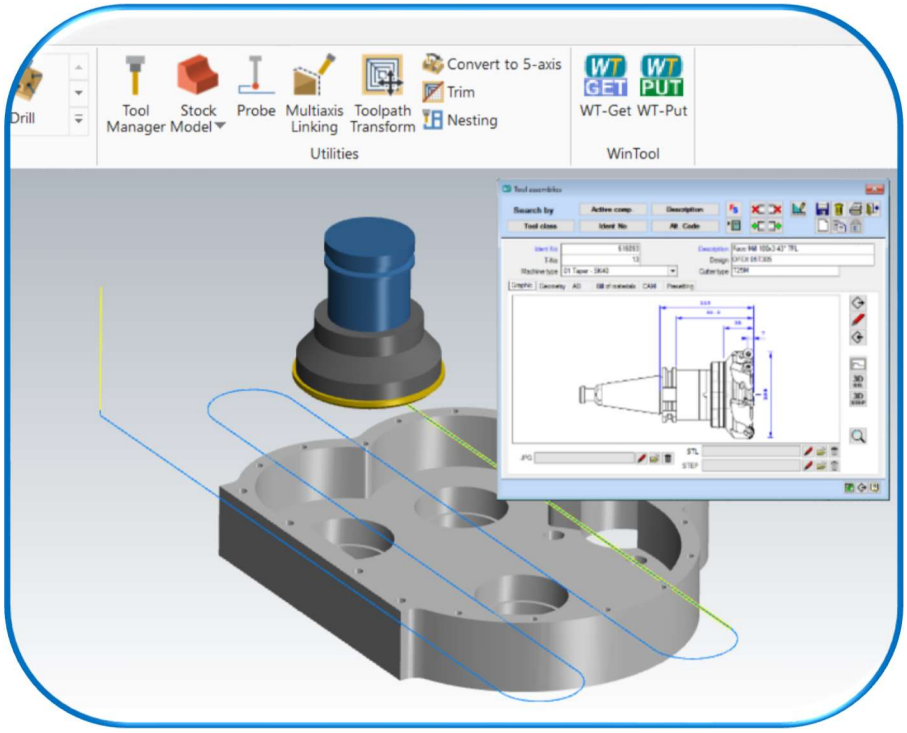
WT-Mastercam Interface



Manual

WinTool Interface 4.2.1 for Mastercam

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* 2011 Professional or later
- Mastercam 2022

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

Contents

Summary 5
Job 5
Requirements 5
Supported Tool Types 5
Copyright 5
Installation 6
New licensing structure 6
Installing for WinTool prior 2019.1 6
New Directory Structure 6
Update Installation 6
Update from version 3.1 and newer 6
Update from version 3.0.1 or older 6
New Installation 6
Mastercam 2017 and up 8
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 *WinTool* 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
Software Structure 32
Software-Modules and Data-Exchange 32
WinTool-Mastercam Data Integration 33
Known Issues 34
Missing license error message 34
Uninstall Error-Message in Mastercam 34
No PUT and GET Buttons Available 34
Incorrect Diameter and Length Correction Numbers 35
Annex 36
Configuration File Parameters 36
General Information 36
WT-Mastercam-Interface.cfg 36
Windows Registry values 37
Local Machine 37
Current user 37
Supported Mastercam Tool Types 38
Milling Tools 38
Not Supported Mastercam Tool Types 45
History 46
4.2 46
4.1 46
4.0 46
3.8 46

3.7 / 3.7.1 46
3.6 46
3.5 46
3.4 46
3.3 47
3.2 47
3.1 47
3.0.1 47
3.0 47
2.7 48
2.6 48
2.5 48
2.4 48
2.3 - 3rd Release 48
2.3 - 2nd Release 48
2.3 48
2.2 49

List of Figures 50

Summary

Job

The WT-Mastercam-Interface enables the user to select and transfer tool assemblies from the *WinTool* 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 2011 or later and Mastercam 2018 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
http://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 folder:

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

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.
  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

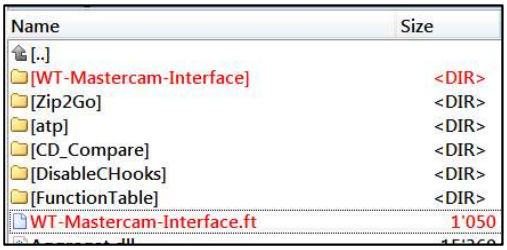
The default location of the setting `UserModelPath` has changed. If you haven't set a `UserModelPath` in the interface configuration, in which case the UserModels folder is in the interface installation directory, you must move the contents of the folder to the new default location `[Public Documents]`\WT-Mastercam-Interface\UserModels.

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\WinTool\WT-Mastercam-Interface-2018

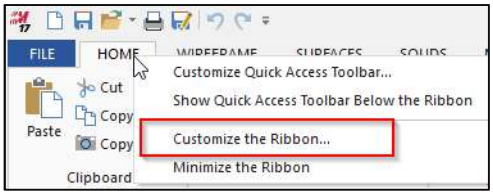
Note: During the installation some WT-Mastercam-Interface software components will be stored in your Mastercam **chooks** directory:



Setup the new WT-Mastercam Interface toolbar as follows:

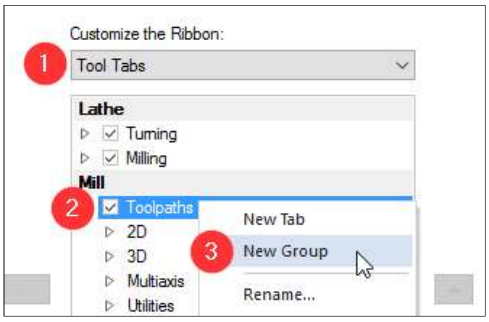
Mastercam 2017 and up

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



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"



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

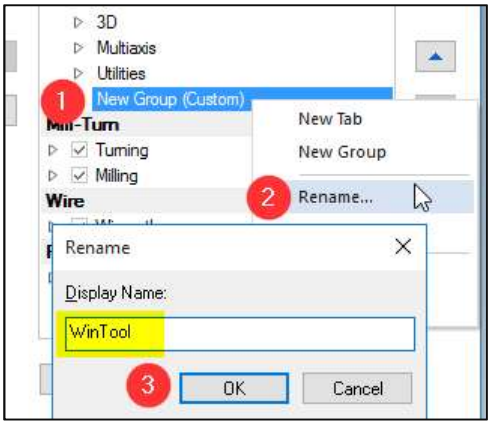


Figure 1 Steps to the WinTool Mastercam Installation

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 >>":

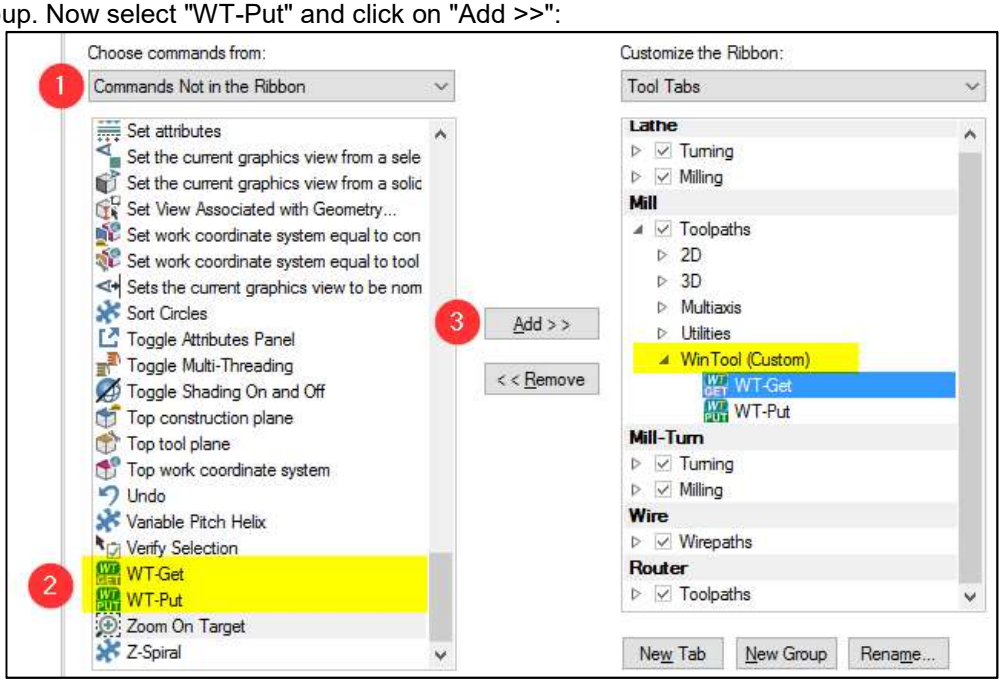


Figure 2 Choose commands and add to Tool Tabs

Configuration

Mastercam Settings

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

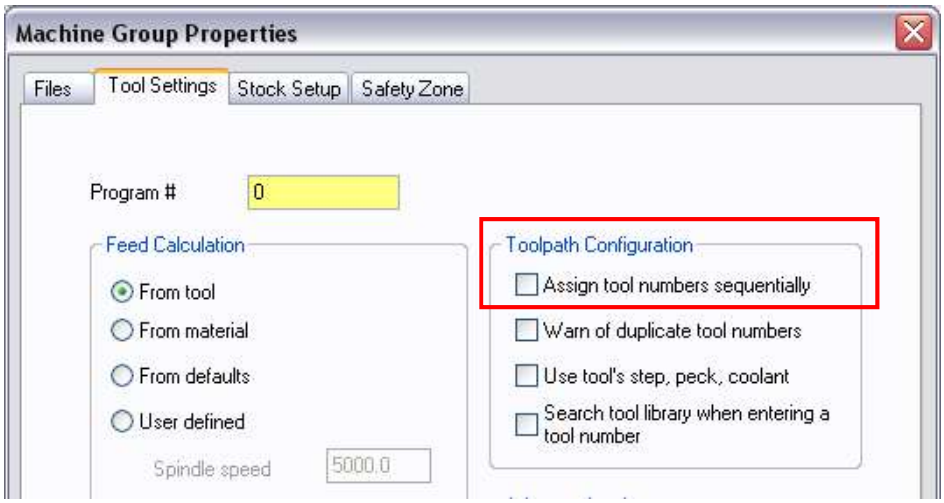


Figure 3 Mastercam Tool Settings

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:

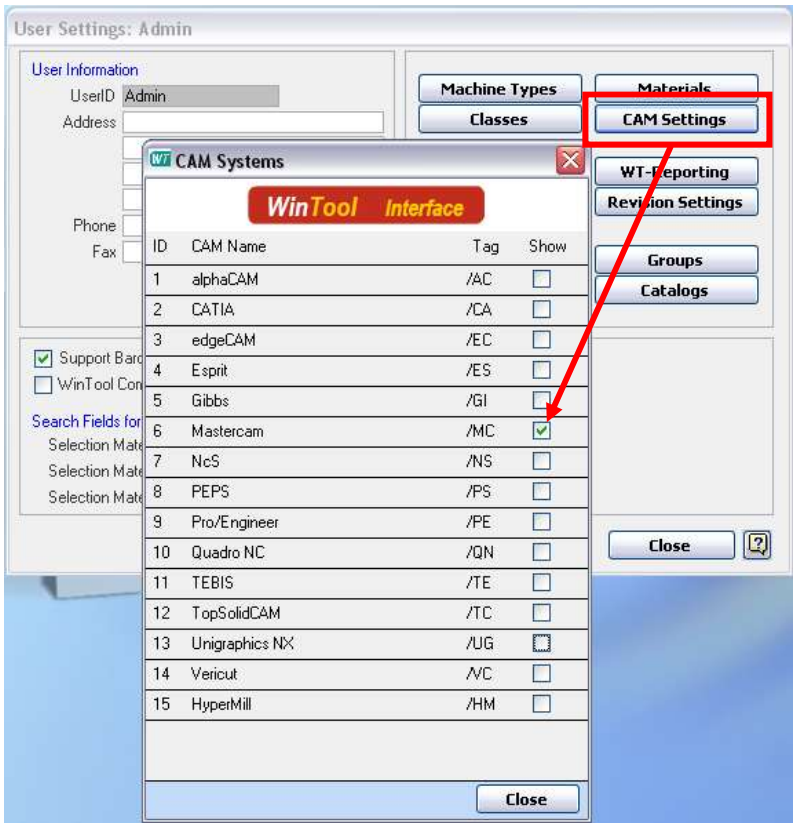


Figure 4 Choose Mastercam in User Settings

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

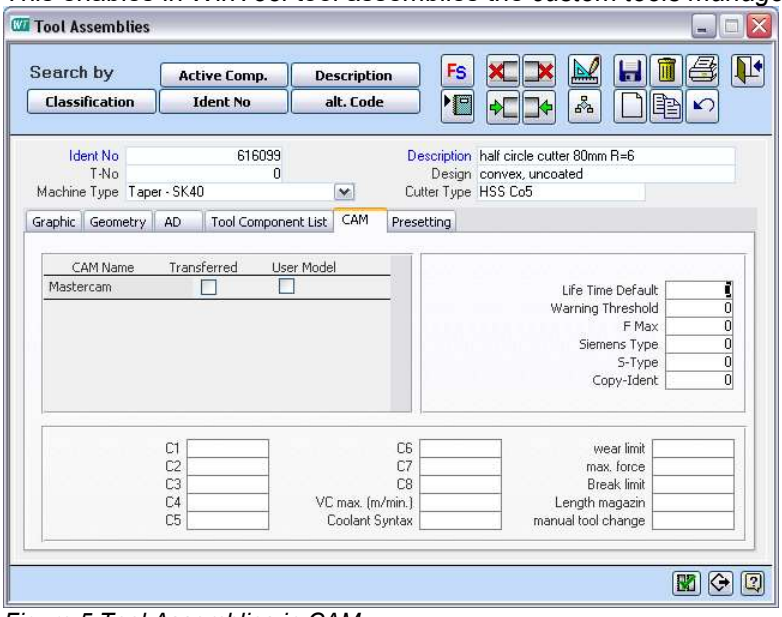


Figure 5 Tool Assemblies in CAM

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:

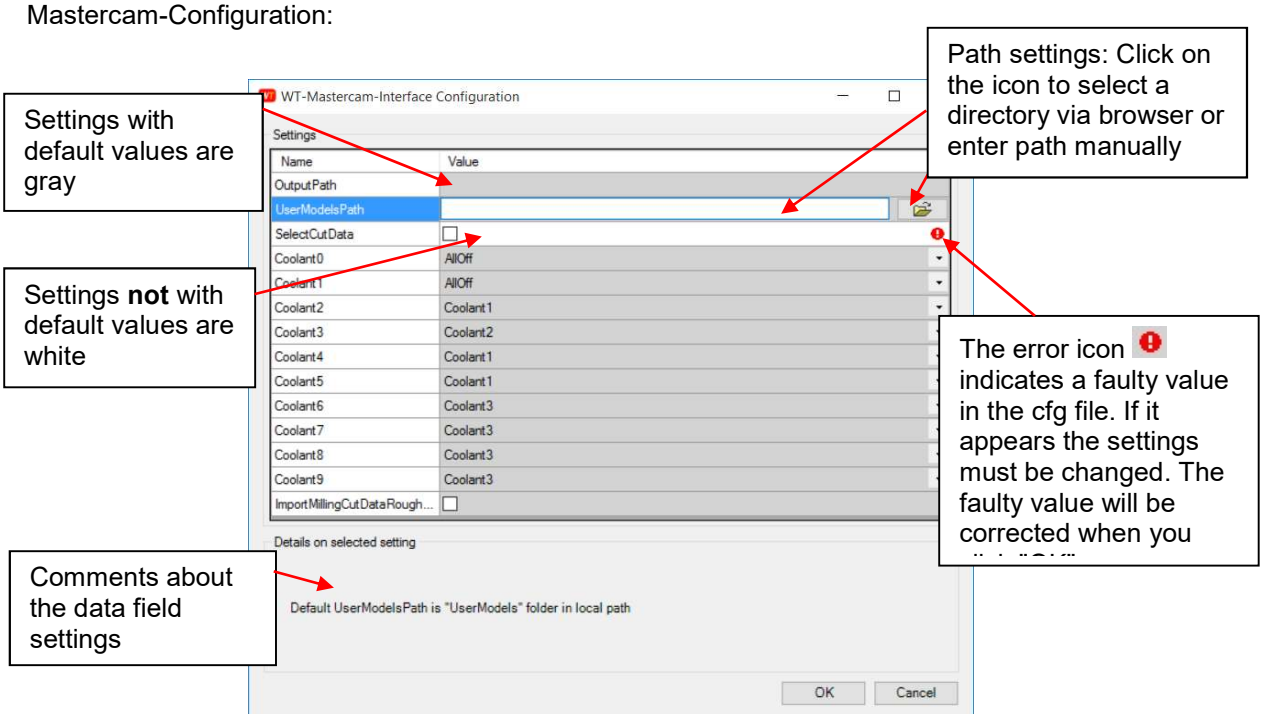


Figure 6 WT-MasterCAM-Interface configuration

<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 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-Interfaces 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	For a tool list the interface imports all cutting conditions available for *one material only*. If the work material has been assigned to a tool list in *WinTool* (see folder tab "General Data") then the interface imports the tools with the cutting condition for this material automatically. If the work material has *not* been assigned to a tool list in *WinTool*, 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. 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.

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”:

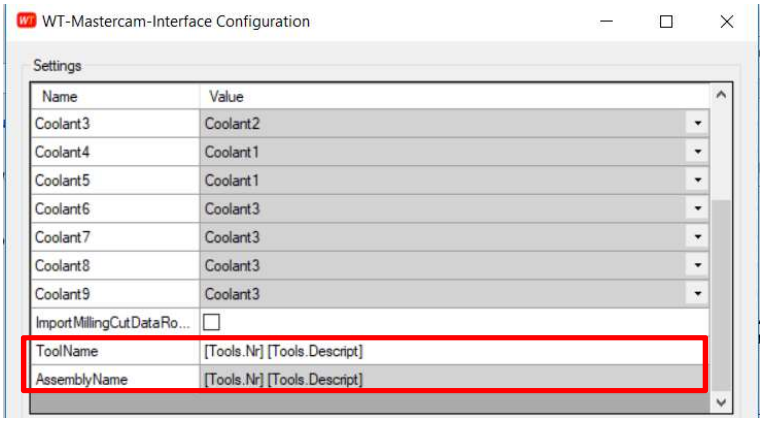


Figure 7 WT-MasterCAM-Interface configuration for naming conventions

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

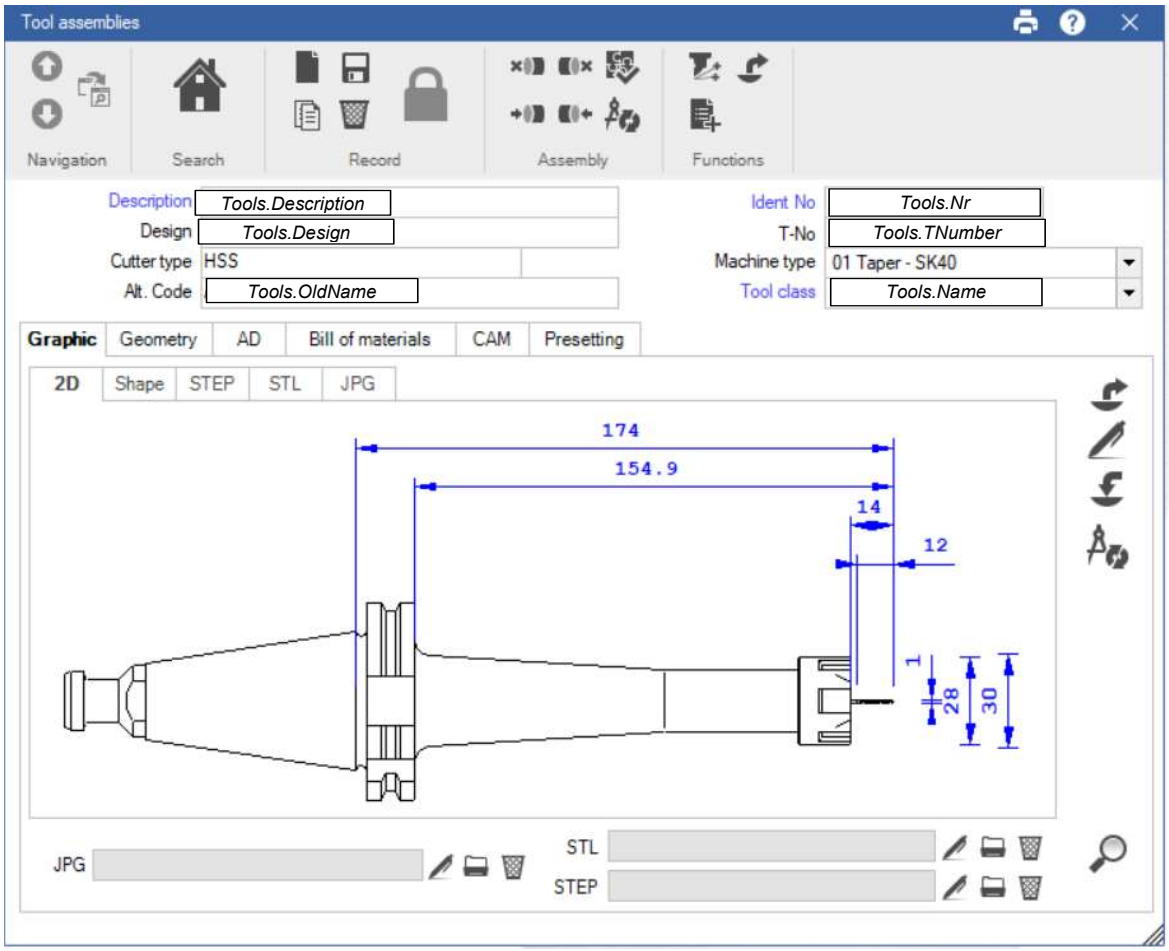


Figure 8 Main Placeholders

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 have to be put in square brackets.
- ToolName and AssemblyName text is limited by Mastercam to 100 characters each.
- For Identification, ToolName have to 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 * (CuttingCondition.Ae / CuttingCondition.Diameter)
Rough Z step (%) = 100 * (CuttingCondition.Ap / 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.

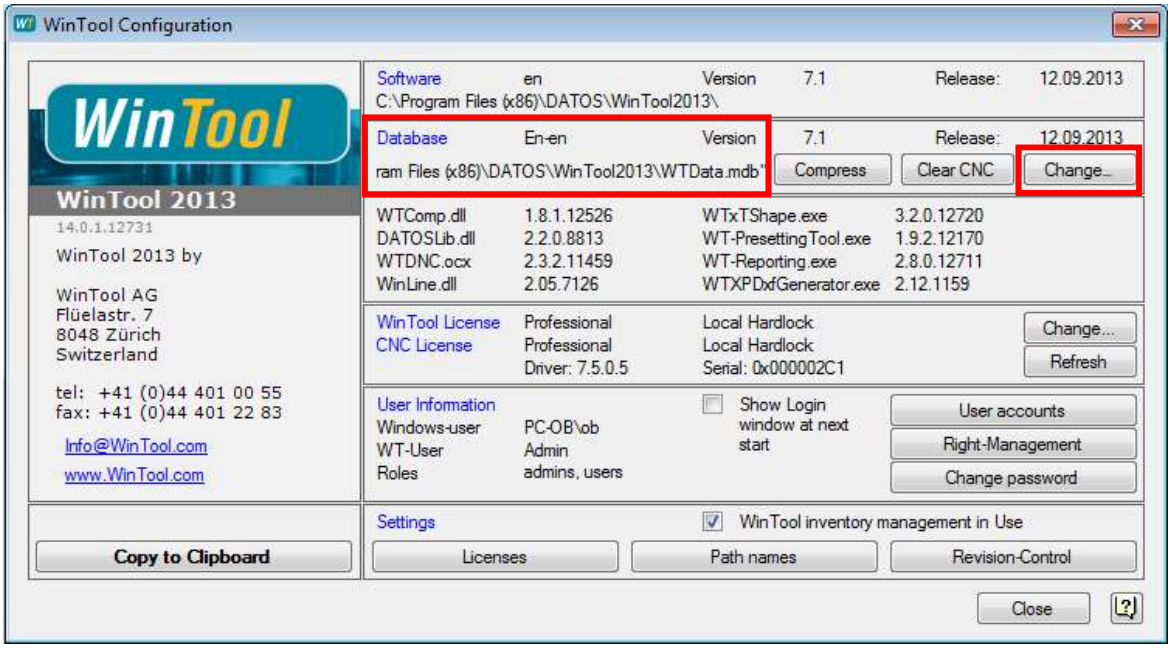


Figure 9 Change the WT-database

For instructions on how to link 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" 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*:

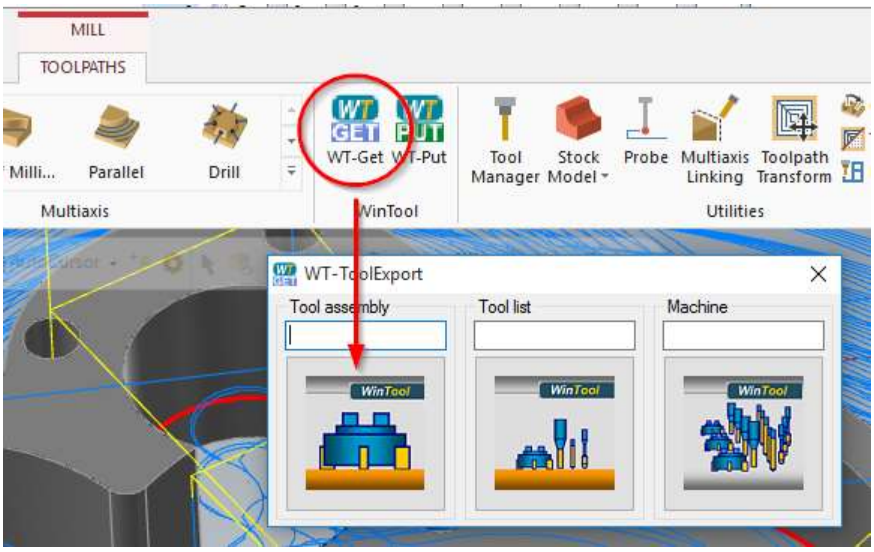


Figure 10 Select Tool assembly in the menubar „TOOLPATHS“

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

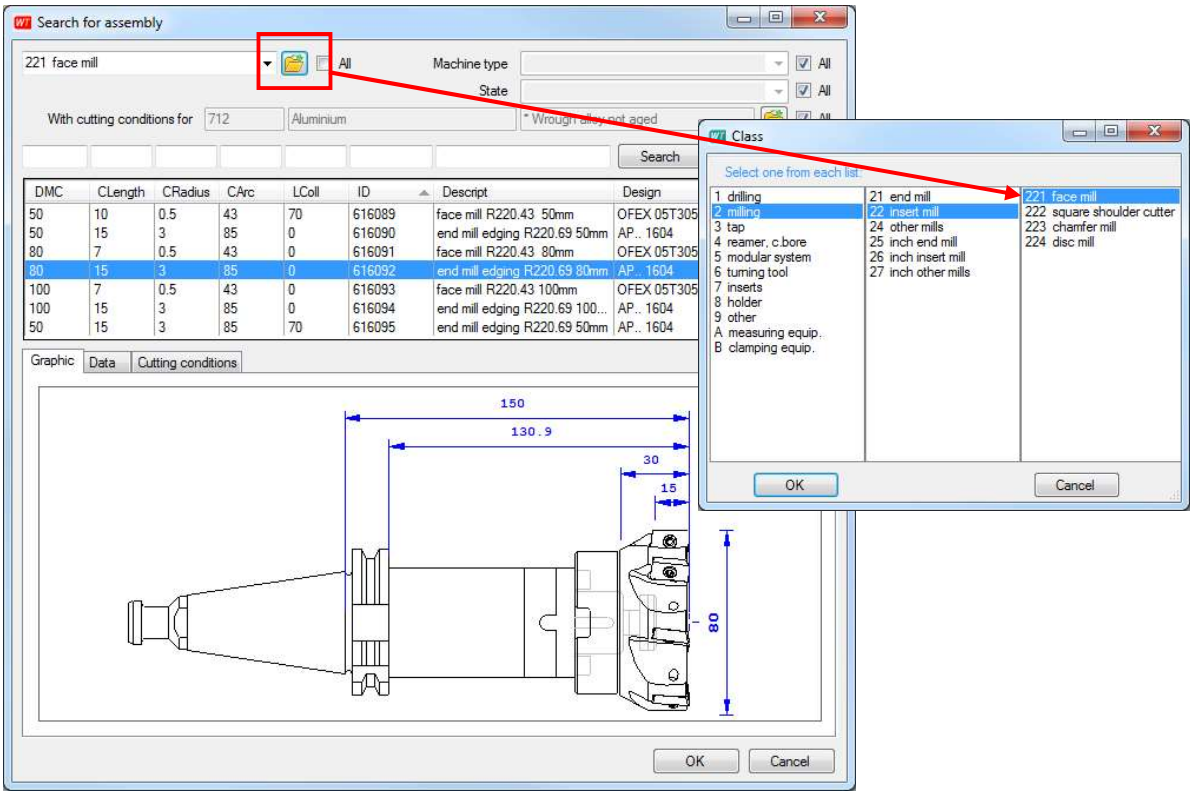


Figure 11 Search a tool assembly class from list

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”.

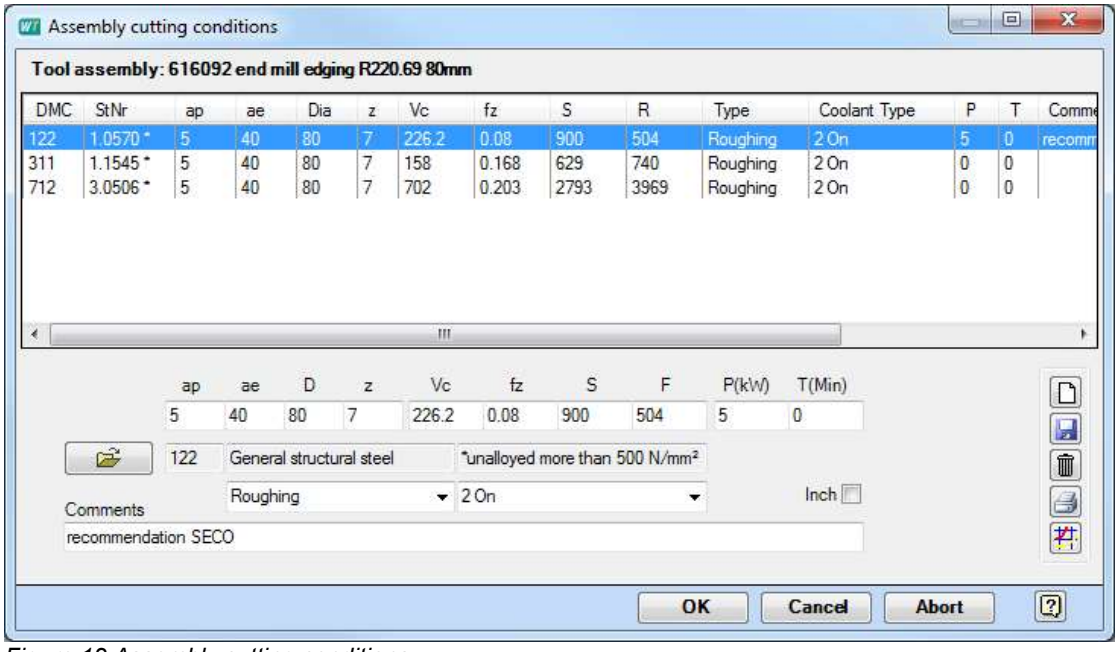


Figure 12 Assembly cutting conditions

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.

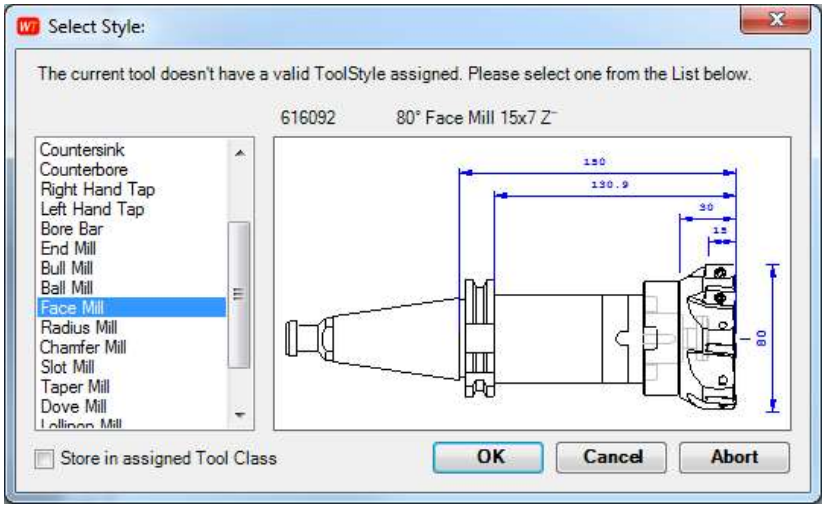


Figure 13 Select a Mastercam tool type from list

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:

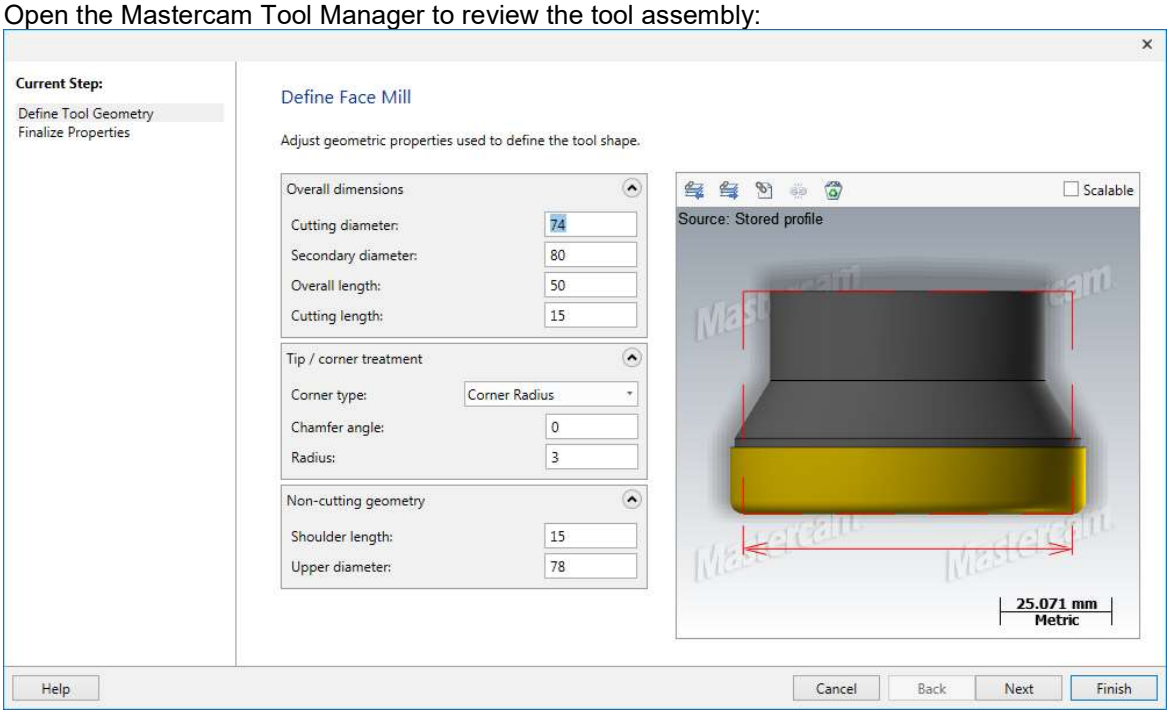


Figure 14 3D Toolpaths model

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

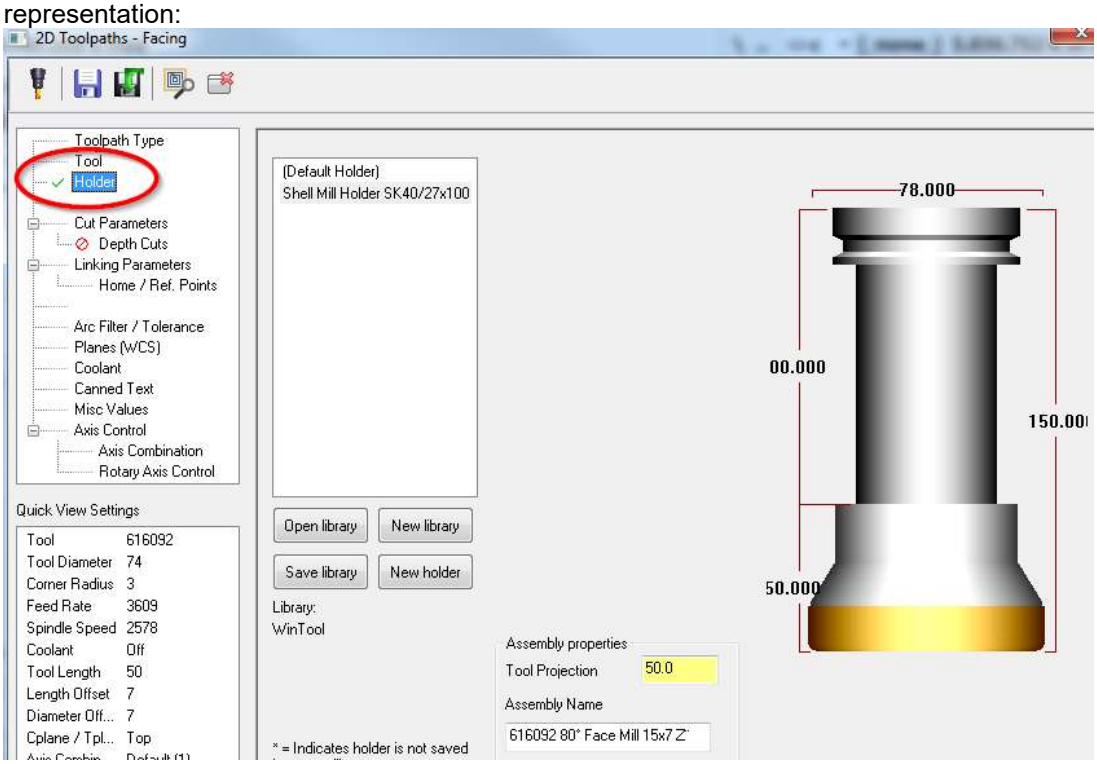


Figure 15 2D Toolpaths model

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

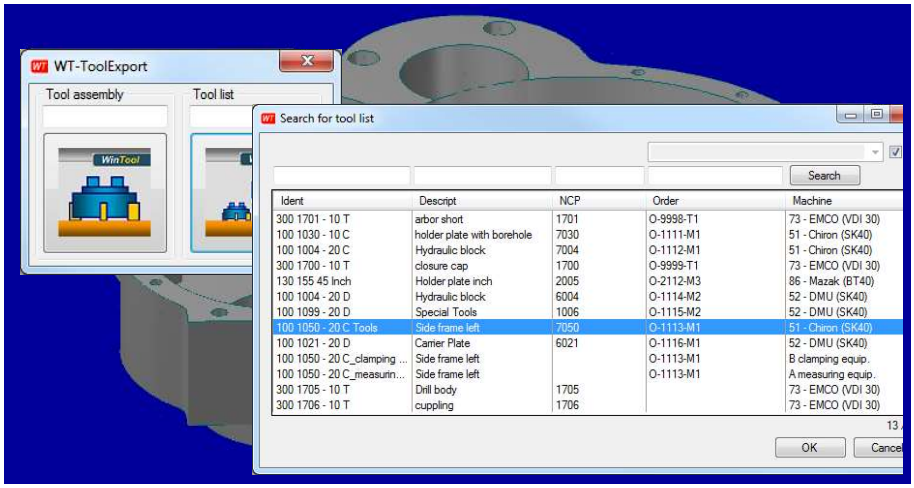
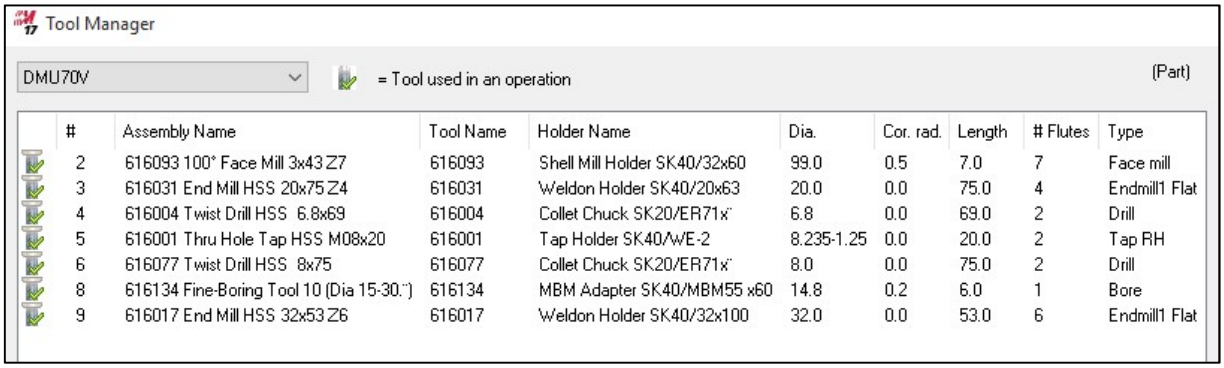


Figure 16 Search to transfer the Tool List

Review all imported tools in Toolpaths > Tool Manager:



#	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 17 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.

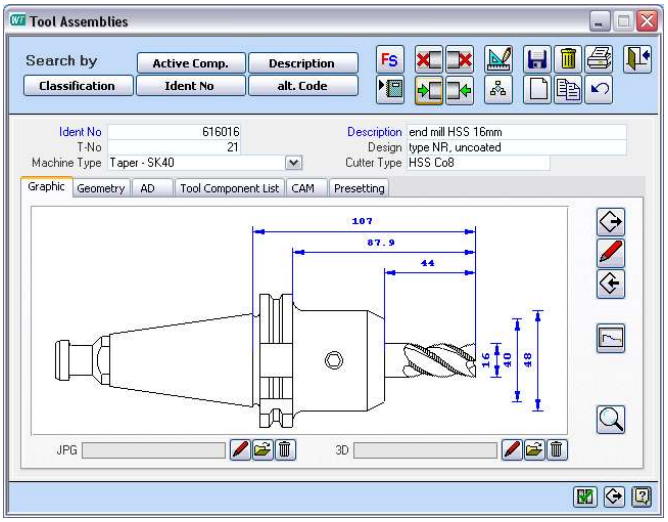
Tool Numbers

T-Number Assignment

If you import *WinTool* tool assemblies to Mastercam, the interface software will assign automatically 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:

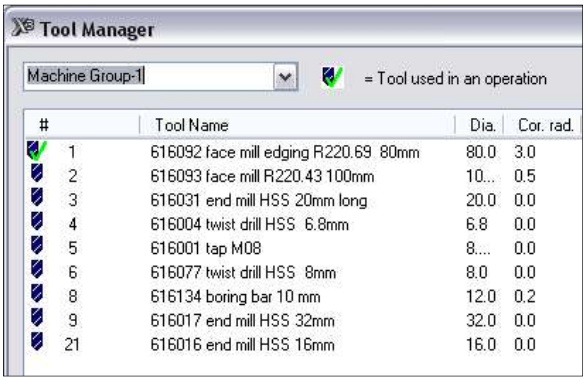


Figure 18 List of T-No for Tool Assemblies

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.

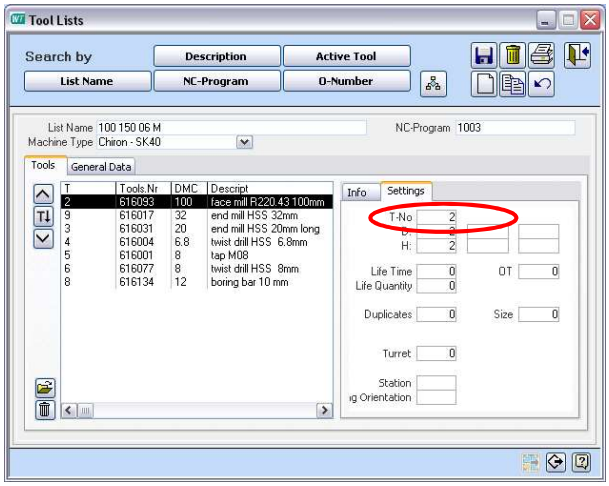


Figure 19 Check the T-No.

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:

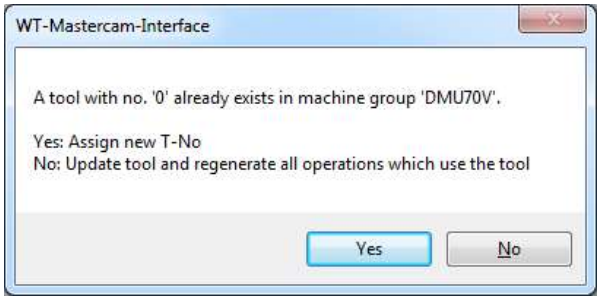
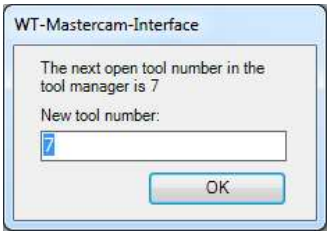


Figure 20 Notification - Tool number

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.

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 in to the Mastercam tool library.

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

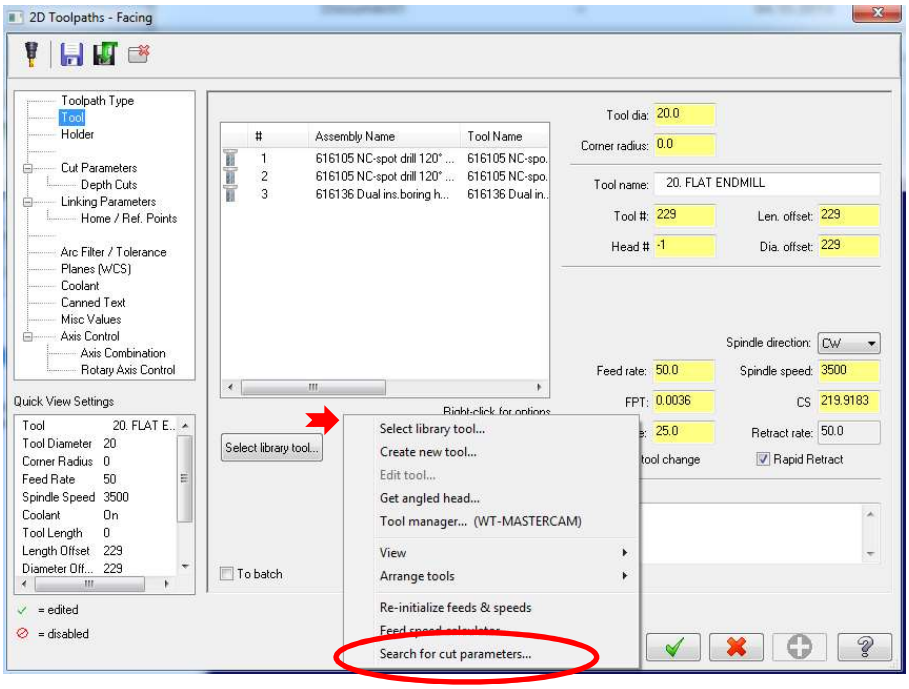


Figure 21 Instruction to search for cut parameters

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":

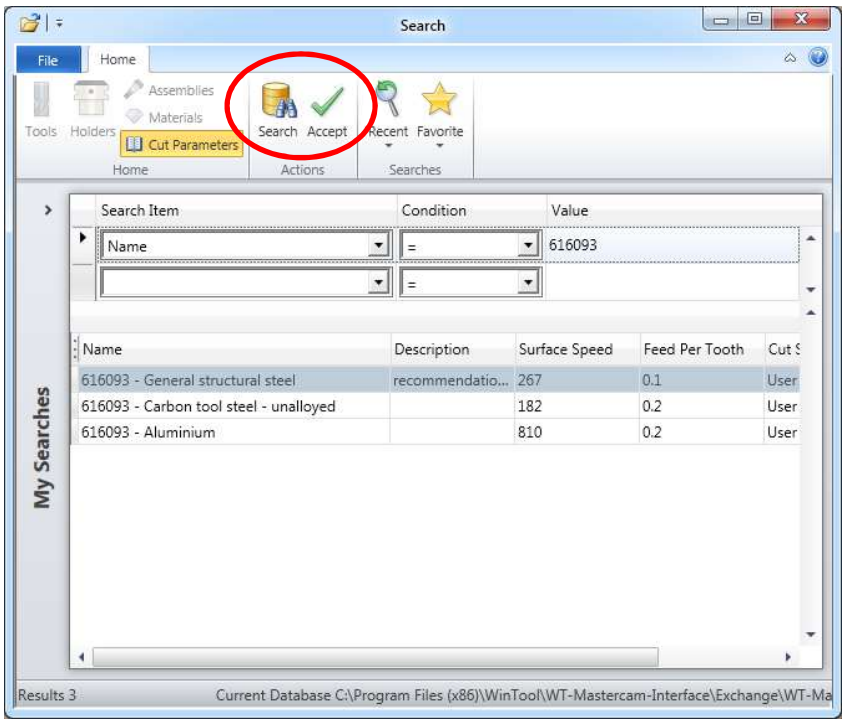


Figure 22 Enter tool assembly and click on Search icon

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:

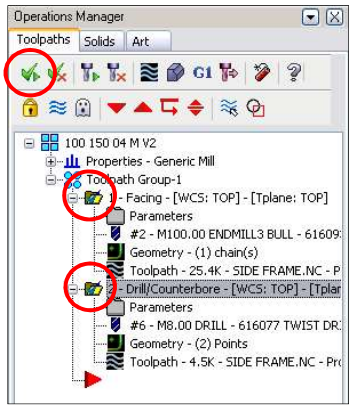


Figure 23 Select tools to be transferred

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

Edit the tool list header information:

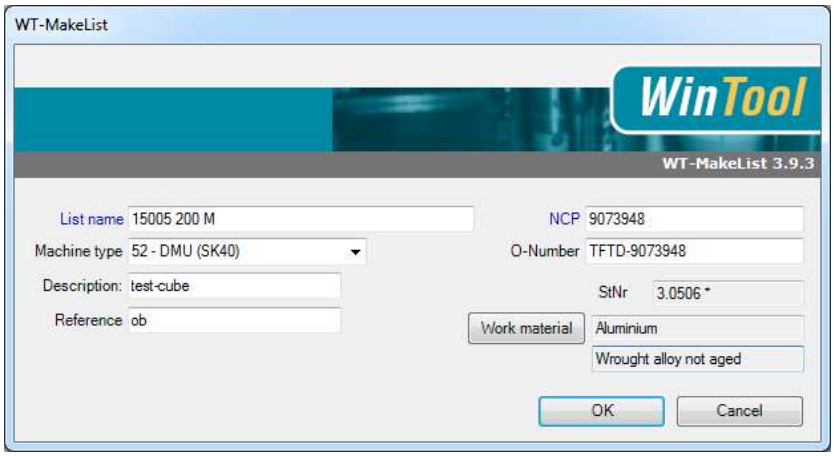


Figure 24 Edit List name in WT-MakeList

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 25 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:

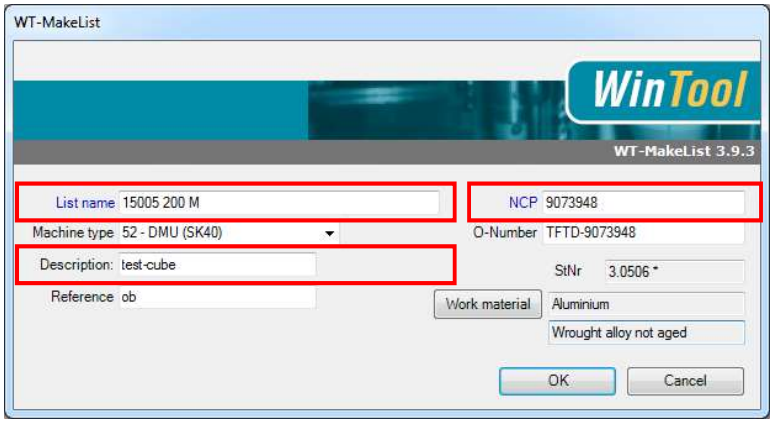
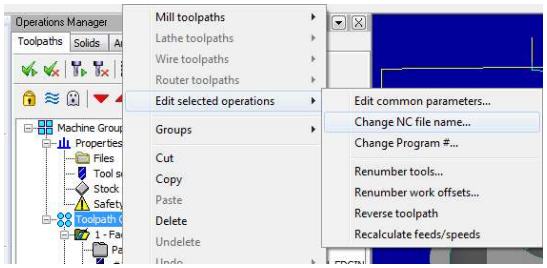


Figure 26 Entry fields will be filled in automatically

List 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".

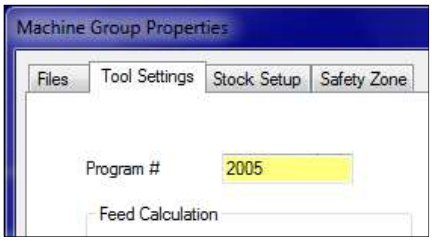
Figure 27 Change NC file name

Description

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

Figure 28 Tab folder Files - Machine data

NCP



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

Figure 29 Enter a program number in tab folder ´Tool Settings´

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).

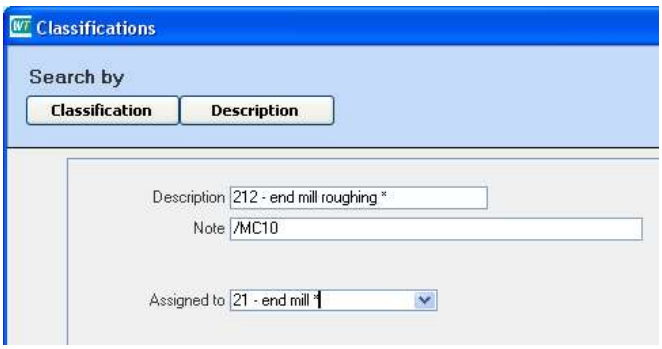


Figure 30 Every Descriptions has an own Note

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.

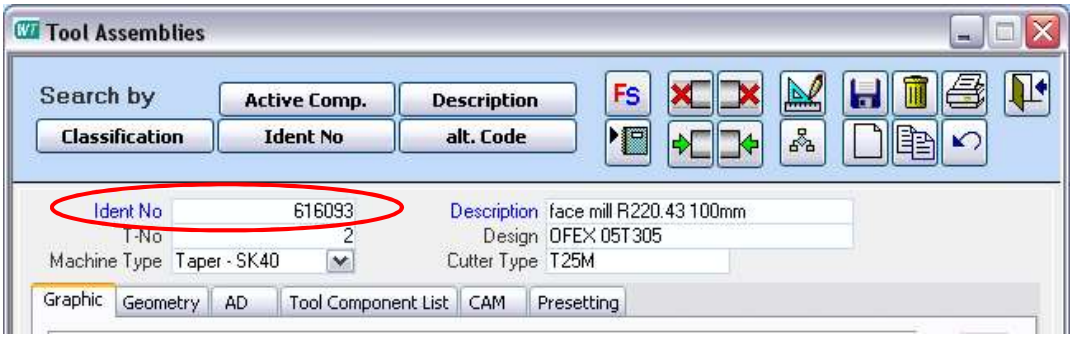


Figure 31 Enter Ident Number and Tool Description

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".

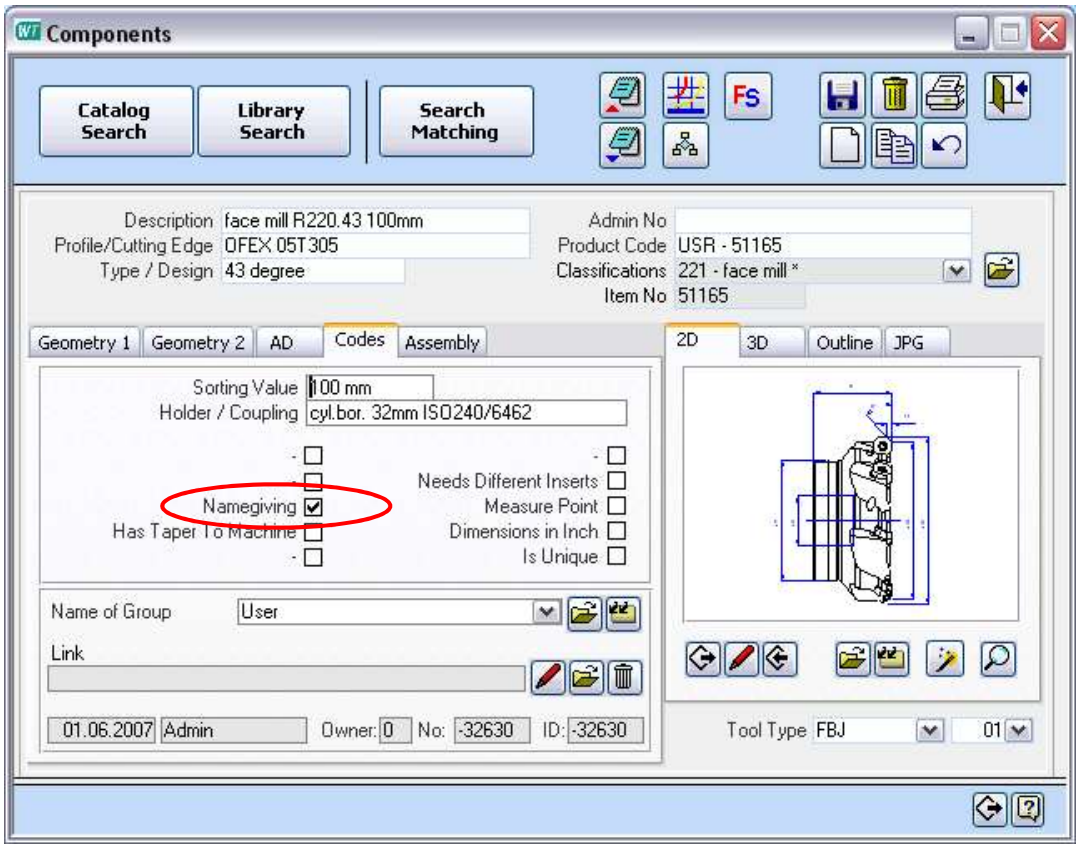


Figure 32 Namegiving must be marked

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 marked with the light green symbol).

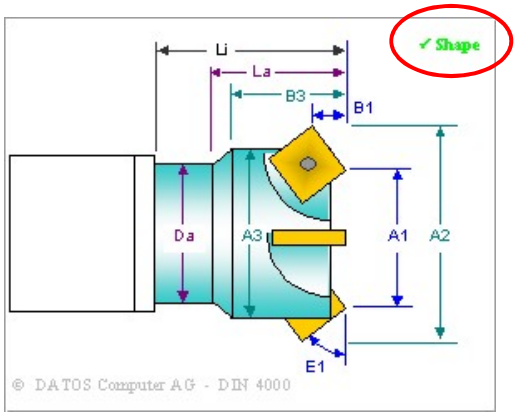


Figure 33 Select Shape for Generator

WinTool can generate for all regular tools 3D representations as long as 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.

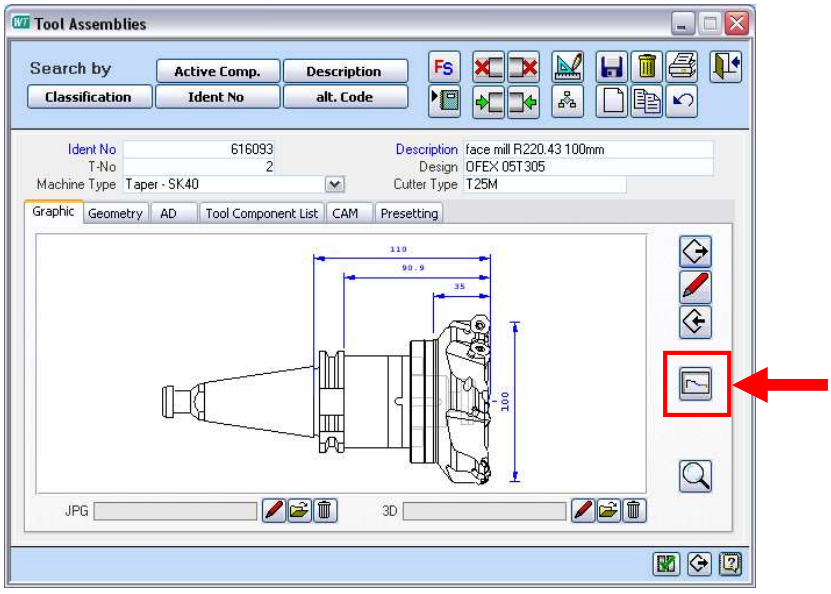


Figure 34 Verify Tool Contour by click on the icon

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 (eg. `616099.dxf`). Then flag the *WinTool* tool assembly in the folder tab CAM:

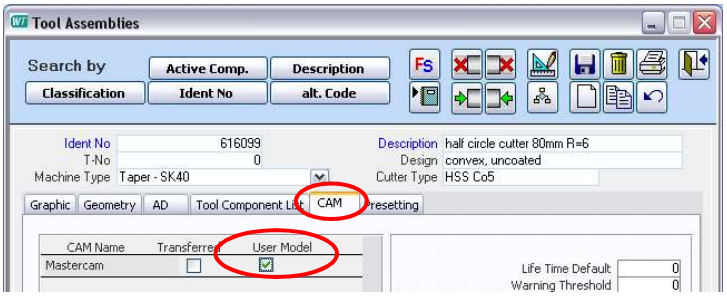


Figure 35 Select User Model in the menu "CAM"

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.

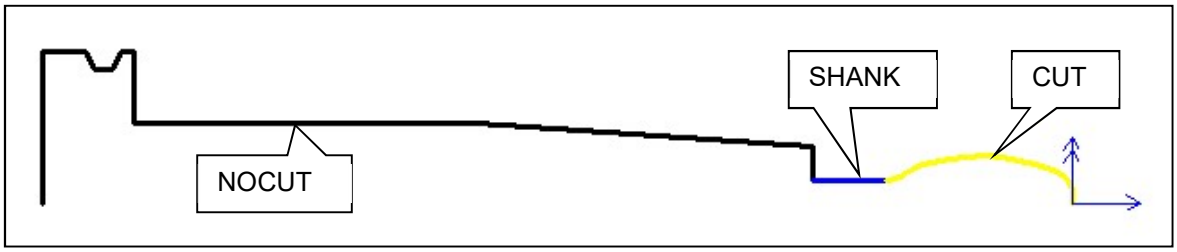


Figure 36 Create a Tool Contour DXF

- 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".

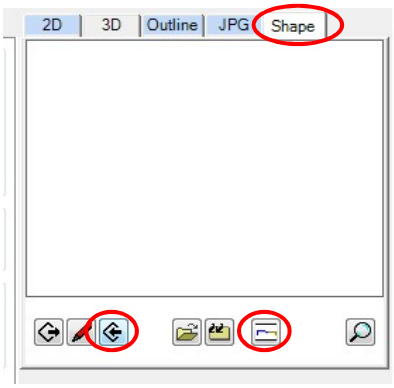


Figure 37 Instruction to activate the tab "Shape"

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 supports also special cutting tools that are axially symmetric. This is useful if no suited 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)

Software Structure

Software-Modules and Data-Exchange

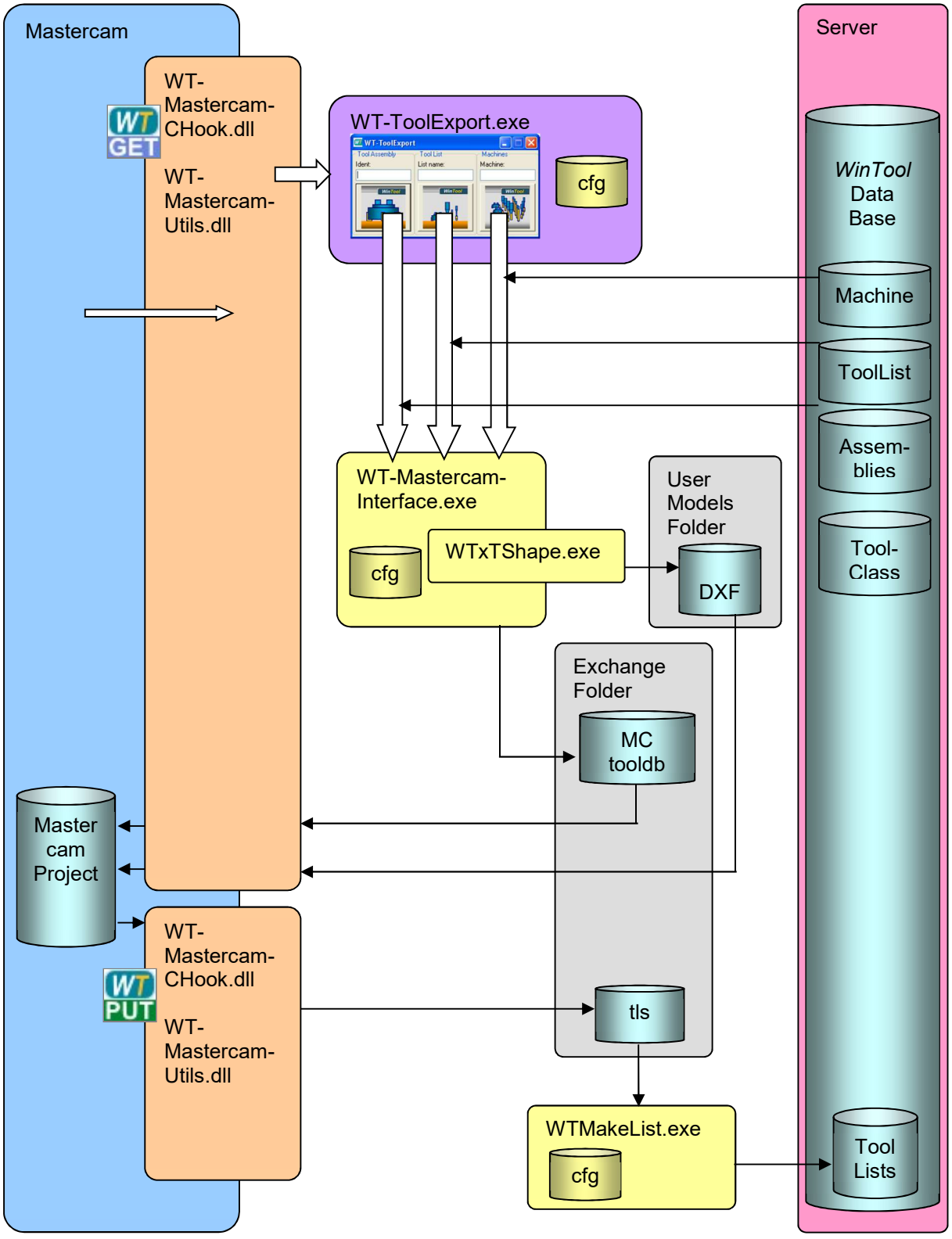


Figure 38 Software structure

WinTool-Mastercam Data Integration

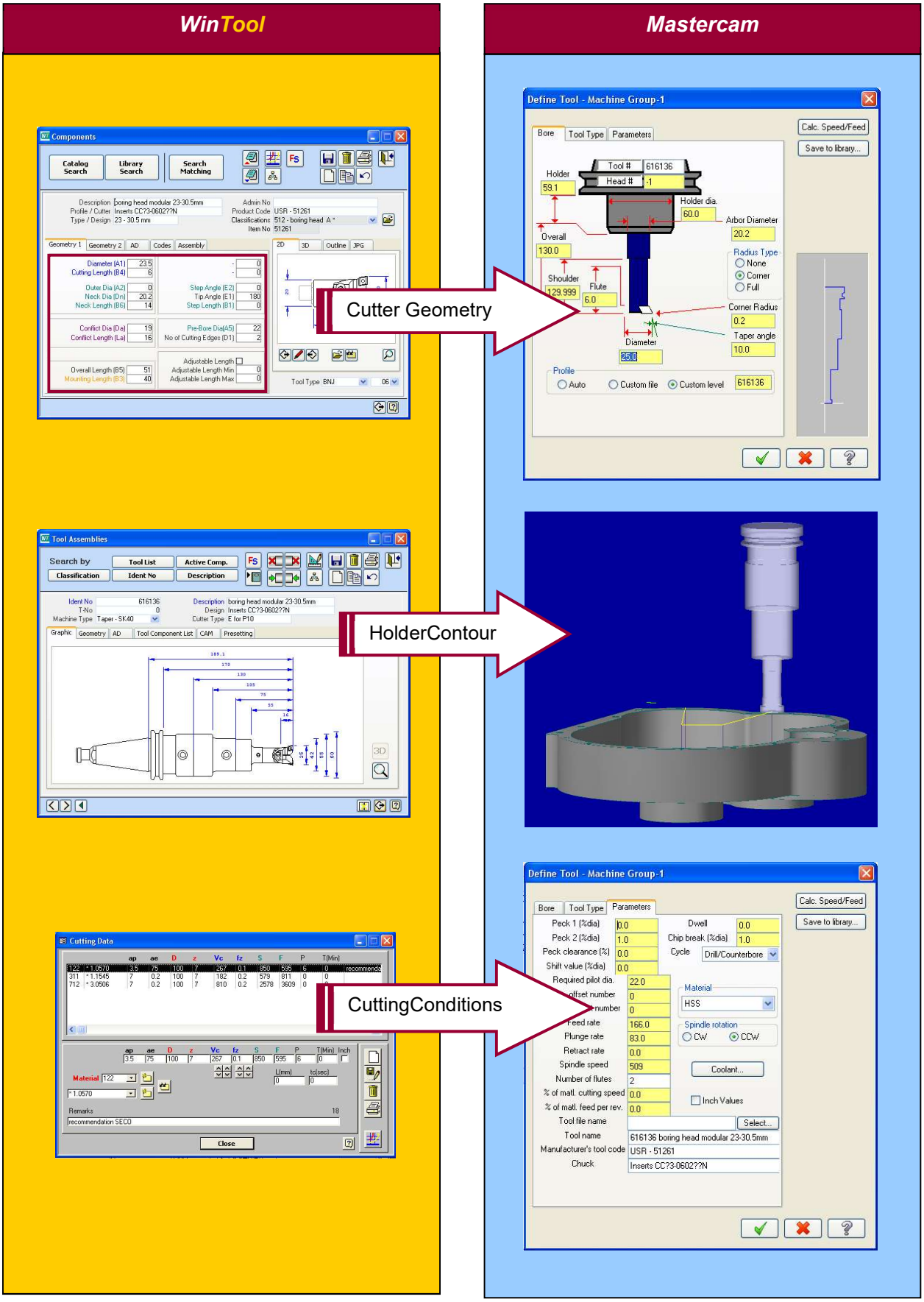
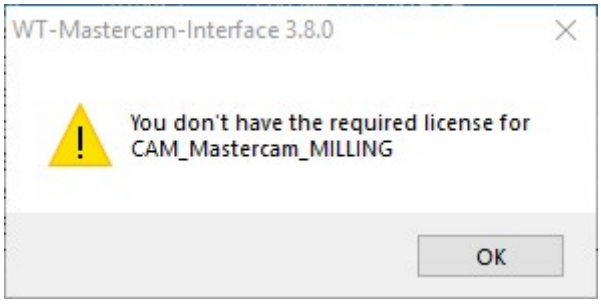


Figure 39 Integrate WinTool in Mastercam

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:

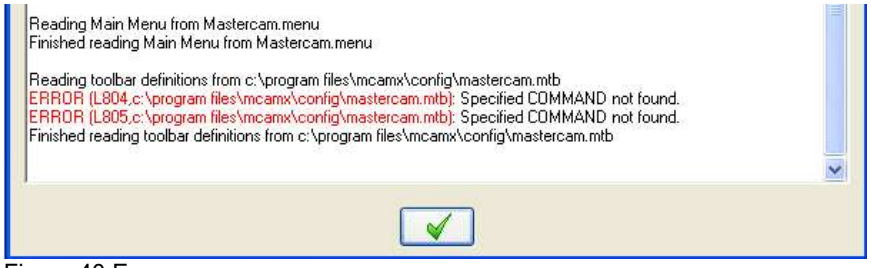


Figure 40 Error message

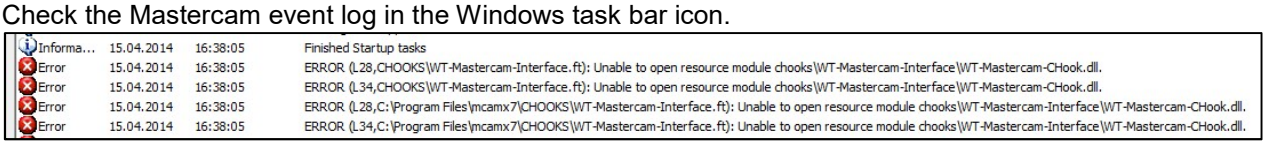
The message is caused from an emptied out *WinTool* toolbar (GET and PUT have been removed) in Mastercam. However, the uninstall program (Windows Add/Remove Program) does not delete the empty toolbar in Mastercam.

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.



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 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 41 Edit the control definition

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

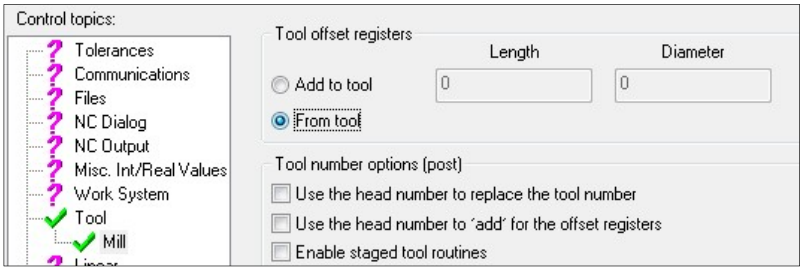


Figure 42 Edit the control definitions

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". 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".
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-2018.exe

64-bit Windows registry key:
HKEY_LOCAL_MACHINE\SOFTWARE\Wow6432Node\Microsoft\Windows\CurrentVersion\App
Paths\WT-Mastercam-Interface-2018.exe
```

(Parameter is set during installation)

Current user

```
HKEY_CURRENT_USER\Software\WinTool\WT-Mastercam-Interface
OutputPath = C:\Users\Public\Documents\WT-Mastercam-Interface\Exchange
```

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

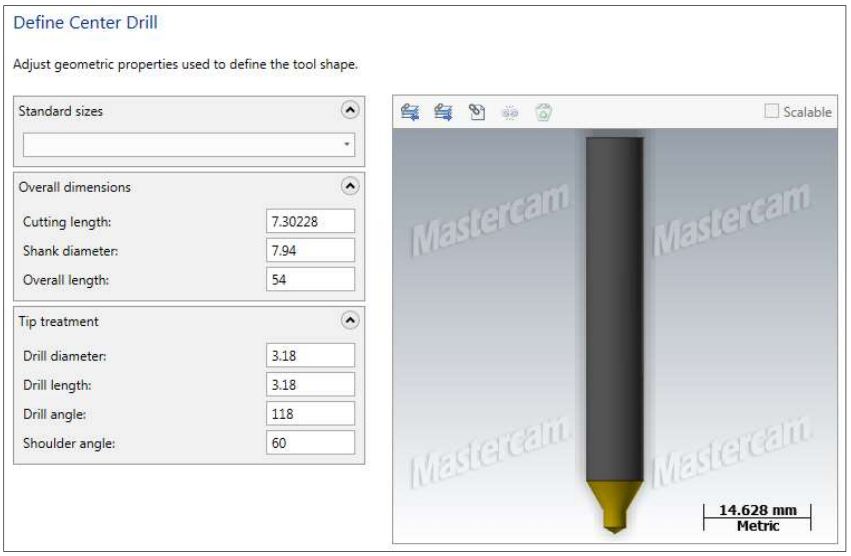
```
HKEY_CURRENT_USER\Software\WinTool\WT-Mastercam-Interface
UserModelsPath = C:\Users\Public\Documents\WT-Mastercam-Interface\UserModels
```

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

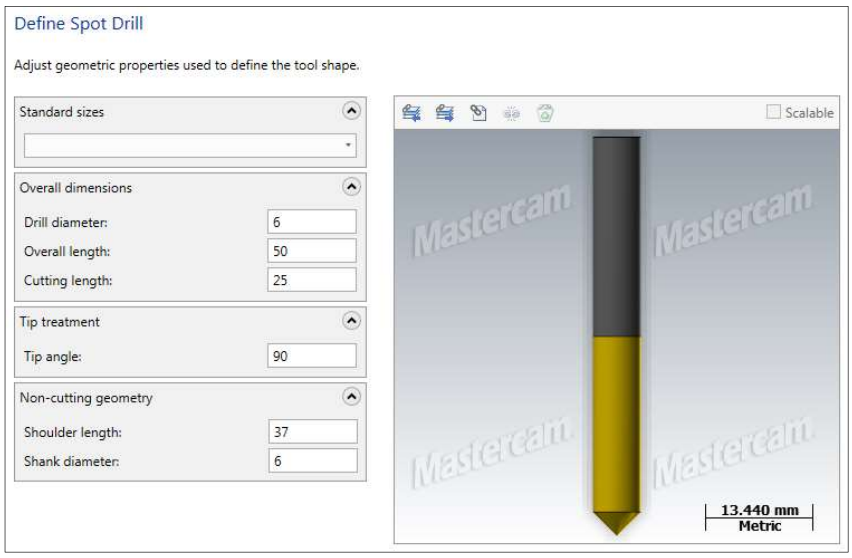
Supported Mastercam Tool Types

Milling Tools

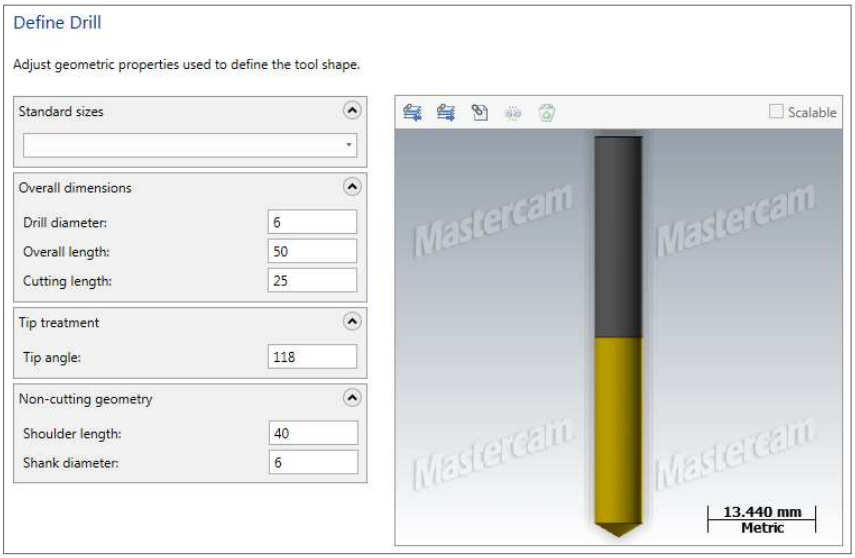
Center Drill (/MC1)



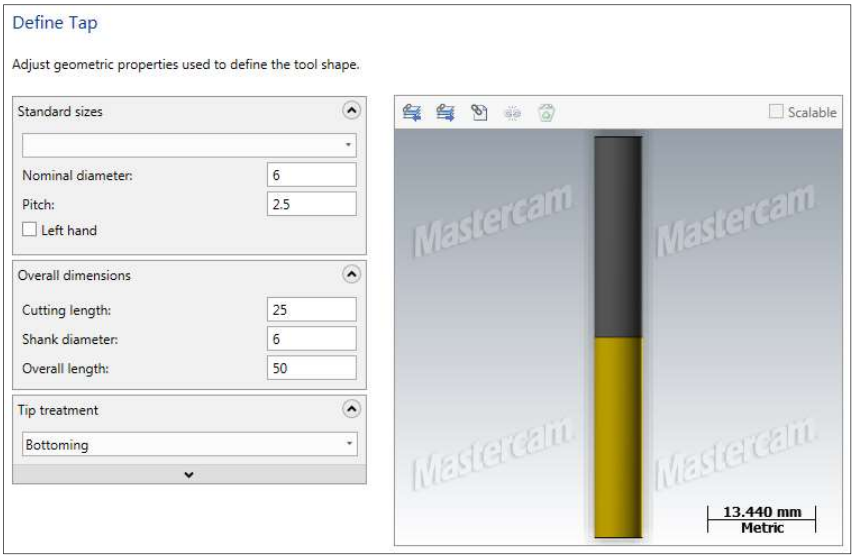
Spot Drill (/MC2)



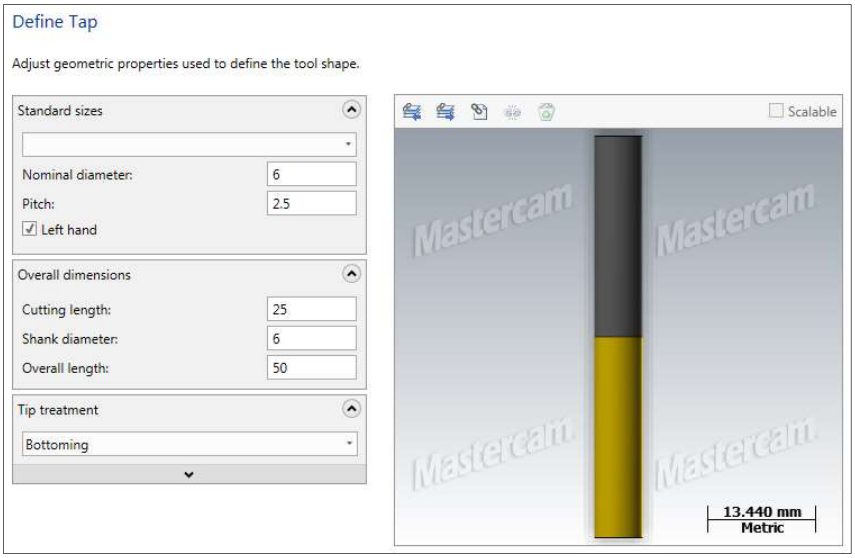
Drill (/MC3)



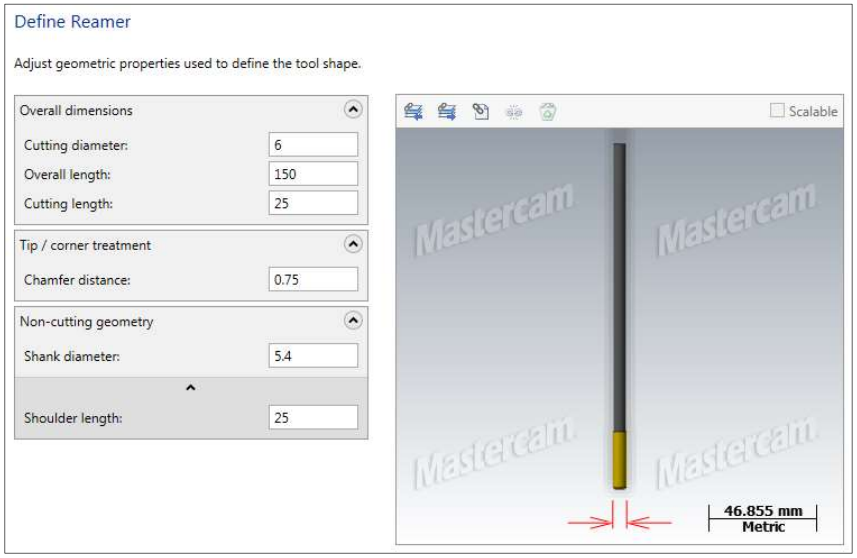
Tap RH (/MC4)



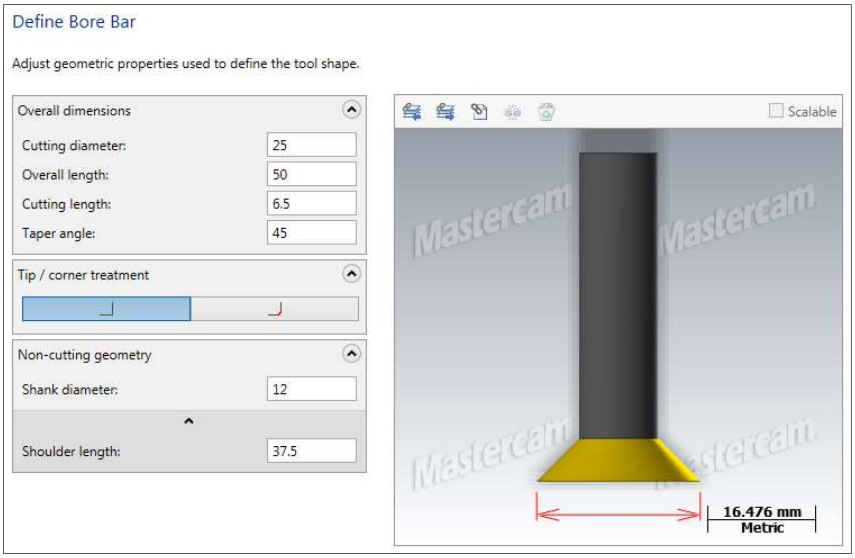
Tap LH (/MC5)



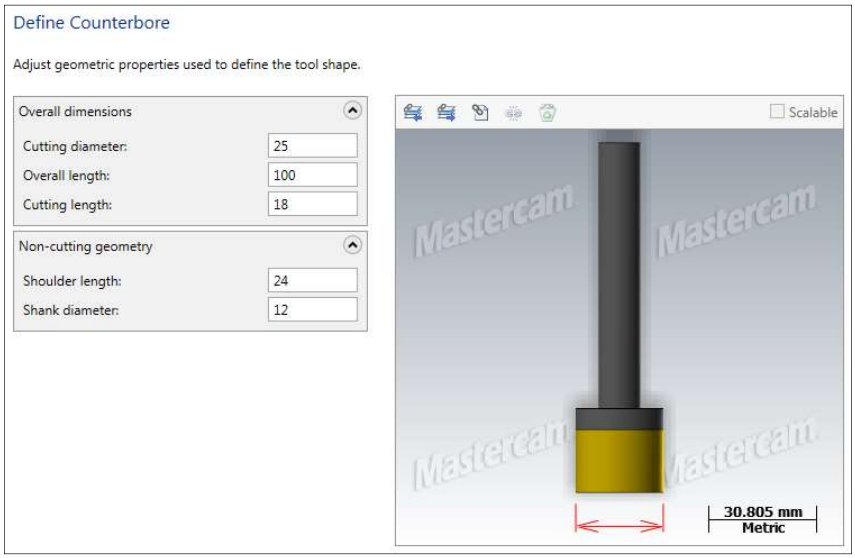
Reamer (/MC6)



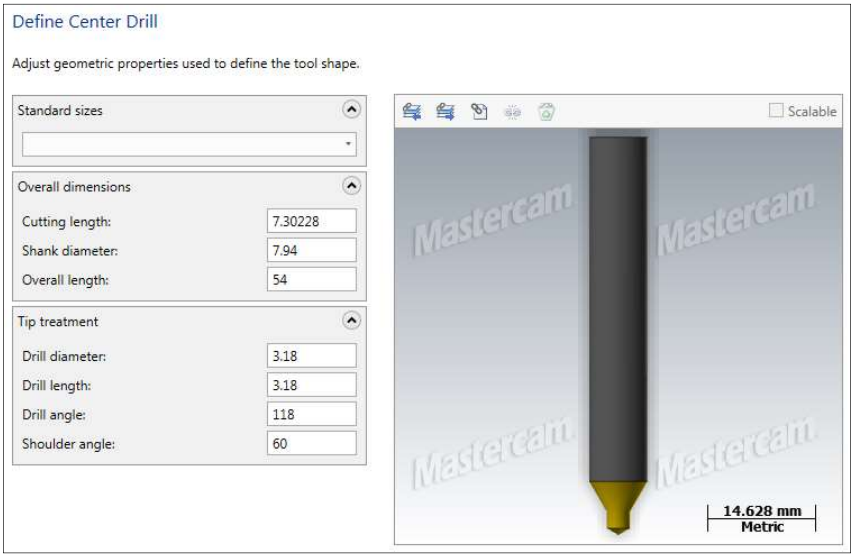
Bore Bar (/MC7)



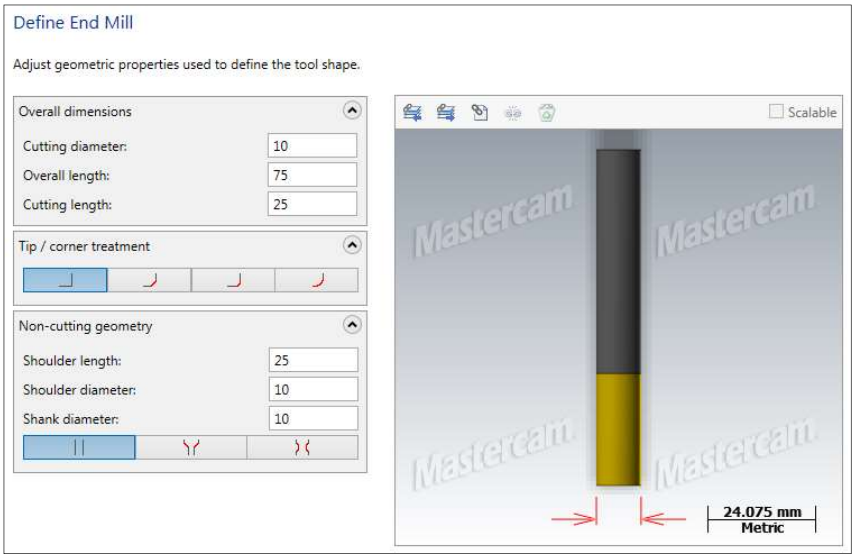
Counter Bore (/MC8)



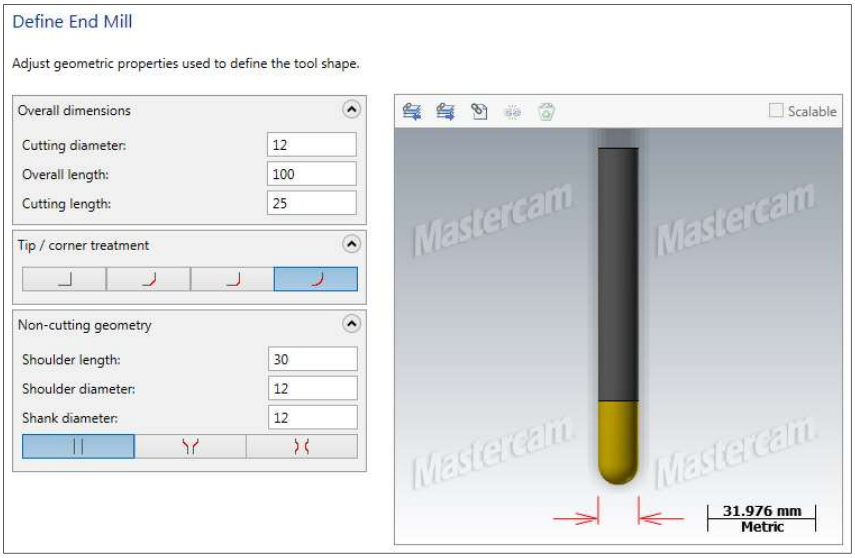
Counter Sink (/MC9)



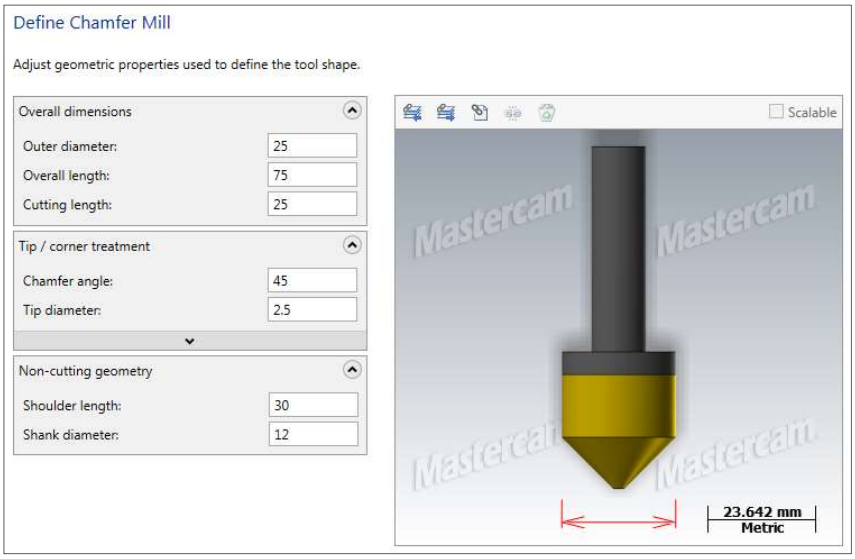
End Mill (/MC10)



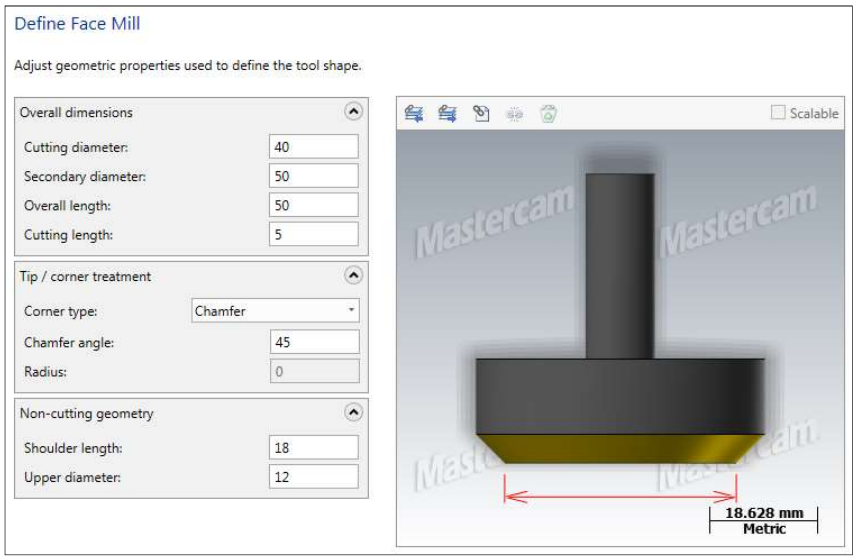
Sphere Mill (/MC11)



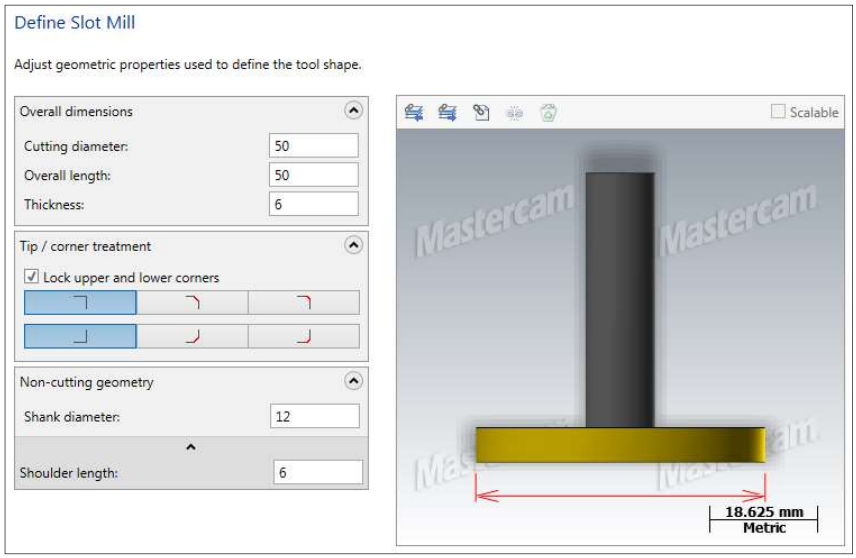
Chamfer Mill (/MC12)



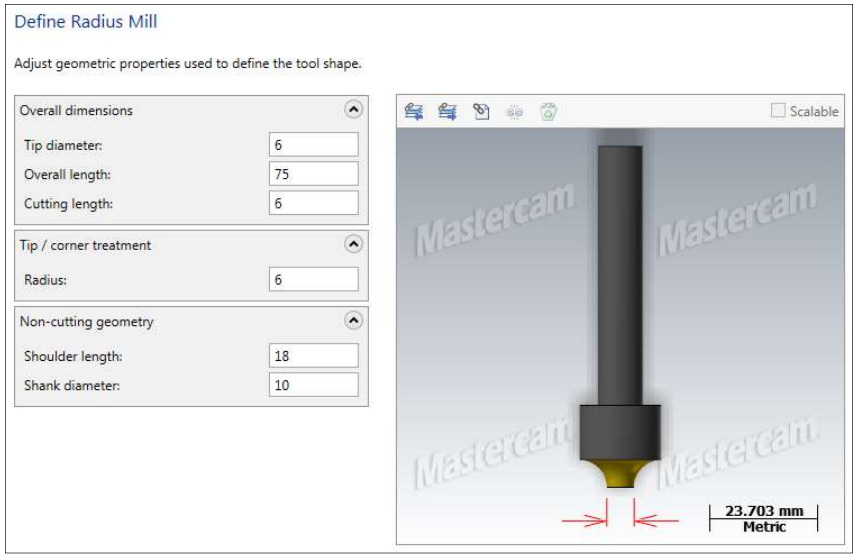
Face Mill (/MC13)



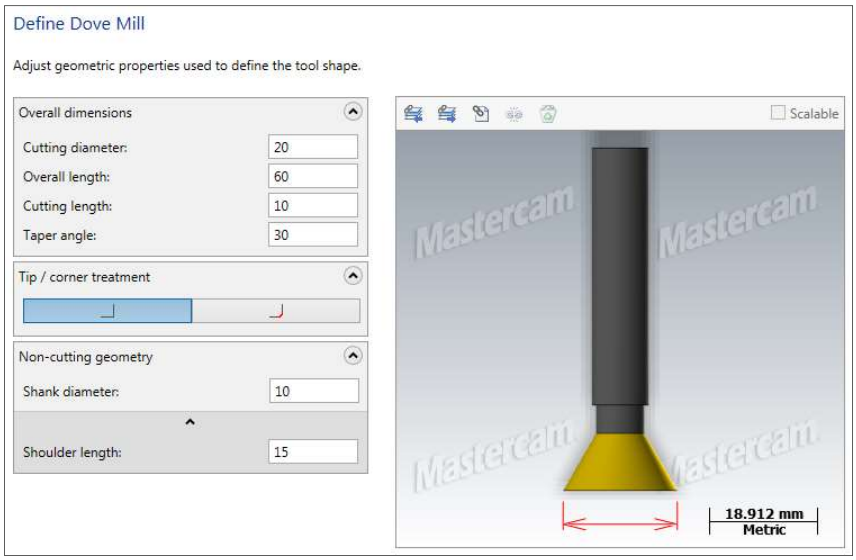
Slot Mill (/MC14)



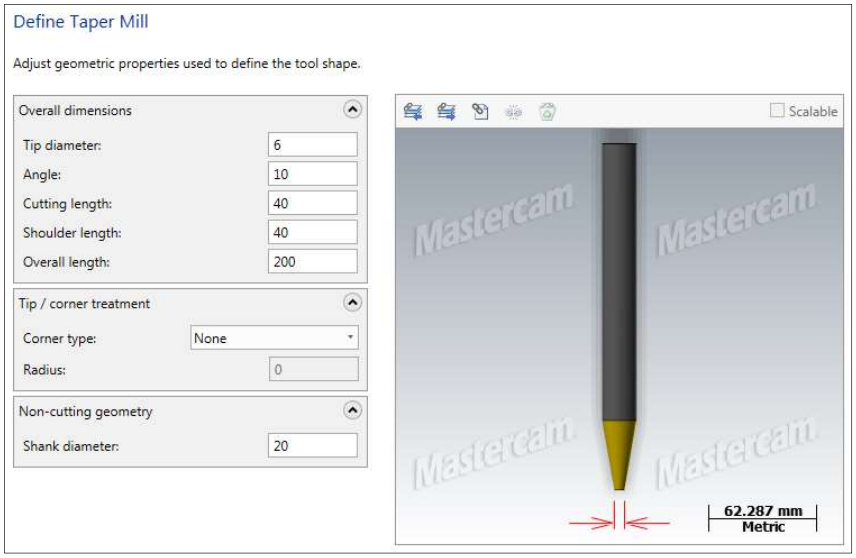
Radius Mill (/MC15)



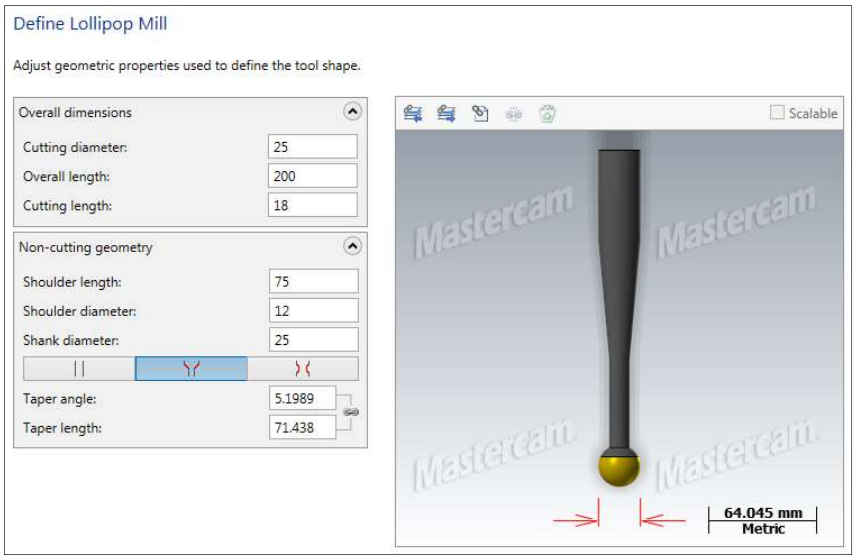
Dove Mill (/MC16)



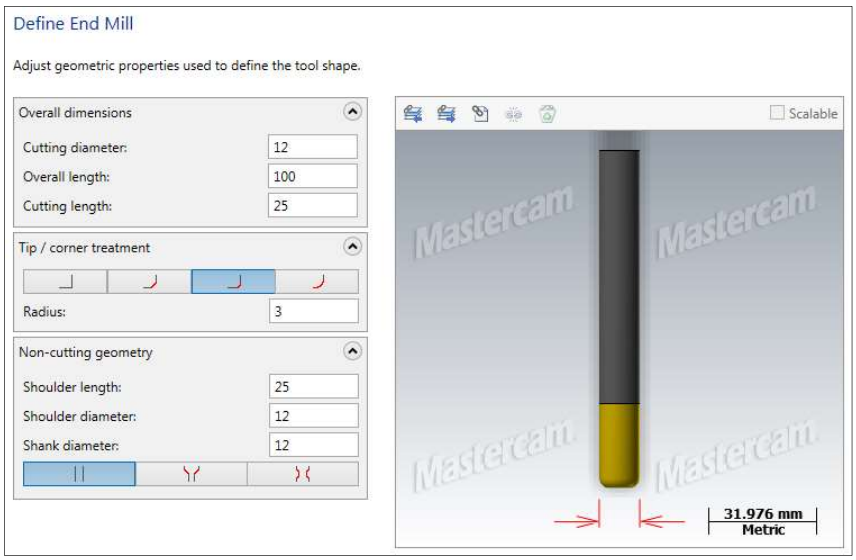
Taper Mill (/MC17)



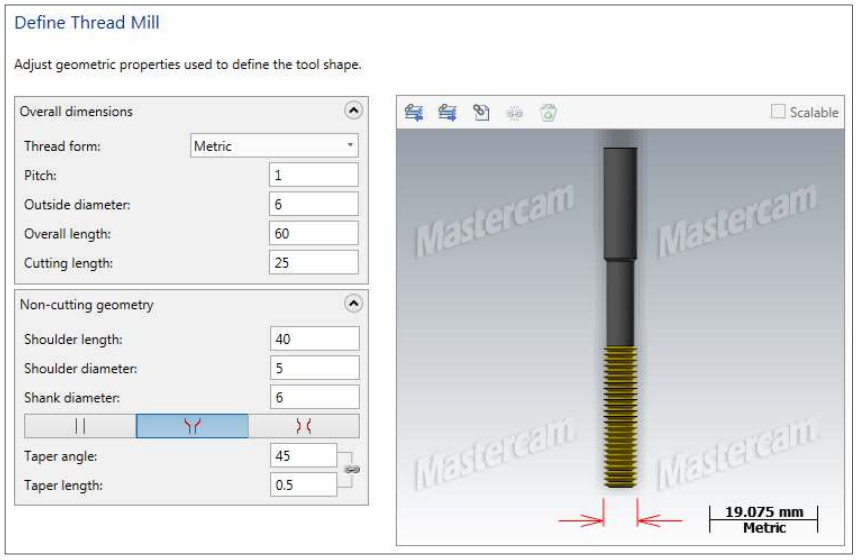
Lollipop Mill (/MC18)



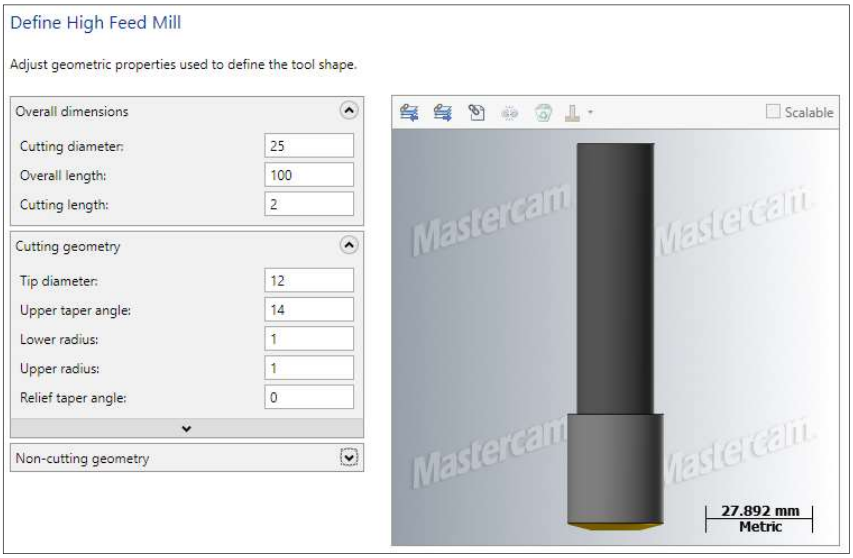
Bull Mill (/MC19)



Thread Mill (/MC24)

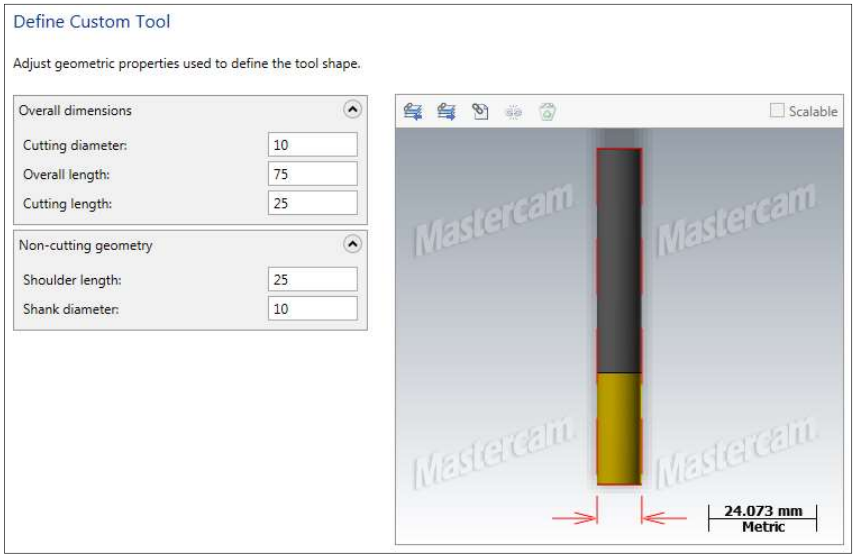


High Feed Mill (/MC26)



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
- Bradpt Drill
- Barrel Mill (Incompatible definition)
- Grooving tools
- Threading tools

History

4.2.1

- Support for tool assemblies containing multiple same parts and different measure points
- Fixed issue with sort field capitalization
- Allowing import with databases that contain classes with empty comment

4.2

- Compatible with Mastercam 2022
- Compatible with WinTool 2021.1.1
- Improved Tool Import to Tool Manager from Mastercam
- General Tool Import Improvement

4.1

- Improvement on Cutting Conditions Import
- Multiple improvements on Tool Import regarding correct values for Mastercam 2020 and 2021
- Correction of PUT button to export tools into WinTool correctly
- Compatible with WinTool 2020.3

4.0

- New technology called ToolNet replacing former CHook
- Improved general shape Import into Mastercam
- Improved General Turning and Boring Bar Import
- General Turning Tool Parameter improvements
- Added new Configuration option “ImportPitchValueFromComponent”
- Compatible with Mastercam 2021
- Compatible with WinTool 2020.2.1

3.8

- Improved Tool Name / Assembly Name settings handling (see page 13)

3.7 / 3.7.1

- Added Tool name / Assembly name functionality
- Added optional Lathe Tools functionality (only Mastercam 2018)
- Compatible with Mastercam 2018 / 2019

3.6

- Compatible with Mastercam 2018
- Possibility to install the Interface for Mastercam 2017 along with the Interface for Mastercam 2018

3.5

Mastercam 2017

- Corrected Mastercam error when tool assembly contour contains invalid geometries
- Face mill: Corrected error "Tip angle/tool diameter don't intersect"
- Settings: Added customizable mapping of WinTool to Mastercam coolant types
- Importing coolant from cutting condition instead of tool assembly
- Settings: Added switch to enable import of "Rough Step %" values
- Adjusted import of cutting condition name into Mastercam tool database

Mastercam 2017 and X9

- Changed tool name to format "Tool.IdentNo Tool.Description" for post processors

3.4

- Compatible with Mastercam 2017
- Supporting new Mastercam 2017 tool type "High Feed Mill"

- Corrected holder length (parameter 20007:13) in NCI output

3.3

- Compatible with *WinTool* 2011-2015
- Compatible with Mastercam X9
- Supporting new X9 features:
  - "Thread mill" tool type import
  - Importing arc segments to holder profile
  - Importing neck/shank parameters to end mill tool types
- Improved tool shape import
- Adjusted "Slot Mill" and "Dove Mill" import
- Corrected helix type import
- Adjusted tool type names and tool type order in selection window
- Changed tool name and tool description import
- Importing diameter to cut parameters "Min/Max Diameter" in Mastercam tool library

3.2

- Compatible with Mastercam X8
- Mastercam X8: Importing tool assembly coolant type
- Corrected import of chamfered mill with mask "04-09"
- Corrected "Mastercam is missing" error during uninstallation of X7 interface
- Changed default installation path in X7 setup to allow simultaneous installations of X7 and X8 interface

3.1

- Compatible with Mastercam X7 SP1, SP2, MU1, MU2
- Compatible with *WinTool* 2011-2014
- Separated program files and user data
- Corrected transfer of diameter and length offset numbers greater than 32767
- Included newest version of WT-MakeList (see detailed changes in WT-MakeList manual)
- Included newest version of WT-ToolExport:
  - Saving selection state of "preferred only" filter
  - Improved readability with high DPI settings
  - Compatible with *WinTool* 2014
- Single tool assembly import: Transferring ident-no for t-no if "T-No=Ident No" is activated in the machine type
- Tool assembly head # fixed to -1

3.0.1

- Included newest version of WT-MakeList due to issue with SQL Server

3.0

- Compatible with *WinTool* 2013
- Support for Mastercam X7 and new Mastercam tool database format
  - Tools are imported as Mastercam tool assembly for improved simulation and collision check
  - All cutting conditions for work materials are imported with tools
- Corrected import of D and H no. of tool list tools
- Corrected import of T-No. if a tool assembly is more than once in a tool list
- Corrected import of shoulder angle of center drill (/MC1)

- Corrected import of chamfer mill (/MC12)
- Included newest version of WT-MakeList (see detailed changes in WT-MakeList manual)
- Total operation time of used tool assemblies are saved in *WinTool* tool list

2.7

- Compatible with *WinTool* 2012
- Added support for ring groove mill
- Corrected import of drills with tip angle = 0
- Included newest version of WT-ToolExport:
  - Resizable search windows
  - Compatible with *WinTool* 2012

2.6

- Better process to update tools with same T-Nr. in Tool Manager and Operations Manager
- Support for *WinTool* 2011 and Mastercam X6 32/64 Bit
- Included newest versions of WT-ToolExport and WT-MakeList module
- WT-ToolExport: Start-up time with large databases is quicker
- Added tool type "Ignore" (/MC00) for tool assemblies that must be ignored on transfer
- Improved error handling

2.5

- Added new WT-ToolExport module
- Added WT-Mastercam-Interface configuration window
- Minor update of cutting condition selection window
- Fixed issue with installer

2.4

- Support of Mastercam X5 and X4 (updated)
- Cutting condition selection window during tool import
- Pre-selection of cutting conditions after import of first tool
- Mastercam tool type mapping can be done during transfer if missing
- Transferring tool holder description to field “chuck”
- Outer Dia for MC13 is transferred correctly
- WT-MakeList 3.7 with better selection method of Material and O-Number
- Improved error messages
- Updated transfer file
- Updated manual
- 30 day trial license options
- *WinTool* Professional must be started when using the WT-Mastercam-Interface

2.3 - 3rd Release

- Support for *WinTool* 2009 and *WinTool* 2010
  (updated ToolExport and MakeList)

2.3 - 2nd Release

- Support for Mastercam X4

2.3

- Manual completely reworked

- Cutting conditions no longer negative, direction corrected
- ae, ap values supported
- Tool number management and messages optimized
- Level numbers for tool contours >100’000’000
- Update to WT-Mastercam-Interface 2.3.2.5721
- Mastercam contour display always correct now
- Better support of custom cutter DXF
- Mastercam values exported to WT-MakeList window
- Support for Mastercam X3

2.2

- Release for Mastercam X2

List of Figures

Figure 1 Steps to the WinTool Mastercam Installation....8
Figure 2 Choose commands and add to Tool Tabs....8
Figure 3 Mastercam Tool Settings....9
Figure 4 Choose Mastercam in User Settings....9
Figure 5 Tool Assemblies in CAM....10
Figure 6 WT-MasterCAM-Interface configuration....11
Figure 7 WT-MasterCAM-Interface configuration for naming conventions....13
Figure 8 Main Placeholders....14
Figure 9 Change the WT-database....16
Figure 10 Select Tool assembly in the menubar „TOOLPATHS“....17
Figure 9 Search a tool assembly class from list....17
Figure 12 Assembly cutting conditions....18
Figure 13 Select a Mastercam tool type from list....18
Figure 14 3D Toolpaths model....19
Figure 15 2D Toolpaths model....19
Figure 14 Search to transfer the Tool List....20
Figure 15 List of all imported tools....20
Figure 16 List of T-No for Tool Assemblies....21
Figure 19 Check the T-No.....21
Figure 18 Notification - Tool number....22
Figure 19 Instruction to search for cut parameters....23
Figure 20 Enter tool assembly and click on Search icon....23
Figure 21 Select tools to be transferred....24
Figure 24 Edit List name in WT-MakeList....24
Figure 25 Notification....24
Figure 24 Entry fields will be filled in automatically....25
Figure 25 Change NC file name....25
Figure 26 Tab folder Files - Machine data....25
Figure 27 Enter a porgram number in tab folder ´Tool Settings´....25
Figure 30 Every Descriptions has an own Note....26
Figure 29 Enter Ident Number and Tool Description....27
Figure 30 Namegiving must be marked....27
Figure 31 Select Shape for Generator....28
Figure 32 Verify Tool Contour by click on the icon....28
Figure 35 Select User Model in the menu "CAM"....29
Figure 36 Create a Tool Contour DXF....29
Figure 35 Instruction to activate the tab "Shape"....30
Figure 36 Software structure....32
Figure 39 Integrate WinTool in Mastercam....33
Figure 40 Error message....34
Figure 41 Edit the control definition....35
Figure 40 Edit the control definitions....35