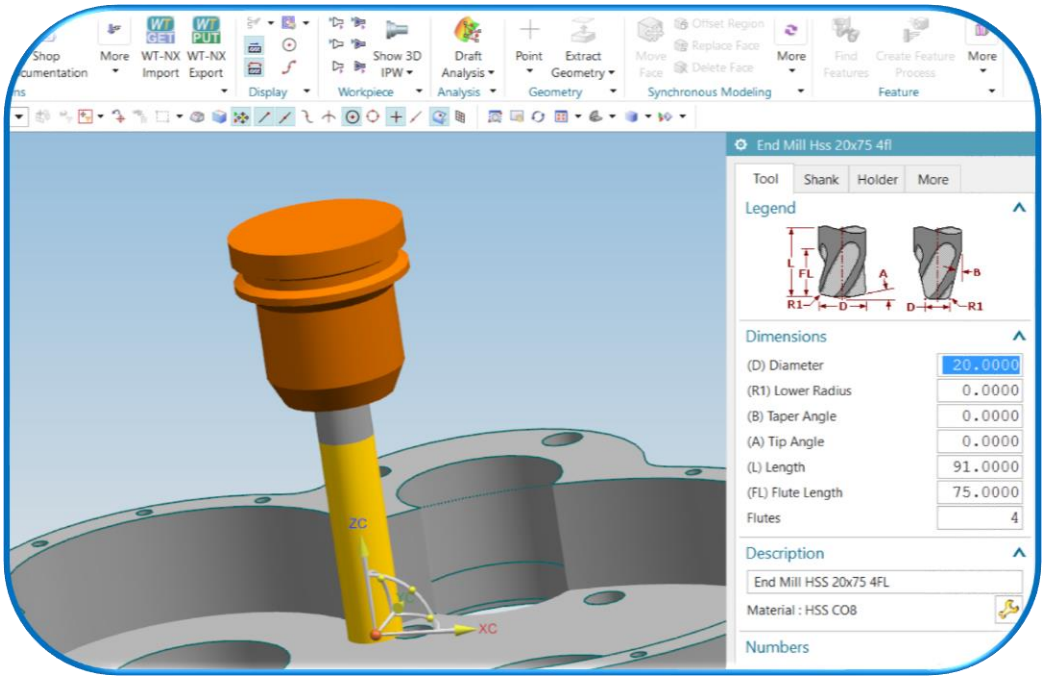
WT-NX-Interface



Manual

WinTool Interface 4.11.1 for NX

The WT-NX-Interface enables the user to select and transfer assemblies from the *WinTool* database to the NX CAM environment, including full graphic representation and cutting conditions for the different work materials.

A complete list of every used tool assembly will be stored in the *WinTool* database for further use as setup sheet, documentation and queries.

Requirements

- *WinTool* 2011 Professional or later
- NX 7.5, NX 8.0, NX 8.5, NX 9.0.3 (Details on page 4), NX 10, NX 11, NX 12

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Summary

Job

The WT-NX-Interface enables the user to select and transfer assemblies from the *WinTool* database to the NX CAM environment. Full graphic representation for each assembly is supported and the cutting conditions for the different work materials are transferred from the *WinTool* technology library. A complete list of every used tool assembly will be stored in the *WinTool* database for further use as setup sheet, documentation and queries.

Requirements

This interface requires *WinTool* Professional 2011 and NX 7.5 or higher and can be used under Windows only.
The user must be granted write access on the NX directories

- UGII_CAM_LIBRARY_TOOL_METRIC_DIR
- UGII_CAM_LIBRARY_TOOL_ENGLISH_DIR
- UGII_CAM_LIBRARY_FEEDS_SPEEDS_DATA_DIR

The actual directories can be seen by going to "Help" > "Log file".

Note:

- NX 9 requires maintenance release 9.0.3 for correct import of tool type "Form Drill" and "Probe".
- Import of tool type "Probe" requires NX 8.0 or higher

NX 7 or earlier

To use the interface with NX versions 7 and earlier, the VB.Net macro must be adjusted manually. For detailed instructions, refer to chapter "Adjust tool import macro for NX 7 and below" on page 26.
Note: *WinTool* AG doesn't offer support if the macros are changed by the user.

Principals

Single assemblies, a list of assemblies or all assemblies related to a specific CNC are transferred from *WinTool* to NX. The activity is started out of NX by calling a VB.Net macro linked to the "get"-icon shown in the NX menu. Tool data is exported in NX format and the tool is directly linked to the active NX part.

The final list of used assemblies is exported from NX to *WinTool* by using the "put"-icon being linked to another VB.Net macro.

Tool types supported

Most of the NX Tool Types are supported by this interface. A detailed list is found in a later section of this manual. The NX model for the assembly is transferred from the customer tool parts library. For limitations of the shape module see corresponding documentation.

Important Changes

NX 8.0 or later

Import of shank

NX 8.0 introduced shanks in milling and drill tool classes. The interface imports shank data starting from the front of the tool.

- It imports the neck if it exists and is different in diameter than the cutting diameter
- If no neck, but a tapered shank exists, the interface is importing the tapered shank

Face Mill/Chamfer Mill

In NX 8.0, the tool type "face mill" is replaced by "chamfer mill". The WT-NX-Interface supports this new type. If NX 8.0 is used, the classification in the *WinTool* tool types must be adjusted to ensure that the tools are transferred correctly.
The following table shows the tool types that must be changed. Check if the NX types in the tool classes are correct:

WinTool tool class	NX tool type of WinTool tool class		NX tool type
	Old type	New type	
221 face mill	**/UG0212**	**/UG0205**	Chamfer mill
222 square shoulder cutter	**/UG0212**	**/UG0201**	End mill
223 chamfer mill	**/UG0201**	**/UG0205**	Chamfer mill

Note: See chapter "User Classification" on page 19 for details on how to change the classification. Chapter Supported NX tool types on page 29 shows all supported types.

NX 7.5 or earlier

Starting with *WinTool* 2012, sample databases already contain the new tool types. If NX 7.5 or an older version is used, all *WinTool* tool classes using NX tool type "/UG0205" must be adjusted to import the tool assemblies. The NX tool type classes "221 face mill" and "223 chamfer mill" must be changed to the old type (see table above). Tool type "225 inch step drill" must be changed to "/UG0212".

Note: See chapter "User Classification" on page 19 for details on how to change the classification. Chapter Supported NX tool types on page 29 shows all supported types.

Licensing and Copyright

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Installation

Directory Structure

WT-NX-Interface 4.7 introduces a clear separation of program files and user data.
All user data is centrally placed the [Public Documents]\WT-NX-Interface folder:

User data	Location
Default location of UserModels folder	[Public Documents]\WT-NX-Interface\UserModels
Default location of Exchange folder	[Public Documents]\WT-NX-Interface\Exchange
Default location of CustomToolPath folder	[Public Documents]\WT-NX-Interface\CustomTool
Configuration files: WT-NX-Interface.cfg WT-MakeList.cfg	[Public Documents]\WT-NX-Interface

New Installation of the WT-NX-Interface

Be sure to be local administrator to install software on a PC.

Install *WinTool Professional* first and check the functionality.

Install WT-NX-Interface next. Run setup.exe to install the WT-NX-Interface software into a new folder.
C:\Program Files\WinTool\WT-NX-Interface\

Note: If the software is running on a server, use separate folders for each Windows user or configure specific folders for data exchange (see below).

The software is now installed with the default settings. Continue with the configuration as described in the sections below.

Update Installation of the WT-NX-Interface

Update from version 4.6 and newer

Copy the existing configuration files WT-UG-Interface.cfg or WT-NX-Interface.cfg and WT-MakeList.cfg from the WT-NX-Interface folder to a safe place.

Uninstall the existing interface and delete remaining files and folders. Install WT-NX-Interface similar to a new installation. No adaptations in your NX environment are necessary.

Open the interface configuration via "Start" > "All Programs" > "WinTool" >
"WT-NX-Interface"> "WT-NX-Interface Configuration" and check the settings.

Move WT-MakeList.cfg back to the interface installation folder.

Update from version 4.6 or older

Install WT-NX-Interface similar to a new installation. After the installation, follow one of the two steps to re-cover 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-NX-Interface.
  Check the interface configuration using the configuration window (see page 8) and the configuration file "WT-MakeList.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-NX-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-NX-Interface\UserModels.
Finally, remove the Exchange and UserModels folder in the installation directory of the previous installation.

Sample Database

The WT-NX-Interface always uses the *WinTool* database currently linked to your *WinTool Professional* installation. To test the interface and get familiar with its functionality, please copy a virgin database (WTData) from your latest *WinTool* Professional installation CD.

Re-link this database with the local *WinTool* application using the function "Change" in the *WinTool* Administration section. *WinTool* will thereafter shut down automatically:

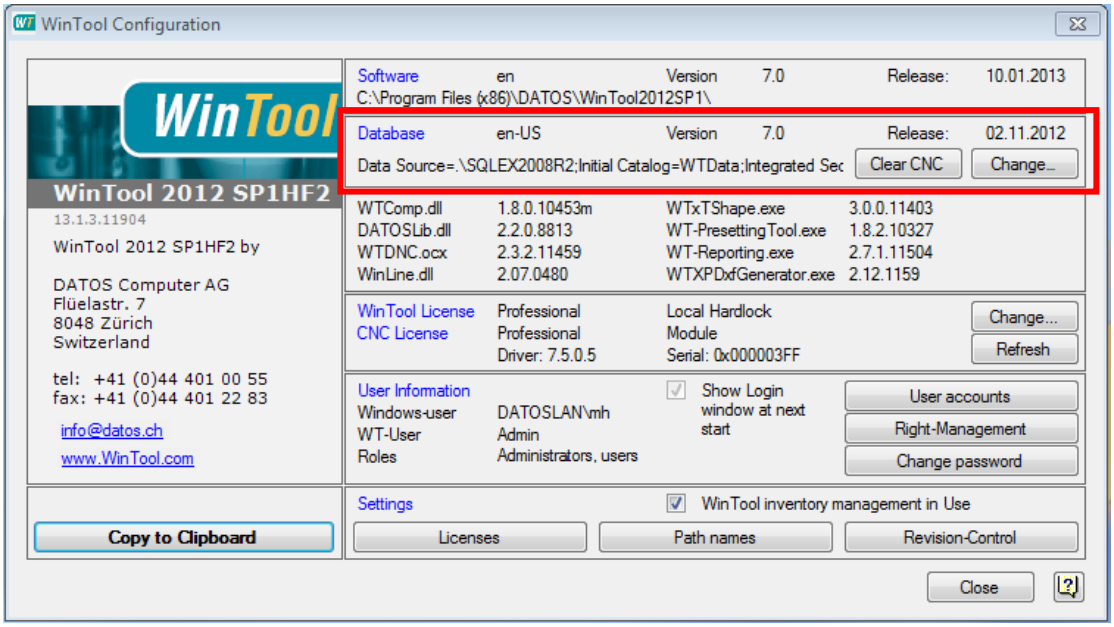


Figure 1 Change the Database in WT-Configuration

Use the NX Sample "Side-Frame-mm" stored in the interface "Sample" folder to make your first tests. Before you can verify the NC-Program with full assembly representation you have to "get" the tools from the *WinTool* sample database. Use the WT-NX-Interface and "get" the tool list "Side-Frame".

STL models of the tool holders of the *WinTool* tools 636106 and 636107 are in the sample folder

Configuration of WT-NX-Interface

The configuration window allows you to check and change the settings of the WT-NX-Interface.
Open the configuration window in "Start" > "All Programs" > "WinTool" > "WT-NX-Interface"> "WT-NX-Interface Configuration"

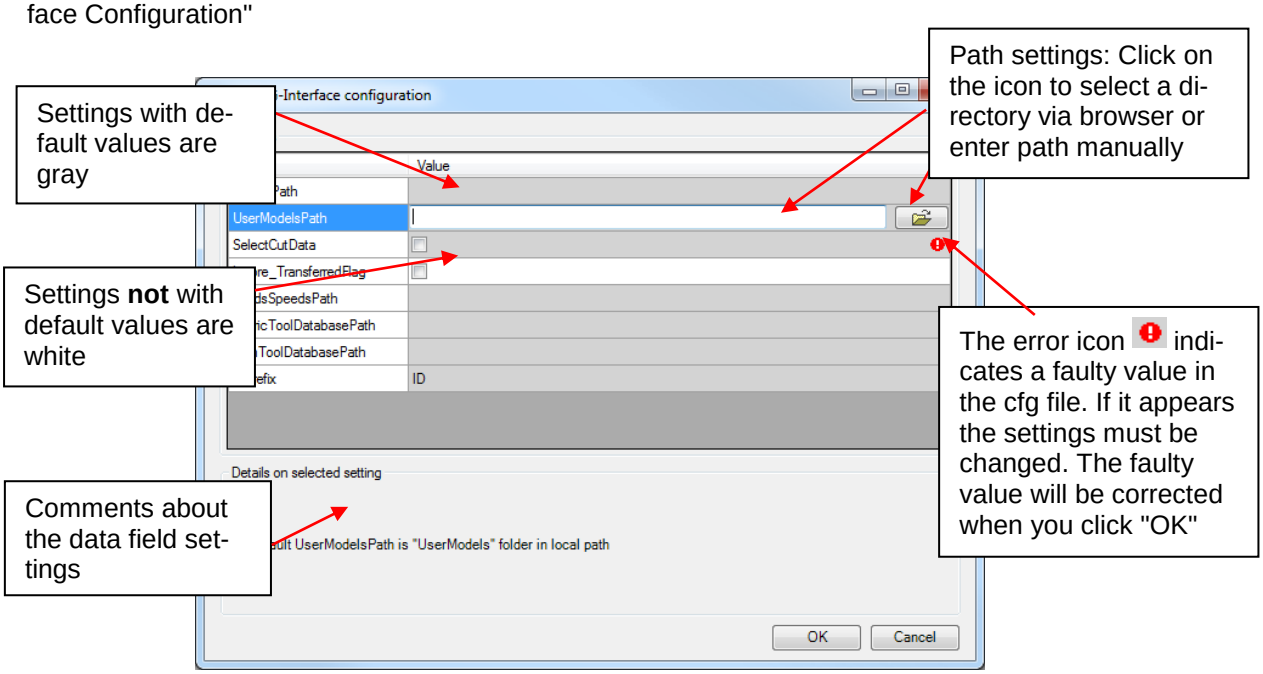


Figure 2 Instruction for the WT-NX-Interface configuration

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

The configuration window reads and stores settings in the file `WT-NX-Interface.cfg` which is located in `[Public Documents]\WT-NX-Interface` .This file can also be edited with a text editor.

Additional information on these parameters is found in the section "Configuration file parameters" of this manual.

Some of these settings are also stored as Registry Variables. The parameters set in the configuration file will be loaded to the Registry Variables when you run the software next time.

Note: Total length of path and filenames must not exceed 255 characters.

Exchange path

Change the configuration of the data exchange folder if required. The file with the current NX database directories is stored in this folder.
`OutputPath = C:\Users\Public\Documents\WT-NX-Interface\Exchange`

Note: Changes to this configuration are not required if you installed the interface software with default settings.
Note: Each user must have an individual data exchange path. The information is stored only temporarily and should not be written over by other users who work simultaneously.

User Models path

Change the path to a location in the network where all NX users can access them. The 2D tool shapes (dxf and geo files) are stored in this folder
`UserModelsPath = C:\Users\Public\Documents\WT-NX-Interface\UserModels\`

Note: This configuration is not required if the default installation path was used to install the interface software.

Cutting Condition Selection

If `SelectCutData` is deactivated (`False` = default), all cutting conditions are transferred.

If this parameter is active (`True`), cutting conditions for work materials are transferred.
For tool assemblies, tool lists and machine tools 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	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.
machine tools data	The interface imports all cutting conditions available for one material only. The cutting condition window of the first tool of the machine will be displayed and a cutting condition must be selected manually. The material for this value will be registered by the interface and is used to preselect the cutting condition for all succeeding tools of the machine. 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 it manually.

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

Transferred Flag

If the transferred flag is activated, no 2D tool shape is created if the tool was transferred before. (For a description of the "transferred flag" see later chapter.)

Set the "Ignore_TransferredFlag" to "false" after a longer period of problem free operation.
```
ignore_TransferredFlag = true
```

Note: The "transferred Flag" should be ignored for installation and tests.

User specific library paths

The settings `FeedsSpeedsPath, MetricToolDatabasePath` and `InchToolDatabasePath` are obsolete since version 4.7.

T-Prefix for Tool ID

The Tool Identification is defined as a unique numerical value within *WinTool*.

A prefix can be defined. It will be automatically set in front of each Tool Identification (e.g. Tool Identification is "`606012`" and prefix is "ID" the Tool Libref will be "`ID606012`" and the associated 3D Model must be "`ID606012.prt`").

Change the prefix characters if required:
```
T-Prefix = ID
```

Note: No changes are required if "`ID`" as prefix is accepted.

Note: The 3D model that will be created gets the same name as the tool. That means including the prefix!

Custom Tool Definitions Path

Location of the custom tool definition files. Change the path to a location in the network where all NX users can access them. Default path is
```
C:\Users\Public\Documents\WT-NX-Interface\CustomTool
```

Configuration of WTMakeList.cfg

See WT-MakeList manual for details: "Start" > "All Programs" > "WinTool" > "WT-NX-Interface">"WT-MakeList-Manual"

Configuration of NX

Setup buttons

Start up NX to find the file which contains directories with NX customizations. Open "Help" > "Log file" (German edition: "Hilfe" > "NX Protokolldatei") and search within the log file for the variable `"UGII_CUSTOM_DIRECTORY_FILE"` which stores the path to the customization file, e.g.
`"D:\CAM\NX 8.5\ugii\menus\custom_dirs.dat"`

Make sure you have write access for this file and open it with a text editor, e.g. Notepad.

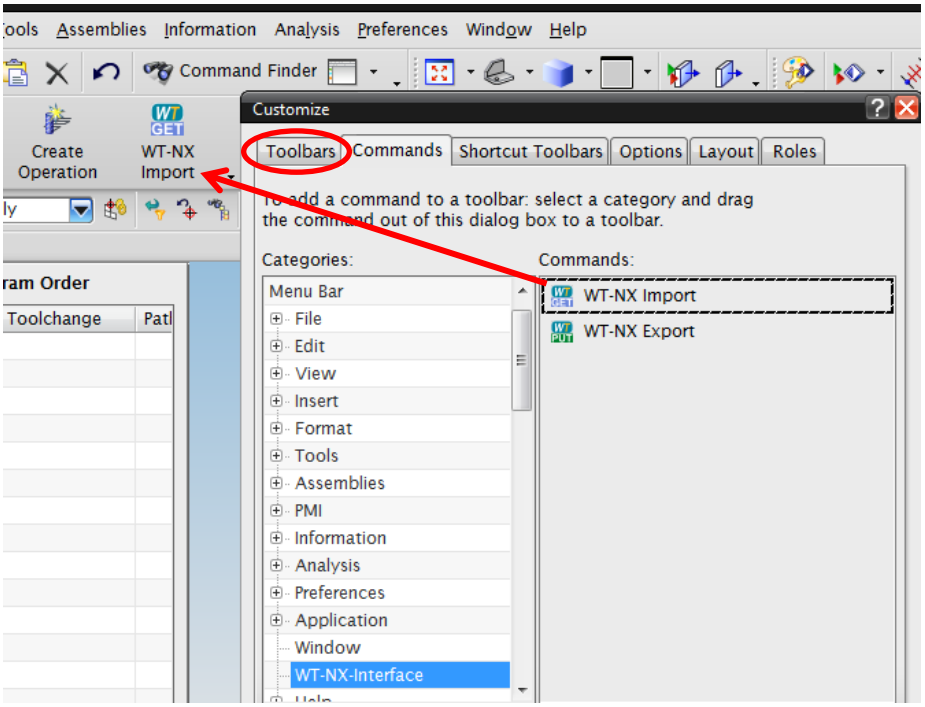
Add the following line at the bottom of the file and save it:
`$WT_NX_Integration_Path`

Info: `WT_NX_Integration_Path` is an environment variable set by the interface installation procedure. It stores the directory which contains the menu files, buttons and journals that start the WT-NX-Interface functions "Get" and "Put". The directory is `<WT-NX-Interface installation directory>\NX Integration\`

Restart NX and open the NX manufacturing application. Right click anywhere on the menu bar and choose "Customize...". Select the "Commands" tab.

NX 8.5 and earlier

The buttons are in the "WT-NX-Interface" menu bar:

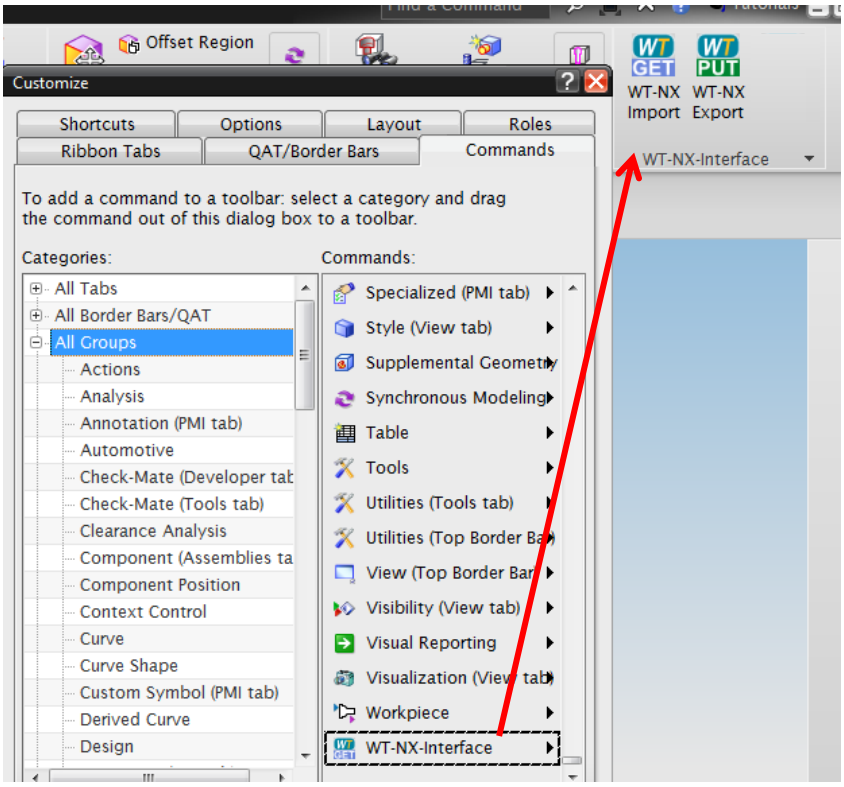


Drag and drop the buttons in to an existing toolbar or create a new one in the "Toolbars" tab and drag the buttons into it.

Figure 3 Drag & Drop into a toolbar

NX 9 and later

The "WT-NX-Interface" group is located in "All Groups":



Drag and drop the "WT-NX-Interface" group into a ribbon.

Figure 4 Drag & Drop into a ribbon

Custom Tool Definitions

Custom tool definitions contain user specific datafield definitions which override the definitions of the WT-NX-Interface tool types. WT-NX-Interface will search the configured "CustomToolPath" directory for custom tool definitions.

Changing existing tool type

To change an existing tool type, the files have to be named the same as the configuration within the *WinTool* Class configuration, either preceded with "UG" or with "NX" (both will work) and have the extension ".xml".
Example: NX0201.xml corresponds to the NX tool type /UG0201.

Creating new tool type based on existing tool type

If the numbers of the file name don't correspond with a *WinTool* NX-Interface tool type, a new NX tool type is created and displayed in the tool type selection. The new tool type is based on an already existing NX tool type with the same parameter values T, ST, UGT and UGST as in the file.
Example: NX9948.xml with <FieldMapping>attributes T="02" ST ="01" UGT="01" UGST="01" creates a new tool type with the type /UG9948, based on the tool type /UG0201.

It will be displayed as a new tool type:

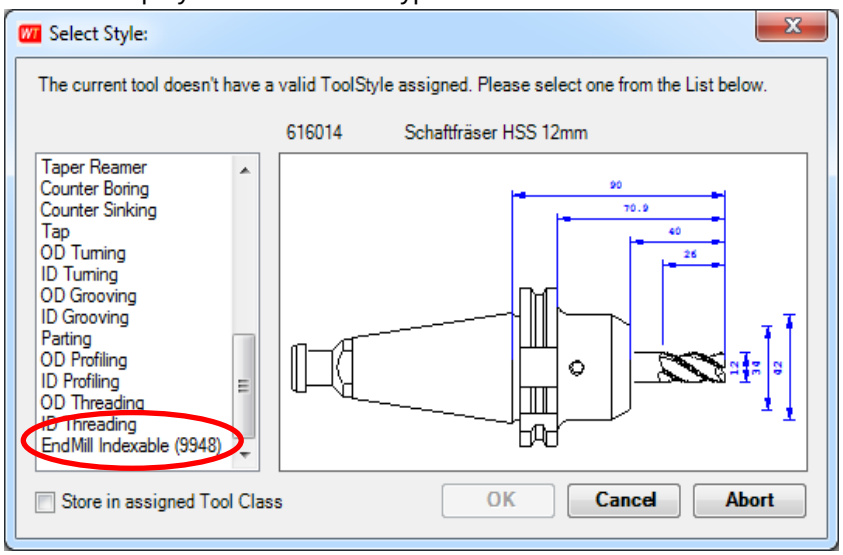


Figure 5 Select a Tool Style from List

Note: Chapter Supported NX tool types shows all tool types

The example custom tool definition "NX0201.xml" which applies to the tool type EndMill (/UG0201) is in the "Sample Data" folder in the WT-NX installation directory.

Format and language definition

Tool definition files have the following format:
Sample:

```
<?xml version="1.0" encoding="utf-8" ?>
<FieldMapping T="02" ST ="01" UGT="01" UGST="01">
  <NXParam name="DIA" value="WT(Tools.DMC)" />
  <NXParam name="FN" value="WT(PartsNG.Teeth)" />
  <NXParam name="HEI" value="WT(PartsNG.CLength)" />
  <NXParam name="HLD" value="IIF(WT(Tools.MachineNr)=21, 'HSK_63', '')" />
  <NXParam name="OA" value="WT(Tools.CMainArc)+90" />
  <NXParam name="RELT">
    <SELECT value="WT(PartsCut.CArc)">
      <CASE equals="85" value="0" />
      <CASE equals="82" value="1" />
      <ELSE value="99" />
    </SELECT>
  </NXParam>
  ...
</FieldMapping>
```

XML Structure

The whole XML content will be wrapped within a "<FieldMapping>" tag. The FieldMapping tag must contain the parameters "T", "ST", "UGT" and "UGST" that define the NX target tool type.
NX target parameters have to be defined as

```
<NXParam name="targetName" value="formula" />
```

If a parameter isn't defined within the file and a default definition within the interface for the target tool type exists, the default definition from the interface will be taken.
The following NX parameters will be taken from the interfaces default definition even if no NX target tool type is defined within the interface - so it isn't necessary to define them explicitly:
FORMAT, LIBRF, DESCR, MATREF, MATDES, TLNUM, ADJREG,CUTCOMREG, HLDDES

To be able to output a value based on a certain numerical *WinTool* value or formula, the SELECT statement can be used:

```
    <SELECT value="formula">
      <CASE equals="value/formula" value="value/formula" />
      ...
      <ELSE value="value/formula" />
    </SELECT>
```

Usable functions

Following functions can be used within formulas. They are case-insensitive:

WT(Table.Field)
The value will be taken from the specified Table/Field from within the *WinTool* database
to distinguish between namegiving, cutting and holder part of a tool, the substitutions "Tools", "PartNG", "PartCut" and "PartHolder" can be used.
Boolean fields (e.g.WT(Tools.NonMetric)) are evaluated as numbers: True = 1, False = 0
Adjustment Values
"PartNG", "PartCut" and "PartHolder" include the corresponding ToolParts table values "AdjDMVal", "AdjZVal", "AdjArcVal" and "AdjXVal". Example: WT(PartNG.ToolPart.AdjZVal) to use the adjustment length of the namegiving component.
Note for users of *WinTool* 2014 and newer: Use IsNull(WT(PartNG.ToolPart.AdjZVal)) to check if the value is empty before using it. Otherwise mathematical operations will fail when *WinTool* 2014 or newer is used. Example:
"IIF(WT(PartNG.CFreeArc)=0, IIF(IsNull(WT(PartNG.ToolPart.AdjZVal)), WT(PartNG.Zinfluence), WT(PartNG.ToolPart.AdjZVal)), WT(Tools.CLength))"

IIF(Expression,TrueValue,FalseValue)

This function is used to select a certain value based on another value. Expression, TrueValue and False-Value can contain other functions.
An expression must use one of the following operators to compare values:

Comparison type	Syntax	Remarks
Greater than	gt	Only numerical values
Lesser than	lt	Only numerical values
Greater than or equals	gt=	Only numerical values
Lesser than or equals	lt=	Only numerical values
Equals	=	Numerical and string values
Not equals	!=	Numerical and string values

Example: IIF(WT(PartNG.Zinfluence) gt= 0, WT(PartNG.Length), WT(PartNG.Zinfluence))

Math Expressions

Supported math operators are: +, -, /, *, sin(), cos(), tan().
Brackets '(' and ')' can be used, e.g. (0.5 + WT(PartNG.Zinfluence)) * sin(45)

Defining Text

To tell the interface that something written as a value within the definition should be directly taken as text, the according formula part has to be surrounded by single quotes (' ')
To concatenate text, use @. Example: WT(Tools.Descript) @ '-Tool'

Use of WT-NX Interface

Import tool assemblies in NX

In NX, click on the get Button and select a tool assembly, a tool list, or all tools for a machine type.

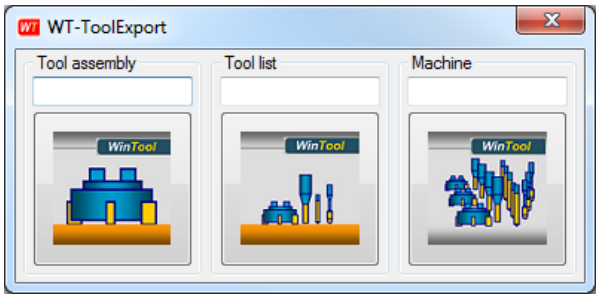


Figure 6 WT-ToolExport menu

If no valid user classification is set for a tool assembly, you will be asked to assign a tool type.

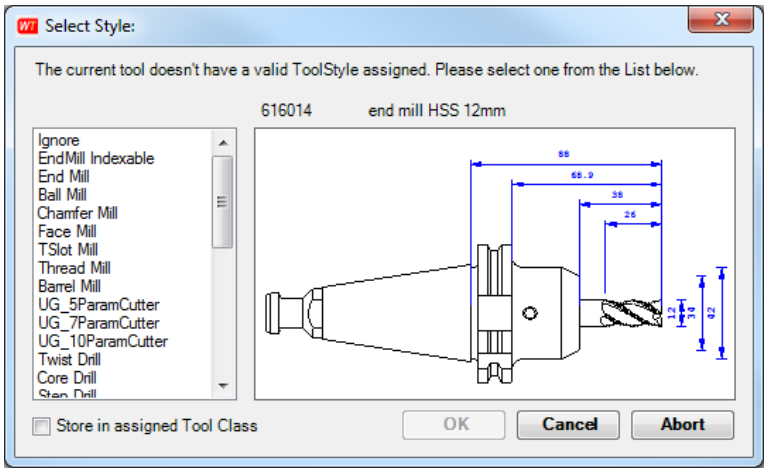


Figure 7 Select a Tool Style

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 NX, 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).

Click “OK” to confirm. “Cancel” doesn’t transfer the current tool assembly, “Abort” stops the whole transfer process.

If the setting `SelectCutData` (see page 9) is enabled, the cutting conditions are shown and you can select the condition which will be transferred with the tool assembly. Press "Cancel" to transfer no condition, "Abort" to abort the transfer.

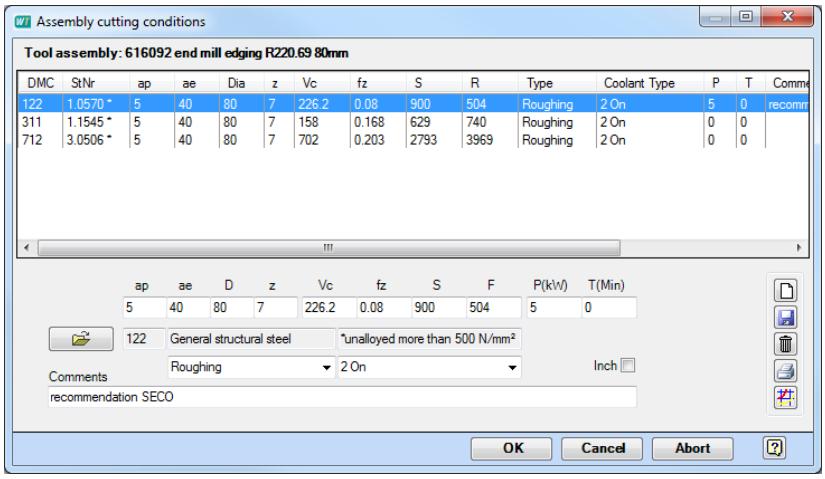


Figure 8 Select an assembly cutting condition from list

The tool assemblies are imported now and can be used in NX.

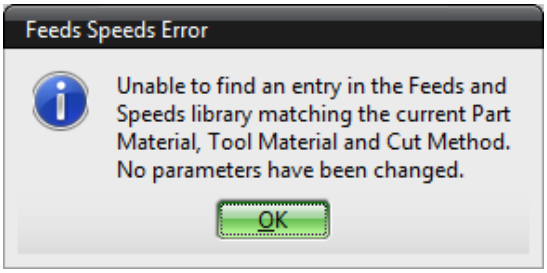
Import Tool Assembly Cutting Conditions into Operation

The following steps are based on NX version 7.5. They can be different on other versions.

To use the cutting conditions associated with a tool assembly in an operation, the following settings must be checked in the operation:

1. The geometry of the operation must use the material of the cutting condition. To select a material go to "Geometry" → "Geometry" → "Edit button" → "Material" → "Edit button"
2. The "Cut Method" of the "Method" must be set to the cutting method of the cutting condition. To select the cutting method, go to "Path Settings" → "Method" → "Edit button" → "Cut Method"→ Edit button. Select the appropriate cutting method, e.g. for a milling operation "WT_MILLING"
   Note: If the "Edit button" of the "Method" is not active, create a new method or select a different, editable method in the list.

The cutting condition data can now be loaded:
Go to "Path Settings" → "Feeds and Speeds button" → "Automatic Settings" → "More" and click "Reset from Table button" to load the data.



If this message is displayed, go through steps 1 and 2 and check if the material and the cutting method are correct. Then try to load the cutting condition data again.

Figure 9 Error message

Export used tools to WinTool

All tools that are used in an operation will be exported into a *WinTool* tool list by pressing the Put button.

The following window will open:

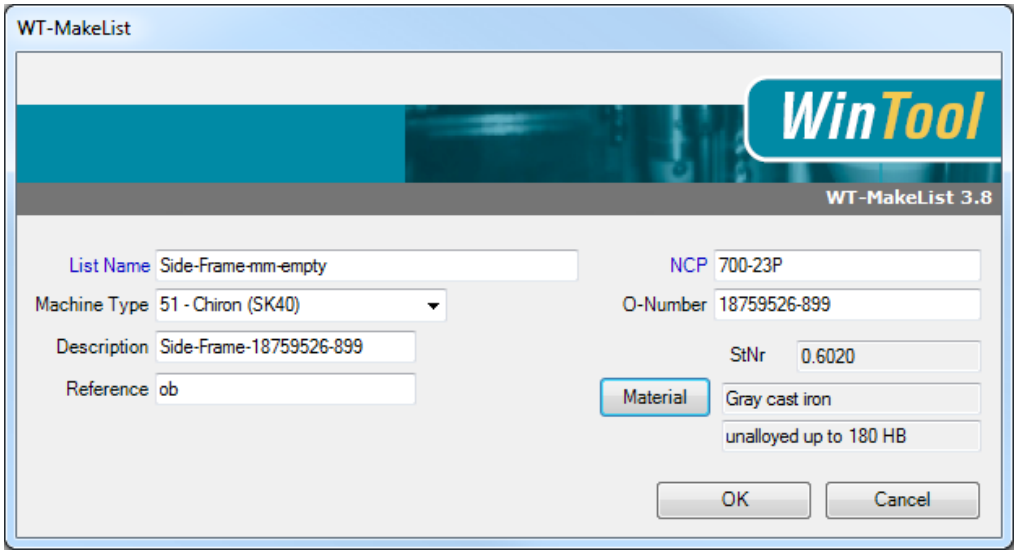


Figure 10 Enter a List name in the WT-MakeList

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

Preparations in WinTool Database

User Classification

Add the corresponding NX tool type in the data field "Codes for CAM" of each *WinTool* classification, which is used for assemblies in the format:
`/UG0201`

In *WinTool* select in menu "Tools" "Settings" > "Classes", then select the classification.

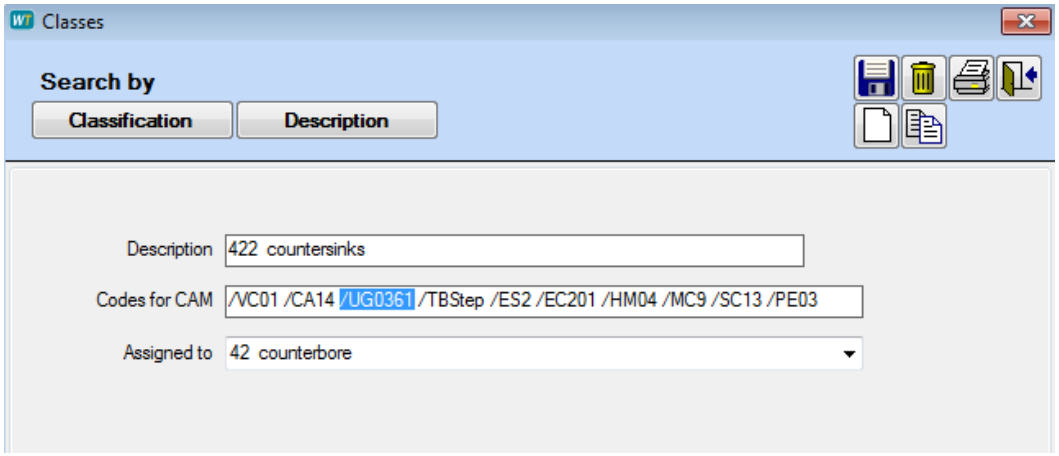


Figure 11 Add Codes for CAM and Description

See samples in the supplied database and in section "Supported NX tool types" (see Annex).

Work materials

WinTool comes with a database of over 1000 work materials structured in about 100 different material classes. Select the work materials you use and remember the assigned material classification. Later on this will help you to easily find and select your work materials to link in your *WinTool* cutting database.

Note*:* Missing work materials can be added in *WinTool* in menu "Tools" "Settings" > "Materials".

Technology library

For each tool assembly, cutting data for different work materials can be stored in a table. If for a desired work material cutting data is stored, the corresponding values are transferred by the WT-NX-Interface automatically.
If multiple cutting data is stored for one work material, all of them will be transferred. If the setting `SelectCutData = true`, the cut data is transferred depending on the work material and the import mode.

Note: If the files are already present in the destination folder, all technology data will be overwritten.

Tool Geometry Data

Be sure to record the tool component data according to the *WinTool* documentation.

Only classified tool assemblies will be transferred. Each tool assembly requires a "namegiving component" and a "cutting component", which will be set in the component window.

Each tool assembly must be linked to a *WinTool* machine type.

Special data fields for assemblies

Tool Orientation Angle (OA) - C14

This angle is calculated automatically. It is possible to overwrite the calculated by a user defined value. The value must be entered in the *WinTool* user field C14 of the assembly (tag "CAM").

Insert position (INSP) - C15

The position is automatically set to topside (value = 1). If the insert should be positioned underside (value = 2), enter 2 in the *WinTool* user field C15 of the assembly.

Transferred Flag

Whenever a tool is exported from *WinTool* to NX, the geo models will NOT be deleted in the user models path, thereafter. Therefore it will not be necessary to generate these models again and again with each transfer. If the "transferred flag" in the WinTool assembly window\folder “Geometry” is set, a geo file will not be created again.

Note: If a tool assembly will be rebuild in *WinTool* using the button “rebuild tool assembly data and drawing”, the transfer flag will be automatically deleted.

Note: You can use the *WinTool Shape* module to check the 3D assembly prior to importing it to NX by using the shape module that creates a geo file of the tool assembly contour. You find the Shape module in the export function of the *WinTool* tool assembly window.

Custom Tool Assembly Contour

If a contour of a tool assembly cannot be created automatically with the Shape-Generator, you can create the DXF file manually.
Note: Only the tool types "Form Mill" (/UG0251) and "Form Drill" (/UG030401) use the geometries of the CUT layer to create the cutting tool in NX.
The HOLDER and SHANK geometries are used in all tool types except for turning tool types.

- Open the tool assembly and use the Shape-Generator button 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.
- Then modify it with Vector or any other DXF editor until it is exact. You must use the layers CUT, NOCUT, and SHANK:

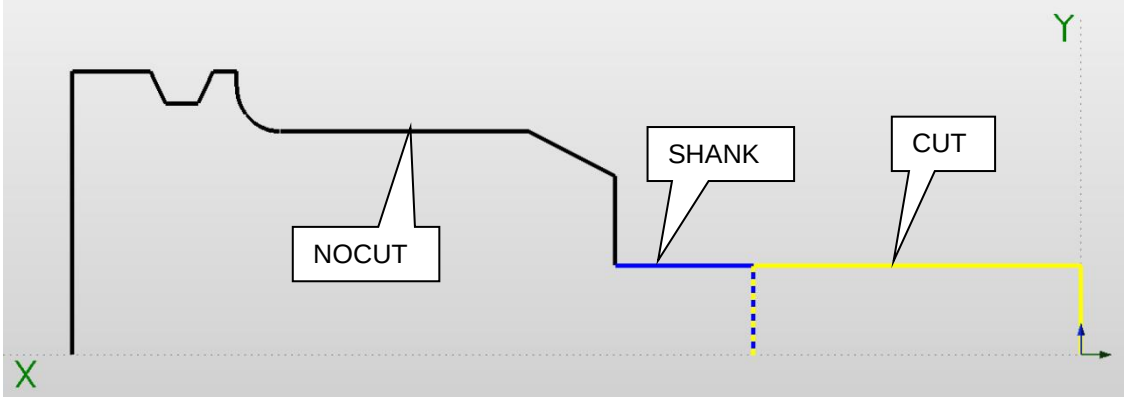


Figure 12 Tool Assembly Contour

- The CUT layer is independent from the shank and holder closed contour. 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.
- The SHANK and NOCUT layer together must form a closed contour. 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.
- Assign the new DXF to the *WinTool* tool assembly: Check the box “User Model” in the tab “CAM” in the row containing "Unigraphics NX". If it is missing, please activate Unigraphics NX in "Settings" > "CAM settings" on the main *WinTool* screen.

Custom 3D Tool Assembly

If the 3D model of the assembly cannot be created automatically because of functional limitations of the shape module, no NX Part will be created. If you need a realistic NX Part for simulation, you need to create it manually. Proceed as follows:

- Create an assembly within *WinTool* as usual.
- On the page “Geometry” of the *WinTool* assembly activate the flag "User model" in the row "Unigraphics/Siemens NX". This will indicate that an independent NX Part is available which should be used together with this assembly.
- **Note**: The model name in the WT-NX-Interface must be the same as the assembly ID and prefix as configured.)

To create a NX Tool Assembly Part, all the components of the *WinTool* assembly must have a NX model. If it doesn't have one it must be created first, based on a STL model.

Create the missing NX Component models as follows:

- Create a new model in NX. Use the Item No. as file name.
  Save the file in the NX location for custom specific tool assemblies. The location is stored in the NX variable `UGII_CAM_LIBRARY_TOOL_GRAPHICS_PATH` which is by default `<NX installation folder>\MACH\resource\library\tool\graphics`
- To import the STL, go to "File" > "Import" > "STL"
- The axes must be oriented like this:

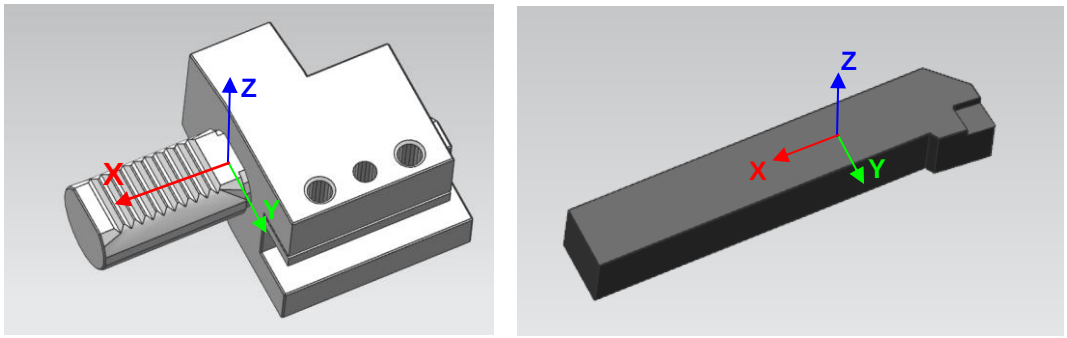


Figure 13 3D simulation holder and turning tool

- Place all geometries on layer 1. Use "Format" > "Move to Layer" if necessary.
- If all NX Components are created, the create the NX Tool Assembly Part as follows:
- Create a new model in NX. Use the prefix plus the assemblies ID # as filename (i.e. "`ID615015.prt`")
  Save the file in the NX location for custom specific tool assemblies. The location is stored in the NX variable `UGII_CAM_LIBRARY_TOOL_GRAPHICS_PATH` which is by default `<NX installation folder>\MACH\resource\library\tool\graphics`
- Import all NX Components of the Tool Assembly, starting with the holder:
  Go to "File" > "Import" > "Part". Select "WCS" as "Destination Coordinate System" and click "OK". Then select the NX Component. Place the component by setting the "Output Coordinates" or by moving it via mouse. Repeat until all components are imported
- The complete Tool Assembly should look like this:

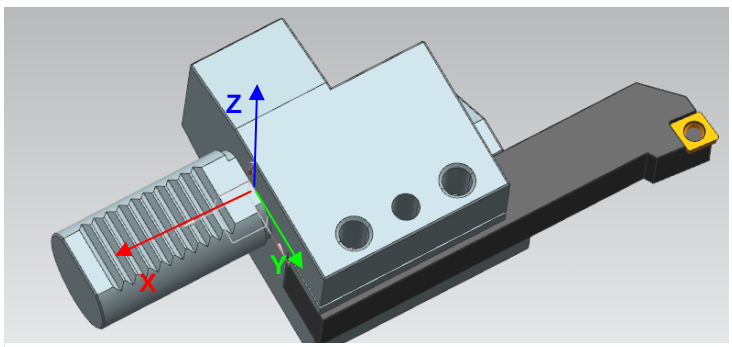
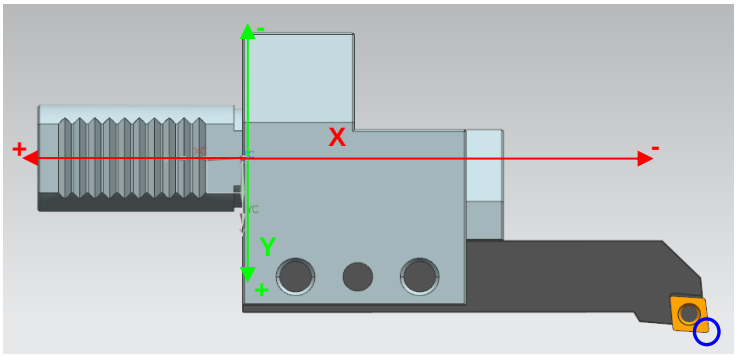


Figure 14 3D simulation assembly tool

- Add the following attributes to the Tool Assembly
  CAM_TOOL_ATT_X(1) = Tool tip X coordinate
  CAM_TOOL_ATT_Y(1) = Tool tip Y coordinate



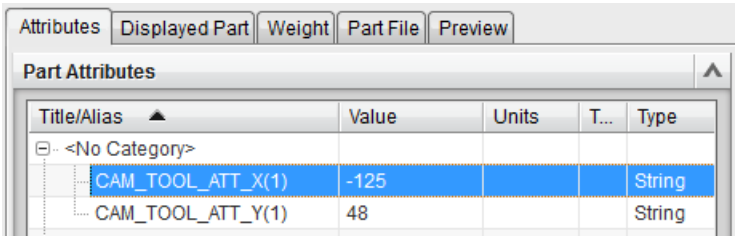


Figure 15 Add attributes to Tool Assembly

Linking NX Parts to the WinTool Database

Manually created models can be linked to the database to manage them easily and to automatically start NX with these parts. Use the hyperlink field implemented for components as well as for assemblies.

Software Structure

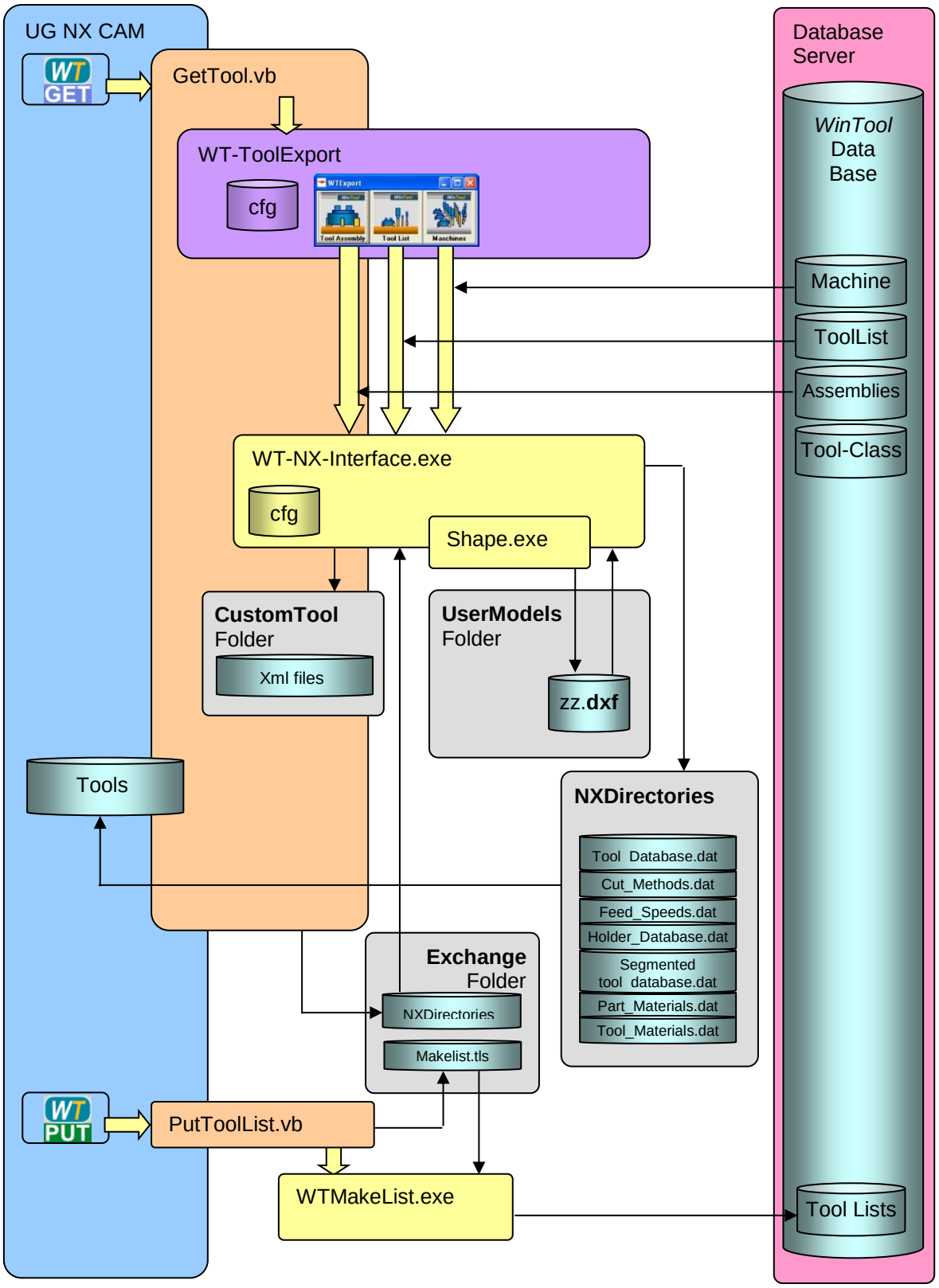


Figure 16 Software Structure

NXDirectories = Database file directories of NX:
UGII_CAM_LIBRARY_TOOL_METRIC_DIR, UGII_CAM_LIBRARY_TOOL_ENGLISH_DIR, UGII_CAM_LIBRARY_FEEDS_SPEEDS_DATA_DIR

Annex

Troubleshooting

Machine Tool View adjustment

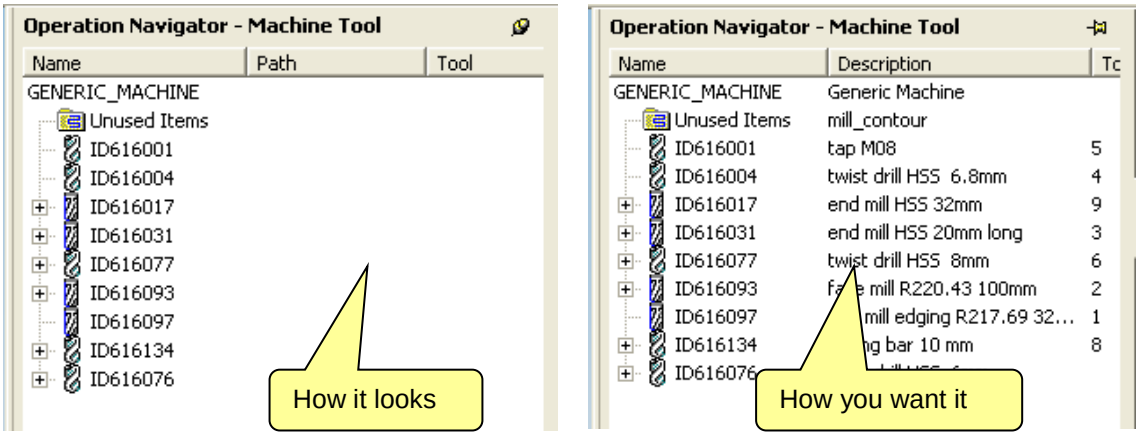


Figure 17 Is - Should; Machine Tool view

Place mouse over column-description and use right click to activate configuration. Unselect unused columns.

Reload NX Parameters

NX sometimes does not reload data files. When a tool is not transferred or the 3D model is not created it often helps to reload the NX configuration file to adjust the environment.

Activate the "Machining" application (Start). Select in the menu section:

```
Preferences
    Manufacturing
```

Activate the "Configuration“ tab and select "browse" for "Configuration file". Copy the file "cam_general.dat" to "cam_general_datos.dat" and select the copy to be loaded.

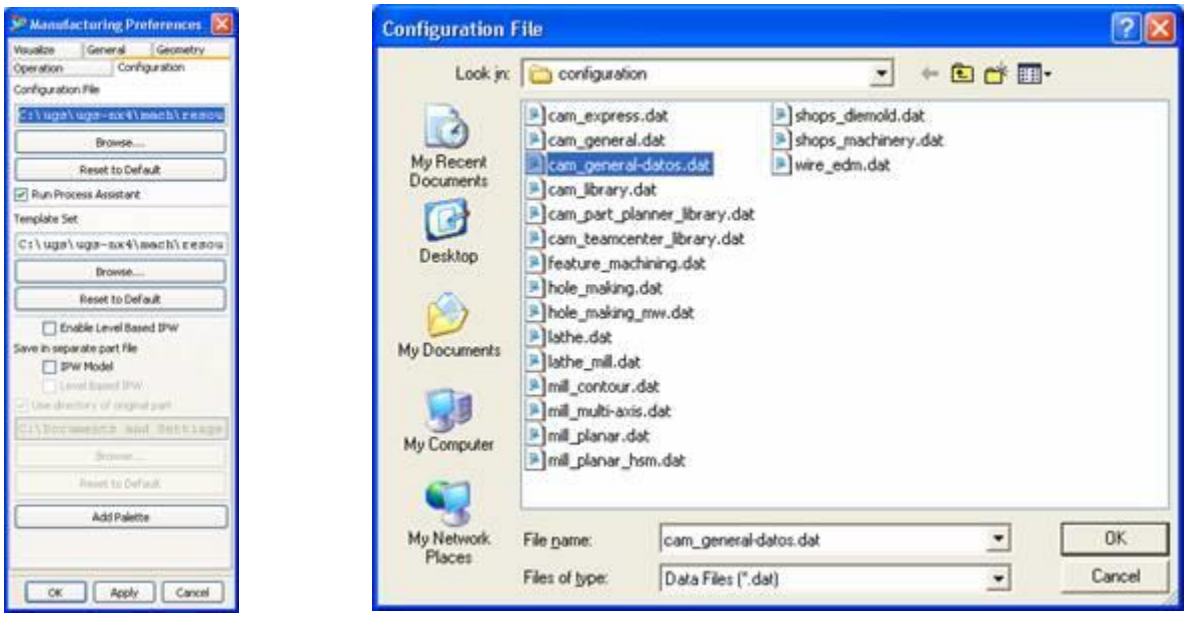


Figure 18 Activate the Configuration File

Next time you want to reload the configuration file you can switch back to the original file. NX will not reload the file if the name is the same as the already loaded.
Sometimes it also helps to additionally reselect the "template set".

Adjust tool import macro for NX 7 and below

The macro "GetTool.vb" contains instructions that are only supported in NX 7.5 and newer. If you want to use the interface with version 7 and below, you must apply the following changes:

1. Open the file "GetTool.vb" in a text editor. Make sure you have write access. It is located in `<WT-NX-Interface installation directory>\NX Integration\application\`
2. Go to line 122 and replace the following lines:
   ```
   camTool = NXObjectManager.Get(tool)
   camTool.UpdateFromLibrary()
   ```
   with this line:
   ```
   theUfSession.Cutter.UpdateFromLib(tool)
   ```
3. Save the file

Error Messages

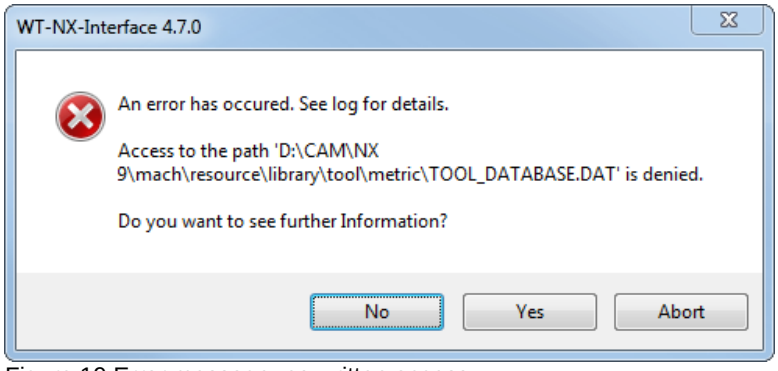


Figure 19 Error message_no written access

Cause: The current user doesn't have write access on the NX database directory.

Solution: Write access must be granted to the directory. See the Requirements for a complete list of NX directories that need write access.

Figure 20 Error message_older version

Cause: WT-NX-Interface is used with NX version 7.5 or below.

Solution: The tool import macro must be adjusted. For detailed instructions, see chapter "Adjust tool import macro for NX 7 and below" on page 26.

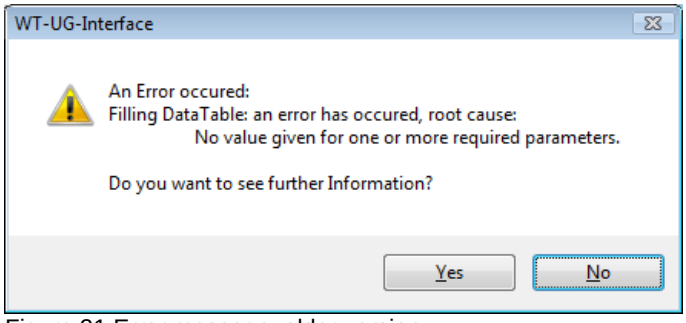


Figure 21 Error message_older version

Cause: A version of *WinTool* below 2009 is used.

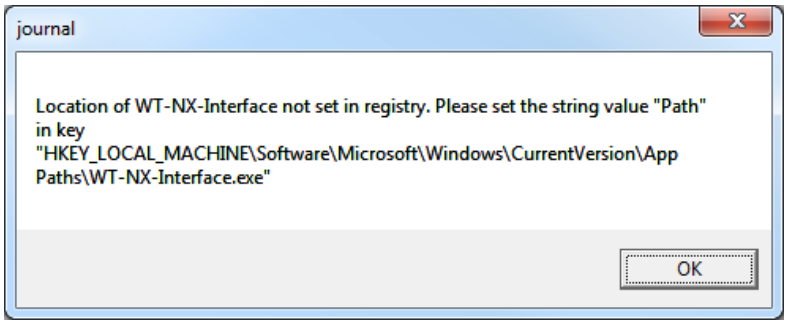


Figure 22 Notification_older version

Cause: The registry stores the installation path to the WT-NX-Interface. If the path is not found, this error message appears.

Solution: Open the registry editor:

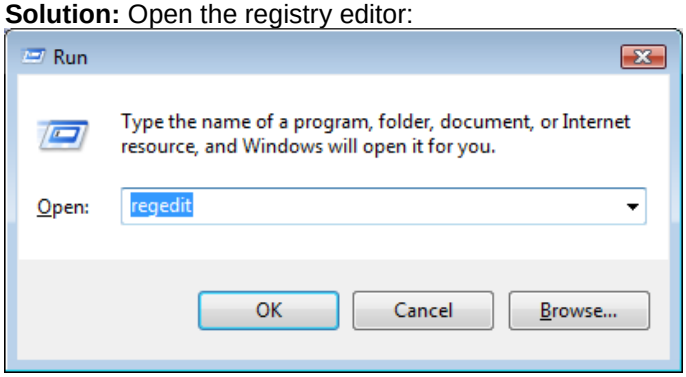


Figure 23 Error message_solution

Create the String value and set the installation directory as data:

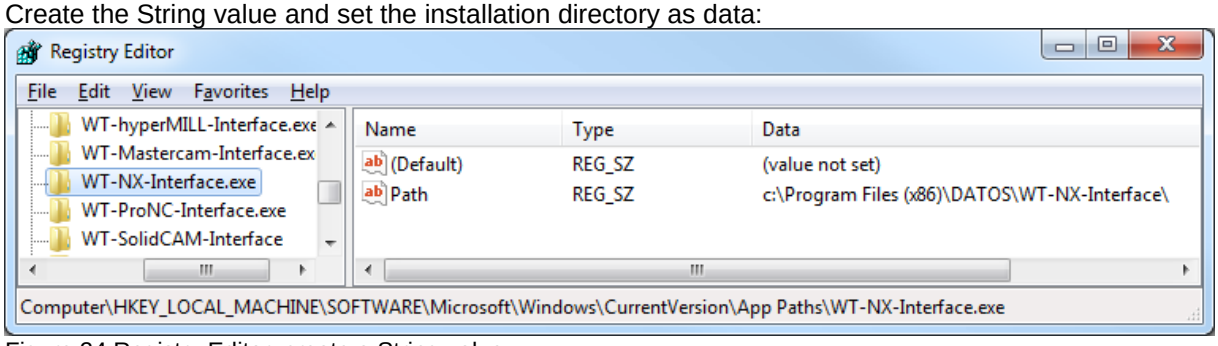


Figure 24 Registry Editor_create a String value

NX File structure

File types

*.dat
These files contain the actual data in the database

*. def
These files contain the information on configuring the dialogs

*. tcl
These files contain a script in the TCL language controlling the functions found in the dialogs. These files are called the event handlers and it is in these event handlers where other TCL files are included.

The actual file paths can be seen by going to Help-→ NX log file.

NX Windows Environment Variables
UGII_BASE_DIR
UGII_ROOT_DIR
UGII_LICENSE_FILE

NX internal Environment Variables directory
$UGII_ROOT_DIR/ugii_env.dat

Important NX internal Environment Variables
UGII_CAM_BASE_DIR=${UGII_BASE_DIR}\mach\

UGII_CAM_CUSTOM_DIR=${UGII_CAM_BASE_DIR}**custom**
UGII_CAM_RESOURCE_DIR=${UGII_CAM_BASE_DIR}**resource**
UGII_CAM_LIBRARY_DIR=${UGII_CAM_RESOURCE_DIR}**library**

# tool library
UGII_CAM_LIBRARY_TOOL_DIR=${UGII_CAM_LIBRARY_DIR}**tool**
UGII_CAM_LIBRARY_TOOL_ASCII_DIR=${UGII_CAM_LIBRARY_TOOL_DIR}ascii\
UGII_CAM_LIBRARY_TOOL_GENIUS_DIR=${UGII_CAM_LIBRARY_TOOL_DIR}genius\
UGII_CAM_LIBRARY_TOOL_INCLASS_DIR=${UGII_CAM_LIBRARY_TOOL_DIR}inclass\
UGII_CAM_LIBRARY_TOOL_ENGLISH_DIR=${UGII_CAM_LIBRARY_TOOL_DIR}english\
UGII_CAM_LIBRARY_TOOL_METRIC_DIR=${UGII_CAM_LIBRARY_TOOL_DIR}**metric**

# Part files for library cutting tools
UGII_CAM_LIBRARY_TOOL_GRAPHICS_PATH=${UGII_CAM_LIBRARY_TOOL_DIR}**graphics**

# Technology:
UGII_CAM_LIBRARY_FEEDS_SPEEDS_DIR=${UGII_CAM_LIBRARY_DIR}**feeds_speeds**
UGII_CAM_LIBRARY_FEEDS_SPEEDS_ASCII_DIR=${UGII_CAM_LIBRARY_FEEDS_SPEEDS_DIR}**ascii**

Feeds and speeds: feeds_speeds.dat
Cutting methods: cut_methods.dat
Part Materials: part_materials.dat

# machine tool library
UGII_CAM_LIBRARY_MACHINE_DIR=${UGII_CAM_LIBRARY_DIR}**machine**
UGII_CAM_LIBRARY_MACHINE_ASCII_DIR=${UGII_CAM_LIBRARY_MACHINE_DIR}**ascii**

# Setting the value to "1" forces NX to load the tools in the tool library into memory
# The interface setup creates this Windows environment variable which is used by NX
UGII_CAM_LIBRARY_TOOL_ASCII_LOAD_LIMIT = 1

WT-NX-Interface Configuration file parameters

Following configuration parameters can be set in the WT-NX-Interface.cfg file (lines starting with a “#” sign will be ignored). By running the module, the values will also be stored in the corresponding Windows Registry. Restart NX to make them effective.

OutputPath	Path to write the exchange files to.
UserModelsPath	Path to write the user model files to.
SelectCutData	Enable/disable (true/false) material based cutting condition selection
ignore_TransferredFlag	Ignore transferred flag in CAMtools table (true/false)
T-Prefix	Defines a string prefix to be added to LIBREF output column
CustomToolPath	Path which contains the custom tool definition files

Supported NX tool types

NX Nr.	NX Tooltype English	NX Tooltype German	*WinTool* Classification
01-01	OD Turning	Aussen - Drehen	/UG0101
01-02	ID Turning	Innen - Drehen	/UG0102
01-11	OD Grooving	Aussen - Stechen	/UG0111
01-12	ID Grooving	Innen - Stechen	/UG0112
01-14	Parting	Abstechen	/UG0114
01-21	OD Profiling	Aussen halbrund stechen	/UG0121
01-22	ID Profiling	Innen halbrund stechen	/UG0122
01-31	OD Threading	Aussen - Gewindedrehen	/UG0131
01-32	ID Threading	Innen - Gewindedrehen	/UG0132
02-01	End Mill	Schaftfräser ohne WPL	/UG0201
02-02	End Mill Indexable	Schaftfräser mit WPL	/UG0202
02-03	Ball mill	Gesenkfräser	/UG0203
02-05	Chamfer mill (>= NX 8.0)	Fasenfräser (>= NX 8.0)	/UG0205
02-31	Thread mill	Gewindefräser	/UG0231
02-12	Face mill (< NX 8.0)	Messerköpfe (< NX 8.0)	/UG0212
02-21	T-Slotter	T-Nutenfräser	/UG0221
02-51	Form Mill Uses cutting contour of tool assembly shape DXF to create segments	Formfräser Der schneidende Teil der DXF Kontur wird importiert	/UG0251
02-90	UG_5ParamCutter	5-Parameter-Fräser	/UG0290
02-91	UG_7ParamCutter	7-Parameter-Fräser	/UG0291
02-92	UG_10ParamCutter	10-Parameter-Fräser	/UG0292
02-93	Barrel mills	Ballenfräser	/UG0293
03-01	Twist Drill	Spiralbohrer	/UG0301
03-03	Core drills	Aufbohrer	/UG0303
03-04	Step drills	Stufenbohrer	/UG0304
03-04	Form Drill, based on "Step Drill", but uses cutting contour of tool assembly shape DXF	Stufenbohrer, bei dem die Stufen aus der schneidenden DXF Kontur importiert werden	/UG030401
03-21	Spot Drill	NC - Anbohrer	/UG0321
03-22	Center Drill	Zentrierbohrer	/UG0322
03-32	Boring	Aufbohrwerkzeug	/UG0332
03-42	Taper Reamer	Kegelreibahle	/UG0342
03-51	Counter Boring	Flachsenkung	/UG0351
03-61	Counter Sinking	Kegelsenker	/UG0361
03-71	Taps	Gewindebohrer	/UG0371
03-41	Reamer	Maschinenreibahle	/UG0341
04-02	Probe	Messtaster	/UG0402
	Ignore (not in NX)		/UG00

History

4.11.1

- Added compatibility with NX 12

4.11

- Added compatibility with NX 11
- Corrected custom tool import error

4.10

- Added compatibility with NX 10
- Added support for NX tool type "Probe"
- Tap tools import: Corrected tip length and added designation
- Milling tools with a tip angle: Importing tip angle in "End Mill" and "End Mill Indexable" tool types.
- Corrected import of material name in cutting condition data

4.9

- NX 8.0 and newer: Importing tool neck diameter as shank in NX instead of as part of the holder.
- Corrected import of face mills with button inserts
- Improved import of custom DXF profiles for tool types "From Drill" and "Form Mill"
- Corrected custom tool definitions:
  - Using formulas of unused fields in tool class, e.g. "User-1"
  - Corrected unnecessary decimal places when using a formula
- Added tool point angle to tool class "Core Drill"
- Setting NX tool database value ZOFF to 0
- Importing "Adjust Register" and "Cutcom Register" on single tool assembly and machine tools import
- Added "Coolant Through" flag import

4.8

- Corrected import of tool direction
- Corrected tool list export to *WinTool* when using "T-Prefix" setting
- Corrected cutting condition import

4.7

- Supports *WinTool* 2011 – 2014
- Separated program files and user data
- Added custom DXF tool assembly contour support
- Saving NX database files in NX default database directories, therefore removed WT-NX-Reset button
- Supporting user defined tool assembly datafields using custom tool definition xml
- Added new tool types "Form Mill" (`/UG0251`) and "Form Drill" (`/UG030401`) which use the tool assembly cutting contour to create tool segments/steps
- Corrected import of W type inserts
- 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

4.6

- Supports *WinTool* 2013, *WinTool* 2012 and *WinTool* 2011
- Added compatibility with NX 9 and 8.5
- Corrected tip length of step drill and tip diameter of countersink
- Minor fixes in holder geometry calculation and tool list creation
- Simplified configuration of interface buttons in NX
- NX environment variable UGII_CAM_LIBRARY_TOOL_ASCII_LOAD_LIMIT is set to 1 during installation
- Included newest version of WT-MakeList (see WT-MakeList manual for details)

4.5

- Support for *WinTool* 2012
- Included newest version of WT-ToolExport:
  - Resizable search windows
  - Compatible with *WinTool* 2012

4.4

- Renamed interface to WT-NX-Interface
- Updating instead of adding new tool assembly if it is already in imported in NX
- Support for *WinTool* 2011
- Included newest versions of WT-ToolExport and WT-MakeList module
- WT-ToolExport: Start-up time with large databases is quicker
- Corrected import of chamfer mills with corner radius > 0
- Added tool type "Ignore" (/UG00) for tool assemblies that must be ignored on transfer
- Improved error handling

4.3

- Added compatibility with NX 8.0
- Added support for new NX tool type "Chamfer mill"
- Installing WT-MakeList 3.8, updated WT-MakeList tls file version to 2.2.1

4.2

- Improved holder geometry transfer
- Transferring cutcom and length adjust numbers if tool list is imported
- Added support for drills with flat tip
- Transferring upper and lower radius of t-slot mill

4.1

- Added new WT-ToolExport module
- Added WT-NX-Interface configuration window
- Minor update of cutting condition selection window
- Added compatibility with NX 5.0
- Added material based cutting condition transfer (SelectCutData = true)

4.0 Beta

- Using NX Open .Net API instead of C++ API
- Added "Reset" button
- Generating tools with holders instead of toolshapes (.prt)
- Technology data is always transferred
- No support of inch tools

- No support of quiet mode
- Disabled progress windows and "put" for highlighted operations
- Removed obsolete .cfg parameters and 3D Samples for Turning Tools

3.2.9.8144

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

3.2.9

- Support for NX5 and NX6 (signed DLLs)

3.2.8

- No more user models are overwritten
- Diameter now correctly set when tool lists are transferred

3.2.5

- Turning tools and extended Tool_Database implemented
- Lollipop mill supported if classified as barrel mill
- End mill taper angle from WinTool field E1 transferred with higher priority than from G5

3.2.2

- TAPA for Face Mills correctly supported
- HEI value is automatically reduced to FL if any angle is present

3.2.1

- Exchange format changed (no more user9 / user10 fields, new UG fields added)
- Corner radius of End Mills now transferred to COR1 as well as to LCOR
- Quarter round mills supported (with negative corner radius)
- For Shell Mill diameter WinTool diameter A2 is used instead A1
- Tip Length for center drills correctly supported
- User field C14 supported for manual setting of Orientation Angel "OA"
- User field C15 supported for manual setting of Insert position "INSP" (up/down)
- X64 support included (as additional file version)
- Optional export limitation of assemblies within a highlighted operation

3.1

- Correction with tool list import
- Angle E1 correctly supported
- Corner radius for tools with insert supported

3.0

- New shape module implemented (see separate manual).
- Parameters transferred correctly now for Spot drills, Taps and Face mills
- User model flag supported
- Transferred flag supported
- Tool_database.dat is no longer deleted after import but before next data exchange
- Layer 5 automatically is set to selectable (visible)
- Parameter value LOAD_LIMIT=1 is checked during import
- No more message if transfer was successful
- Message text is configurable now (for better translation)
- Shorter timeout when selection is interrupted
- Inch values and parts are supported

- Pathnames with spaces supported
- Fast and extended tools-search with new ToolExport module
- Fully SAL and multy-language support with new ToolExport module

2.8

- Non metric tools supported (parts for simulation is built as inch model)
- Cutter radius supported for face mills
- Pitch and tip angle supported for taps
- Diameter correctly transferred for spot mills
- Layer 5 is automatically set to selectable
- Error message included if UGII_CAM_LIBRARY_TOOL_ASCII_LOAD_LIMIT not set to 1
- Messages translated to English
- Timeout after interruption of tool selection reduced
- No more message if transfer was successful
- Shipped with new Shape module 1.8 (supporting backside cutting and problem free taps)
- Updated manual

2.7

- Spaces allowed in installation- and exchange path
- Support for turning tools, spot drills and barrel mills
- Technology databases are updated during data exchange
- WT-ToolExport module implemented as replacement for not fully satisfying dll's
- No more problems with user rights and SQL-server access
- New WTMakeList module 3.0 implemented (better support for defaults and user rights)
- Shipped with new Shape module 1.7 (see separate documentation)

2.6

- Shipped with new Shape module 1.6.9
- Tool-ID Prefix implemented (for NX 3 users)
- Additional UG Tool Types supported
- Fees&Speeds supported (file replace method)
- Optional "Transferred-Flag" omitting
- Proven User-Model handling
- Geometry definition unified according latest Parts-Geometry-Documentation

List of Figures