WinTool Interface for CATIA V5 R19 – R25

Version 2.9

Changes

Version 2.9

- Updated tool types "end mill" and "reamer": Tool neck diameter is imported as a straight neck
- Updated tool type "end mill": Non cutting diameter is imported directly from data field A4

Version 2.8

- Support for *WinTool* 2011 – 2015
- Extended CATIA support to V5 R25
- Corrected and improved boring bar import
- Moved sample data to public documents folder

Version 2.7

- Support for *WinTool* 2011 – 2014
- Separated program files and user data into different directories
- Tool assembly import: Added setting "ToolName" which sets ident-no or alt. code as tool name
- Removed redundant references in DATOS.catvba
- Corrected issue in CATPart tool models where arcs are imported the wrong direction
- Adjusted non cutting diameter import of end mills
- Importing "Ball-end tool" flag of tool types "end mill", "conical mill" and "t-slotter"
- Corrected automatic tool number assignment when language of CATIA is German
- 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
- Single tool assembly import: Transferring ident-no for t-no if "T-No=Ident No" is activated in the machine type

Version 2.6

- Support for *WinTool* 2013, 2012 and 2013
- CATPart creation: Corrected issue where inch tool assembly CATPart models are scaled incorrectly
- Import as CATIA tool assembly: Corrected issue where holder parameters are scaled incorrectly if an inch tool assembly is transferred
- Tool list export: Calculating operating time of used tool assemblies
- Tool assembly import: Corrected issue where WT-ToolExport window is not in foreground
- Improved import of drill tool parameters "tip angle", "tip length" and "body diameter"
- CATIA process data can be used to set values for tool list export (see Export Additional CATIA Data to Tool List)
- Included newest version of WT-MakeList (see detailed changes in WT-MakeList manual)

Version 2.5

- Support for *WinTool* 2012

- Corrected issue in exchange file when regional setting with comma as delimiter is used
- Updated DXF contour specifications to support non-cutting front diameters(see page 20)
- Included newest version of WT-ToolExport:
  - Resizable search windows
  - Compatible with WinTool 2012

Version 2.4
- Support for *WinTool* 2011 and CATIA V5 R21
- Added interface configuration window
- Installing WT-MakeList 3.8.2, updated WT-MakeList tls file version to 2.2.1
- Installing WT-ToolExport 2.1
- Updated transfer of CATIA tool type "MfgConicalMillTool" (/CA15) to use taper angle (G5)
- Added tool type "Ignore" (/CA00) for tool assemblies that must be ignored on import
- Improved error handling and messages
- Removed setting "WinToolAppPath" from configuration

Version 2.3
- Added material dependent cutting condition selection
- Installing WT-MakeList 3.7, updated WT-MakeList tls file version to 2.2
- Transferring tool assembly description as tool name in CATIA
- Setting next free tool number to a imported tool assembly if its *WinTool* T-No is 0
- Tool assemblies do not require a "holder" component anymore
- Adjusted T-Slotter import
- Improved check of configuration file
- Disabled automatic user model (CATPart) attachment to CATIA tool assembly
- Making sure that only one axis system is added to CATPart user models
- Updated manual
- Setup installs .net framework if necessary
- Removed AIX support

Version 2.2.3
- Ensured compatibility with *WinTool* 2009 and *WinTool* 2010

Version 2.2.2
- Using always "." as decimal point delimiter in geo files instead of current region setting delimiter

Version 2.2.1
- Fixed bug in user model generation when Installed on PC with comma decimal separator

Version 2.2
- Added adjustable Coolant Type
- User model of tool can be modified by editing corresponding .dxf shape file
- Tool assembly import with German region setting is working
- More precise user model is generated to avoid open contours
- Improved user model generation of tool assemblies
- Improved calculation of Non Cutting Diameter
- Adjusted transfer of TNr when tools of a machine are exported

Version 2.1

- Extended tool holder support: Tools can additionally be transferred to CATIA as tool assemblies to reduce collision calculation time.
- Improved calculation of Tool Core Diameter, Nominal Diameter, Non Cutting Diameter, Overall Length
- Added support for Tool Cutting Material, Cutting Coolant, Tool Weight
- Integrated Tool List transfer to *WinTool* for each Manufacturing Program
- Tools can be added more than once
- More robust error handling during transfer process
- Usage of extended CATIA transfer file format

Version 2.0

- New implementation of interface in C#
- Tool style and cutting data can be selected during transfer of tool assemblies into CATIA.
- Tool assemblies in tool lists are linked with generated .CATPart file.

Version 1.8

- MfgEndMillTool, MfgFaceMillTool, MfgBoringBarTool: Changed calculation of MFG_TOOL_CORE_DIAMETER

Version 1.7

- MfgEndMillTool: Changed calculation of MFG_TOOL_CORE_DIAMETER

Version 1.6

- T-Number for tool is taken from default T-Number of tool
- Number of flutes is transferred to CATIA
- Non cutting diameter supported (MFG_Tool_Core_Diameter)
- Better support for T-slot mills

Version 1.5.7

- WTCatiaToolimport.CATScrip: Axis System is automatically set.

Version 1.5.5

- Default T-No from assembly is transferred to CATIA field "Tool number".
- CATPart is re-created for assemblies marked as "transferred" if CATPart is missing in destination folder. This functionality is not supported for AIX implementation.
- Diameter and FL correctly supported for CenterDrill and MultiDiamdrill types.
- Tip Length properly calculated for MFGDrill, Chamfering, CenterDrill and MultiDiamdrill.

Version 1.5

- Latest and extended version of Shape module integrated
- Last selected work material is kept as default for next selection
- New WT-Tool-Export module implemented for easier selection of assemblies and problem free access of SQL database
- Extended support of decimals (comma or point) depending on regional settings.
- No more message appears when the tool is transferred
- Axes system is created automatically during automatic creation of assembly Parts.
- The tool axes is now not shown any more to avoid too much any information on screen
- New name for WTCatiaToolImport.CATScript (former name WT2Catia)

- *WinTool* 2007 compatibility (database access)
- CATIA V5 R16 compatibility (tool list export, empty T-number)
- Inch tools are fully supported now
- Corrected transfer of geometry for mills (A1, A2)
- Installation parameters now valid for each user on a machine (changed names for system variables required)
- CATProduct and CATPart models allowed as user made simulation models
- Last selected Work material is now used as default for next selection
- User model Flag supported (former U15 field entry is replaces by the new method)
- No more overflow with big spindle rotation values in case of small drills
- Tool import is now allowed before mfg program section is created
- Data path with spaces is now allowed
- Additional CATScript available for tool list export for a set of NC-programs

Version 1.4
- Parameter optimized for Counter Sink and Boring Bars
- Tip length corrected for Drilling Tool
- „\" are allowed in path names (UNIX)
- Data path with spaces allowed
- Error message implemented if destination folder does not exist
- extended Setup

Version 1.3
- Default for Installation path standardized
- Extended manual
- External Shape application implemented
- SQL Server support
- Minor bug fixes