland</p> <p>tel: +41 (0)44 401 00 55 fax: +41 (0)44 401 22 83</p> <p>Info@WinTool.com www.WinTool.com</p> <p>Copy to Clipboard</p>	<p>Software en Version 22.3.0.c0d489a Release: 08/12/2022 C:\Program Files (x86)\WinTool\WinTool2022.3</p>
	<p>Database en-US Version 9.6 Release: 04/11/2022</p> <p>Provider=Microsoft.Jet.OLEDB.4.0;Data Source="C:\\" Compress Clear CNC Change...</p>
	<p>WTComp.dll 1.9.0.6805 WTShape.exe 3.6.1 WTDNC.ocx 2.5.1.c84e565 WT-PresettingTool.exe 2.3.7.7412 WinLine.dll 2.08.3359 WT-Reporting.exe 2.15.8 WTDxfGenerator.exe 3.1.2</p>
	<p>WinTool License Demo 2324359722:6000539:10010:ToolLibrary Licenses CNC License Demo 2324359722:6000539:10020:CNCArchive Driver: 7.40.4997 Serials: 2324359722, 3040581710</p>
	<p>User Information <input type="checkbox"/> Show Login window at next start</p> <p>Windows-user Admin WT-User Admin Roles admins, users</p> <p>User accounts Right-Management Change password</p>
<p>Settings</p> <p>Logistics <input checked="" type="checkbox"/> WinTool inventory management in Use</p> <p>Unit system <input checked="" type="radio"/> Metric <input type="radio"/> Inch</p> <p>Path names Revision-Control</p>	

Figure 12 Change the WT-database

For instructions on how to link to a different database refer to the documentation about the *WinTool* DB-Manager.

The following chapters refer to the sample data in this database.

Import Tools

Start *WinTool* Professional first.

Then start Mastercam and open the sample part "Side Frame" located in the "Samples" folder in the "[Public Documents]\WT-Mastercam-Interface-2025" directory and select a Machine Group.

Select the tab "TOOLPATHS". Click on the button WT-GET to open the WT-ToolExport window and select *Tool Assembly*:

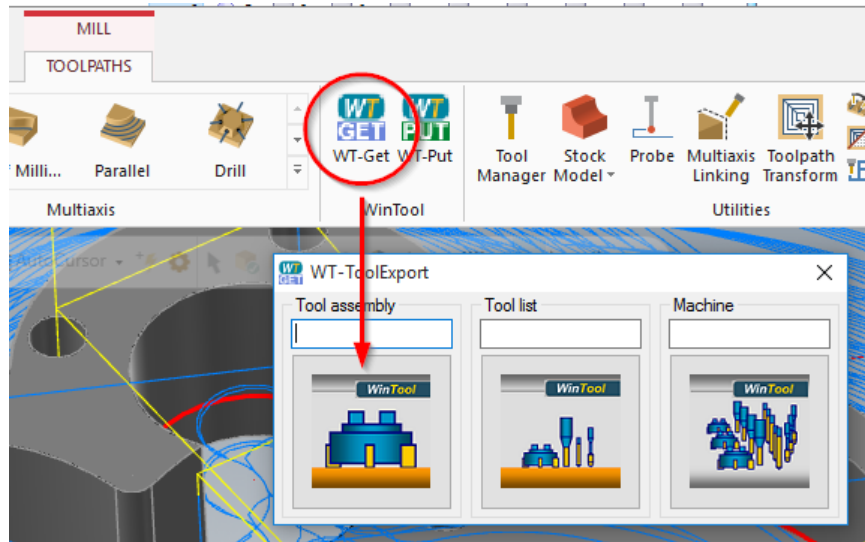



Figure 13 Select Tool assembly in the menubar „TOOLPATHS“

Click on  to open the tool classification screen. Select the classification "221 face mill" and highlight tool 616092.

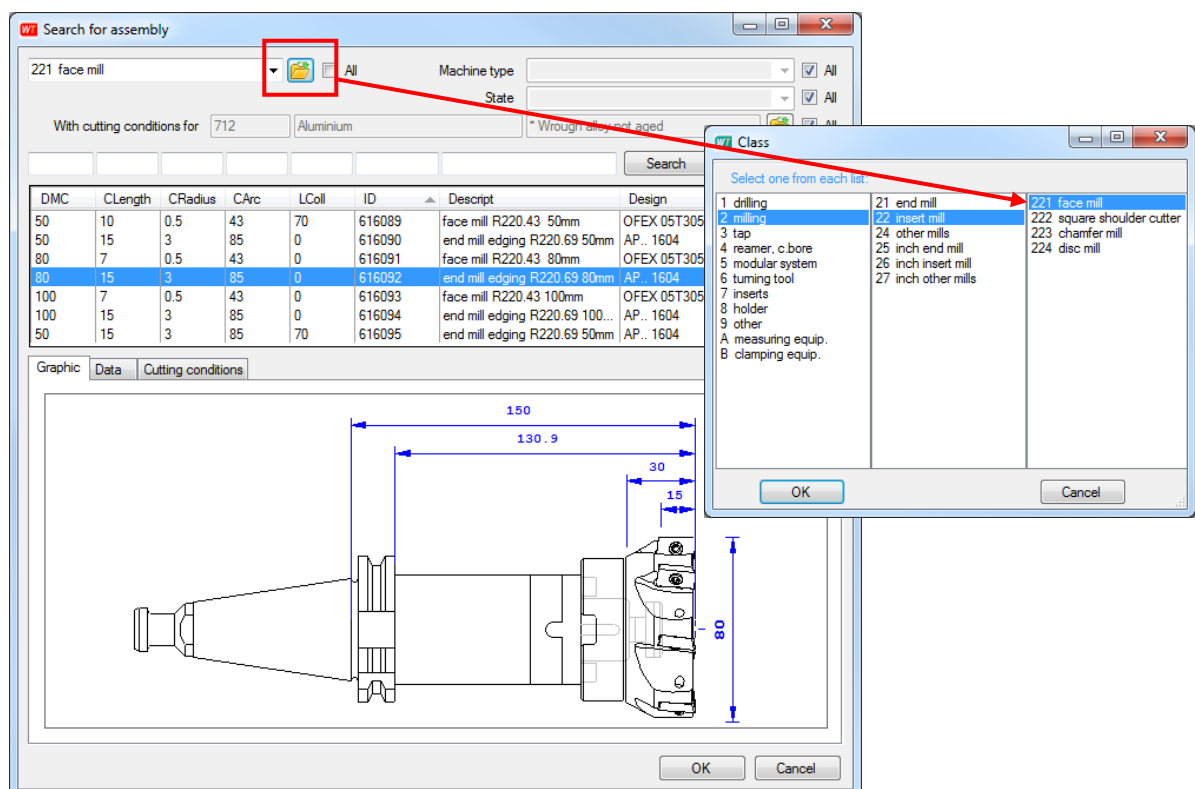


Figure 14 Search a tool assembly class from list

If the cutting conditions import is turned on (`SelectCutData` is enabled) select the value, you want to transfer with the tool assembly and click "OK".

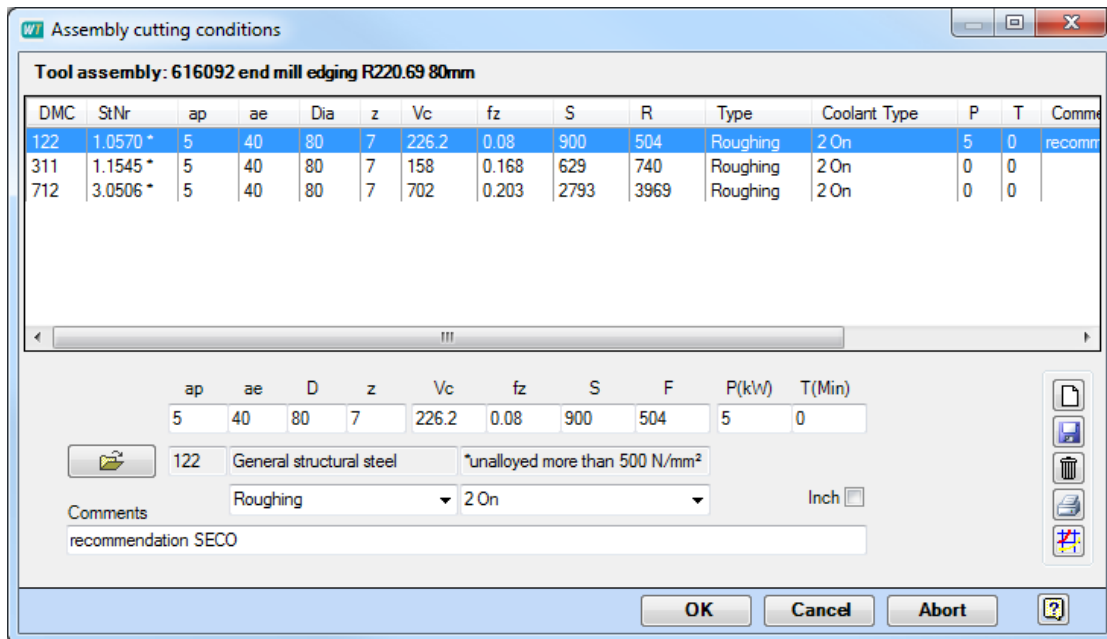


Figure 15 Assembly cutting conditions

If you click on "Cancel" it will not transfer any cutting conditions with the tool. "Abort" will stop the entire tool data transaction to Mastercam.

If no Mastercam tool type has been assigned to a WinTool classification, yet you must do it now. This will map the WinTool classification to the Mastercam tool type. Select the correct Mastercam tool type from the selection list.

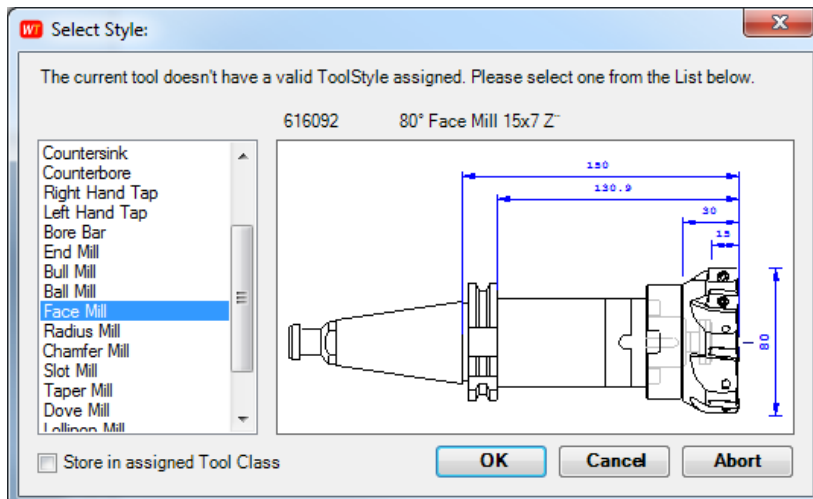


Figure 16 Select a Mastercam tool type from list

If you select "Ignore" to assign to a tool classification, the tool assemblies in this classification will not be transferred at all. This is useful for data that must not be transferred to Mastercam, e.g., measuring equipment.

In most cases it makes sense to assign the mapping permanently to a tool classification. Then you must also check the box "Store in assigned Tool Class" (recommended).

Open the Mastercam Tool Manager to review the tool assembly:

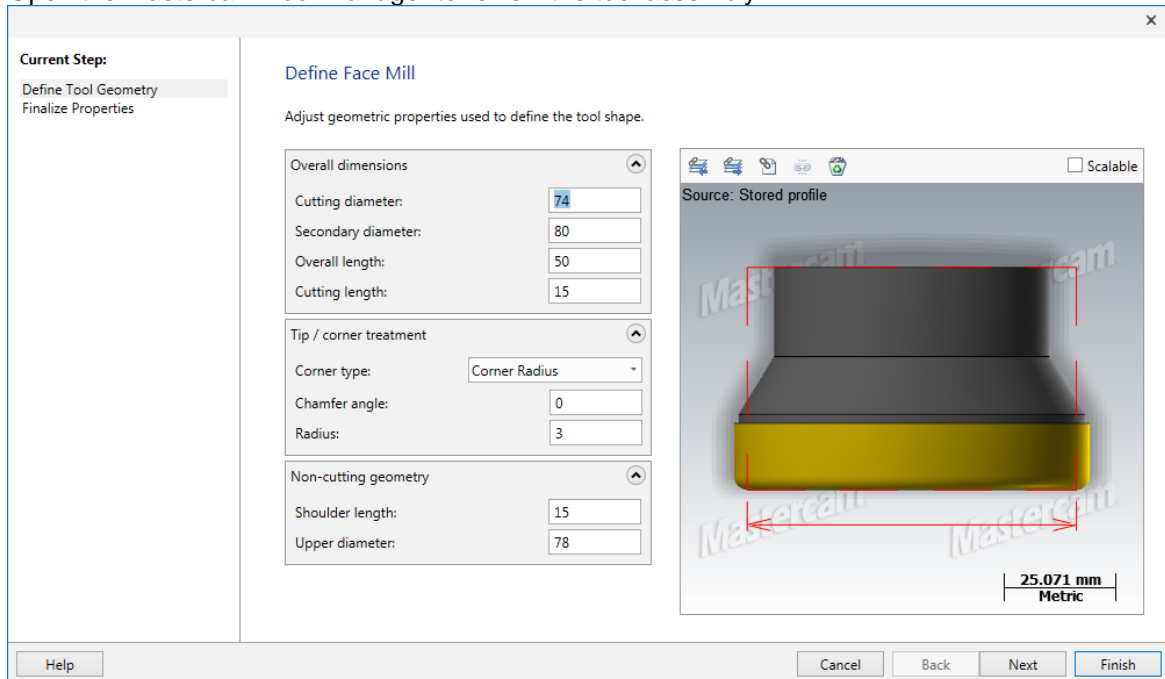


Figure 17 3D Toolpaths model

Create an operation using the transferred tool assembly and click "Holder" to see the WinTool tool representation:

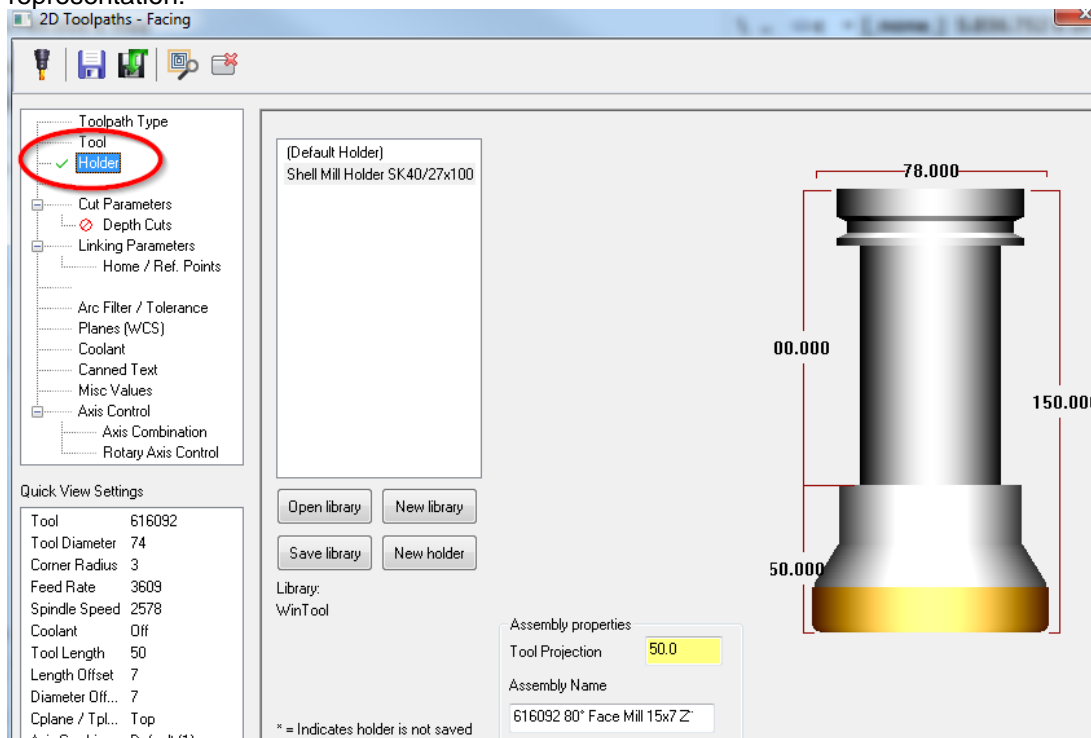


Figure 18 2D Toolpaths model

Select "GET" again to transfer the Tool List "100 1050-20 C Tools" to Mastercam:

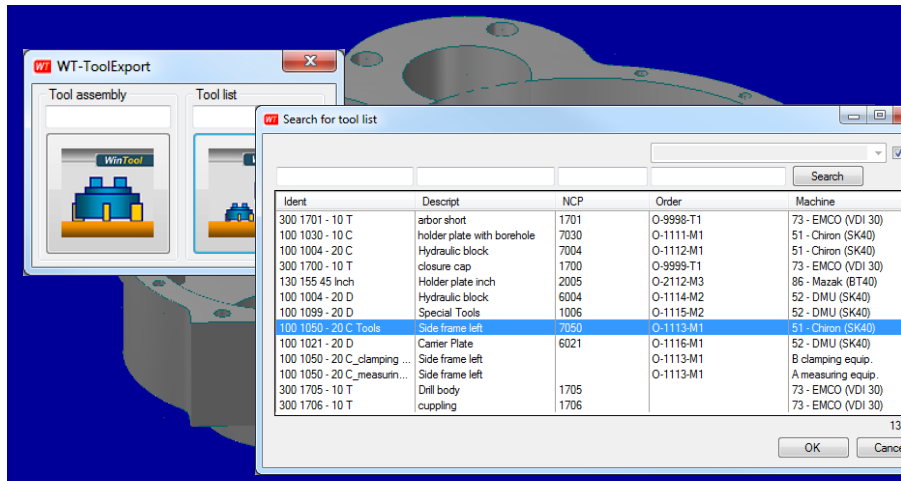


Figure 19 Search to transfer the Tool List

Review all imported tools in Toolpaths > Tool Manager:

Tool Manager									
DMU70V									
= Tool used in an operation									
(Part)									
#	Assembly Name	Tool Name	Holder Name	Dia.	Cor. rad.	Length	# Flutes	Type	
2	616093 100° Face Mill 3x43 Z7	616093	Shell Mill Holder SK40/32x60	99.0	0.5	7.0	7	Face mill	
3	616031 End Mill HSS 20x75 Z4	616031	Weldon Holder SK40/20x63	20.0	0.0	75.0	4	Endmill1 Flat	
4	616004 Twist Drill HSS 6.8x69	616004	Collet Chuck SK20/ER71x"	6.8	0.0	69.0	2	Drill	
5	616001 Thru Hole Tap HSS M08x20	616001	Tap Holder SK40/wE-2	8.235-1.25	0.0	20.0	2	Tap RH	
6	616077 Twist Drill HSS 8x75	616077	Collet Chuck SK20/ER71x"	8.0	0.0	75.0	2	Drill	
8	616134 Fine-Boring Tool 10 (Dia 15-30.')	616134	MBM Adapter SK40/MBM55 x60	14.8	0.2	6.0	1	Bore	
9	616017 End Mill HSS 32x53 Z6	616017	Weldon Holder SK40/32x100	32.0	0.0	53.0	6	Endmill1 Flat	

Figure 20 List of all imported tools

Note: In the WinTool sample database only the tools in list "100 1050-20 C Tools" have cutting conditions assigned.

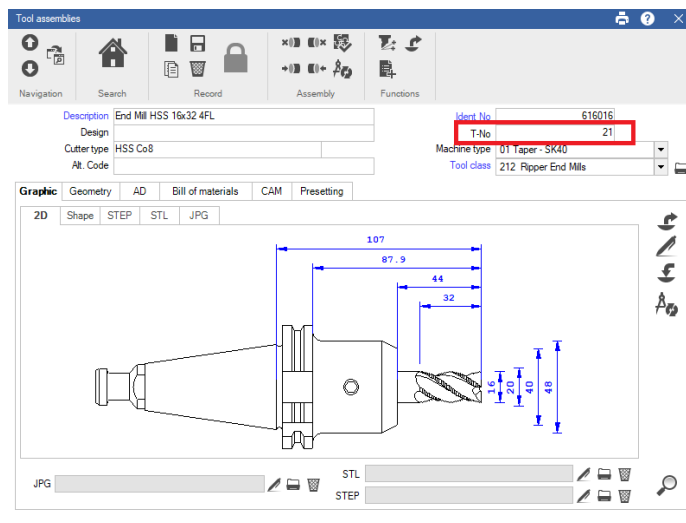
Tool Numbers

T-Number Assignment

If you import *WinTool* tool assemblies to Mastercam, the interface software will automatically assign a Mastercam "Tool#". The number is assigned sequentially starting at 0.

WinTool Tool Assembly T-No

In case you work with specific tool numbers on a machine, you can assign the number to the tool assembly in *WinTool* Professional. In the following example the tool assembly with ID 616016 has T-No = 21 assigned:



If you import this tool to Mastercam the interface will assign Tool# 21 in Mastercam:

#	Tool Name	Dia.	Cor. rad.
1	616092 face mill edging R220.69 80mm	80.0	3.0
2	616093 face mill R220.43 100mm	10...	0.5
3	616031 end mill HSS 20mm long	20.0	0.0
4	616004 twist drill HSS 6.8mm	6.8	0.0
5	616001 tap M08	8...	0.0
6	616077 twist drill HSS 8mm	8.0	0.0
8	616134 boring bar 10 mm	12.0	0.2
9	616017 end mill HSS 32mm	32.0	0.0
21	616016 end mill HSS 16mm	16.0	0.0

Figure 21 List of T-No for Tool Assemblies

If T-No is 0 and the setting "T-No=Ident No" is activated in the assigned machine type, the ident no is transferred.

Note: This is only recommended if the same tool keeps always the same T-Number on all machines using this tool (e.g., Probe has T#999, Spot Drill has T#1, etc.)

WinTool Tool List T-No

In *WinTool* Professional you can also assign T-Numbers in tool lists. If you import a list to Mastercam, the interface software assigns the Mastercam Tool# used in the *WinTool* list.

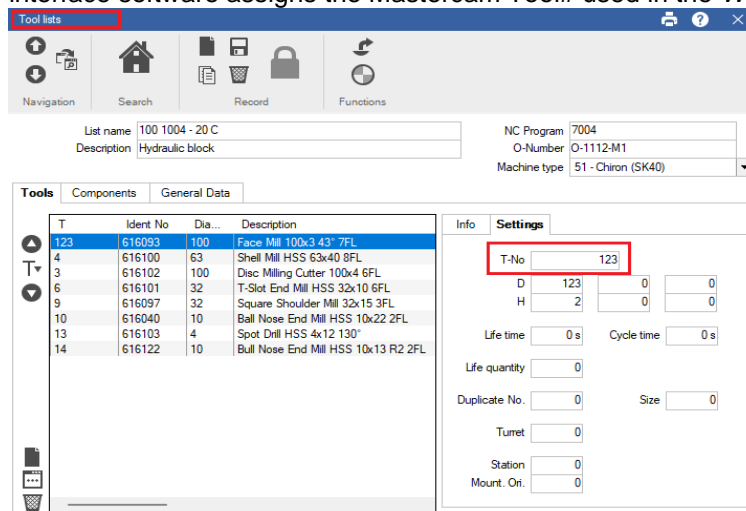
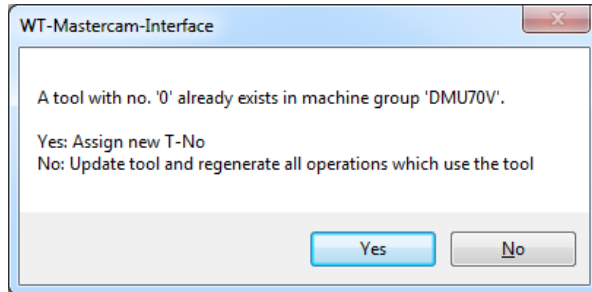


Figure 22 Check the T-No.

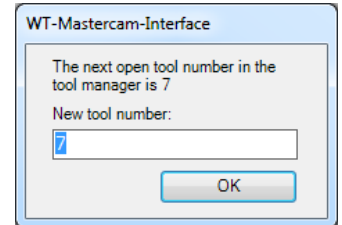
Note: This is recommended if you work with resident tools on machines. Create a "Resident Tool List" for each machine and dedicate T-numbers to each resident tool in this list. If you import this list in Mastercam the tools will be loaded with the dedicated T-numbers.

Duplicate T-Numbers

If a tool number is already used in the Mastercam tool manager, you cannot import the tool with the same Mastercam Tool#. Then the following dialog message appears:



Select **YES** to import the tool with a different Mastercam Tool#. By default, the next open tool number is entered:



D and H numbers are set to the new tool number.

Figure 23 Notification - Tool number

Select **NO** if you want to overwrite the existing tool assembly with the same Tool# in Mastercam. The toolpath of the operation is recalculated automatically after the tool assembly is overwritten.

Coolant Import

The standard Mastercam coolant types "Flood", "Mist" and "Thru-Coolant" are imported in Mastercam X8 and later.

Starting with the interface for Mastercam 2017, you can set a custom mapping, see [Interface Settings for Mastercam 2017 and newer](#)

Coolant Nr	WT English	WT German	Mastercam
1	1 Air	1 Luft	All Off
2	2 On	2 Ein	Flood
3	3 Mist	3 Sprühnebel	Mist
4	4 Flood 1	4 Strahl 1	Flood
5	5 Flood 2	5 Strahl 2	Flood
6	6 On internal	6 Ein innen	Thru-Coolant
7	7 Mist internal	7 Sprühnebel innen	Thru-Coolant
8	8 Flood 1 internal	8 Strahl 1 innen	Thru-Coolant
9	9 Flood 2 internal	9 Strahl 2 innen	Thru-Coolant

Using Cutting Conditions

All cutting conditions are transferred together with the tool assemblies into the Mastercam tool library.

The cutting conditions can be selected in the operation parameters > "Tool" > R-click on right area > "Search for cut parameters".

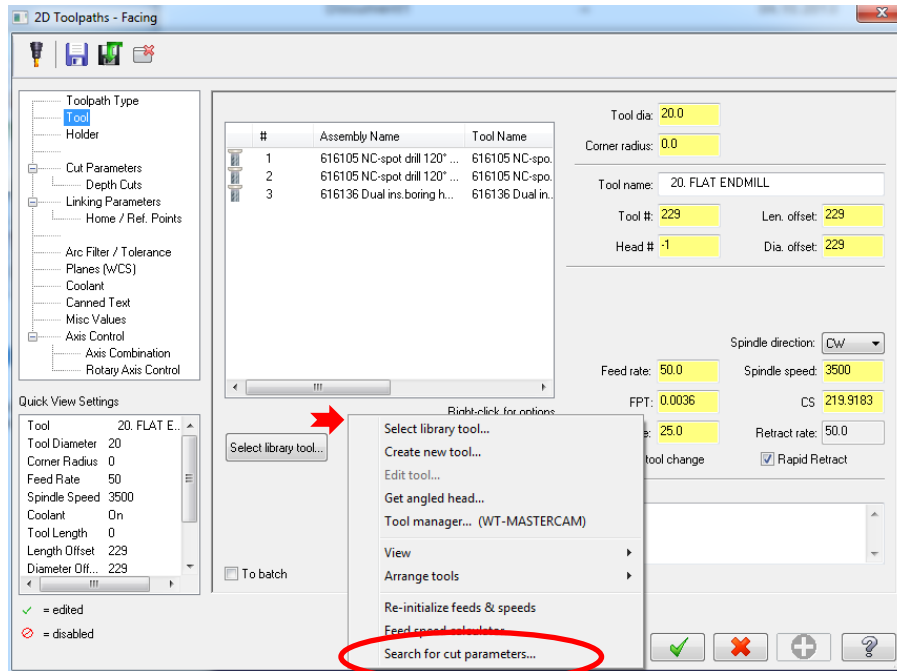


Figure 24 Instruction to search for cut parameters

In the window, remove the search filters and click "Search" to show all cutting conditions.

To show cutting conditions of a specific tool assembly, select "Name" as a search item, enter the tool assembly ident-no in "Value" and click "Search":

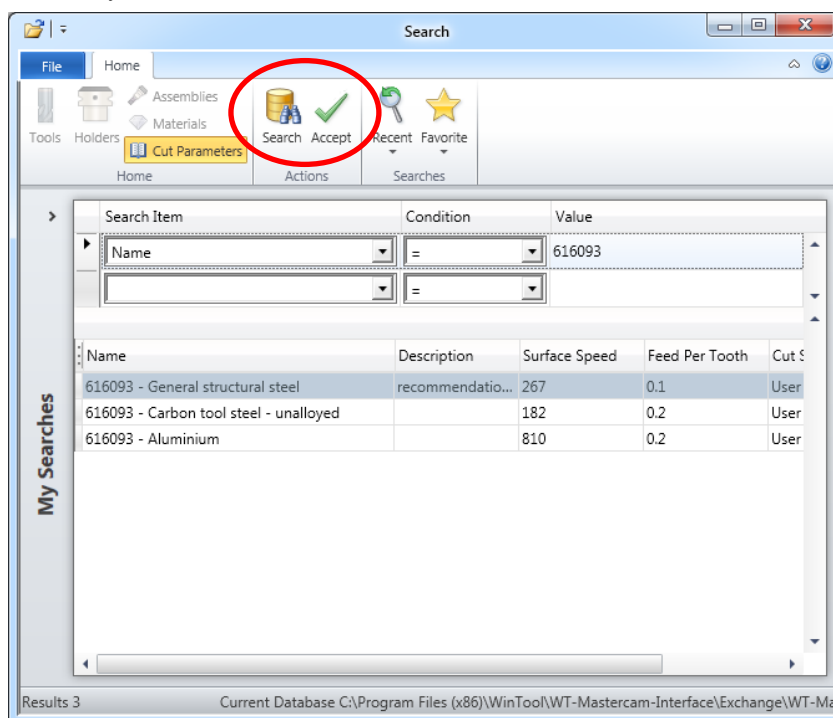


Figure 25 Enter tool assembly and click on Search icon

To assign a cutting condition to the operation, click "Accept".

Export Tool List to *WinTool*

When you have finished the NC program, the list of all the tools used in the Mastercam toolpath Group must be stored back to *WinTool*. This will allow the next person in the production process to continue with the job.

Step-by-Step

To create a *WinTool* tool list from within Mastercam proceed as follows:

Select in your Operations Manager all the tools that are used in the NC-Program and need to be transferred to the *WinTool* tool list:

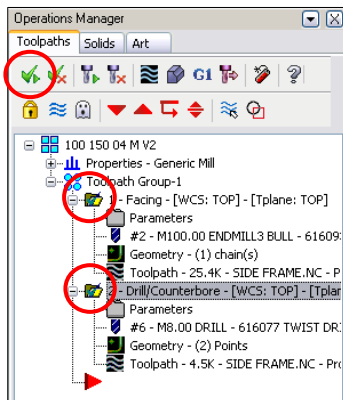


Figure 26 Select tools to be transferred

Select the button WT-PUT in the menu to store the tool list in the *WinTool* database:



Edit the tool list header information:

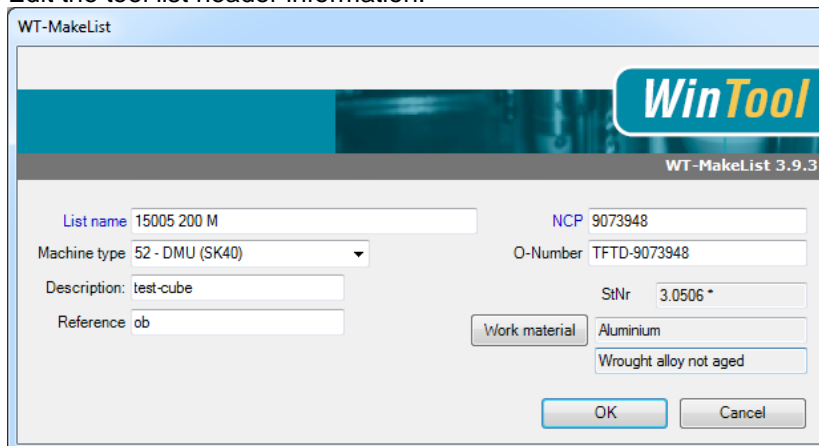


Figure 27 Edit List name in *WT-MakeList*

Select "OK" to store the information in the *WinTool* database.

If a tool list with the same List Name already exists in *WinTool* the following dialog box appears:

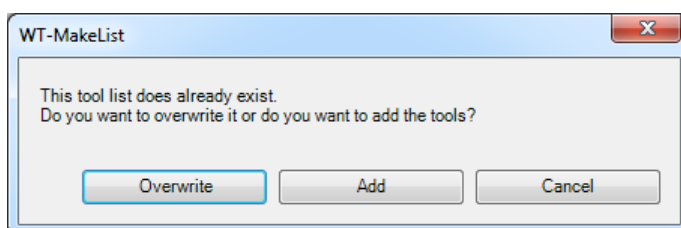
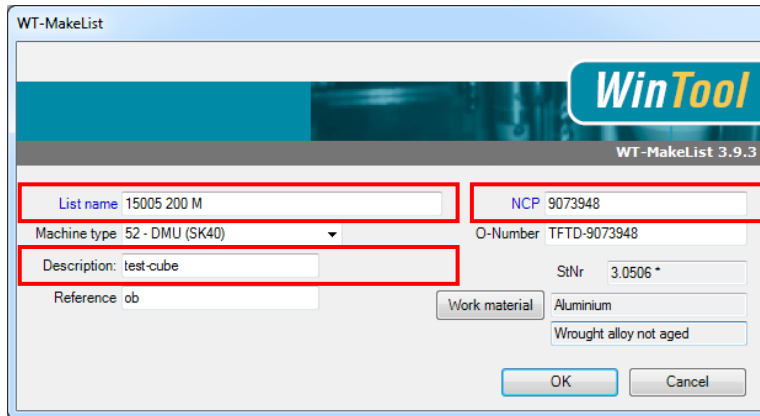


Figure 28 Notification

Note: In the new *WinTool* tool list, the T-Numbers and the sorting will be the same as in the Mastercam Toolpath Group.

Mastercam data fields transfer

Some of the WT-MakeList window data entry fields will be filled in automatically with values used in your Mastercam session:



The WT-MakeList 3.9.3 dialog box is shown with the following fields and values:

- List name: 15005 200 M
- NCP: 9073948
- Machine type: 52 - DMU (SK40)
- O-Number: TFTD-9073948
- Description: test-cube
- StNr: 3.0506 *
- Reference: ob
- Work material: Aluminium
- Wrought alloy not aged

Buttons: OK, Cancel

Figure 29 Entry fields will be filled in automatically

List Name

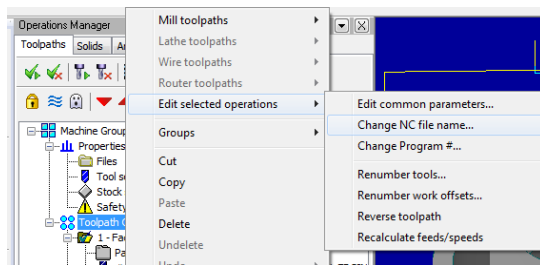


Figure 30 Change NC file name

The data field List Name pulls data from the NC file name. You can change the name as follows: R-click "Toolpath" > "Edit selected operations" > "Change NC file name".

Description

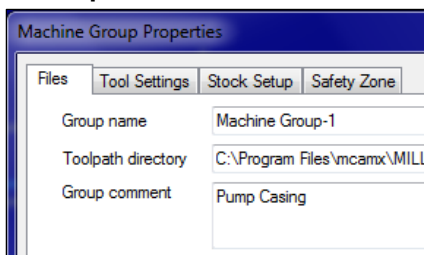


Figure 31 Tab folder Files - Machine data

The data field Description pulls the data from Machine Group Properties > "Files" > "Group Comment".

NCP

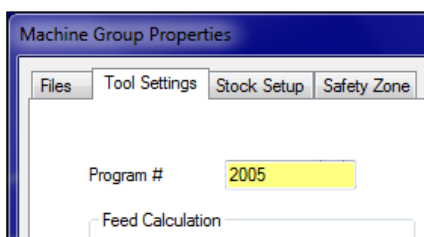


Figure 32 Enter a program number in tab folder 'Tool Settings'

The data field NCP pulls the data from: Mastercam > Machine Group Properties > "Tool Settings" > "Program #".

Preparing Tool Data in WinTool

The WT-Mastercam-Interface works only if the data has been entered correctly in *WinTool*.

Before you import *WinTool* data to Mastercam, read this chapter carefully. The following points must be considered:

- Each *WinTool* classification must be assigned to a Mastercam tool type.
- Each tool assembly must be linked to a *WinTool* Machine Type.
- Each tool assembly must have a "Namegiving", "Cutter", and a "Has Taper to Machine" component.
- The tool geometry of all components of an assembly must be recorded correctly according to the Tool Type-Outline.

User Classification

Each tool classification in *WinTool* must be mapped to the corresponding Mastercam tool type. If the mapping is missing the WT-Mastercam-Interface will ask to assign, then classification during import (see chapter [Import Tools](#)).

You can also map the *WinTool* classification with the Mastercam tool types manually. In *WinTool* select Settings > Class, then select a classification. In the data field "Note" you can assign the corresponding Mastercam tool type.

For the classification "212 - end mill roughing" assign the Mastercam tool type code /MC10 (see chapter [Supported Mastercam Tool Types](#) for a list of Mastercam tool type codes).

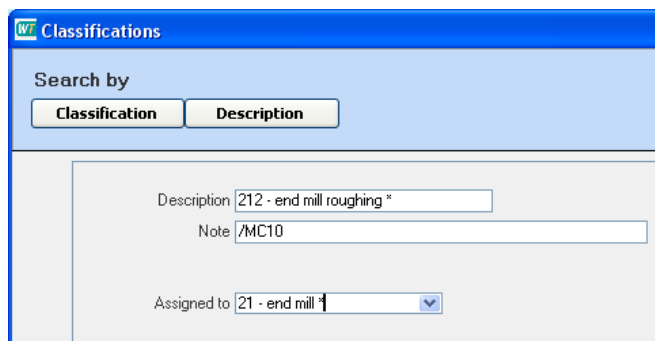


Figure 33 Every Descriptions has an own Note

Machine Configuration

In order to create tool assemblies in *WinTool* you must record the Machine Types in *WinTool*. This is required for a number of reasons:

- Tools can be filtered by machine adapter type during tool import in Mastercam
- *WinTool* can automatically create an accurate 3D milling tool model
- Tool lists can be filtered by machines

WinTool tool assemblies that are not assigned to a *WinTool* Machine Type cannot be imported to Mastercam.

Note: Review *WinTool* documentation for details on how to setup the machine types.

Tool ID and Name

Each tool assembly record in *WinTool* gets a unique numeric Ident No.

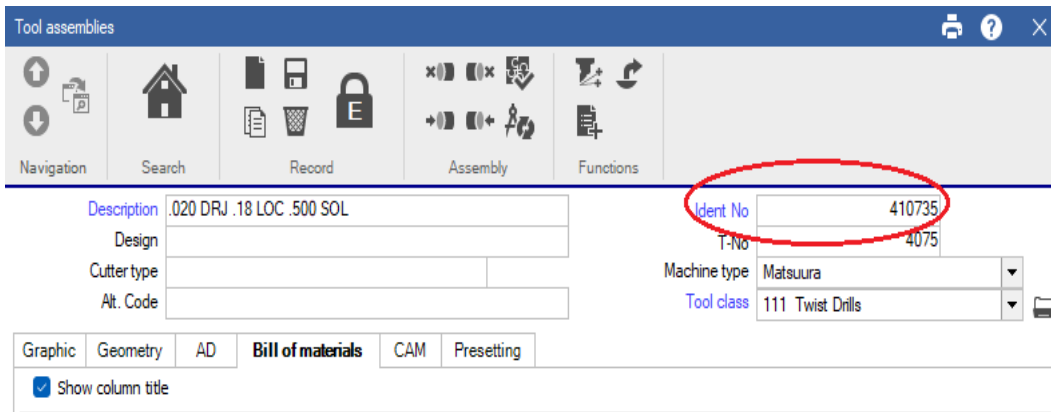


Figure 34 Enter Ident Number and Tool Description

Each *WinTool* tool in Mastercam will get a unique Tool# (see chapter Getting Started above) and a unique tool name. The name is a combination of the *WinTool* Ident No and the Description. Example: 616093 face mill R220.43 100mm

A *WinTool* tool assembly is generated from the data of its components. One of the components must be marked as the "Namegiving" and one as the "Cutting".

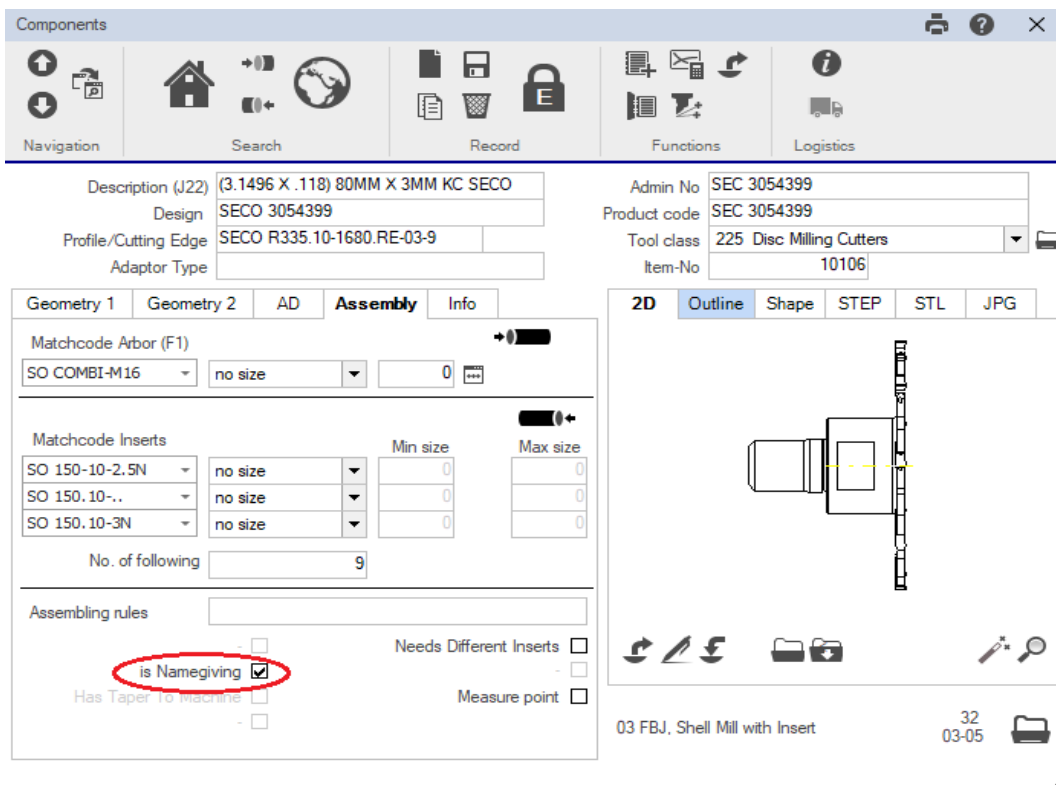


Figure 35 Namegiving must be marked

Note: If none of the components are marked as "Namegiving" / "Cutting", the WT-Mastercam-Interface will fail to import the tool.

Regular Tools

WinTool considers “regular tools” (as opposed to “special tools”) all tools that can be recorded with the Outlines provided in *WinTool* and that are supported by the *WinTool* Shape-Generator (which is marked with the light green symbol).

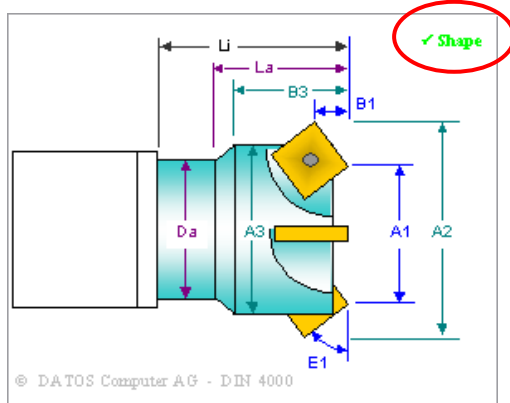


Figure 36 Select Shape for Generator

WinTool can generate for all regular tools 3D representations if they are axially symmetric.

The tool geometry of all components must be recorded fully and correctly according to the *WinTool* Outlines. You can verify the tool contour directly in *WinTool* starting the Shape-Generator.

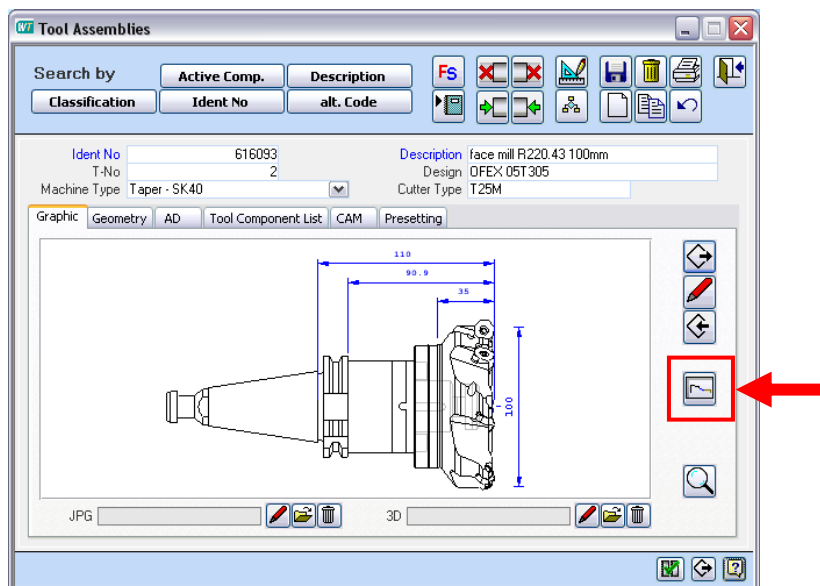


Figure 37 Verify Tool Contour by click on the icon

Special Tool Assemblies

If a contour of a tool assembly cannot be created automatically with the Shape-Generator it is considered a special tool assembly.

Managing Special Tool Assemblies

For Special tool assembly you can edit the holder contour in *Vector* (or any other CAD system) and store it in the [UserModelsPath](#) of the WT-Mastercam-Interface. Save the DXF contour as a DXF and assign the name of the tool assembly Ident No (e.g., [616099.dxf](#)). Then flag the *WinTool* tool assembly in the folder tab CAM:

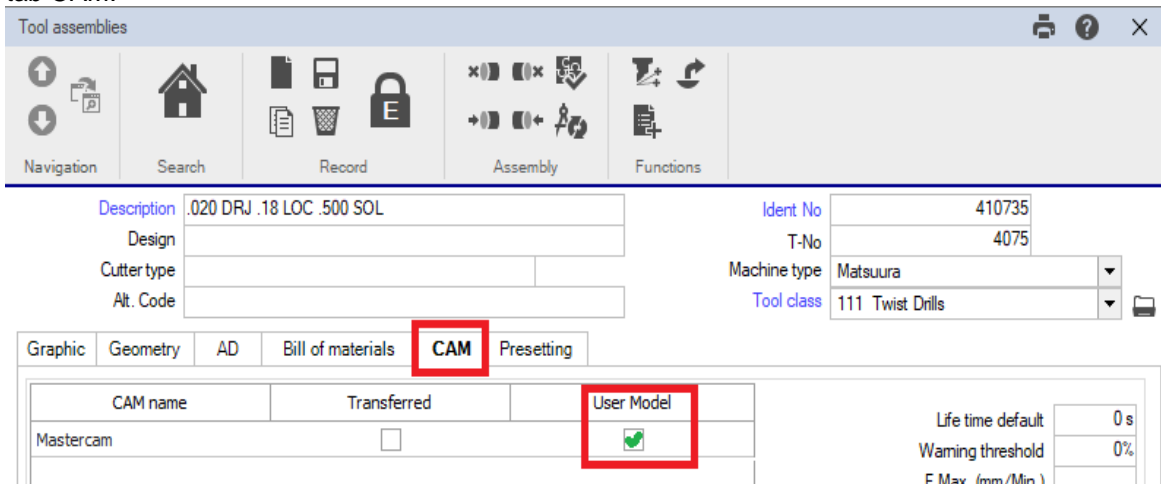


Figure 38 Select User Model in the menu "CAM"

If the User Model flag is active, the WT-Mastercam-Interface ignores the Shape-Generator and takes the customized DXF (e.g. [616099.dxf](#)) from the directory for User Models (see chapter [UserModels Path](#)).

Create a Special Tool Assemblies Contour DXF

- Use the *WinTool* Shape-Generator module to create a DXF contour. Even if a tool is not supported fully by the Shape-Generator, it will create in most cases a contour-DXF, although not with all additional details of the custom tool - but with a lot of useful elements in place already: holder, extensions, reductions, shank, total length, correct layers, etc.

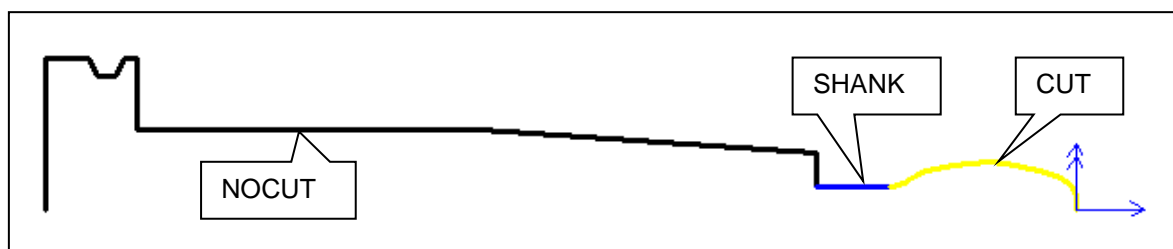


Figure 39 Create a Tool Contour DXF

- Then modify it with *Vector* or any other DXF editor until it is exact. You must use the layers CUT, NOCUT, and SHANK:
- The contour must be one continuous line. It **must** start and end at the X-axis (Y=0). Only the first and the last line of the contour are allowed to start/end at X-axis.
- When you have finished the modification, you must save the file in the User Models Path with the name of the *WinTool* tool assembly Ident No (e.g., [616089.dxf](#)). Already existing files must be overwritten.

Special Components

If you are using *WinTool* 2013 or newer, you can store special contours within the components. If not, you can create a special cutter contour, see paragraph "Special Cutters" below.

Managing Special Component Contours

The *WinTool* shape generator checks if a component contains a special contour and uses it to generate the tool assembly contour. This way, all CAM interfaces use the special cutter contour automatically.

Open the corresponding component in *WinTool* and activate the tab "Shape".

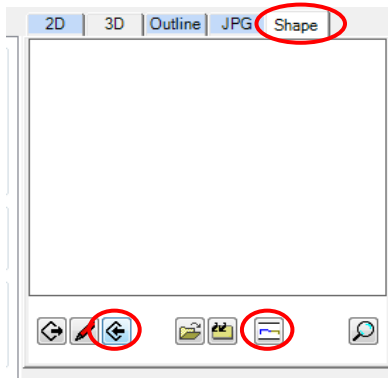



Figure 40 Instruction to activate the tab "Shape"

The component contour generator  creates a contour based on the geometry data of the component and opens it with the standard DXF editor. Customize the contour. You must use the layers CUT, NOCUT and SHANK. Save the DXF file.

Use the import button  and select the DXF file.



Open a tool assembly which uses this component and start the Shape-Generator to verify the tool assembly contour and the special component contour inside it.

Note: If a special cutter contour file exists for the component (see next paragraph) in `<UserModels>\Parts`, it must be removed; otherwise, it will override the contour stored in *WinTool*.

Special Cutters

The WT-Mastercam-Interface also supports special cutting tools that are axially symmetric. This is useful if no suitable Mastercam tool type for the cutter geometry is available.

Managing Special Cutters

Draw the special cutter contour manually and save it in `<UserModels>\Parts` with the name of the *WinTool* tool component Item No (e.g., `51271.dxf`).

If you import in Mastercam a *WinTool* tool assembly that is using a component with a special cutter, the WT-Mastercam-Interface will automatically find the special cutter DXF in the `<UserModels>\Parts` -path and attach it to the tool holder contour generated by the Shape-Generator.

Create a Custom Cutter Contour

Create a DXF-file with your custom cutter contour. The contour must be in the Layer CUT. The tip of the cutting contour must end at the origin (zero point) and must have the same cutting length as entered in the components field (CLength).



Cycle Type / Usage (C7)

The default usage of a tool can be set in the folder tab CAM of a *WinTool* tool assembly. Default Usage (milling) respectively Cycle Type (drilling) is preset for each assembly in the custom field C7. The following values are used (bold = default):

Drilling:

- **0=Simple Drill**
- 1=Boring
- 2= Peck Drilling
- 3=Thread
- 4= Drill 1
- 5= Drill 2
- 6=Special 1
- 7=Special 2

Milling:

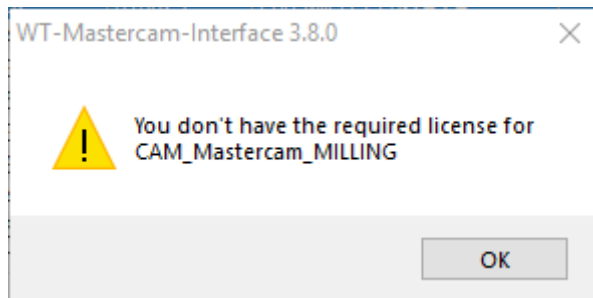
- **0=Rough and finish**
- 1 rough
- finish

Note: You can label the customer fields in the *WinTool* software for each Machine Type individually (Settings > Machine Type, then select machine end edit the labeling of customer fields)

Known Issues

Missing license error message

After installing Mastercam and activating the correct licenses, importing tools continuously shows an error message:



This can happen with Mastercam installed on a system, where no WinTool 2019.1 (and newer) or other Interfaces using the CodeMeter licensing are installed. The issue is a missing wupi.net.dll in the installer. To fix this, request a copy of the wupi.net.dll from WinTool AG; copy/paste it from another installation, or install WinTool 2019.1 (or newer).

Uninstall Error-Message in Mastercam

After uninstalling the WT-Mastercam-Interface the following message might appear during start-up of Mastercam:

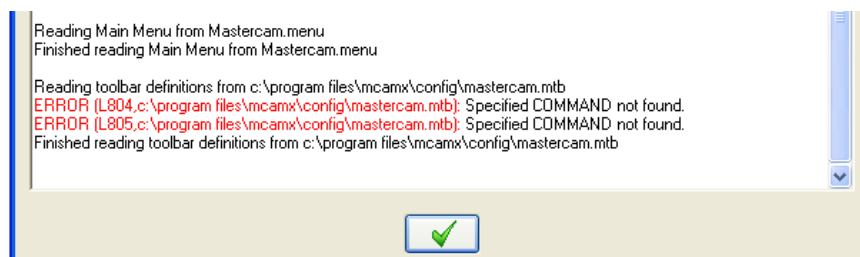


Figure 41 Error message

The message is caused from an emptied out *WinTool* toolbar (GET and PUT have been removed) in Mastercam. However, the uninstall program (Windows Add/Remove Program) does not delete the empty toolbar in Mastercam.

If you install a new WT-Mastercam-Interface version just add the new toolbar (Get, Put) in Mastercam and the message will disappear.

If you want to remove the WT-Mastercam-Interface completely, remove the empty toolbar manually in Mastercam in Customize > Toolbars, then select the toolbar and delete it.

No PUT and GET Buttons Available

Check the Mastercam event log in the Windows task bar icon.

Informa...	15.04.2014	16:38:05	Finished Startup tasks
Error	15.04.2014	16:38:05	ERROR (L28,CHOOKS\WT-Mastercam-Interface.ft): Unable to open resource module chooks\WT-Mastercam-Interface\WT-Mastercam-Chook.dll.
Error	15.04.2014	16:38:05	ERROR (L34,CHOOKS\WT-Mastercam-Interface.ft): Unable to open resource module chooks\WT-Mastercam-Interface\WT-Mastercam-Chook.dll.
Error	15.04.2014	16:38:05	ERROR (L28,C:\Program Files\mcamx7\CHOOKS\WT-Mastercam-Interface.ft): Unable to open resource module chooks\WT-Mastercam-Interface\WT-Mastercam-Chook.dll.
Error	15.04.2014	16:38:05	ERROR (L34,C:\Program Files\mcamx7\CHOOKS\WT-Mastercam-Interface.ft): Unable to open resource module chooks\WT-Mastercam-Interface\WT-Mastercam-Chook.dll.

If an error similar to "Unable to open resource module ... WT-Mastercam-Chook.dll" is listed, the currently installed WT-Mastercam-Interface is not compatible with Mastercam.

The Mastercam versions that are compatible with the interface are listed on the first page of this manual.

Incorrect Diameter and Length Correction Numbers

Problem: The tool assembly diameter and length correction numbers of values don't match with the values in *WinTool*.

Solution: Check the currently used machine definition in the machine group. Open "Machine group properties" and click the edit button of "Machine-Toolpath Copy". In the "Machine Definition Manager", click the "Control definition button":

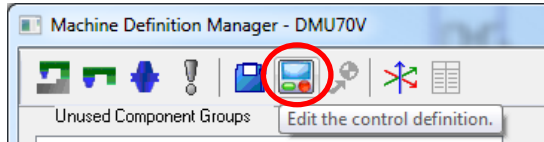


Figure 42 Edit the control definition

Check the "Tool offset registers" setting. Select "From tool" if the D and L number must be imported directly.

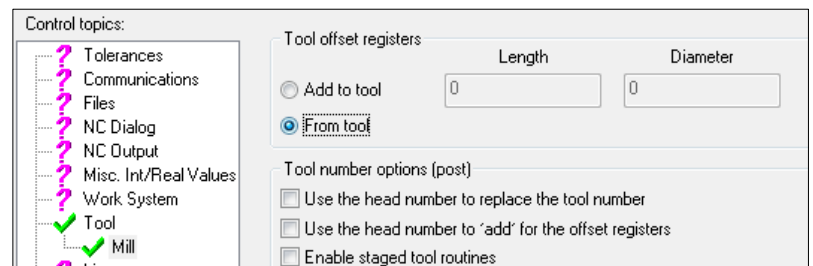


Figure 43 Edit the control definitions

Annex

Configuration File Parameters

General Information

All configurable parameters will be installed with default values unless they are changed in the cfg files. A cfg file can be edited with a text editor: In the cfg file the lines starting with a “#” symbol will be ignored (the symbol “#” defines a “comment line”). If you remove the # symbol the line will be activated.

Some parameters have a default value stored as a system variable. As soon as Mastercam is started up the default values will be overwritten with the values configured in the cfg file.

WT-Mastercam-Interface.cfg

```
[WT-Mastercam-Interface]
# Exchange Path configuration
# -----
OutputPath=
#   Default OutputPath is "Exchange" folder in local path

UserModelsPath=
#   Default UserModelsPath is "UserModels" folder in local path

SelectCutData=True
```

OutputPath	Folder path in which the WT-Mastercam-Interface stores the data exchange files. The system automatically registers the WTMastercamExportPath system variable with this value (see next page). Default OutputPath is the "Exchange" folder in the folder "[Public Documents]\WT-Mastercam-Interface-2025". Note: Use a different exchange path for each user
UserModelsPath	Folder path in which the WT-Mastercam-Interface stores the contour DXF files. The system automatically registers the system variable WTMastercam-UserModelPath with this value (see next page). Default UserModelsPath is "UserModels" folder in the folder "[Public Documents]\WT-Mastercam-Interface-2025".
SelectCutData	If "True", the interface imports cutting conditions for work materials. A selection window opens if there are multiple or no cutting conditions for the material, or if a single tool assembly is transferred. If not set, the value is "False". This transfers all cutting conditions.

Windows Registry values

Local Machine

Installation path of interface

32-bit Windows registry key:

HKEY_LOCAL_MACHINE\SOFTWARE\Microsoft\Windows\CurrentVersion\App Paths\WT-Mastercam-Interface-2025.exe

64-bit Windows registry key:

HKEY_LOCAL_MACHINE\SOFTWARE\Wow6432Node\Microsoft\Windows\CurrentVersion\App Paths\WT-Mastercam-Interface-2025.exe

(Parameter is set during installation)

Current user

HKEY_CURRENT_USER\Software\WinTool\WT-Mastercam-Interface

OutputPath = C:\Users\Public\Documents\WT-Mastercam-Interface-2025\Exchange

(You can change this path in the file WT-Mastercam-Interface.cfg:

HKEY_CURRENT_USER\Software\WinTool\WT-Mastercam-Interface

UserModelsPath = C:\Users\Public\Documents\WT-Mastercam-Interface-2025\UserModels

(You can change this path in the file WT-Mastercam-Interface.cfg)

Supported Mastercam Tool Types

Milling Tools

Center Drill (/MC1)

Define Center Drill

Adjust geometric properties used to define the tool shape.

Standard sizes

Overall dimensions

Cutting length: 7.30228

Shank diameter: 7.94

Overall length: 54

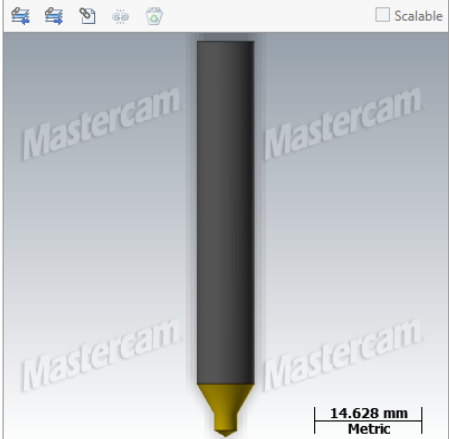
Tip treatment

Drill diameter: 3.18

Drill length: 3.18

Drill angle: 118

Shoulder angle: 60



14.628 mm
Metric

Spot Drill (/MC2)

Define Spot Drill

Adjust geometric properties used to define the tool shape.

Standard sizes

Overall dimensions

Drill diameter: 6

Overall length: 50

Cutting length: 25

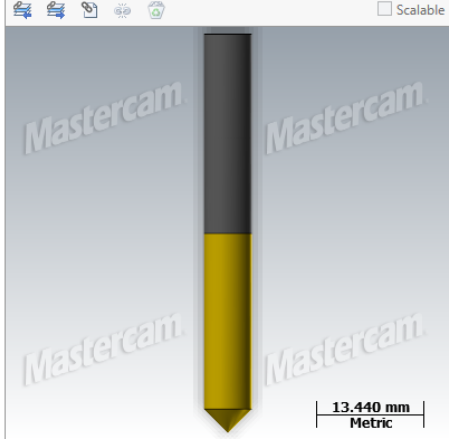
Tip treatment

Tip angle: 90

Non-cutting geometry

Shoulder length: 37

Shank diameter: 6



13.440 mm
Metric

Drill (/MC3)

Define Drill

Adjust geometric properties used to define the tool shape.

Standard sizes

Overall dimensions

Drill diameter: 6

Overall length: 50

Cutting length: 25

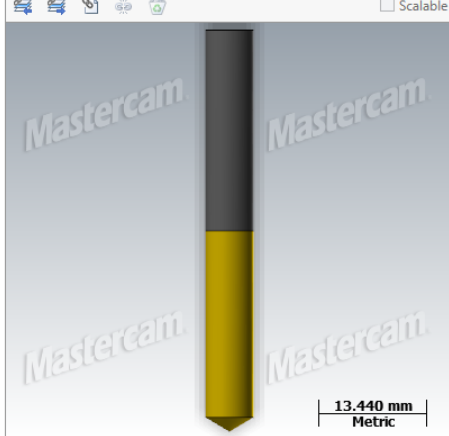
Tip treatment

Tip angle: 118

Non-cutting geometry

Shoulder length: 40

Shank diameter: 6



13.440 mm
Metric

Tap RH (/MC4)

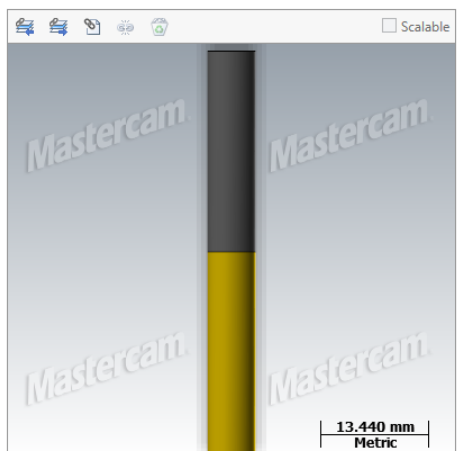
Define Tap

Adjust geometric properties used to define the tool shape.

Standard sizes	
Nominal diameter:	6
Pitch:	2.5
<input type="checkbox"/> Left hand	

Overall dimensions	
Cutting length:	25
Shank diameter:	6
Overall length:	50

Tip treatment	
Bottoming	



13.440 mm
Metric

Tap LH (/MC5)

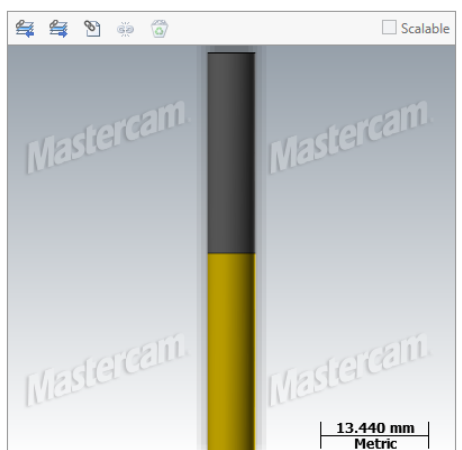
Define Tap

Adjust geometric properties used to define the tool shape.

Standard sizes	
Nominal diameter:	6
Pitch:	2.5
<input checked="" type="checkbox"/> Left hand	

Overall dimensions	
Cutting length:	25
Shank diameter:	6
Overall length:	50

Tip treatment	
Bottoming	



13.440 mm
Metric

Reamer (/MC6)

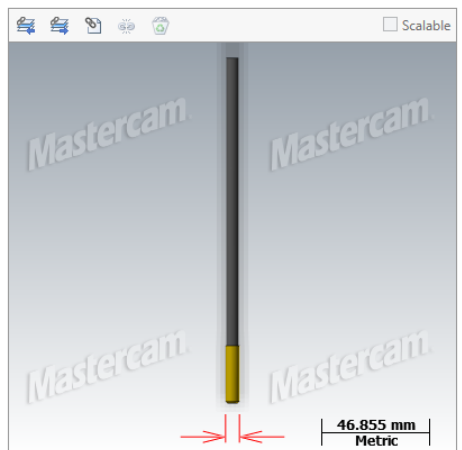
Define Reamer

Adjust geometric properties used to define the tool shape.

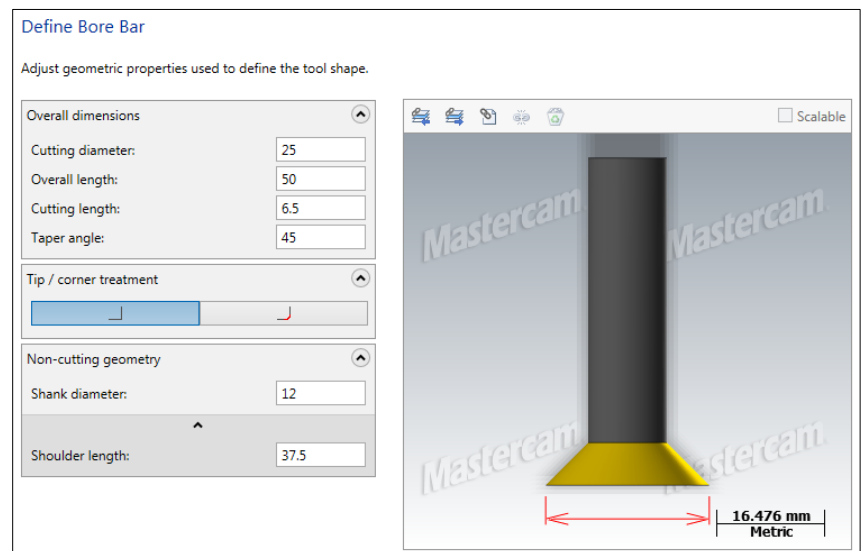
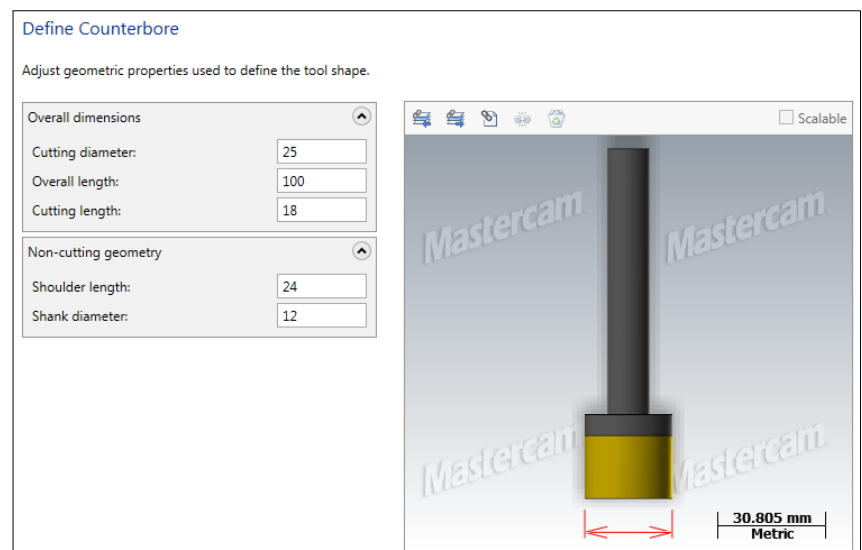
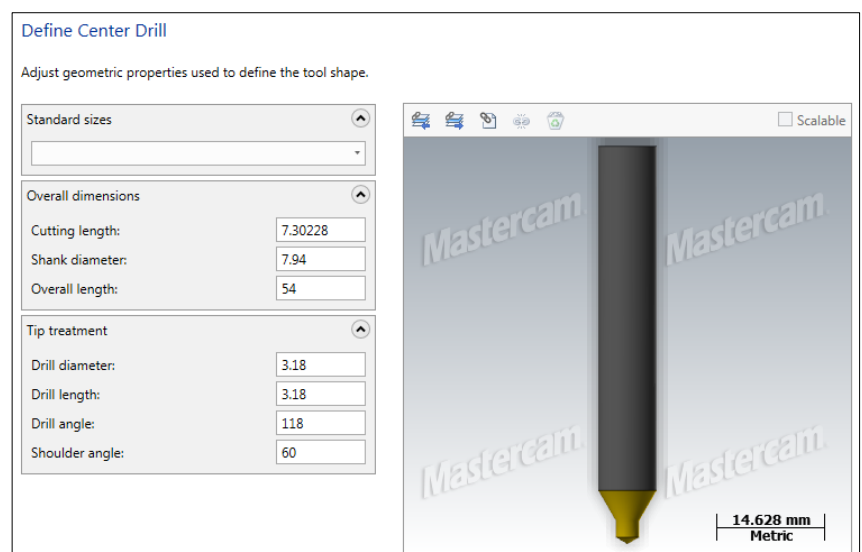
Overall dimensions	
Cutting diameter:	6
Overall length:	150
Cutting length:	25

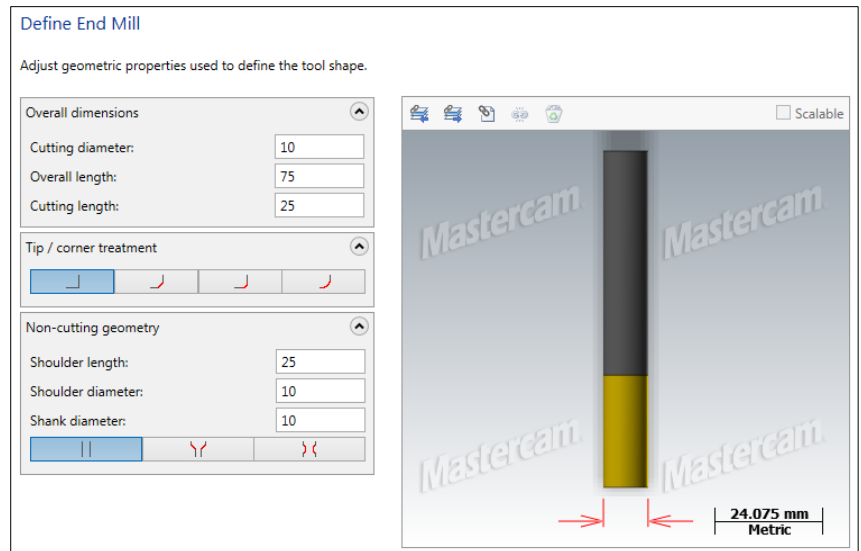
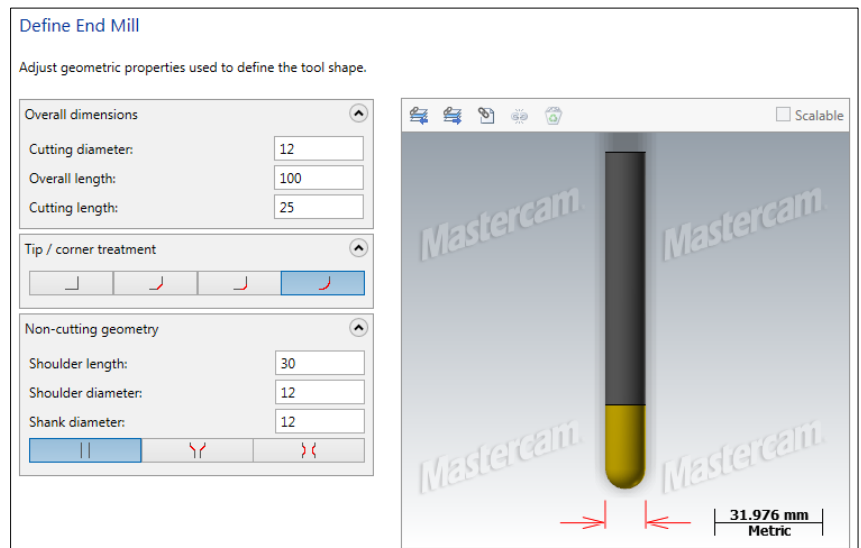
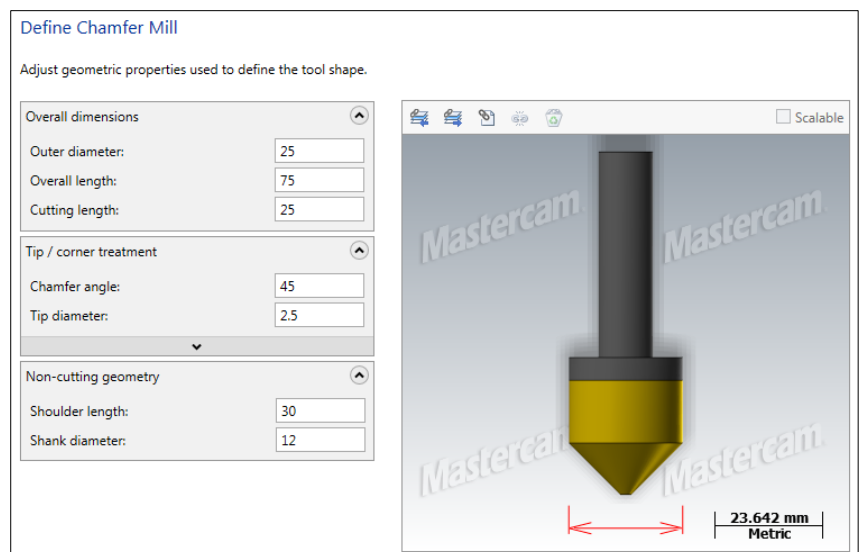
Tip / corner treatment	
Chamfer distance:	0.75

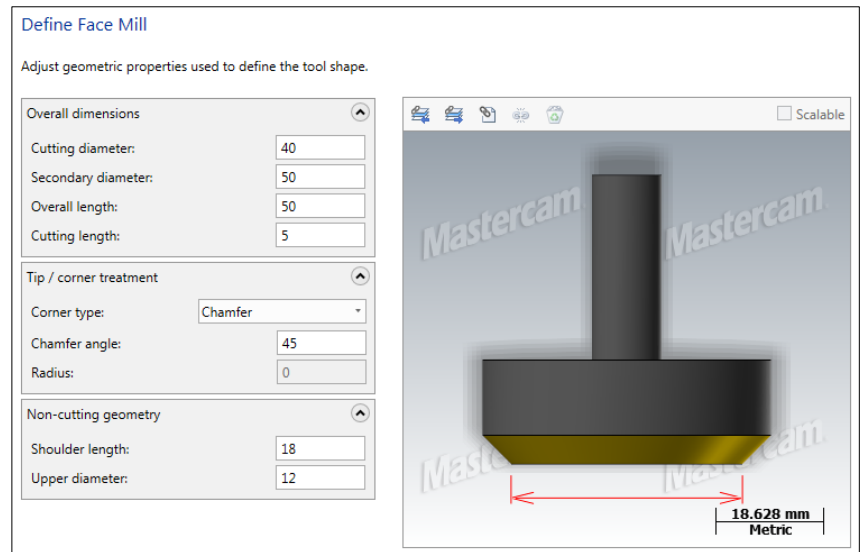
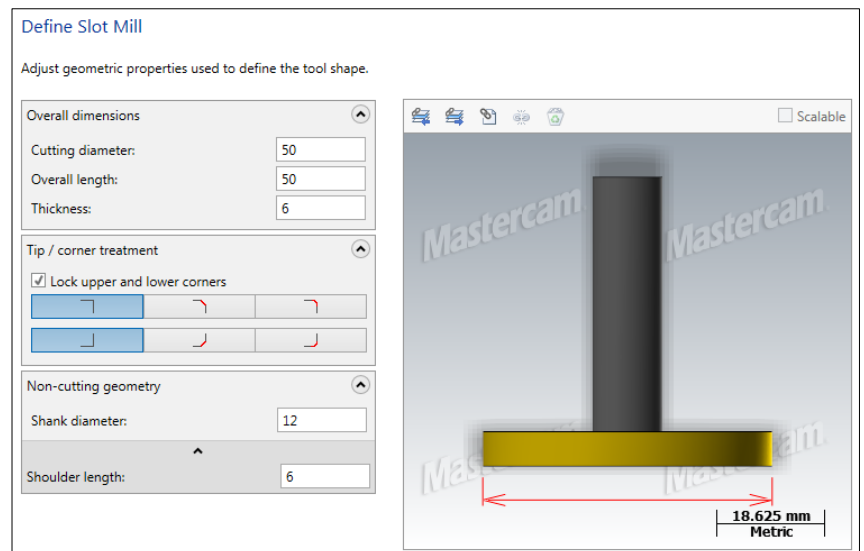
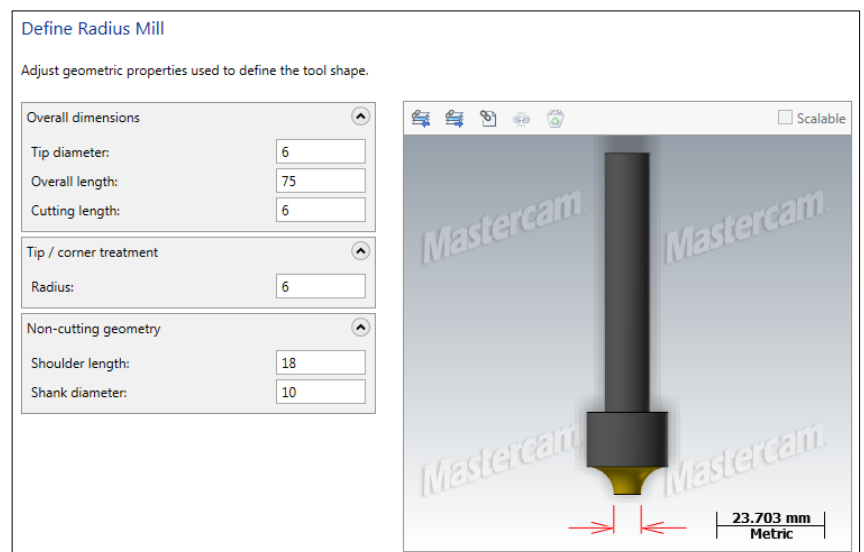
Non-cutting geometry	
Shank diameter:	5.4
Shoulder length:	25

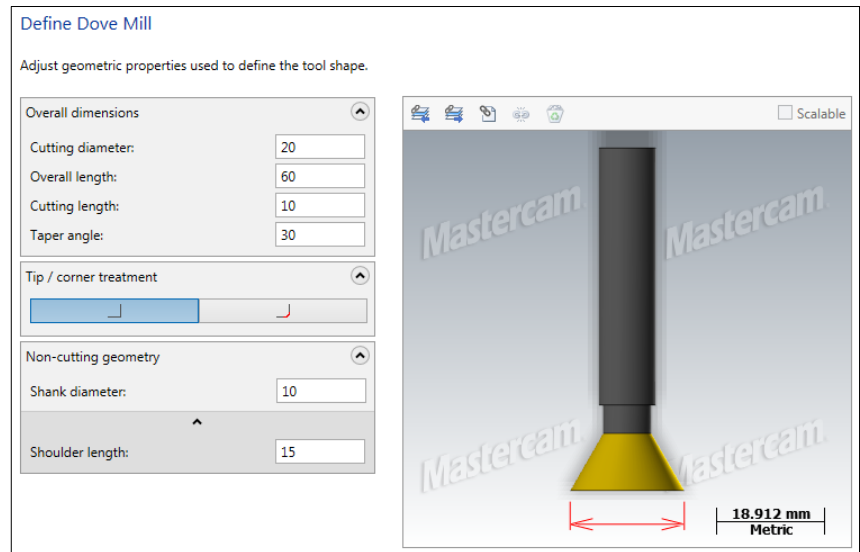
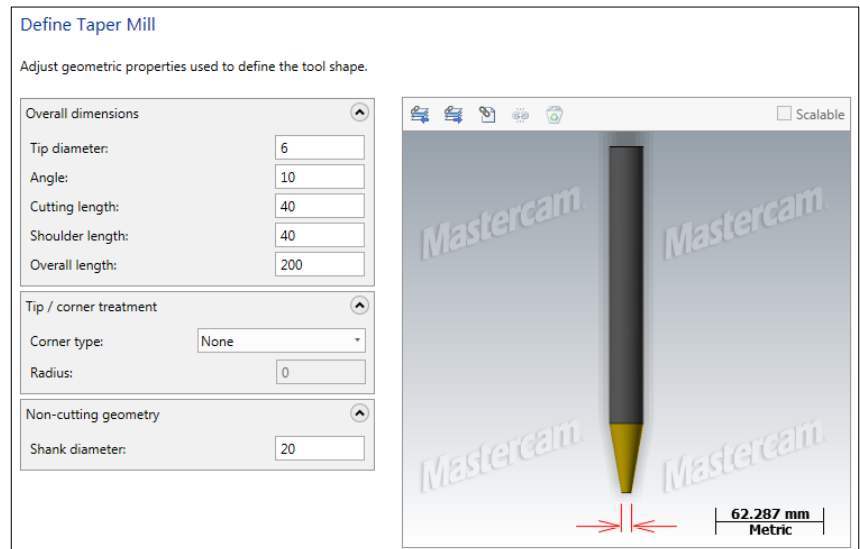
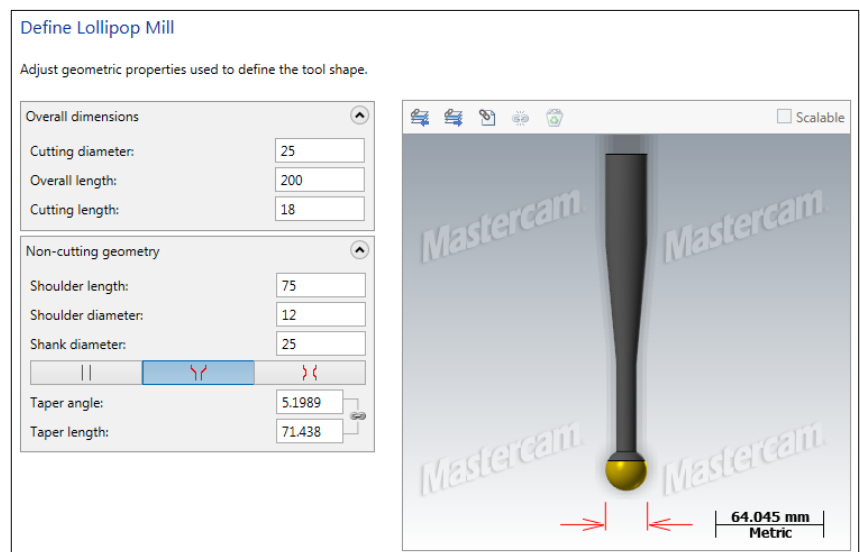


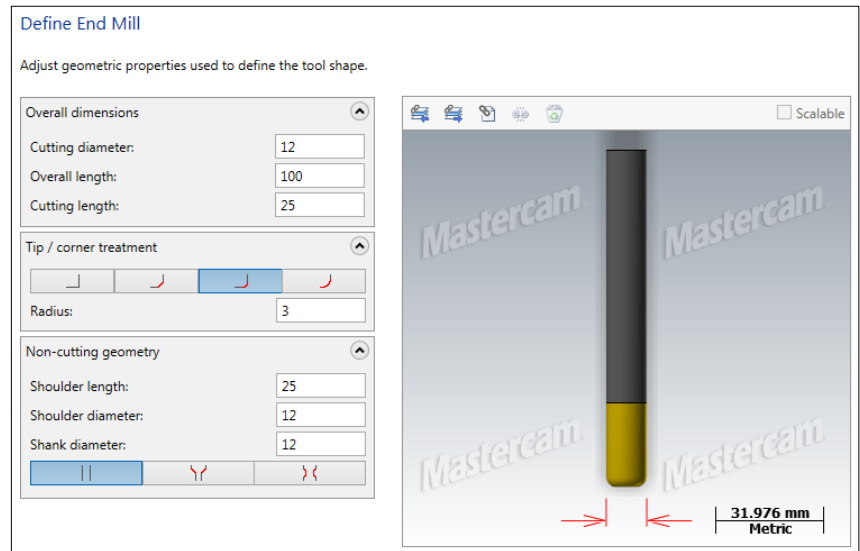
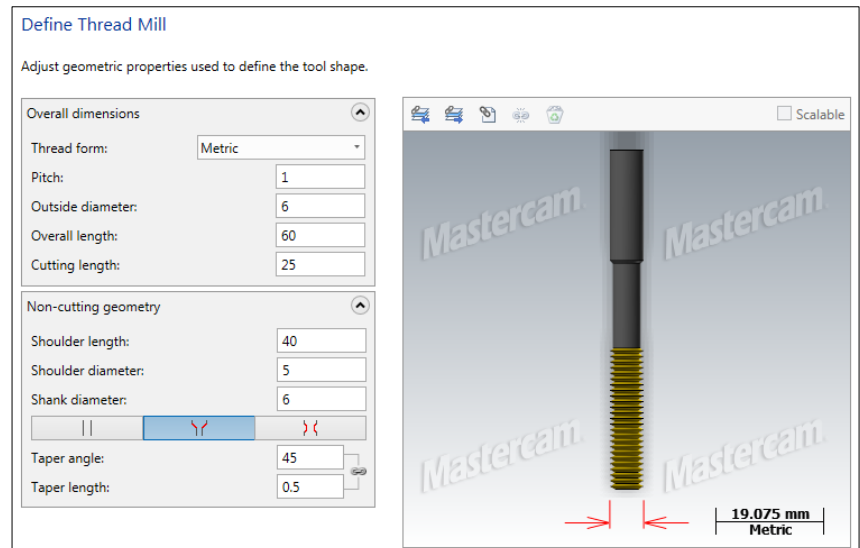
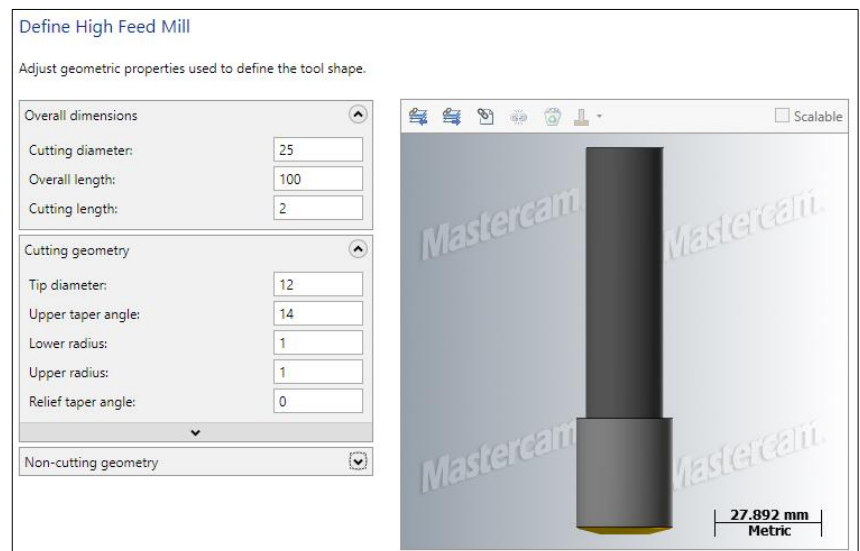
46.855 mm
Metric

Bore Bar (/MC7)**Counter Bore (/MC8)****Counter Sink (/MC9)**

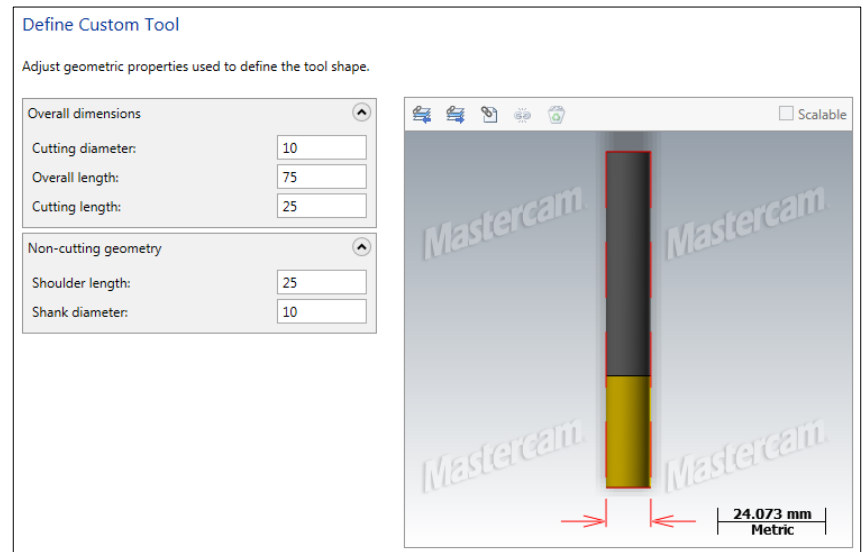
End Mill (/MC10)**Sphere Mill (/MC11)****Chamfer Mill (/MC12)**

Face Mill (/MC13)**Slot Mill (/MC14)****Radius Mill (/MC15)**

Dove Mill (/MC16)**Taper Mill (/MC17)****Lollipop Mill (/MC18)**

Bull Mill (/MC19)**Thread Mill (/MC24)****High Feed Mill (/MC26)**

Lathe Drill	(/MC51)
Lathe Center Drill	(/MC52)
Lathe Countersink	(/MC53)
Lathe Counterbore	(/MC54)
Lathe End Mill	(/MC55)
Lathe Reamer Drill	(/MC56)
Lathe RH Tap	(/MC57)
Lathe LH Tap	(/MC58)
General Turning	(/MC100)
Lathe Boring Bar	(/MC106)

Custom Tool (/MC0)**Not Supported Mastercam Tool Types**

- Engrave Tool
- Brad point Drill
- Barrel Mill (Incompatible definition)
- Grooving tools
- Threading tools

List of Figures

Figure 1 Mastercam chooks directory	6
Figure 2 Customize the Ribbon.....	7
Figure 3 Customize the Ribbon with New Group	7
Figure 4 Steps to the WinTool Mastercam Installation.....	7
Figure 5 Choose commands and add to Tool Tabs	8
Figure 6 Mastercam Tool Settings	9
Figure 7 Choose Mastercam in User Settings	9
Figure 8 Tool Assemblies in CAM	10
Figure 9 WT-MasterCAM-Interface configuration	11
Figure 10 WT-MasterCAM-Interface configuration for naming conventions	13
Figure 11 Main Placeholders.....	14
Figure 12 Change the WT-database	16
Figure 13 Select Tool assembly in the menubar „TOOLPATHS“	17
Figure 14 Search a tool assembly class from list.....	17
Figure 15 Assembly cutting conditions.....	18
Figure 16 Select a Mastercam tool type from list	18
Figure 17 3D Toolpaths model	19
Figure 18 2D Toolpaths model	19
Figure 19 Search to transfer the Tool List	20
Figure 20 List of all imported tools	20
Figure 21 List of T-No for Tool Assemblies	21
Figure 22 Check the T-No.	21
Figure 23 Notification - Tool number	22
Figure 24 Instruction to search for cut parameters	23
Figure 25 Enter tool assembly and click on Search icon	23
Figure 26 Select tools to be transferred	24
Figure 27 Edit List name in WT-MakeList	24
Figure 28 Notification	24
Figure 29 Entry fields will be filled in automatically	25
Figure 30 Change NC file name.....	25
Figure 31 Tab folder Files - Machine data	25
Figure 32 Enter a program number in tab folder 'Tool Settings'	25
Figure 33 Every Descriptions has an own Note	26
Figure 34 Enter Ident Number and Tool Description.....	27
Figure 35 Namegiving must be marked	27
Figure 36 Select Shape for Generator	28
Figure 37 Verify Tool Contour by click on the icon.....	28
Figure 38 Select User Model in the menu "CAM"	29
Figure 39 Create a Tool Contour DXF	29
Figure 40 Instruction to activate the tab "Shape"	30
Figure 41 Error message.....	32
Figure 42 Edit the control definition.....	33
Figure 43 Edit the control definitions	33