

# WinTool Interface for NX

## Version 5.1.1

### Changes

#### 5.1.1

- Added compatibility with NX 2007
- Compatible with WinTool 2022.1.0
- Ensured compatibility with WinTool Versions older than 2020.3.1

#### 5.1.0

- Added compatibility with NX 1953
- Compatible with WinTool 2020.3.1
- Tool List import Improvement
- General Tool import improvement

#### 5.0.1

- Added compatibility with NX 1942
- Compatible with WinTool 2020.3
- General Tool improvement

#### 5.0

- Added compatibility with NX 1899
- Fixed problem where the Tool Database would be Deleted every time a Tool was Imported
- Fixed problem where the Cutting Conditions wouldn't be Transferred
- Implemented new License Mechanism
- Automated generation of a Collision Model when Importing a Tool
- Added default XML's for some Tool types to Improve the general Tool Import

#### 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 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 UG NX 8.0
- Added support for new UG NX tool type "Chamfer mill"
- Included new WT-MakeList module

#### 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-UG NX-Interface configuration window
- Minor update of cutting condition selection window
- Added compatibility with UG 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 (se 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