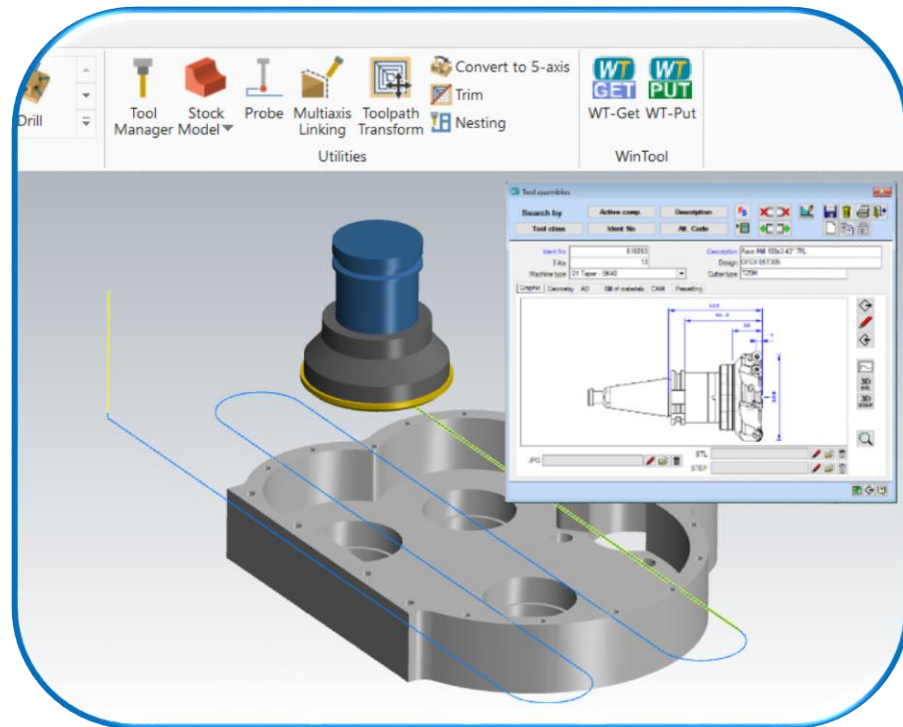


# WT-Mastercam Interface



Manual

**WinTool Interface 3.5 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 X9/2017

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

## Contents

Summary .....	4
Job .....	4
Requirements .....	4
Supported Tool Types .....	4
Limitations .....	4
Copyright .....	4
Installation .....	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.....	6
Mastercam X9 .....	7
Configuration .....	8
Mastercam Settings.....	8
WinTool Settings .....	8
Interface Settings .....	10
Configuration Window .....	10
Output Path .....	10
UserModels Path.....	10
Selection of cutting conditions .....	11
Tool List Exchange Path .....	11
Interface Settings for Mastercam 2017 and newer .....	12
Coolant0-Coolant9 .....	12
Importing Rough Step Percentage Values.....	12
Getting Started .....	13
Sample Database .....	13
Import Tools .....	14
Tool Numbers.....	18
T-Number Assignment .....	18
WinTool Tool Assembly T-No .....	18
WinTool Tool List T-No .....	18
Duplicate T-Numbers .....	19
Coolant Import.....	19
Using Cutting Conditions.....	20
Export Tool List to WinTool .....	21
Step-by-Step .....	21

Mastercam data fields transfer .....	22
Preparing Tool Data in WinTool .....	23
User Classification.....	23
Machine Configuration .....	23
Tool ID and Name .....	24
Regular Tools .....	25
Special Tool Assemblies .....	26
Managing Special Tool Assemblies .....	26
Create a Special Tool Assemblies Contour DXF .....	26
Special Components .....	27
Managing Special Component Contours .....	27
Special Cutters .....	28
Managing Special Cutters .....	28
Create a Custom Cutter Contour .....	28
Cycle Type / Usage (C7) .....	28
Software Structure.....	29
Software-Modules and Data-Exchange .....	29
WinTool-Mastercam Data Integration .....	30
Known Issues .....	31
Uninstall Error-Message in Mastercam .....	31
No PUT and GET Buttons Available .....	31
Incorrect Diameter and Length Correction Numbers .....	31
Annex .....	32
Configuration File Parameters .....	32
General Information .....	32
WT-Mastercam-Interface.cfg.....	32
Windows Registry values .....	33
Local Machine .....	33
Current user .....	33
Supported Mastercam Tool Types .....	34
Not Supported Mastercam Tool Types .....	41
History .....	42
List of Figures .....	45

## **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 X9 or 2017.

### **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.

### **Limitations**

Turning and grooving tools are currently not supported.

### **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](mailto:info@wintool.com)

## Installation

### 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	<code>[Public Documents]\WT-Mastercam-Interface\UserModels</code>
Default location of Exchange folder	<code>[Public Documents]\WT-Mastercam-Interface\Exchange</code>
Configuration files: WT-Mastercam-Interface.cfg WT-MakeList.cfg WT-ToolExport.cfg	<code>[Public Documents]\WT-Mastercam-Interface</code>

### 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 10) 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 on with administrator rights to install the software on your PC.

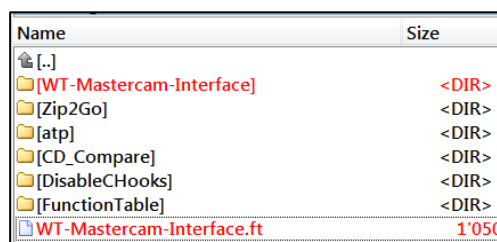
Install *WinTool* Professional first before installing the WT-Mastercam-Interface.

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-2017`

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

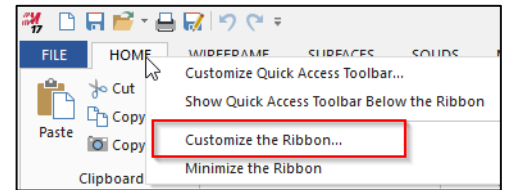


Name	Size
[..]	
[WT-Mastercam-Interface]	<DIR>
[Zip2Go]	<DIR>
[atp]	<DIR>
[CD_Compare]	<DIR>
[DisableCHooks]	<DIR>
[FunctionTable]	<DIR>
WT-Mastercam-Interface.ft	1'050

Setup the new WT-Mastercam Interface toolbar as follows:

## Mastercam 2017

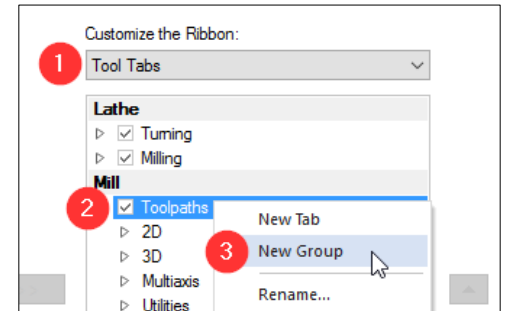
Startup Mastercam. Right-click on the tab "HOME" and select "Customize the Ribbon..."



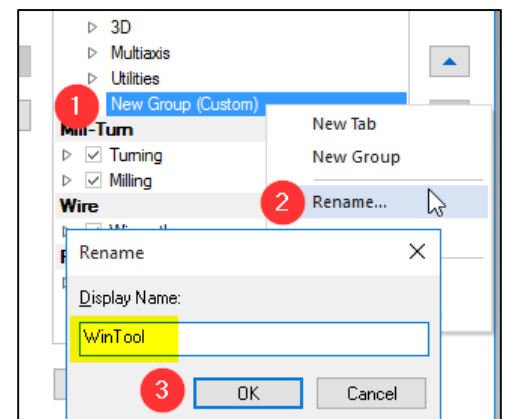
On the right side of the window select the "Tool Tabs" from the list.

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.



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 >>":

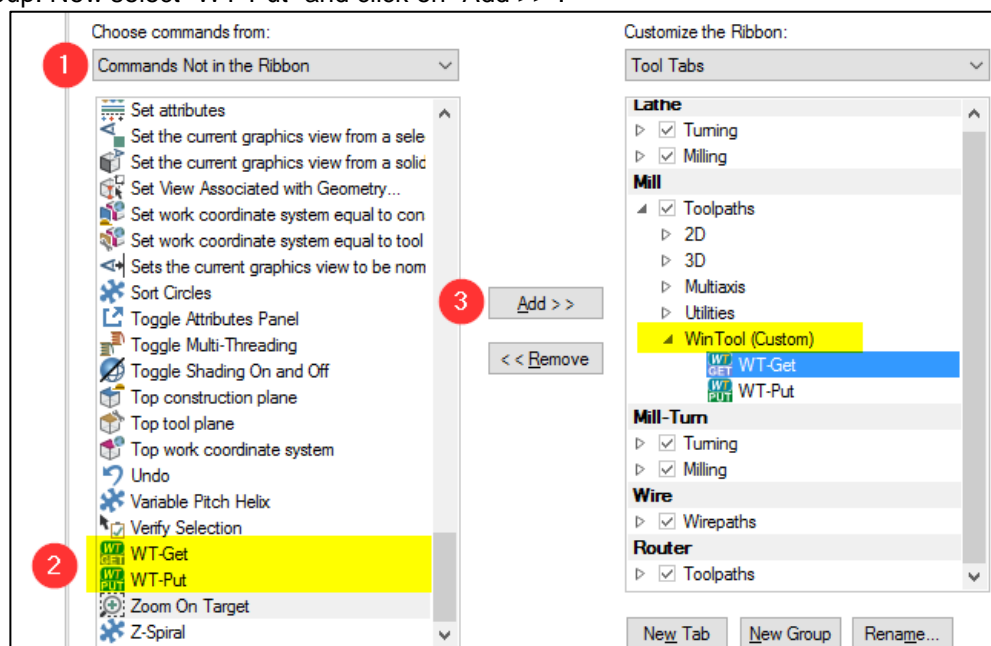
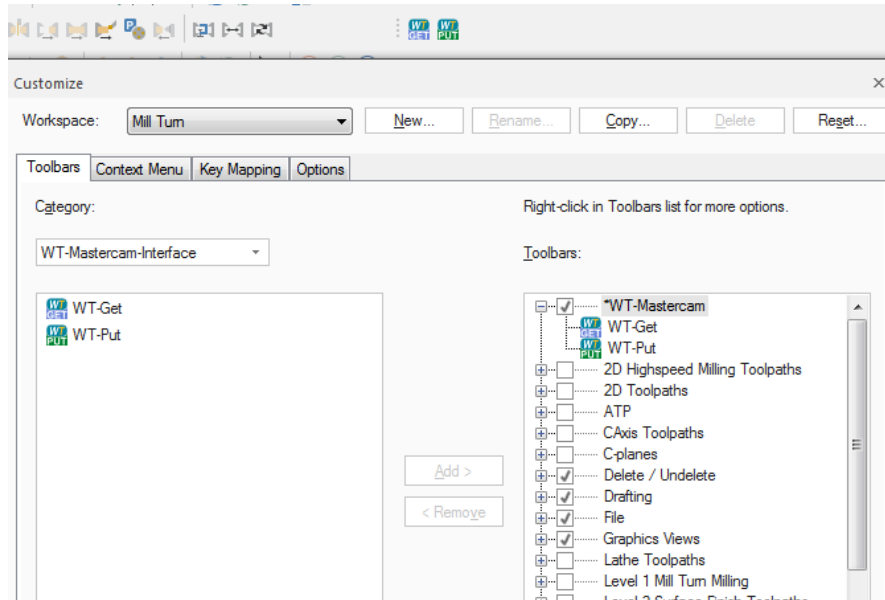


Figure 1 Steps to the WinTool Mastercam Installation

Figure 2 Choose commands and add to Tool Tabs

## Mastercam X9

Startup Mastercam, then activate the functions *WinTool* GET and *WinTool* PUT by creating a new toolbar using "Settings" > "Customize". Thereafter, select the workspace in which you want place the toolbar, right-click in "Toolbars" and click "New Toolbar". Drag the toolbar to the ribbon bar. Then select the category "WT-Mastercam-Interface" and drag the GET and PUT icons to the toolbar.



Select "OK". The WT-Mastercam-Interface is now installed with default settings.

## Configuration

### Mastercam Settings

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

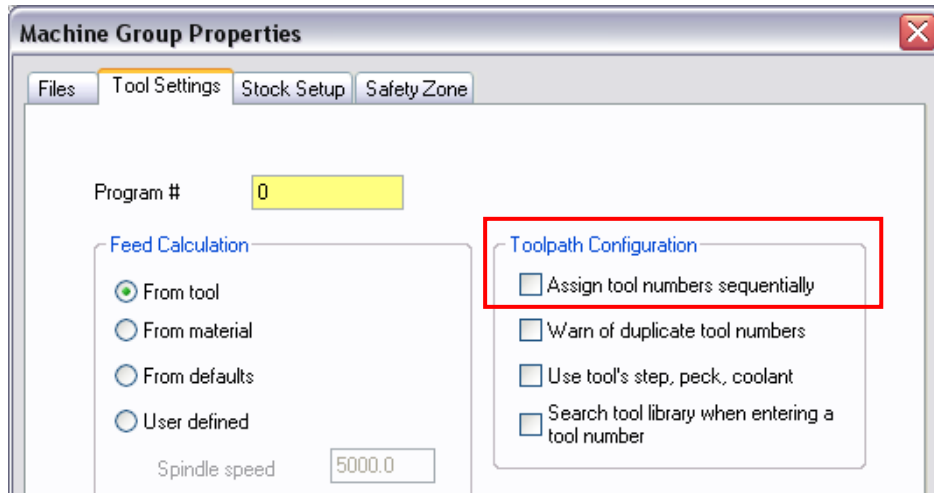


Figure 3 Mastercam Tool Settings

### 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:

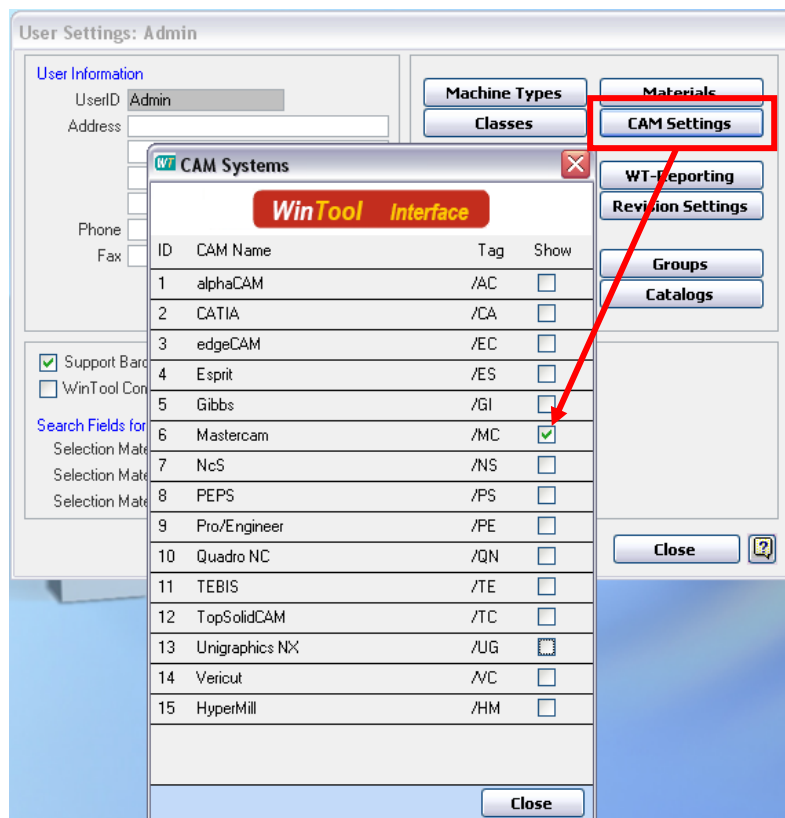
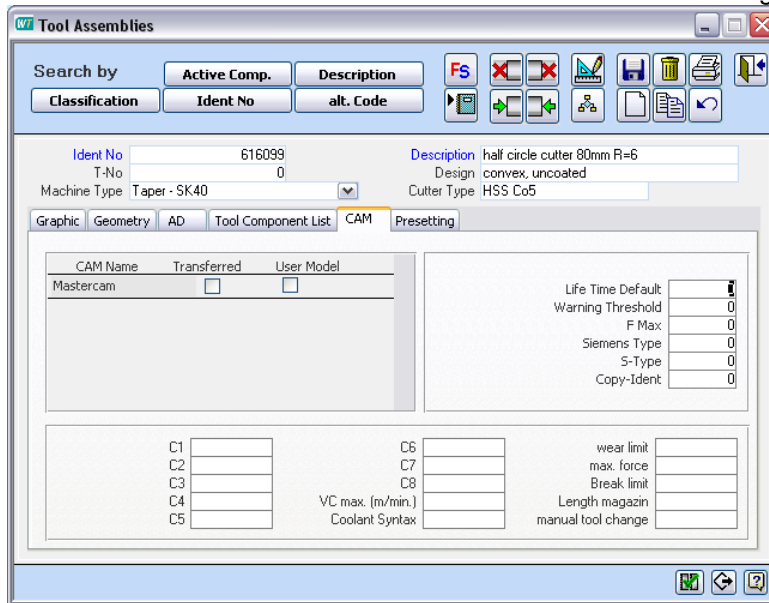


Figure 4 Choose Mastercam in User Settings



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



**Tool Assemblies**

Search by: **Active Comp.** **Description** **Classification** **Ident No** **alt. Code**

Ident No: 616099  
T-No: 0  
Machine Type: Taper - SK40

Description: half circle cutter 80mm R=6  
Design: convex, uncoated  
Cutter Type: HSS Co5

Graphic Geometry AD Tool Component List **CAM** Presetting

CAM Name	Transferred	User Model
Mastercam	<input type="checkbox"/>	<input type="checkbox"/>

Life Time Default:   
Warning Threshold: 0  
F Max: 0  
Siemens Type: 0  
S-Type: 0  
Copy-Ident: 0

C1:   
C2:   
C3:   
C4:   
C5:

VC max. (m/min.):   
Coolant Syntax:

C6:   
C7:   
C8:

wear limit:   
max. force:   
Break limit:   
Length magazin:   
manual tool change:

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 13).

## 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:

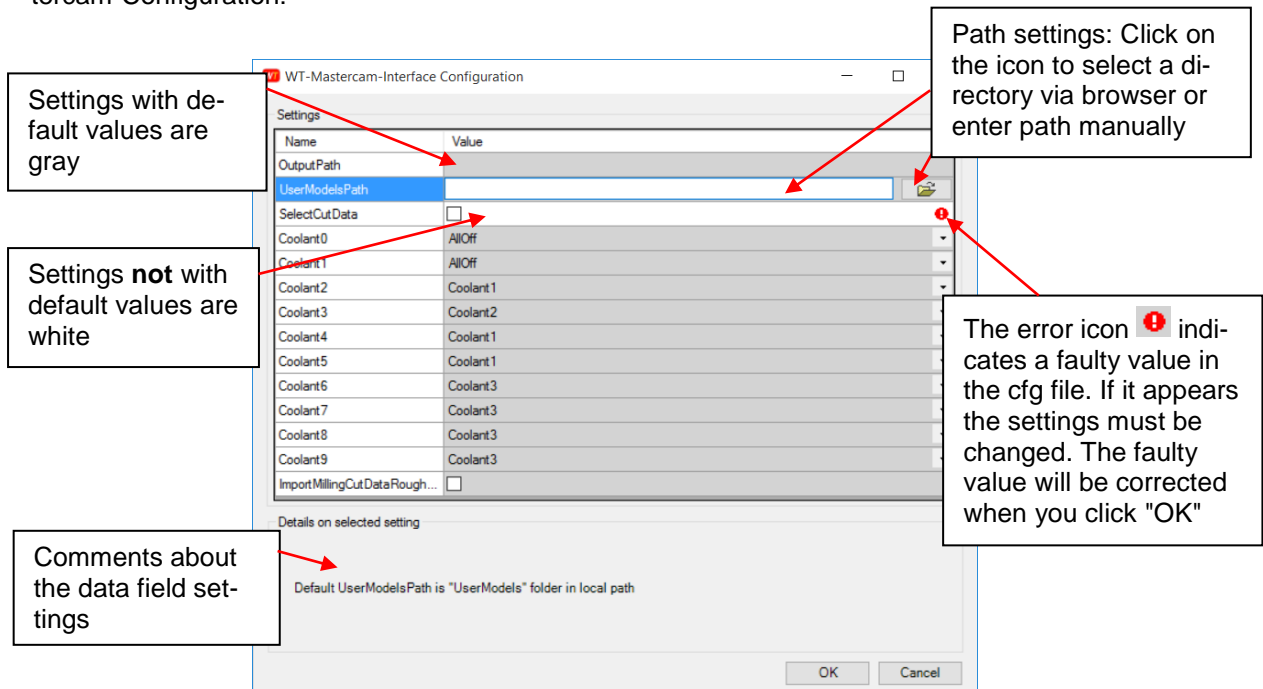


Figure 6 WT-MasterCAM-Interface configuration

<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 10).

## 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 10).

**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 20), 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 10.

## 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 % =  $100 * (\text{Cutdata.Ae} / \text{Cutdata.DM})$
- Rough Z Step % =  $100 * (\text{Cutdata.Ap} / \text{Cutdata.DM})$

## 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.

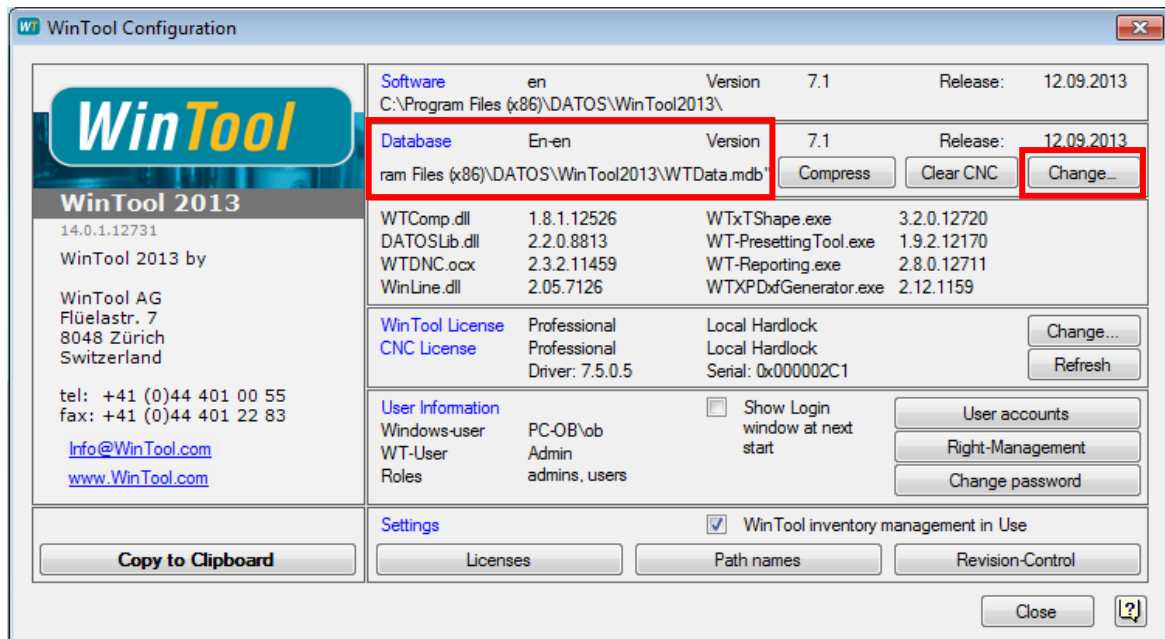


Figure 7 Change the WT-database

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:

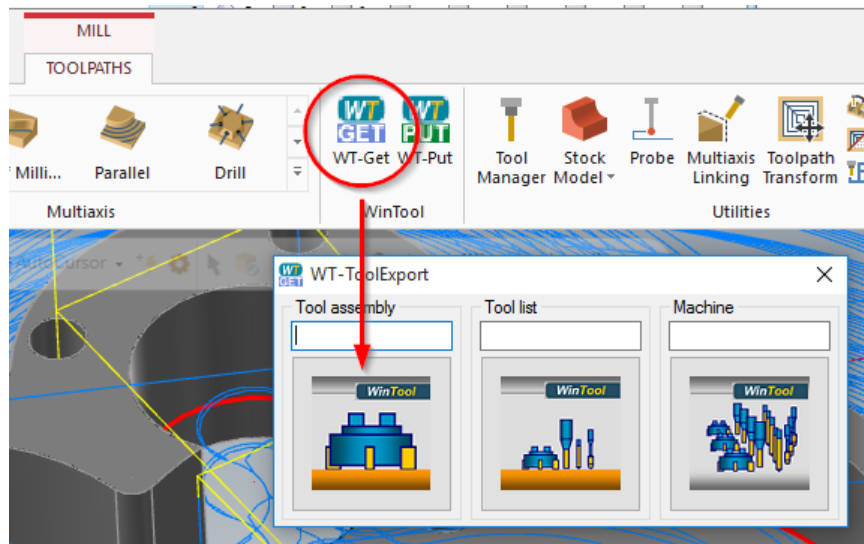



Figure 8 Select Tool assembly in the menubar „TOOLPATHS“

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

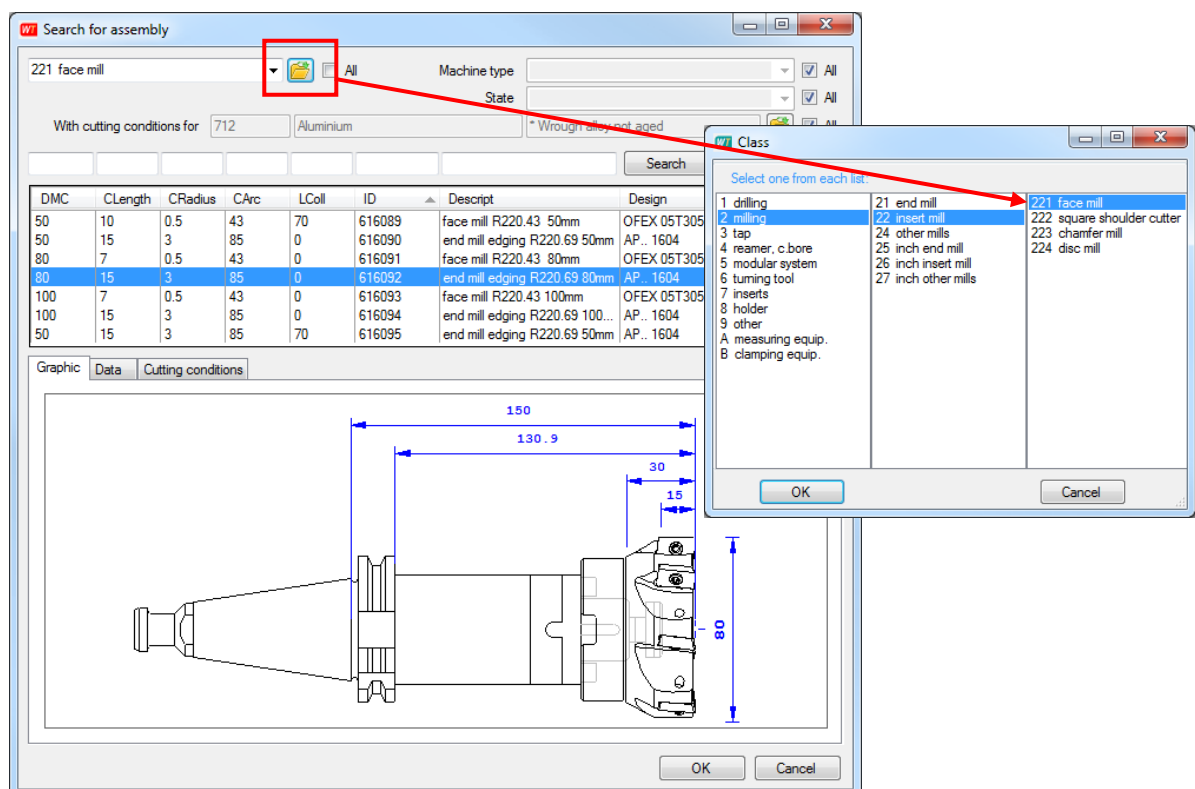


Figure 9 Search a tool assembly class from list

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".

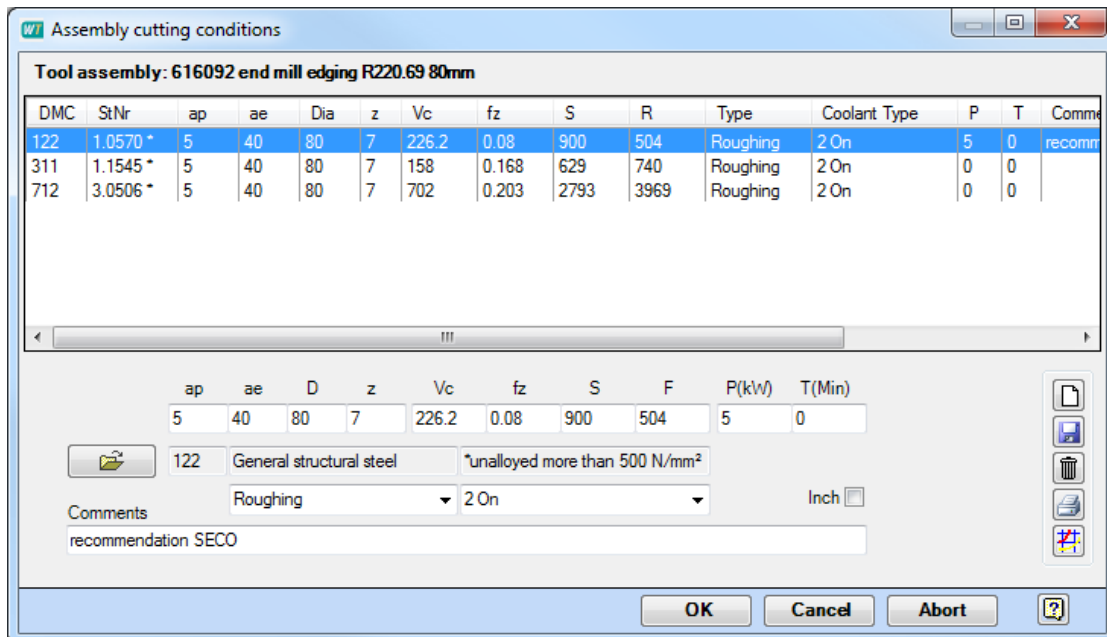


Figure 10 Assembly cutting conditions

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.

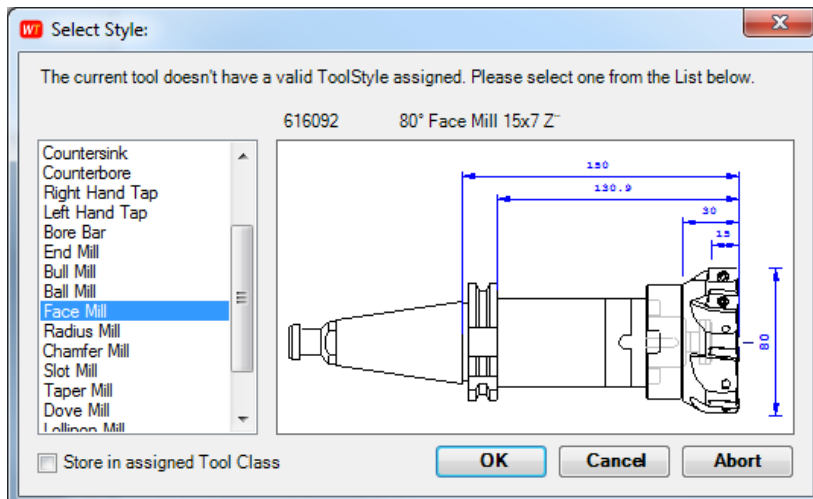


Figure 11 Select a Mastercam tool type from list

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:

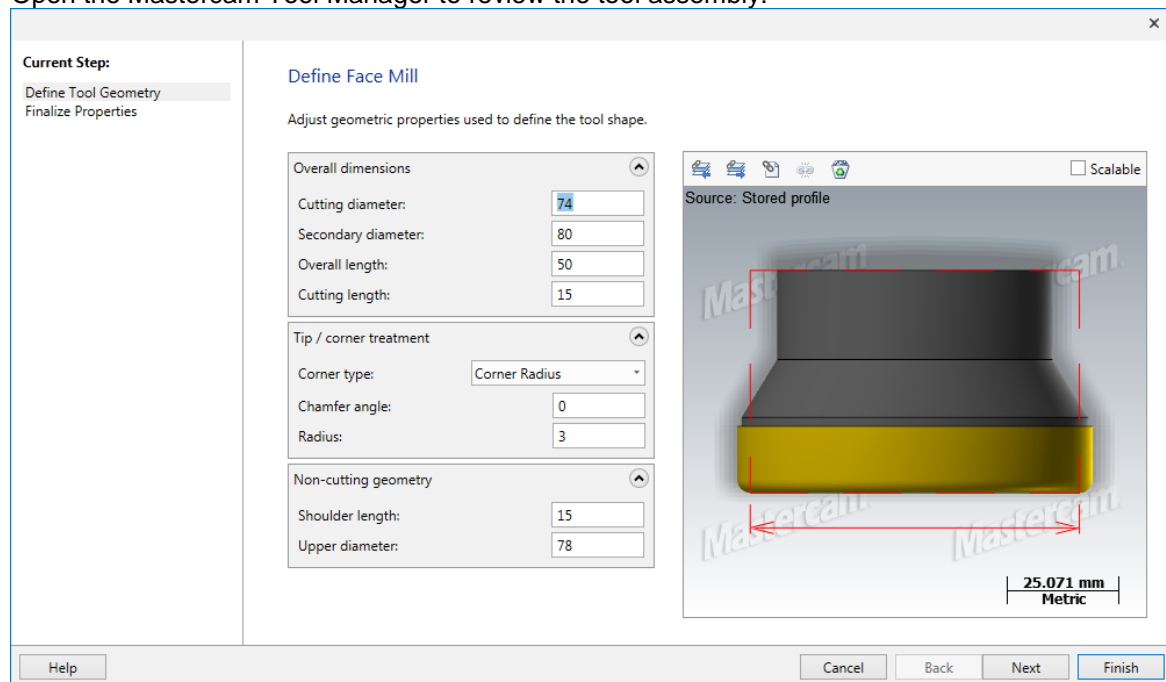


Figure 12 3D Toolpaths model

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

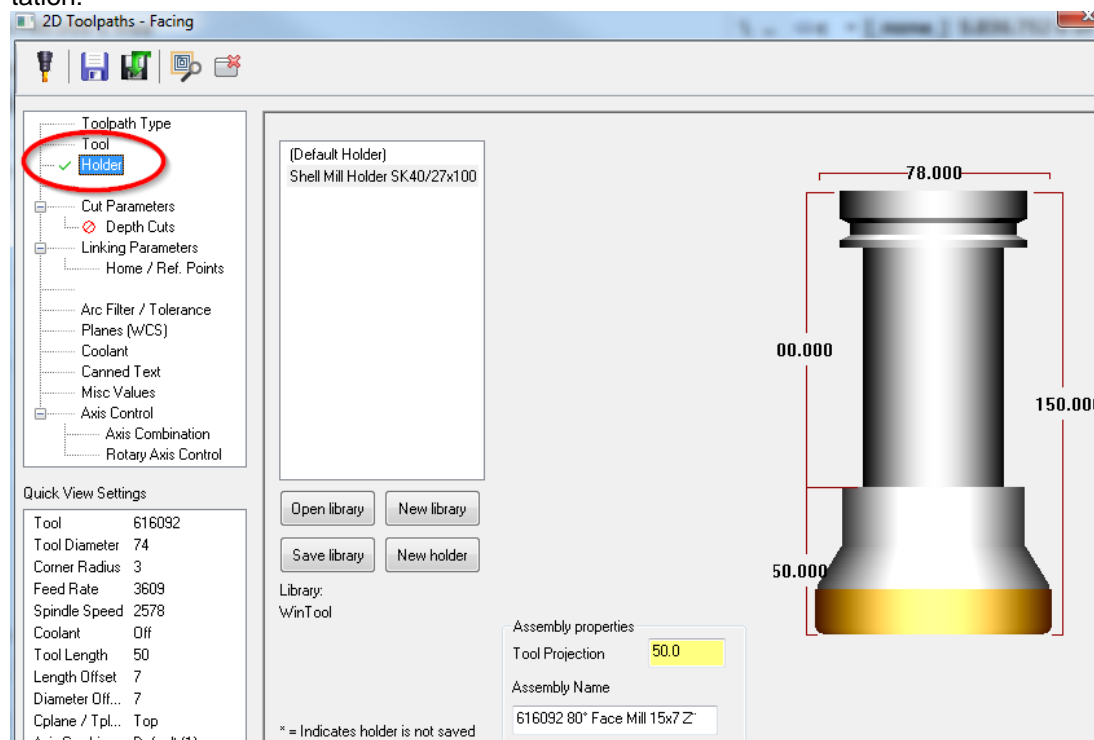


Figure 13 2D Toolpaths model



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

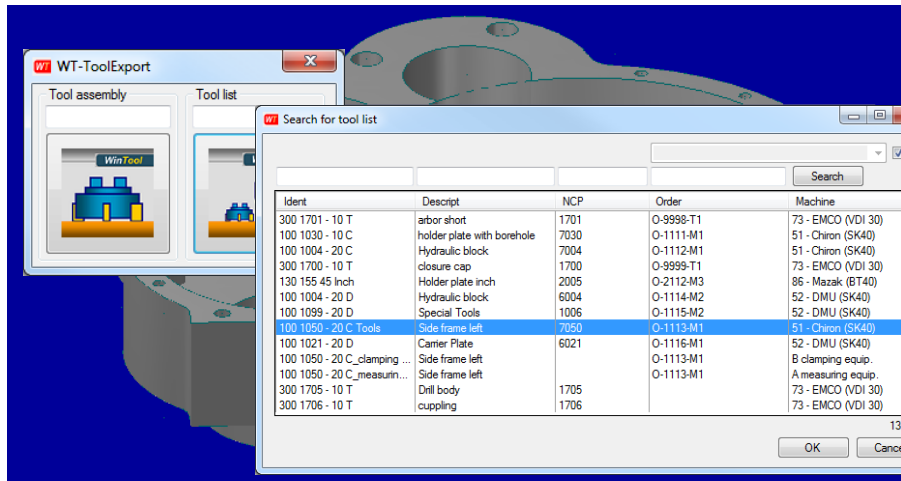
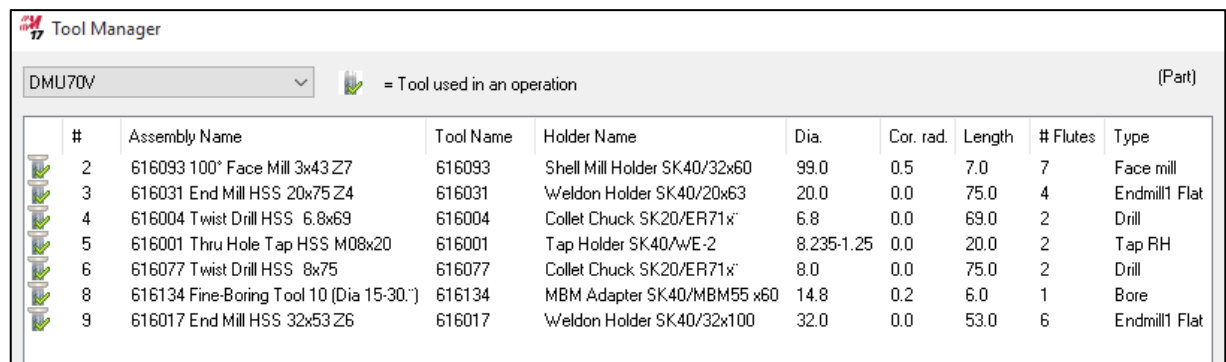


Figure 14 Search to transfer the Tool List

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 15 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.

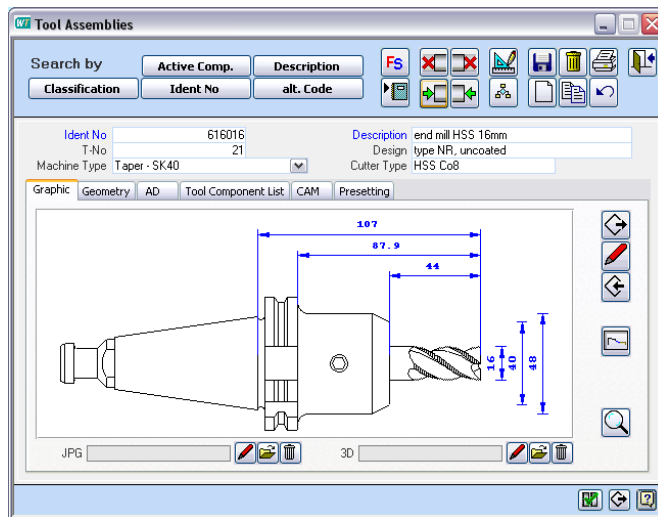
## Tool Numbers

### 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:

#	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 16 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.

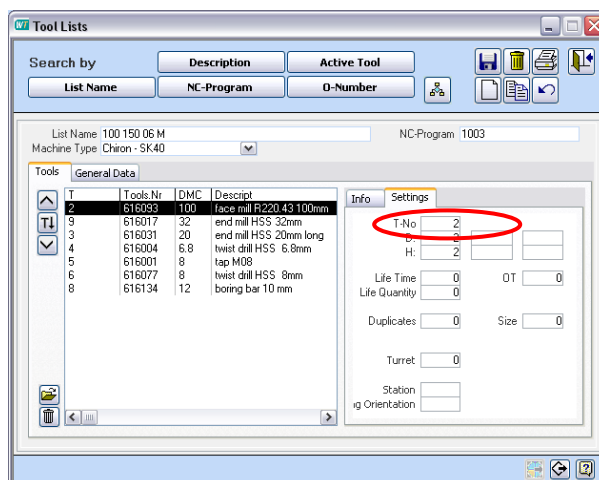
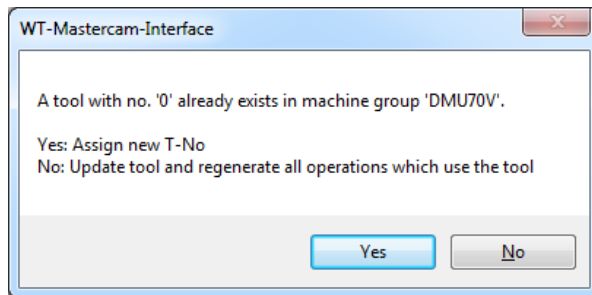


Figure 17 Check the T-No.

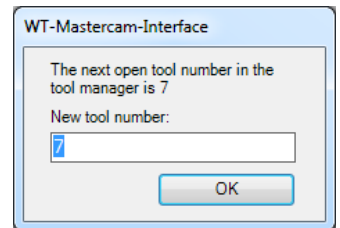
**Note:** This is recommended if you work with resident tools on machines. Create a "Resident Tool List" for each machine and dedicate T-numbers to each resident tool in this list. If you import this list in Mastercam the tools will be loaded with the dedicated T-numbers.

## 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 18 Notification - Tool number

Select **NO** if you want to overwrite the existing tool assembly with the same Tool# in Mastercam. The tool-path 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".

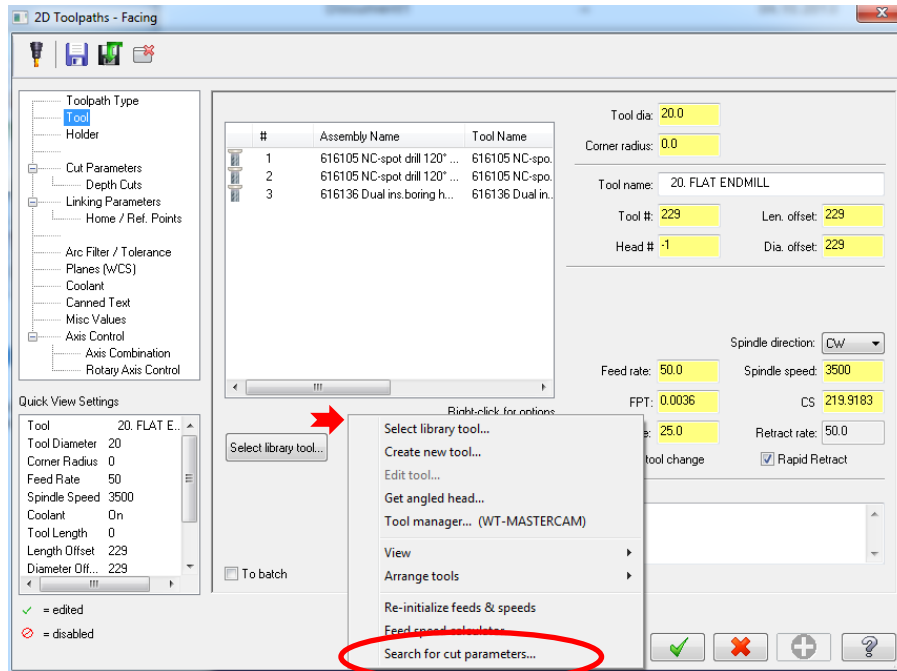


Figure 19 Instruction to search for cut parameters

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":

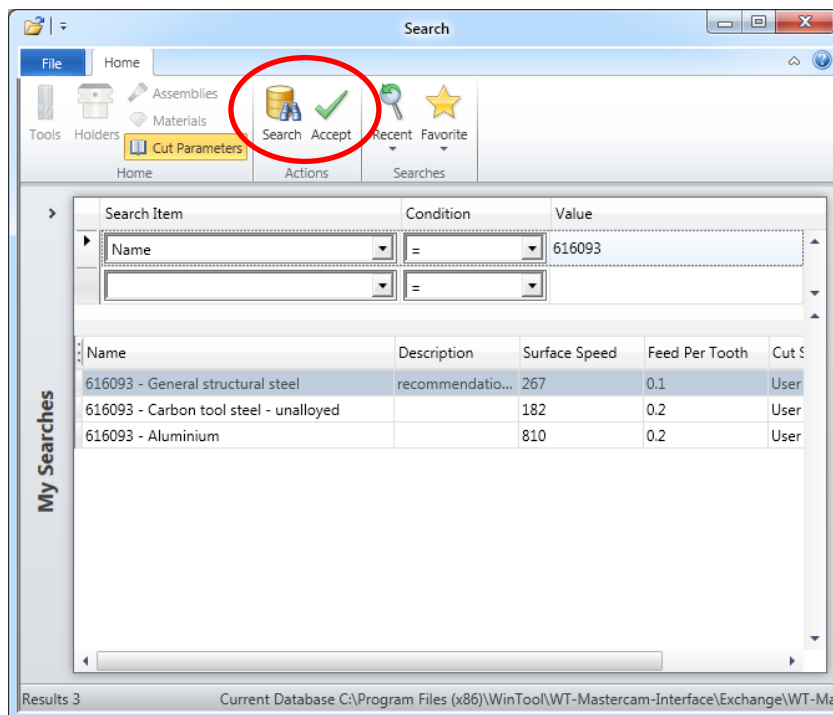


Figure 20 Enter tool assembly and click on Search icon

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:

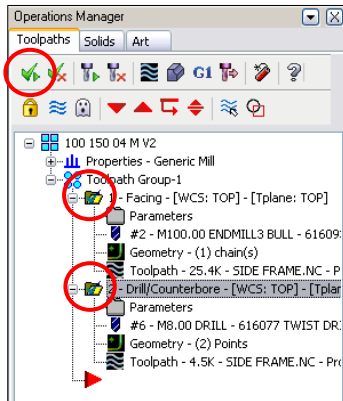


Figure 21 Select tools to be transferred

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



Edit the tool list header information:

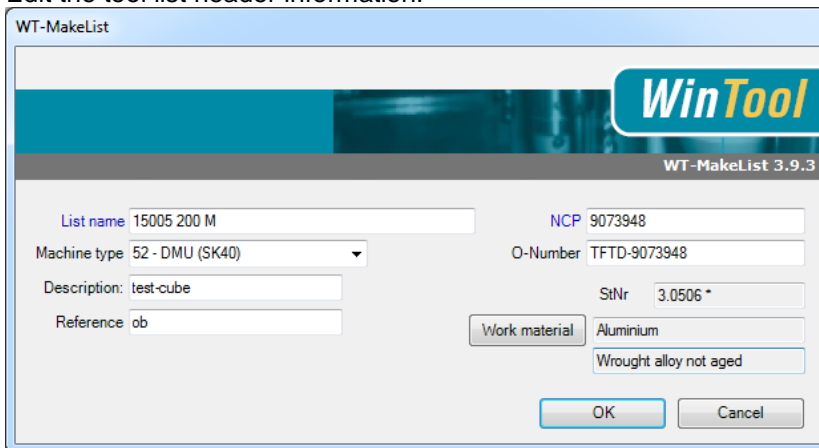


Figure 22 Edit List name in WT-MakeList

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:

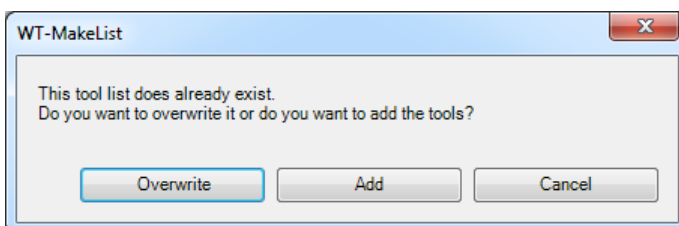


Figure 23 Notification

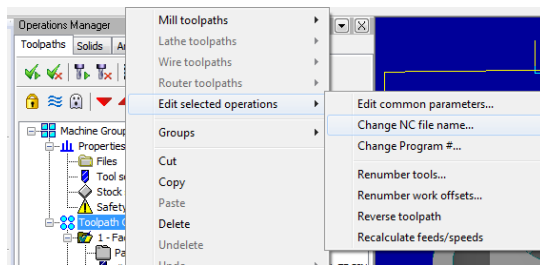
**Note:** In the new *WinTool* tool list, the T-Numbers and the sorting will be the same as in the Mastercam Toolpath Group.

## 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 24 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 25 Change NC file name

### Description

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

Figure 26 Tab folder Files - Machine data

### NCP

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

Figure 27 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).

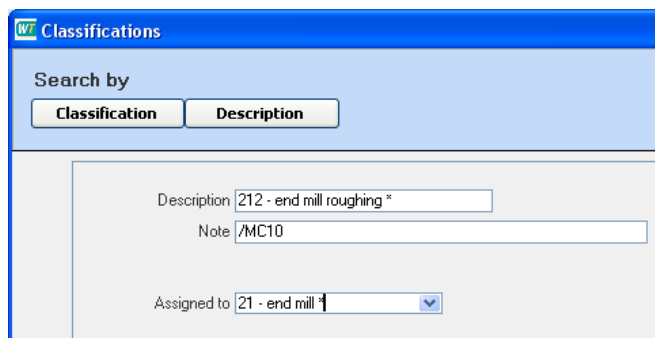


Figure 28 Every Descriptions has an own Note

## 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.

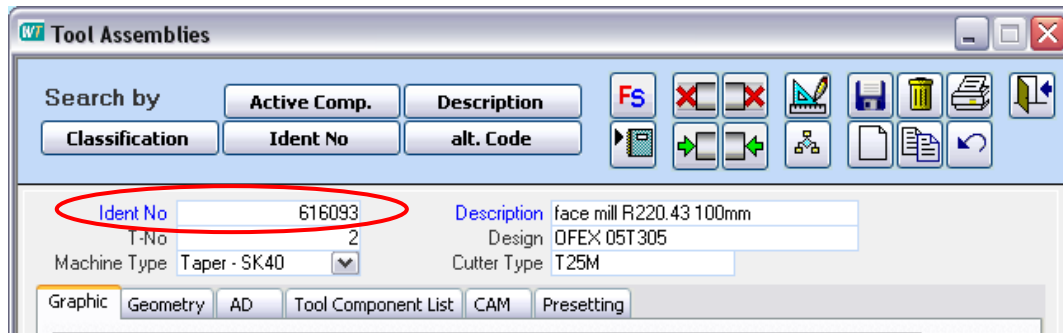


Figure 29 Enter Ident Number and Tool Description

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".

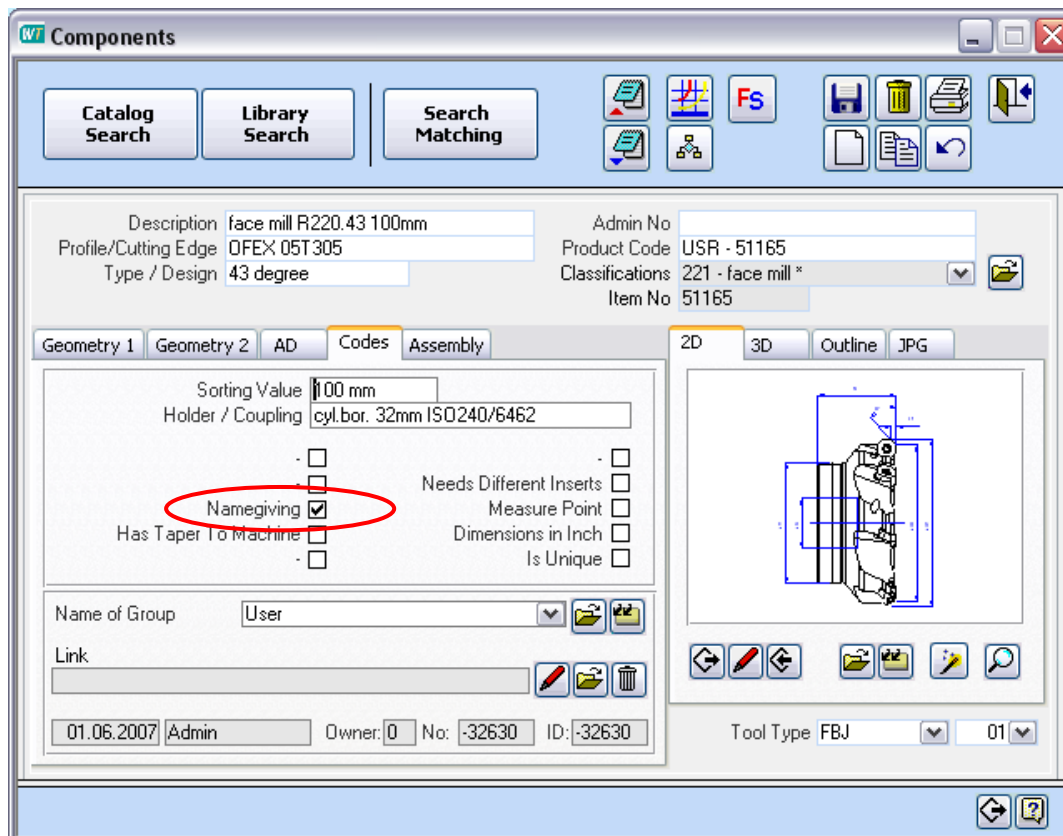


Figure 30 Namegiving must be marked

**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).

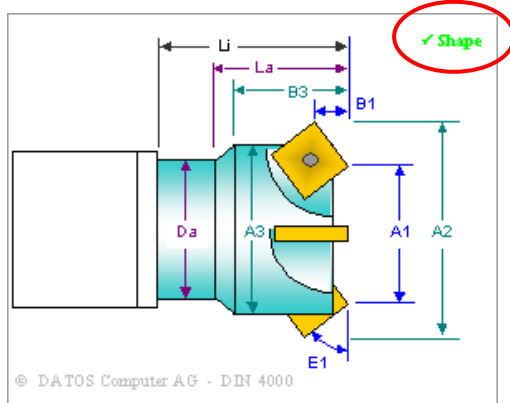


Figure 31 Select Shape for Generator

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.

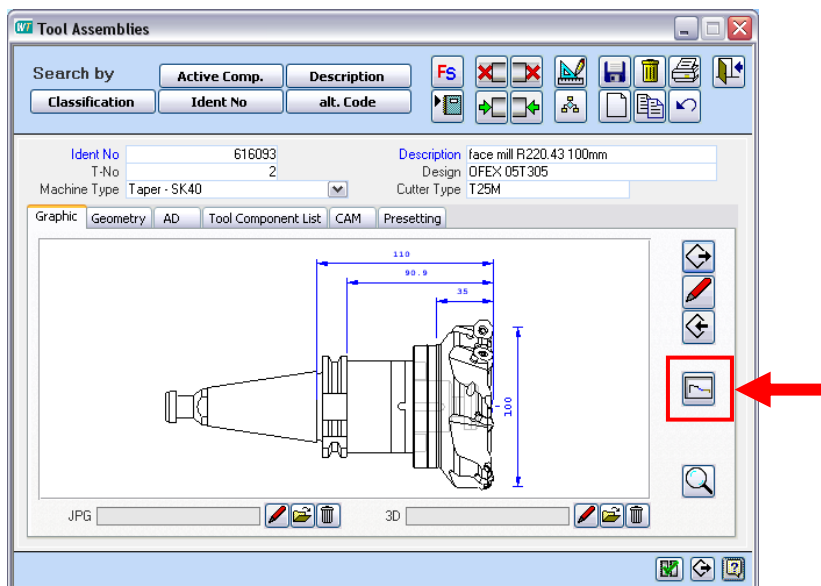


Figure 32 Verify Tool Contour by click on the icon

## 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:

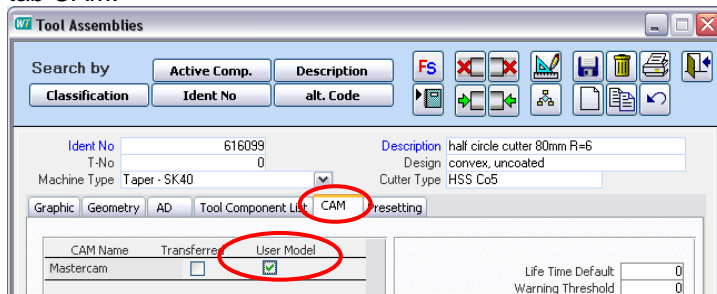


Figure 33 Select User Model in the menu "CAM"

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.

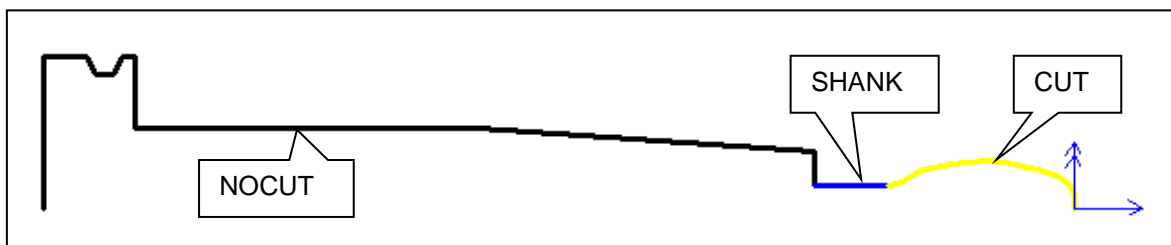


Figure 34 Create a Tool Contour DXF

- 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.

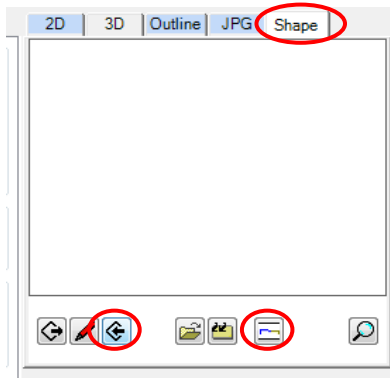
## Special Components


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".



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.

Figure 35 Instruction to activate the tab "Shape"



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

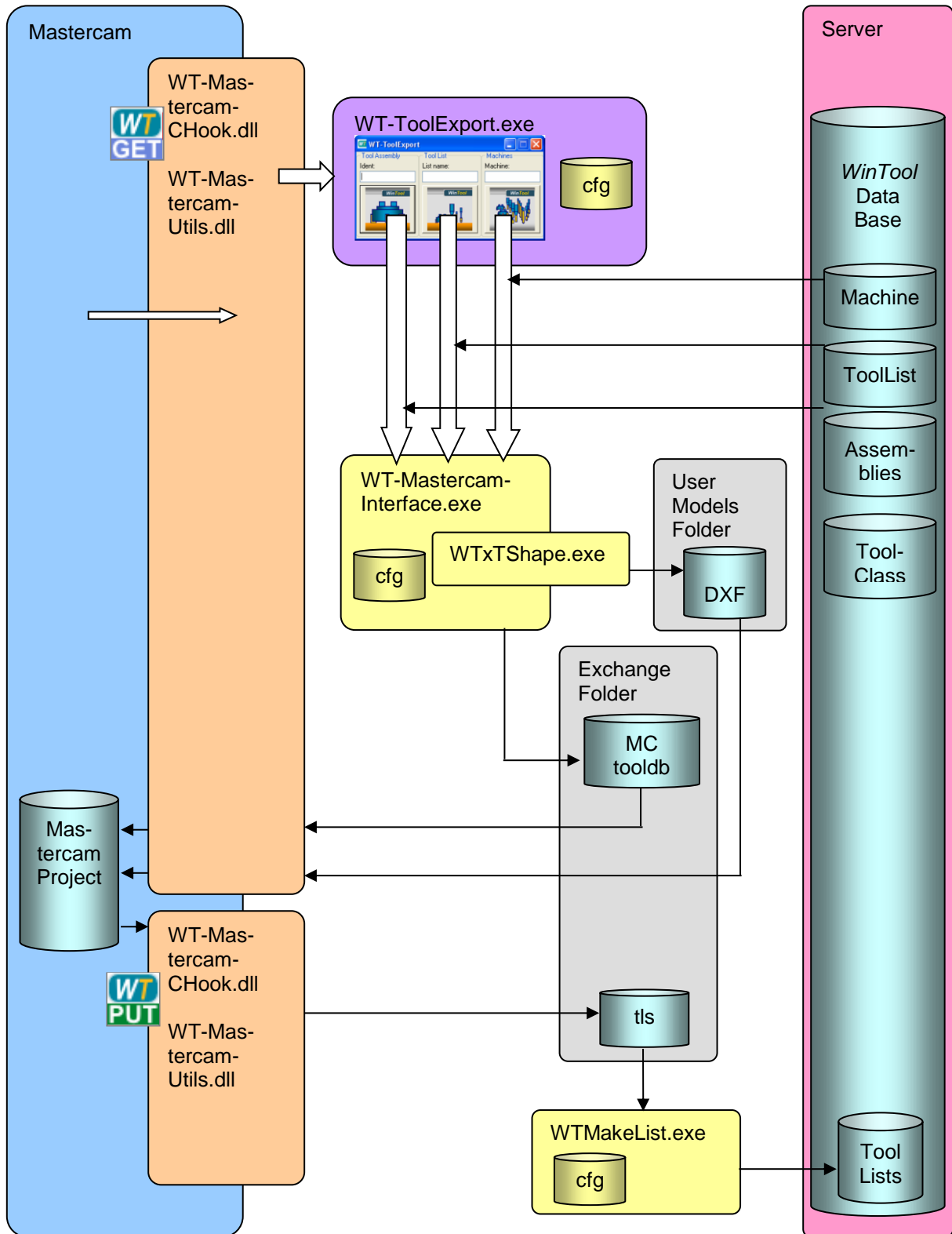


Figure 36 Software structure

## WinTool-Mastercam Data Integration

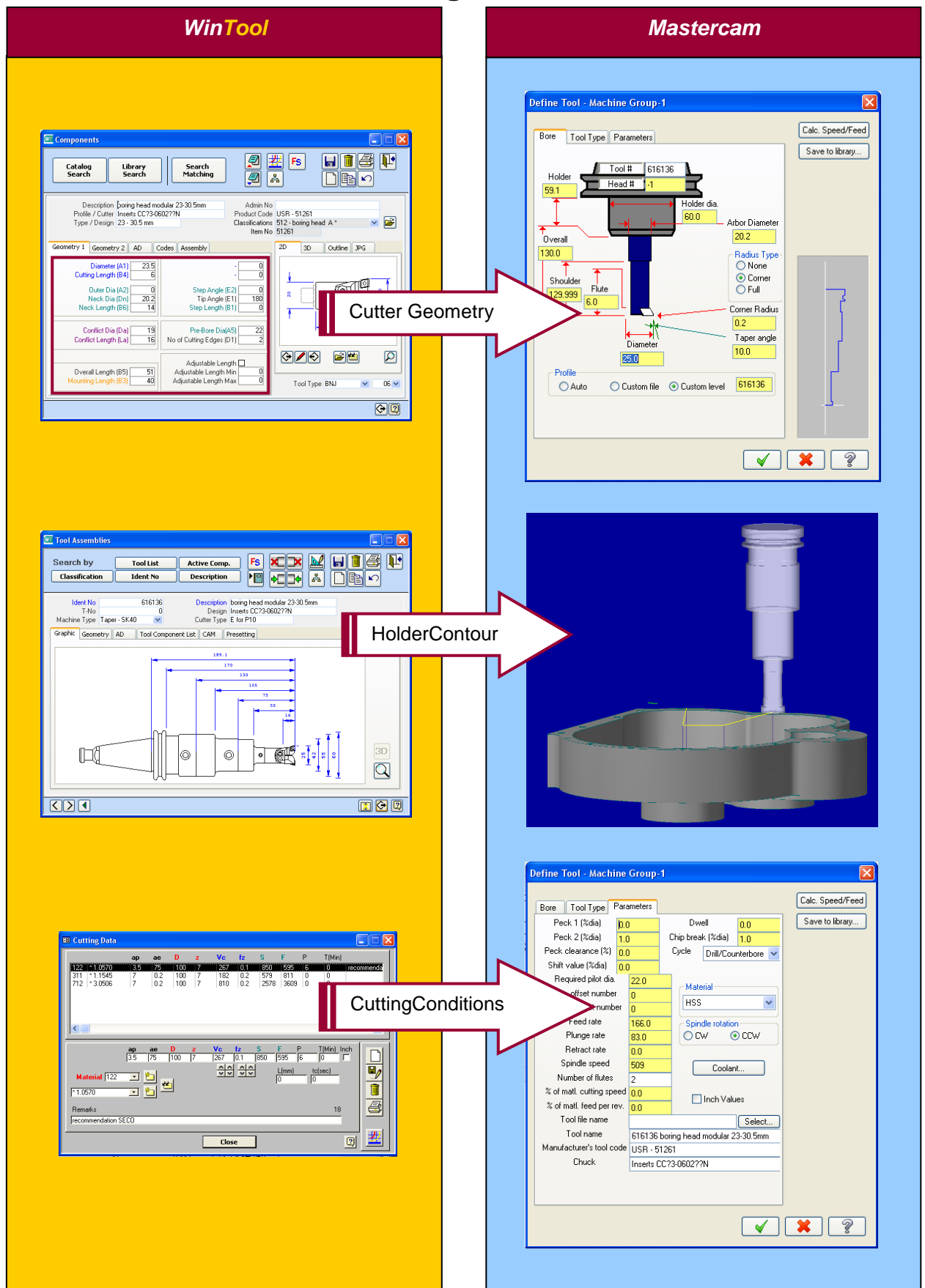


Figure 37 Integrate WinTool in Mastercam

## Known Issues

### Uninstall Error-Message in Mastercam

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

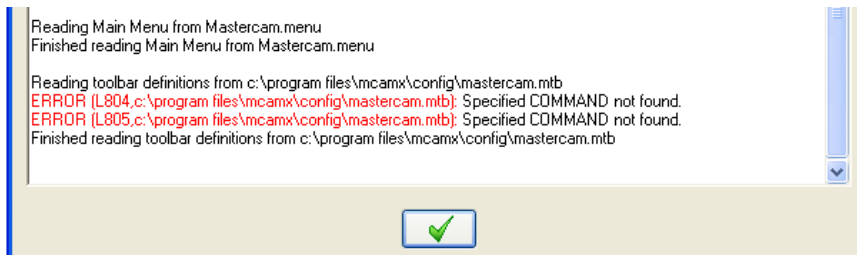


Figure 38 Error message

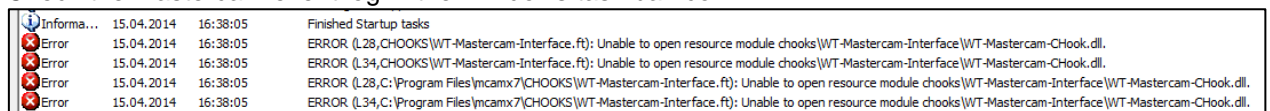
The message is caused from an emptied out *WinTool* toolbar (GET and PUT have been removed) in Mastercam. However, the uninstall program (Windows Add/Remove Program) does not delete the empty toolbar in Mastercam.

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":

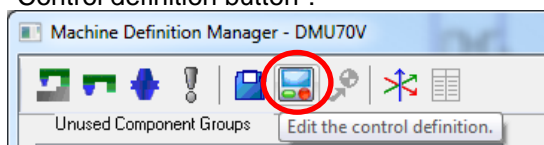


Figure 39 Edit the control definition

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

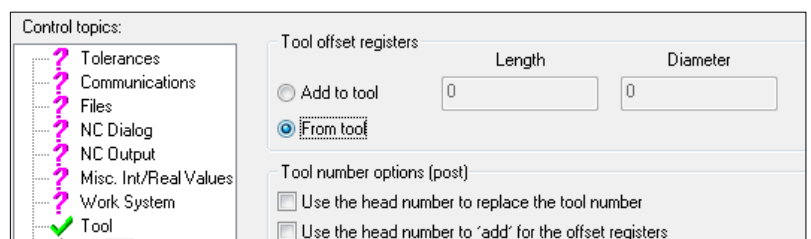


Figure 40 Edit the control definitions

## 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-2017.exe

64-bit Windows registry key:

HKEY\_LOCAL\_MACHINE\SOFTWARE\Wow6432Node\Microsoft\Windows\CurrentVersion\App Paths\WT-Mastercam-Interface-2017.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

### Center Drill (/MC1)

**Define Center Drill**

Adjust geometric properties used to define the tool shape.

Standard sizes	
<input type="text"/>	
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	
<input type="text"/>	
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	
<input type="text"/>	
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

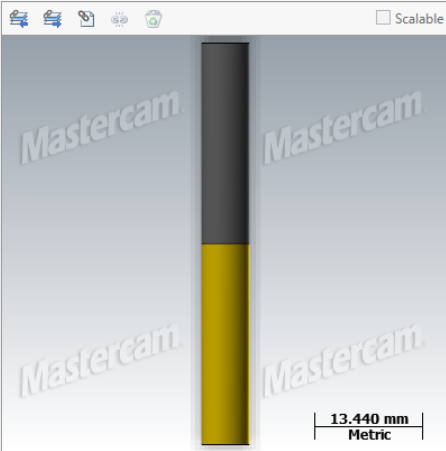
13.440 mm  
Metric

## 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	



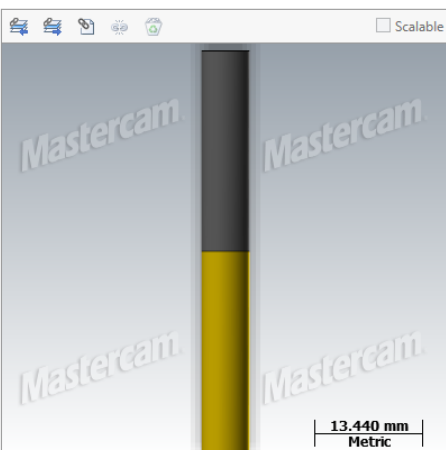
13.440 mm  
Metric

## 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	



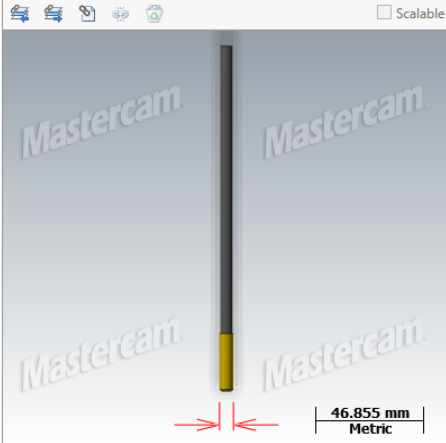
13.440 mm  
Metric

## Reamer (/MC6)

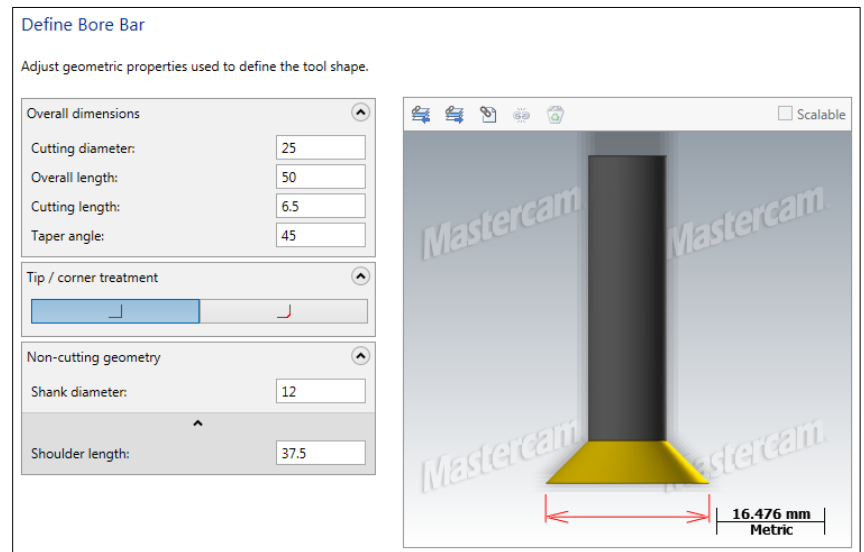
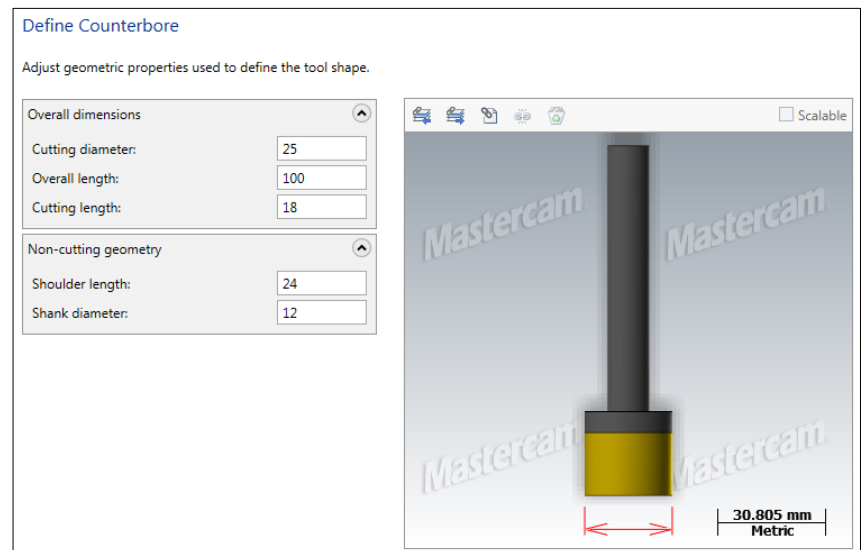
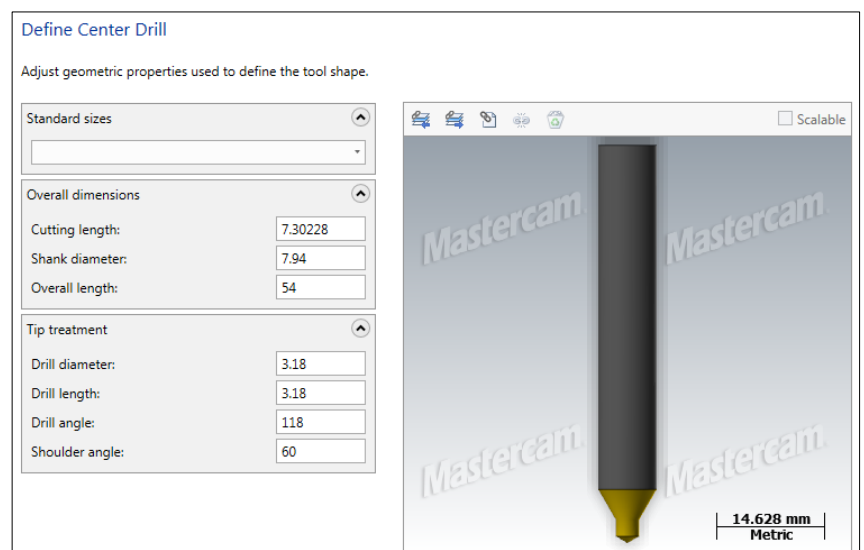
**Define Reamer**

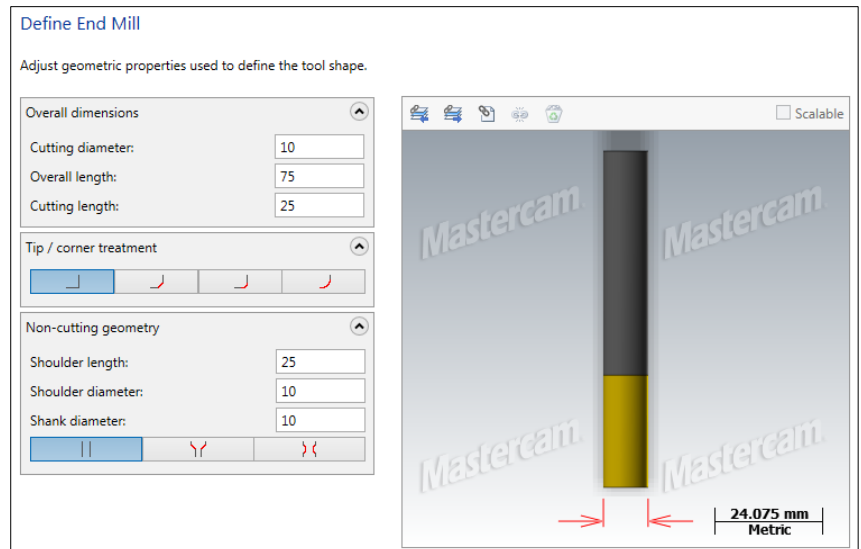
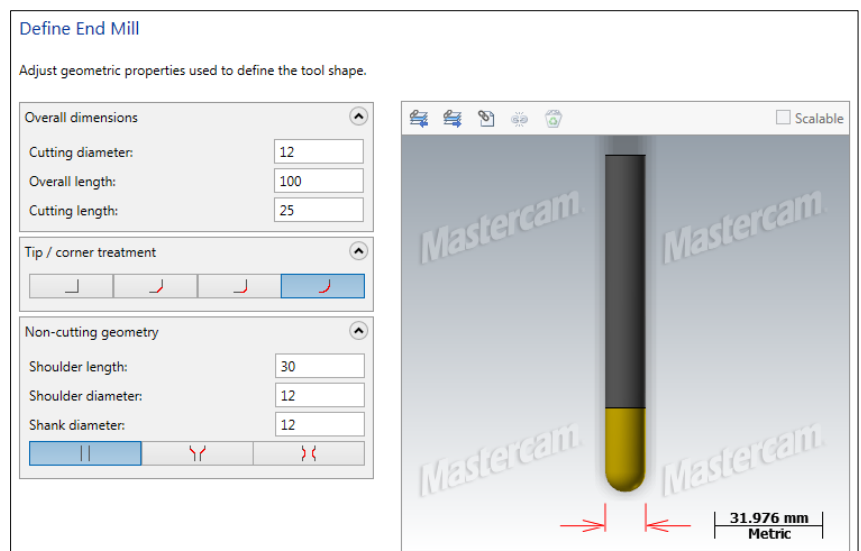
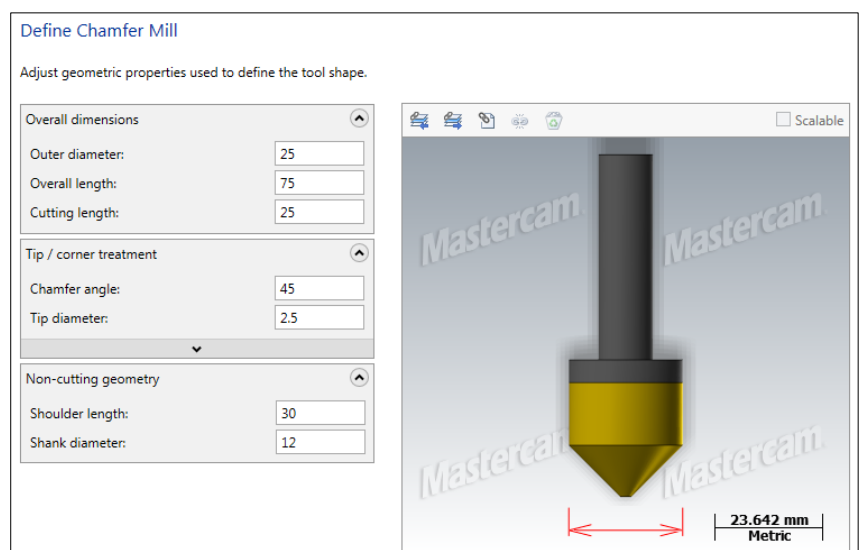
Adjust geometric properties used to define the tool shape.

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

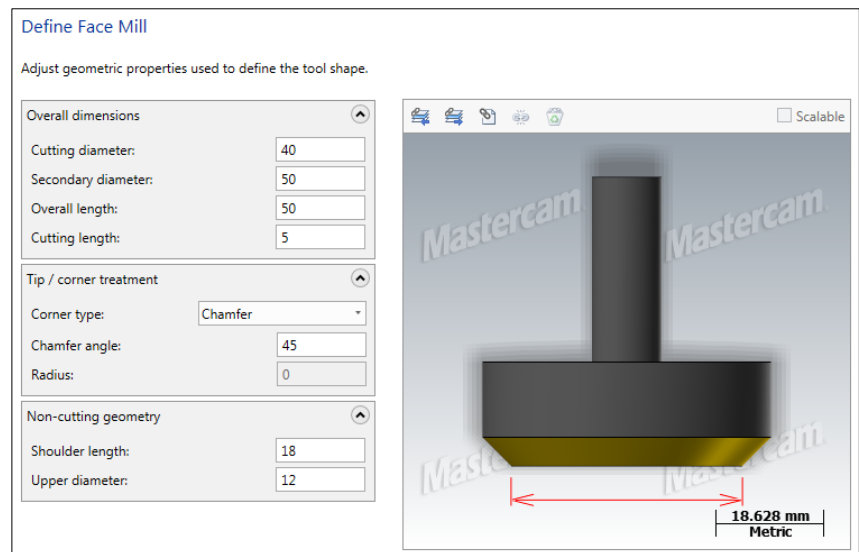


46.855 mm  
Metric

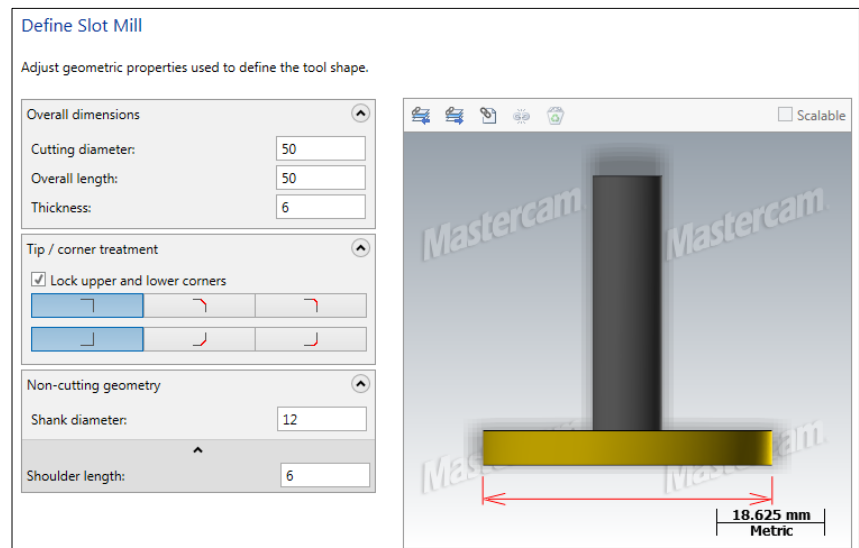
**Bore Bar (/MC7)****Counter Bore (/MC8)****Counter Sink (/MC9)**

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

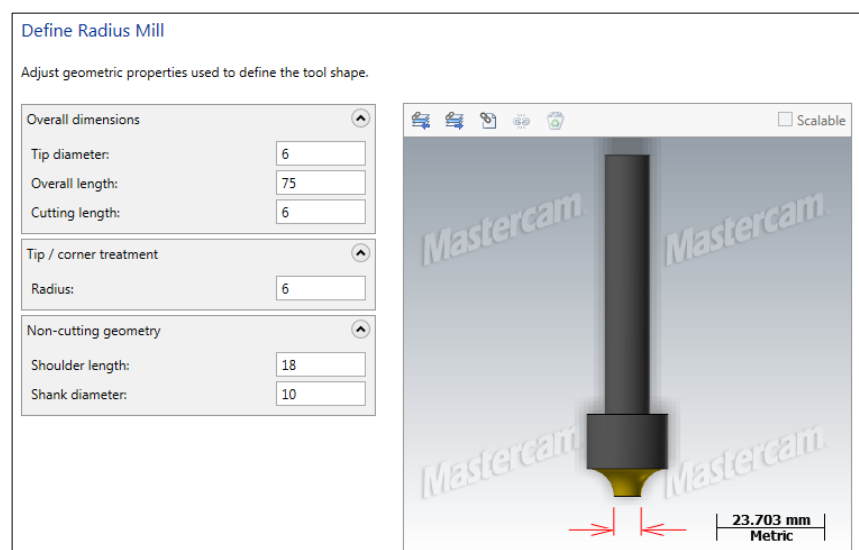
## Face Mill (/MC13)

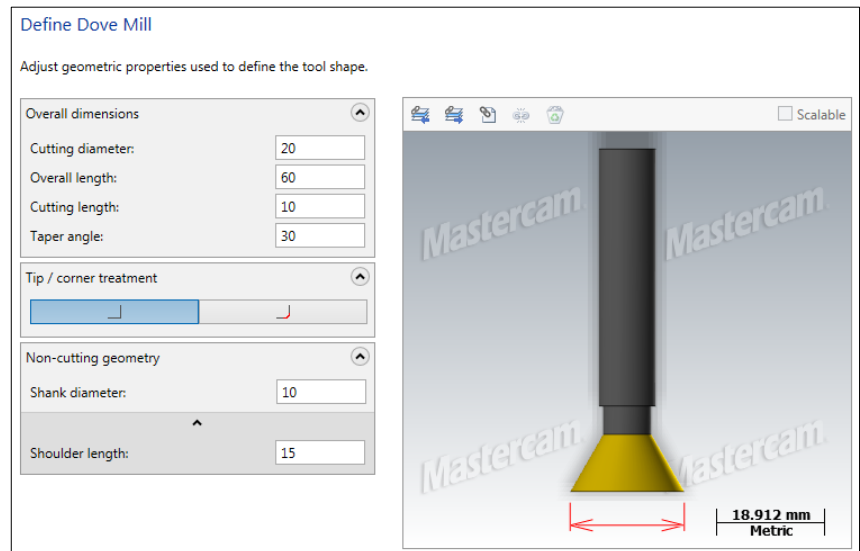
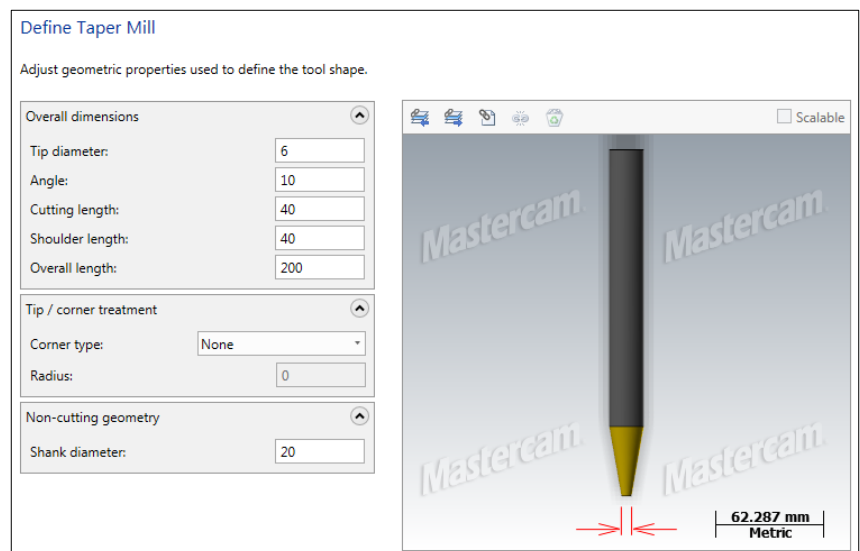
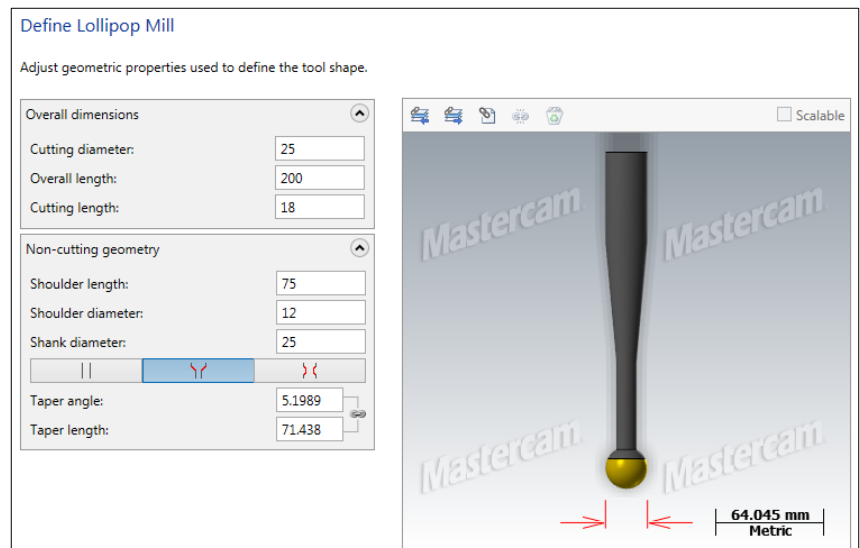


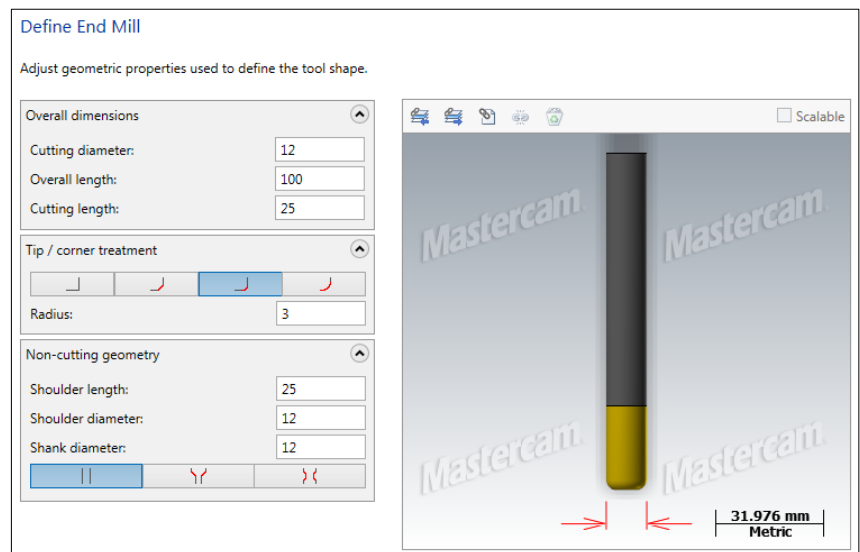
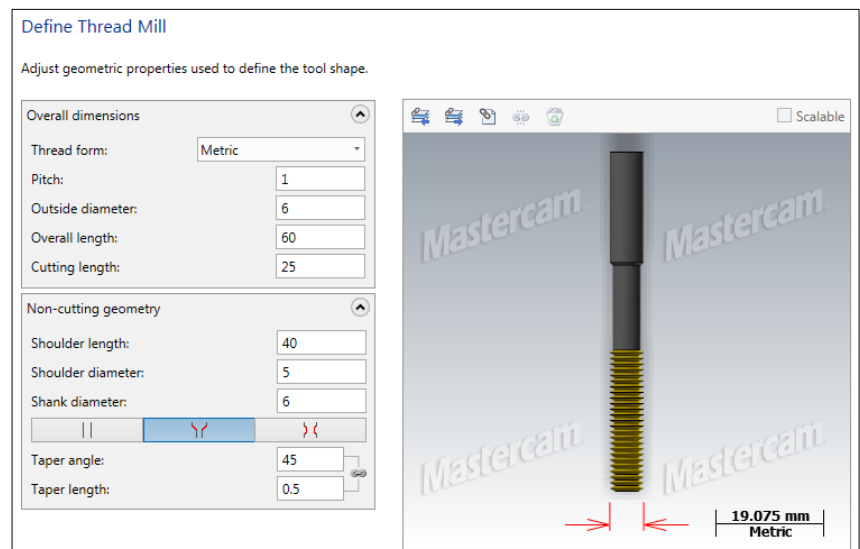
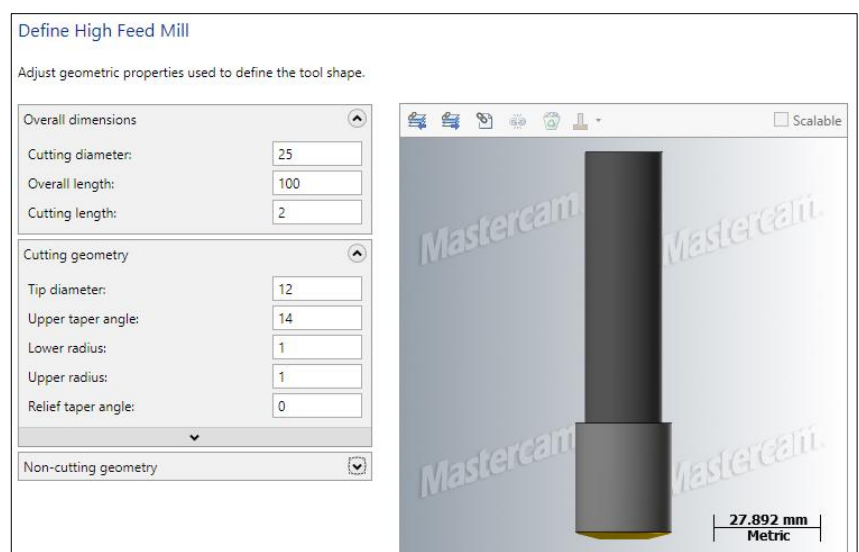
## Slot Mill (/MC14)



## Radius Mill (/MC15)

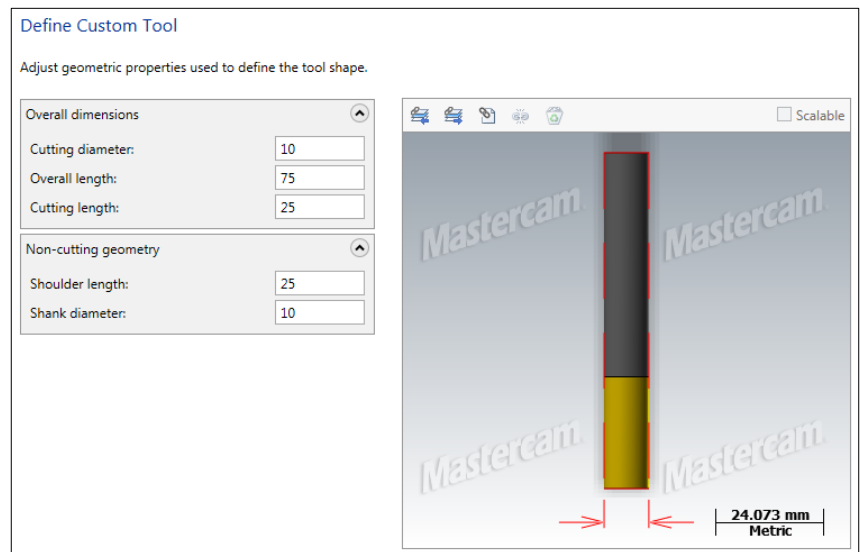


**Dove Mill (/MC16)****Taper Mill (/MC17)****Lollipop Mill (/MC18)**

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



## Custom Tool (/MCO)



## Not Supported Mastercam Tool Types

- Engrave Tool
- Bradpt Drill
- Barrel Mill (Incompatible definition)
- Turning Tools

## History

### 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 - 3<sup>rd</sup> Release

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

## 2.3 - 2<sup>nd</sup> 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.....	6
Figure 2 Choose commands and add to Tool Tabs .....	6
Figure 3 Mastercam Tool Settings .....	8
Figure 4 Choose Mastercam in User Settings .....	8
Figure 5 Tool Assemblies in CAM .....	9
Figure 6 WT-MasterCAM-Interface configuration .....	10
Figure 7 Change the WT-database .....	13
Figure 8 Select Tool assembly in the menubar „TOOLPATHS“ .....	14
Figure 9 Search a tool assembly class from list.....	14
Figure 10 Assembly cutting conditions.....	15
Figure 11 Select a Mastercam tool type from list.....	15
Figure 12 3D Toolpaths model .....	16
Figure 13 2D Toolpaths model .....	16
Figure 14 Search to transfer the Tool List.....	17
Figure 15 List of all imported tools .....	17
Figure 16 List of T-No for Tool Assemblies .....	18
Figure 17 Check the T-No. ....	18
Figure 18 Notification - Tool number .....	19
Figure 19 Instruction to search for cut parameters .....	20
Figure 20 Enter tool assembly and click on Search icon .....	20
Figure 21 Select tools to be transferred.....	21
Figure 22 Edit List name in WT-MakeList .....	21
Figure 23 Notification .....	21
Figure 24 Entry fields will be filled in automatically .....	22
Figure 25 Change NC file name.....	22
Figure 26 Tab folder Files - Machine data .....	22
Figure 27 Enter a program number in tab folder 'Tool Settings' .....	22
Figure 28 Every Descriptions has an own Note .....	23
Figure 29 Enter Ident Number and Tool Description.....	24
Figure 30 Namegiving must be marked .....	24
Figure 31 Select Shape for Generator .....	25
Figure 32 Verify Tool Contour by click on the icon.....	25
Figure 33 Select User Model in the menu "CAM" .....	26
Figure 34 Create a Tool Contour DXF .....	26
Figure 35 Instruction to activate the tab "Shape" .....	27
Figure 36 Software structure .....	29
Figure 37 Integrate WinTool in Mastercam .....	30
Figure 38 Error message.....	31
Figure 39 Edit the control definition.....	31
Figure 40 Edit the control definitions .....	31