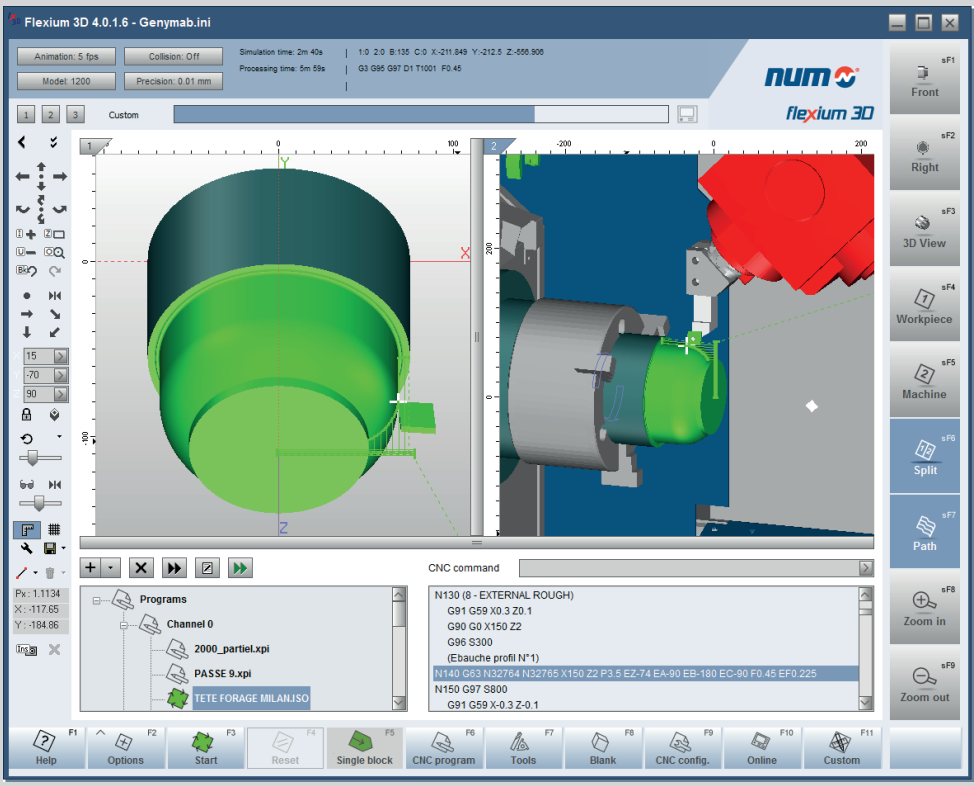
Flexium 3D

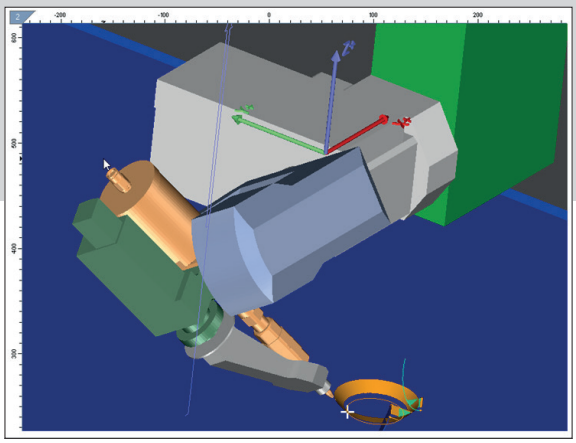
Operating manual

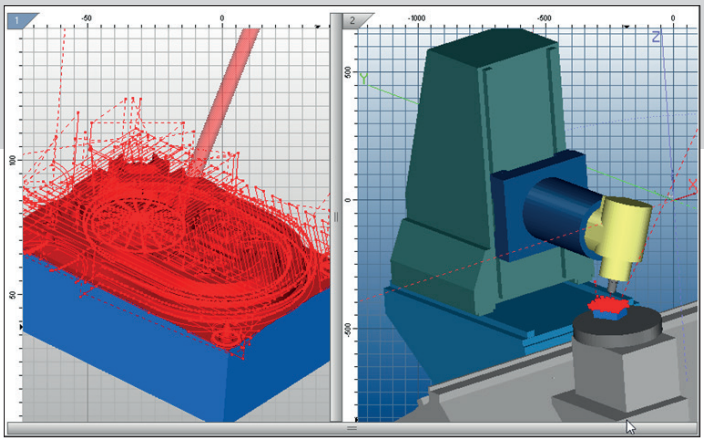
(for release 5.0.00.00)

M00029EN-08

Edition 10.2025







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The programming examples described in this manual are intended for guidance only. They must be specially adapted before they can be used in programs with an industrial application, according to the automated system used and the safety levels required.

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1 Foreword

1.1 Use of the manual

This manual is for use by qualified personnel only.
To reduce the risk due to an improper use of the product, do not perform any servicing other than that contained in this manual unless you are qualified to do so.

To ensure proper use of these products, please always read this User's manual carefully and retain for future reference. Should the unit require maintenance, please contact a NUM authorized service center only.

1.2 Use of symbols in the manual

Symbol	Meaning of the symbol
	Important notes on product use.
STOP	You may find this symbol whenever a wrong operation may damage the product.

1.3 Document revisions

Edition	Evolution index	Code	Meaning of documentation evolution	
11.2010	00	M00029EN	First version of the manual conform to software release 3.3.0.0	
11.2011	01	M00029EN	Manual conform to software release 3.3.10.0	
11.2014	02	M00029EN	Manual conform to software release 4.0.10.00	
02.2016	03	M00029EN	Manual conform to software release 4.0.20.00	
03.2017	04	M00029EN	Manual conform to software release 4.1.00.00 Added a new chapter 15 “3D Path Editor”.	
10.2021	05	M00029EN	Manual conform to software release 4.1.30.00 Added a new chapter 16 “Cylindrical and Non-circular Grinding” New functionalities: • Online Collision detection • New ToolType Grinding Wheel • Resize/Relocate blank • Basic Tool mode • Tool management via database	
11.2022	06	M00029EN	Manual conform to software release 4.2.0.0 • Status panel - Texts and image updating • Function keys shortcut buttons - Texts updating • Advanced - Texts updating • Simulation - Offline - Texts and image updating • Miscellaneous - Texts and image updating • Configuration window entries - Texts and image updating • Create a new tool - image updating • File menu - Texts and image updating • Collision - Texts updating	 2.3.4 2.3.8 3.2.5 3.3.1 3.5 4.2 5.2 5.2.6 9.2.2
09.2023	07	M00029EN	Manual conform to software release 4.2.10.00 • Startup • Position • Data Exchange • Image update • Appendix - Kinematic construction • Appendix - Host Command	 2.3 3.2.5 3.3.3 A B
10.2025	08	M00029EN	Manual conform to software release 5.0.00.00 • Flexium tool editor Window update • Remark - creating a custom tool • Renumbering paragraphs • Note Flexium 3D office version required a USB dongle and a licence file	 5 7 A - B - C 2.2

1.4 How to read the manual

This manual is structured and divided into main sections as indicated here below. We suggest to follow these indicated manual sections in order to easily find your information.

1	Foreword
2	This section contains a *“General description of the Flexium 3D application”*, presenting the main view, the machine view, the function keys, the toolbar and the 3D layout buttons.
3	This section describes the *“Setup Dialog”* configurations.
4	This section describes the *“CNC configuration dialog”* and the window view.
5	This section describes the *“Tool editor window”*, tool page description, tool type creation/ delete, the tool type selection, tool parameters, buttons ....
6	This section describes the *“Blank editor window”*, page description, manually editable values, the 3D view button tool bar and the file menu.
7	This section describes the *“Object editor window”*.
8	This section describes the *“Offset editor window”* used during the simulation by Flexium 3D and the ISO-parser.
9	This section describes the *“Machine editor”*, the page description for construction, collision, miscellaneous, note and preset views contexts.
10	This section describes the *“Machine axis configuration”* window and the modification of the main NC axis parameters of machines stored in 3dm files.
11	This section describes the *“Online simulation”* mode used to visualize real time machine axis movements when a connection to the CNC is established. Online simulation with Online collision detection (OCD)
12	This section describes the *“Text editor”* and the page description.
13	This section describes the *“Import / Export CNC configuration”*, the commands description and the operating steps.
14	This section describes the *Graphical supported “Search mode”*, the commands description and the operating steps.
15	This section describes the “*3D Path Editor”*, the commands description and the operating steps.
16	Cylindrical/Non circular Grinding

1.5 NUM Agencies

The list of NUM agencies is given on http://www.num.com

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2 General description and Main window

2.1 General description

Flexium 3D is a 3D graphical simulation software for part programs written in ISO-Code (DIN 66025 with NUM dialect), which exists in different versions for milling, drilling and/or turning applications as well as water jet and plasma cutting:

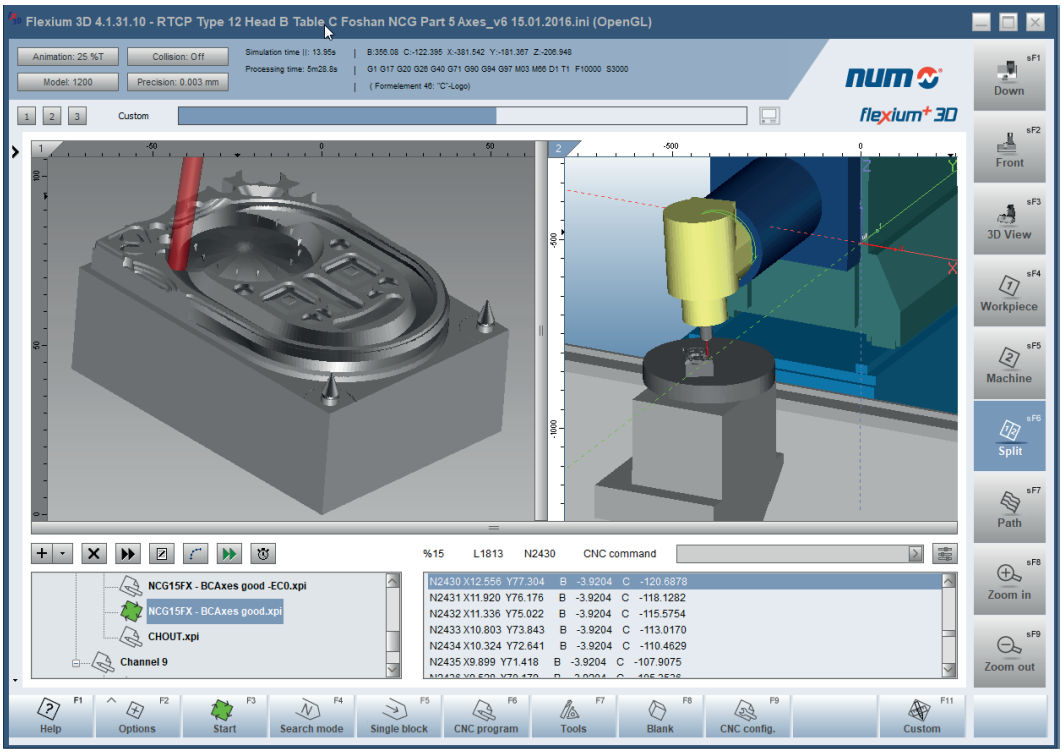
- Offline Version: Flexium 3D as standalone program used in production planning without CNC
- Online Version: Flexium 3D part of Flexium HMI panel software used as pre-simulation or in process simulation simultaneously during part processing
- Machine Version: Flexium 3D part of Flexium HMI panel software used as pre-process simulation.

During the part program simulation the path of the TCP (tool center point) is visualized, the material removal on the work piece is simulated and collision check is made between machine components, part and tools.

Operating the software is oriented on mouse and keyboard usage.

Flexium 3D takes as input the machine configuration (e.g. machine parameter, kinematics and physical parts, tool library, blank and collet definition), offset table and the NC program (part program).

An ISO-parser is integrated inside the host application (simulation), which analyses the part program, makes all tool length and tool radius compensations as well as linear and rotational offset transitions and substitutes mill-, drill- and turning cycle definitions with real movements.



Main window of Flexium 3D

Flexium 3D support different NUM series, and in particular:

- Up to revision 4.2.10.00 : Flexium, Flexium Plus
- revision 5.0.0.0: Flexium Plus, Flexium Pro

The software identify the Flexium series from the parameter file and visualize it in the top right corner of the window (see picture below).

2.2 Hardware requirements for Flexium 3D

If a customer PC is used instead of a NUM iPC please check following hardware requirements for Flexium 3D:

- **Minimum** requirements can be less (no particular CPU, 4GB RAM, no particular graphic card)
- If you have lower resolution than 1280x1024, some dialogs and tool bars might not fit into screen completely
- Of course, if you run Flexium 3D on a PC with less RAM, you could only simulate at lower precision. Also the animation will be slower if you don't have a dedicated graphic card., but the Flexium 3D application will start in any case
- All Windows Operating Systems are supported, so OS doesn`t matter

Recommendation

- CPU INTEL i5 (minimum) / i7 quad core for good performance
- 8GB RAM for satisfying simulation and animation
- Basic resolution 1280 x1024

Flexium 3D office version required a USB dongle and a licence file.

The licence.liz file must be copied to the folder:

- C:\Program Files (x86)\NUM\Licence for 32 bit application
- C:\Program Files\NUM\Licence for 64 bit application

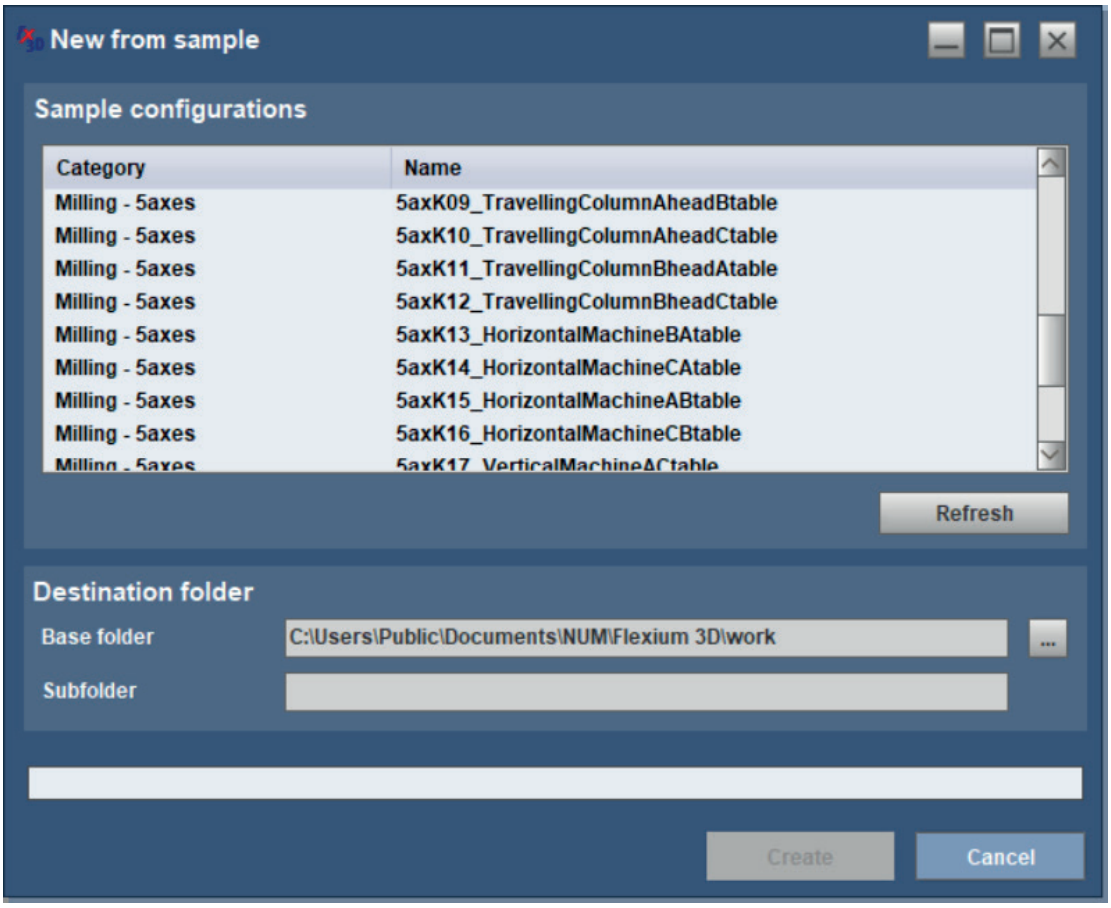
2.3 Start-up Menu

At the first run of Flexium 3D application, when there is no project loaded, the menu in the picture will be display. From this menu is possible to create a new project starting from the sample ones (Flexium Plus series) provided with the installation of the software; the procedure is equivalent to Import CNC Configuration described in chapter 13.

The sample projects, grouped in subfolders (visualized in the menu under the column Category) are located in:

C:\Users\Public\Documents\NUM\Flexium 3D\Samples

The destination folder and subfolder could be selected with the dedicated menu.

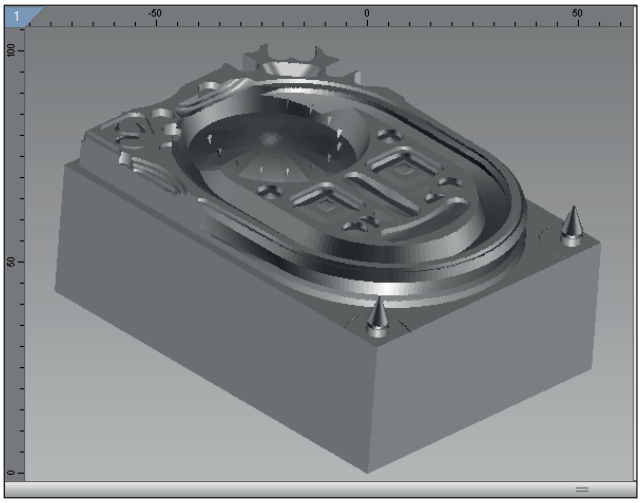


2.4 Main window

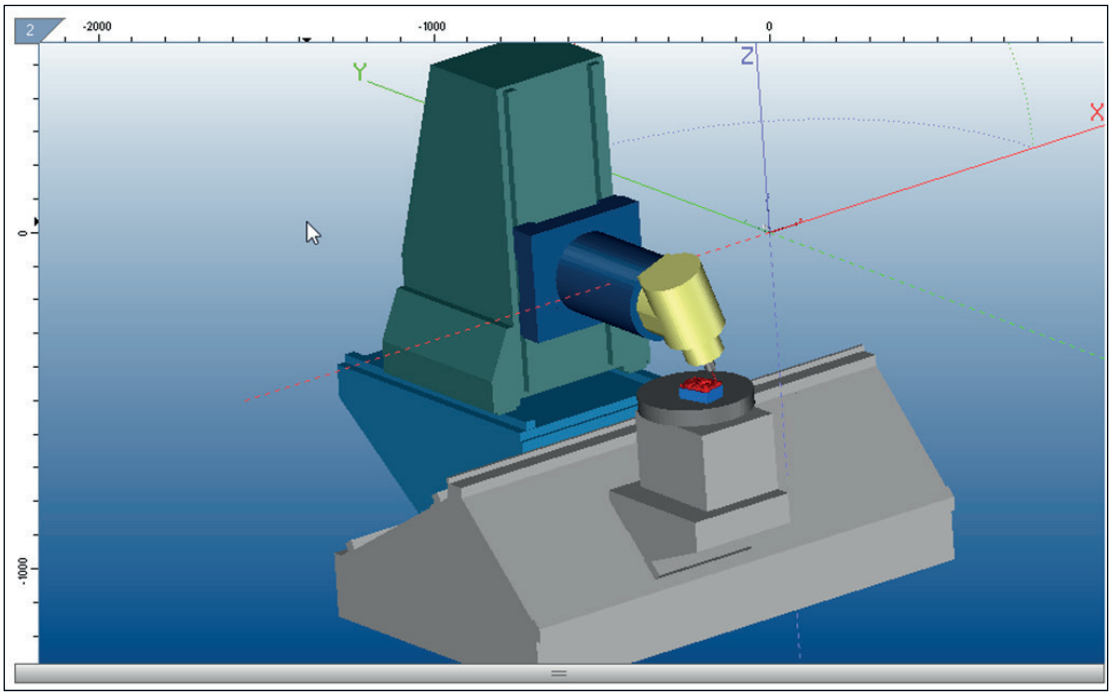
The Flexium 3D main window layout is intended to look similar to Flexium HMI context.
Depending on the configuration of Flexium 3D three layout views are possible:

- *Part View* (fixed part and tool moving around the part)
- *Machine View* (real movements of processing axes)
- *Split Window View* (part and machine).

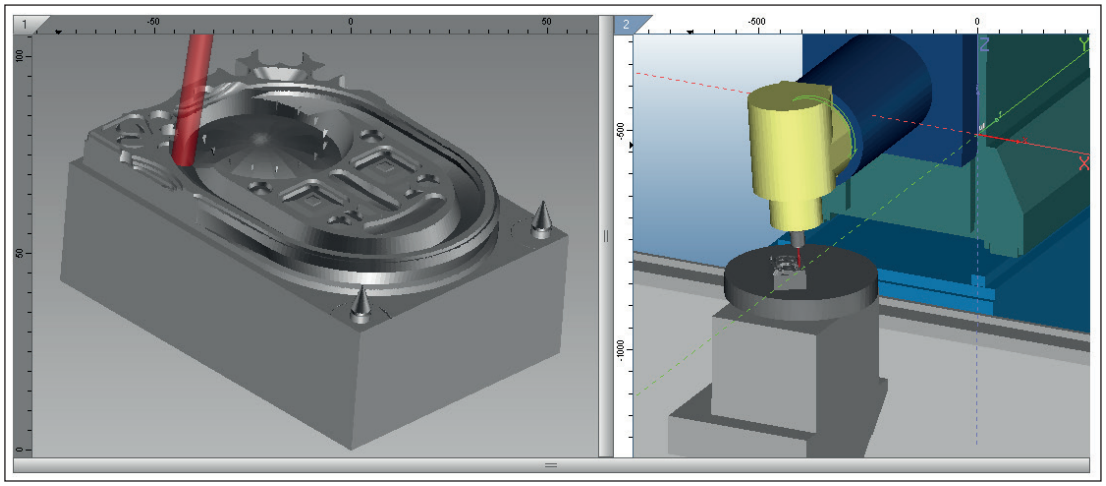
2.4.1 Flexium 3D Main Window: Part View



2.4.2 Flexium 3D: Machine View

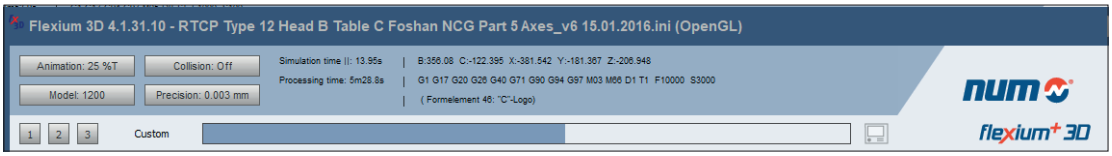


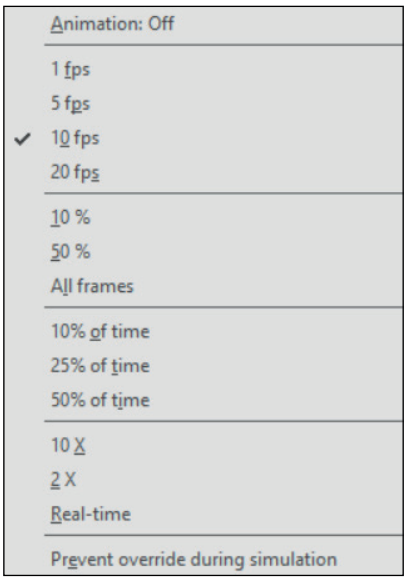
2.4.3 Flexium 3D: Split Window View

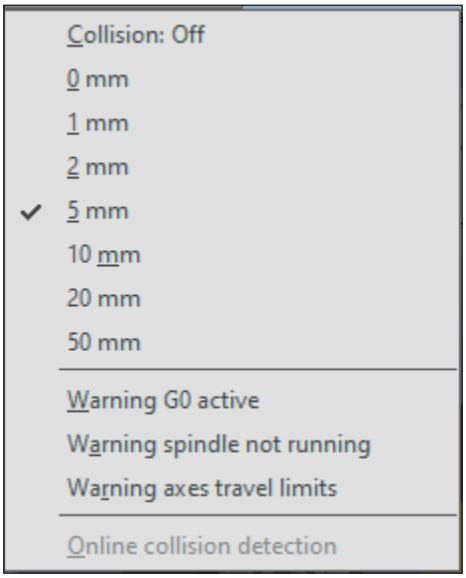


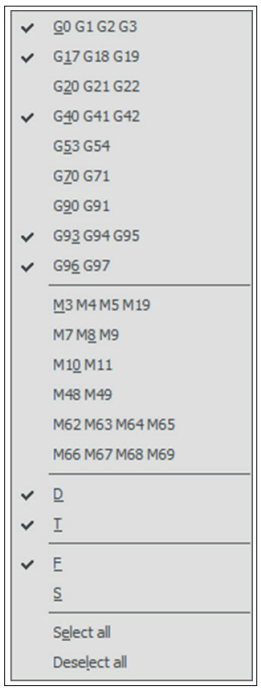
2.4.4 Status panels

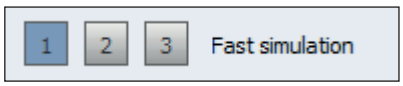
Several status panels are displayed on the main window upper section, presenting various simulation and application states:



Panels description	
Window title	Version number of Flexium 3D/Flexium 3D and currently loaded CNC configuration file. This file *(.ini)* contains a set of resource files like machine description and machine parameter, tool table, blank definition, macro zone files, parameter and variable files and part programs.
Animation	The animation speed during simulation can be configured by clicking the animation button, to one of following possible values.  The animation rate can be defined in frames per second (fps), percentage of frames (%), percentage of time and Real-time factor (X). For example 50% will display every second machine position (including interpolated intermediate positions). *"All frames"* will display every machining position, including interpolated position. 50% of time instead means that 50% of simulation time will be spent for animation If the calculation of the simulation is very fast (no material removal for example) then all % time values will result in a very quick simulation and less animation. **Example:** if you use % of time values to have some animation during slow simulations (a lot of material removal for example) but not spend too much time for the animation. For example if the simulation takes 10 seconds with no animation (just calculation), with 50% of time animation, the total time will be 15 seconds (10 seconds for calculation and 5 seconds for animation).

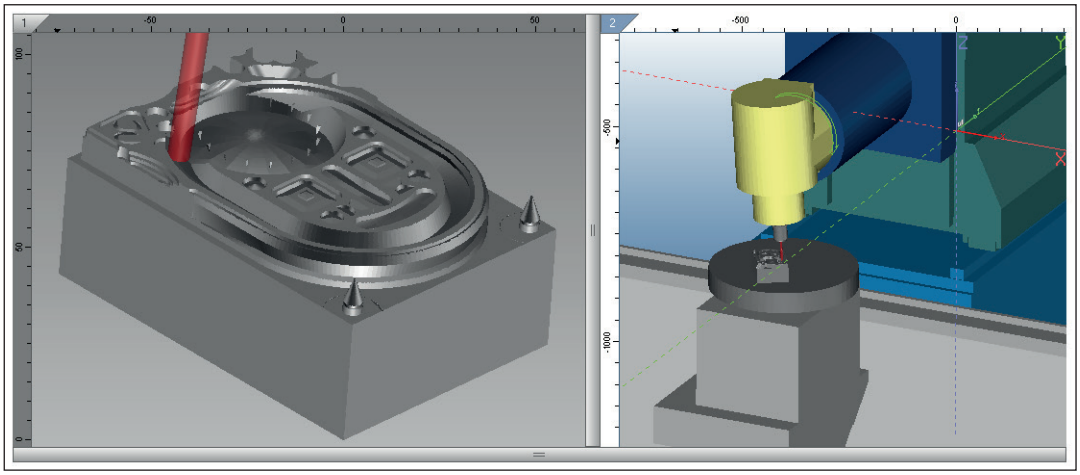
	Real-time will display the animation at real time speed, depending on the programmed feed rate value. The Real-time animation can be speed up by factor 2X and 10X. The option “Prevent override during simulation” will ignore the animation override commands programmed inside the part programs, for example: - (*#ANIMATION 10FPS*) sets the animation to 10 frames per second. - (*#ANIMATION 50%*) sets the animation to half frames (every second frame). - (*#ANIMATION 1X*) sets the animation to real-time. - (*#ANIMATION 0*) turns off the animation.
Model	Define the model resolution (accuracy) used for material removal simulation. The permitted range is 100 to 4000, where1000 would be a good compromise between simulation speed, accuracy and memory load. As a general rule, when doubling the resolution (1000 to 2000 for example), the simulation time increases by factor 2X and memory load increases by factor 4X. The model resolution is used for initial blank (see chapter 6); with big work piece is recommended to increase resolution value. To analyse a small portion of a work piece, is recommended to reduce dimension of the blank and focus only on a specific area.
Collision	Setting for collision detection:  Collision detection can be turned off or activated with a defined safety distance. A safety distance of 1 mm will give a collision warning when the distance between two machine parts that are tested for collision is less than 1 mm.
	There are also options to enable additional collision warnings (only if Collision check is not OFF): • when processing is made in G0 mode (tool is in contact with the part); • processing while current spindle is not turning; • axes exceed travel limits (check only as start position, otherwise standard CNC Alarm occurs).

Precision	Define the maximum accepted error (precision) for axes path interpolation. A lower value will generate more interpolated positions for G02, G03 path and for rotary axes movements. The precision is also used for the approximation of round objects, such as cylinders and round corners, with a limited number of points. The precision has an impact on material removal simulation quality and simulation time.
Simulation time	Display the total simulation time.
Processing time	Display the approximated time for the real processing in minutes and seconds. The processing time is calculated based on axis path, programmed feed rate, feed rate limitations and dwell times (100% feed rate override is assumed).
Collision information	Display the number of collision warning messages.
Axis positions	Display current axes position in machine coordinate system (OM).
G & M parameter display	Display information about currently active CNC functions. The user can customize the information, by right clicking on the panel and selecting the desired groups of functions from the popup menu: 
Status messages	Display the operator massages during simulation, as well as status and error messages.

Preset simulation settings	 Three preset simulation settings can be selected using this panel. The default presets are: • *“Fast simulation”* • *“High accuracy”* • *“Path display only”*.
Status messages	Status messages are displayed during parsing and simulation process as well as error messages.
Progress bar	Display the simulation progress. After the simulation is completed, the user can review the program by clicking on the progress bar and different positions.
Connection status icon	 Display the connection with CNC: • white: no comunication • yellow: initializing • grey: online position idle • cyan: fx server position active (no oscilloscope) • green: oscilloscope position active • red: comunication alarm

2.4.5 3D window view

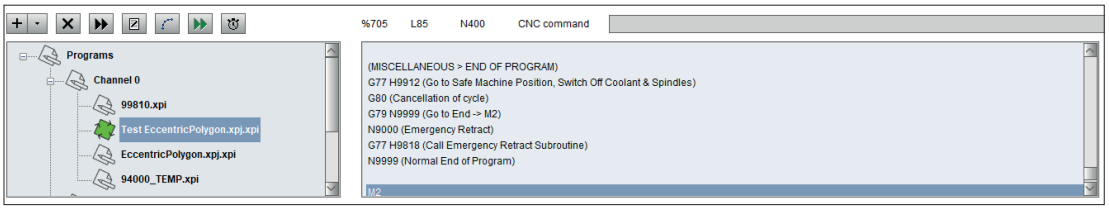
The 3D view (part, machine or split window view) is displayed in the center section of the main window.



The 3D views can be operated using the left or the right side toolbars.

The operator can resize the 3D windows, Zoom, Rotate and Shift the displayed objects.

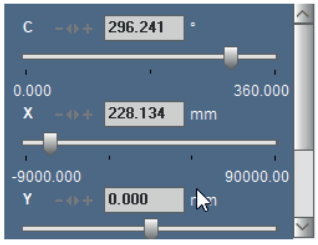
2.4.6 Part program list and NC program contents



The part program files are presented in a tree view (left window). The main program is marked with a green *NC Start icon*. Corresponding/selected part program view is displayed in right window.

This toolbar can be used to perform the following tasks:

	Add a part program to the part program list
	Remove the current part program from the list
	Set current program as main program (or mouse double click).
	Edit the current part program with a text editor
	Parse the current configuration. The simulation will not start automatically. This button can be used to test if there are any error messages coming from the parser (for example syntax errors, missing programs).
	3D path editor. To optimize a present part program
	Estimate Processing time of the part program based on PcParser capabilities. No 3D Simulation necessary.

%705	L69	N400	CNC part program number in use / current line and block number during simulation.
CNC command			MDI commands can be executed by typing a “NC command” and then pressing the Return key (this is not possible while the simulation is in progress).
			Axes ruler with limit switch specification from XPA file. 

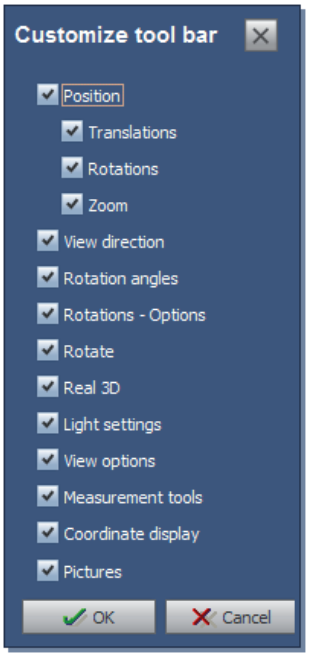
2.4.7 3D view side control panel

The vertical panel on the left side of the main window controls the current 3D window display (work piece or machine view) and allows changing of view position, rotate, zoom in and out, turn on and off some display options (like comparator and grid), make measurements, save pictures, and so on.

Each control button has tool tip information.

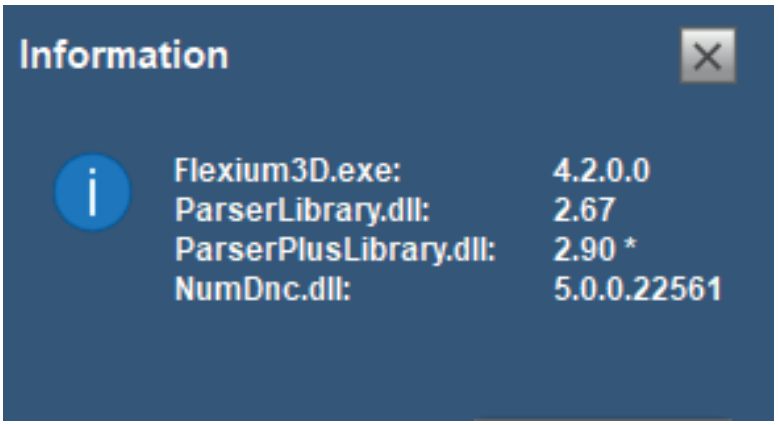
Control panel

Customizing the tool bar

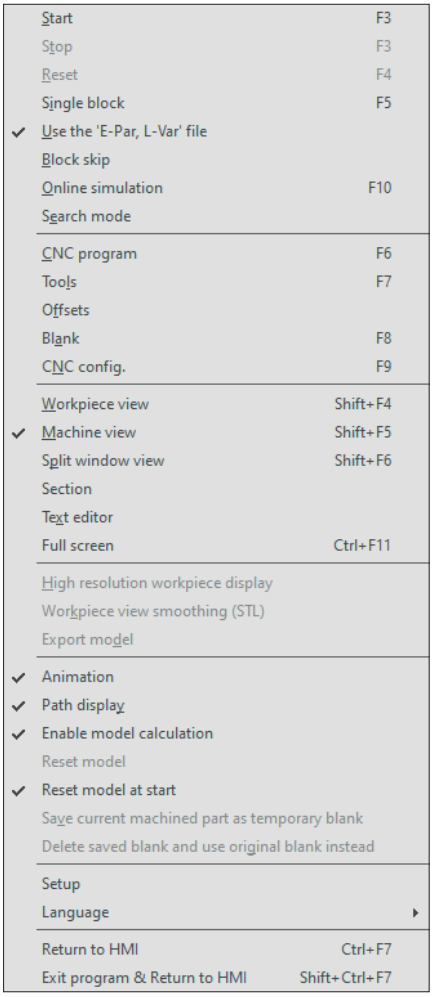


2.4.8 Function keys shortcut buttons

On the bottom bar of the window there is a short-cut buttons toolbar for *F keys*:

Function keys description	
F1	Displays the information page about the current version, serial number and license options; is also visualized if CNC or Dongle licence is used. Pressing dots after SW revision number type of executable, loaded PcParser versions and revision of loaded NUMdnc.dll (used for Online simulation) is displayed 

F2	Open the *"Options"* dialog box as a Pop Up-window:



All buttons from bottom bar F3 to F9 are selectable.
In addition to this you can select:

Use the ‚E-Par, L-Var' file
Use or ignore the E-parameter and L-variable initialization program, configured on the CNC configuration window.

Block skip
Execute or ignore the NC blocks that start with slash character " / "

Search mode
Open the Visual Search mode window, used for restarting the program from a selected path point (see Chapter 14).

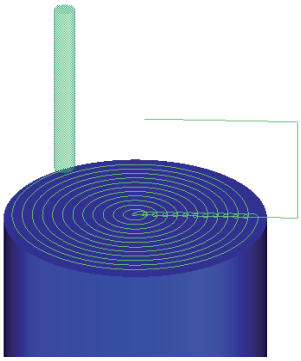
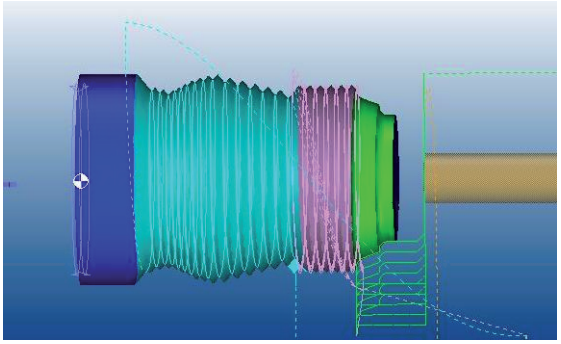
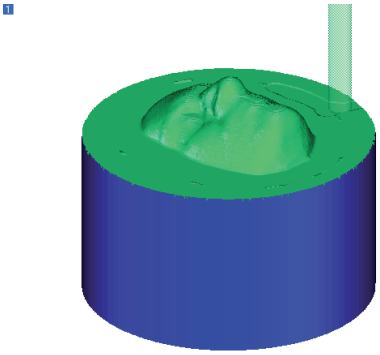
Offsets
Open the Offsets editor window.

Section
Open the clipping plane setup dialog.

Text editor
Open on the left side the program editor, with integrated help for G functions and E parameters.

Full screen
Enable full screen mode of the Machine\Workpiece area; come back with Esc button.

F2	***High resolution work piece display*** Create a picture-perfect image of the 3D work piece view. This option can be used after the simulation has completed.
	Work piece view smoothing (STL) Activate the STL display mode for the processed 3D part.
	Export model Open the model export dialog. Using this dialog, the processed part can be exported as triangle surface model (STL) at different accuracy levels. The edges between different colors on the processed part can be exported as DXF file.
	Workpiece view By selecting this 3D-view, only the workpiece (part) and the tool will be displayed. In this view the part is always the fixed object. The current tool will move around the part. Machining the part is visualized as material removal with the currently used tool. The wire path of the tool center point (TCP) can also be visualized.
	Machine view By selecting this 3D-view the complete machine including work piece and the tool will be displayed in the way the machine description was made (3DM-file). During machining or simulating the real axis movements are visualized.
	Split window view Work piece and Machine view are displayed side by side.
	High resolution work piece display Make a *"picture perfect"* image for the 3D model or a detail of the part . This button is active after the simulation has terminated and if there is a 3D model view visible (it is not active if only the machine view is visible).
	Animation Animation during the simulation.

F2	***Path display*** Display the wired path of the tool, reference point is tool center point (TCP). Rapid movements (G00) will be displayed in discontinuous lines and feed rate movements with continuous lines. Colors are selectable; automatic new color optional at Tool change M06 or subroutine call G77.  Example for milling  Example for wired path of TCP at turning and thread cutting
	Enable model calculation Calculate the 3D model of the work piece (with material removal operations) 
	Reset model Reset the 3D model to the initial blank object. The wire path is also reset.

F2	***Reset model at start*** If this option is checked, then the 3D model and/or the wire path display will be reset every time you restart the simulation. If this option is off, the existing 3D model and/or wire path will not be reset and the new simulated path will be added to the existing one. The animation, path display and model calculation can be turned on and off during the simulation at any time. The 3D model can be reset only after the simulation is completed. There are two more options to save the blank to memory and restore the blank: Reset model ✔ Reset model at start Save current machined part as temporary blank Delete saved blank and use original blank instead
	Save current machined part as temporary blank The current machined part is saved to memory (including the wire path) and will be used as starting blank for the next simulations (if reset model at start is activated) or every time you manually select “Reset model”.
	Delete saved blank and use original blank instead The saved part is deleted from memory. The original blank object will be used when the model is reset.
	Setup Open Setup-dialog (See chapter 3)
	Language 14 languages are supported by Flexium 3D and are online changeable. Deutsch ✔ English Français Italiano Chinese Traditional Chinese Espanol Português Cesky Türkçe Magyar Poliski Română русский
	Return to HMI Set focus to HMI window or start HMI ***Exit Program & Return to HMI*** Close Flexium 3D application and set focus on HMI

F3	***Start the simulation*** While the simulation is running and the “Block cycle” mode is not activated the “Start” button will turn into “Stop” button. If you press “Stop” then the simulation will be paused. You can resume the simulation with “Start”.
	Online simulation mode During the online simulation mode, the F3 button is used to connect and disconnect from the CNC.
F4	***Reset:*** terminates the simulation ***SEARCH mode*** Open new dialog for SEARCH mode to re-engage at any path position
F5	***Activates/deactivates the “Single block” mode*** If this mode is active then the simulation is made line by line by pressing Start (F2) after each line.
F6	***NC program window*** Show or hide the NC program.
F7	***Open Tool Editor*** (See chapter 5).
F8	***Open Blank Editor*** (See chapter 6).
F9	***Open the “CNC configuration” dialog***
F10	***Switches between the offline and online simulation modes*** “... to start up to 10”
F11	***Custom*** Custom Key to startup to 10 applications (e.g. different Flexium 3D configurations or 3rd party programs). Configuration can be made in *Setup -> Miscellaneous -> Custom Key*
F12	**Reserved** Online simulation mode (machine version).

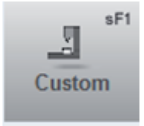
2.4.9 Vertical toolbar shortcut buttons

On the right side of the main window there are several shortcut buttons also accessible with keyboard combination Shift + F key.

2.4.10 View direction buttons

The first three buttons are used to select a standard view direction for the current 3D view. The buttons for a Milling and Turning configuration are:

Buttons for Milling configuration		Buttons for Turning configuration (Work piece view)	
	Front (XZ)		Top (ZX)
	Top (XY)		Front (XY)
	3D View (XYZ)		3D View (XYZ)

Buttons for Milling configuration		Buttons for Turning configuration (Machine view)	
	Front (XZ)		Top (ZX)
	Top (XY)		Front (XY)
	3D View (XYZ)		3D View (XYZ)

2.4.11 3D view layout buttons

The next three buttons are used to change the 3D views layout.

	Work piece view
	Machine view
	Split window view

2.4.12 Additional buttons

	Enable or disable work piece clipping plane. This only works if you have completed a simulation with the model calculation turned on
	Zoom IN the current 3D view
	Zoom OUT the current 3D view.

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Summary

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3 Setup Dialog

The Setup Dialog can be opened by pressing F2 and then *"Setup"* or by right click on different parameters from the top status panels.

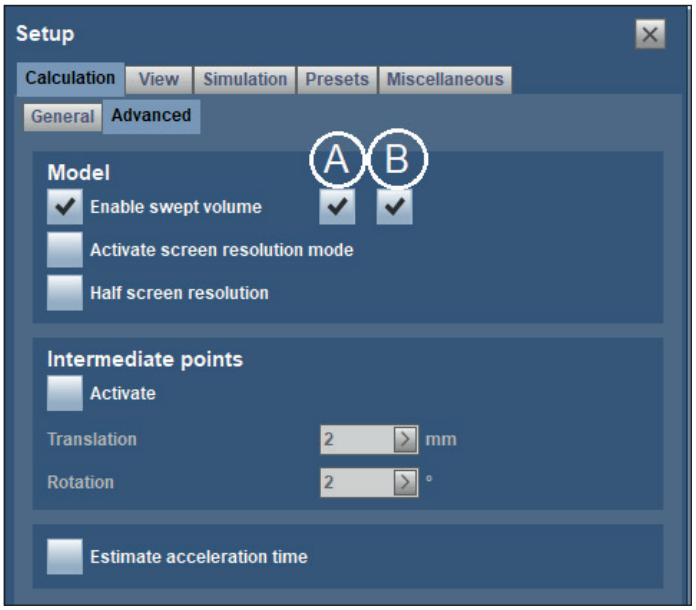
The dialog window has five main pages: *Calculation, View, Simulation, Presets and Miscellaneous*. Most of these pages are in turn organized in several sub-pages.

3.1 Calculation

3.1.1 General

Precision	Precision represents the maximum accepted error for different objects and movements. It is a very important quality factor for the 3D model calculation.
Model	Model calculation activates the 3D model calculation and sets the model resolution. 400 is a low resolution, 800 is medium and above 1200 is a high resolution. A higher resolution determines a smaller pixel size for the 3D model and this means more details on the 3D model. When the resolution is doubled (from 400 to 800 for example) the model will have 4 times more details. In this case the simulation time will increase with 75% up to 300% depending on other factors.
Collision	Collision calculation activates the collision check during the simulation and sets the *"Safe distance"*. For example if the collision check is activated with a safe distance of 1mm there will be a reported collision if the distance between two objects checked for collisions is smaller then 1mm. When the *"Use 3D model for collision calculation"* option is activated, and the collision and model calculation are both enabled, the collision detection calculation will test the collisions with the processed part instead of the blank object. In this way *"false"* collisions will be avoided: the tool holder for example might be in contact with the original blank object but it doesn't intersect the processed part because material has been removed from the blank object.
Safe distance	These configuration values are important for the collision calculation in Offline or in Online (OCD). The meaning is to create a exterior envelope around the specified object to obtain safety distances.

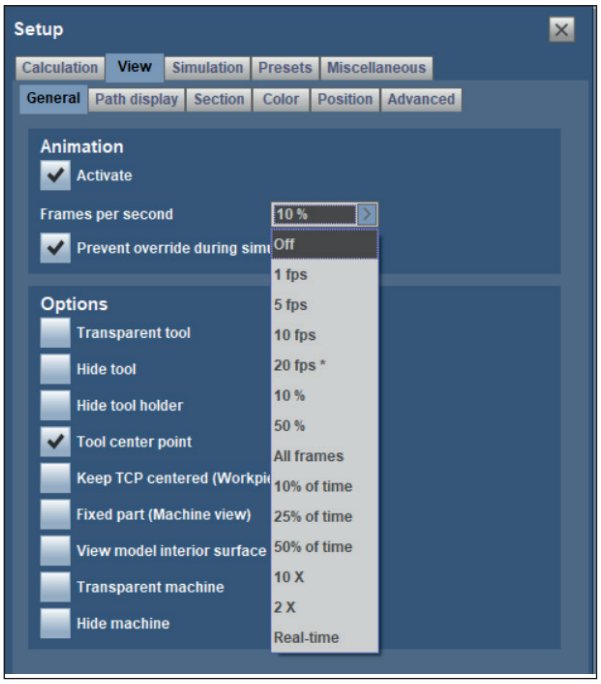
3.1.2 Advanced



Enable swept volume	Enable a swept volume calculation between two cutting positions and it is used for the 3D model calculation. If the option is turned off, the individual cutting positions will be visible on the 3D model (like a saw blade). For better quality this option should be enabled all the time. 2nd checkbox (A) in addition to first one: improved version of swept volume is activated. Only second checkbox checked: deactivate swept volume completely 3rd checkbox (B) enable: improve OD grinding. For turning simulation (with tool centered with blank) disable the option, for better algorithm.
Activate screen resolution mode	Activates “High resolution work piece display” during simulation. Generally for offline simulation the application works with reduced resolution to increase simulation speed.
Half screen resolution	Half screen resolution refers to the *“High resolution workpiece display”* calculation. When this option is activated, the result of the high resolution display will be a picture with half the screen resolution (a pixel from the model is made out of 4 screen pixels). The advantage is that the picture calculation is 2 up to 3 times faster.
Intermediate points or increments	When this option is activated, intermediate positions will be generated for a machine axis movement which is longer than the maximum indicated values. By default, the intermediate points are generated every 2 mm and 2 degrees. Even if this option is deactivated, some intermediate position will still be generated for G02, G03, G21, G22 to respect the maximum accepted error (precision).
Estimate acceleration time	Add to estimation of machining time, the calculation of acceleration time. The acceleration consider is linear, and read out from parameter file.

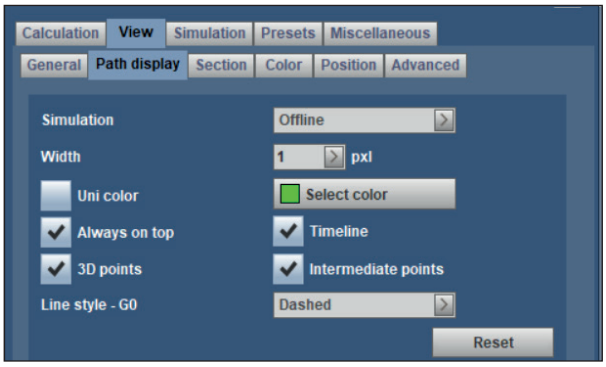
3.2 View

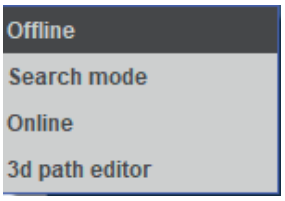
3.2.1 General



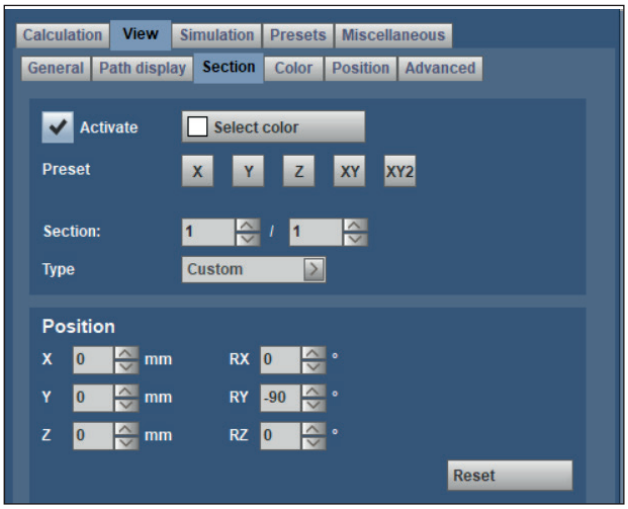
Animation	Activate the animation and sets the frame rate for the animation (frames per second). The option “Prevent override during simulation” will ignore the animation override commands programmed inside the part programs. See chapter 2.3.4 for frame per second description.
Options	***Transparent tool*** Display tool with transparency.
	Hide tool Hide the tool object for better view on the workpiece.
	Hide tool holder Hide the tool holder object.
	Tool center point Mark the tool center point inside the 3D views with a white cross.
	Keep TCP centered (Work piece view) Keep the tool center point in the center of the work piece view. This option prevents invisible processing, outside the view borders, for large work pieces.
	Fixed part (Machine view) Keep the work piece fixed on screen inside the machine 3D view. The machine components move around the work piece.
	View model interior surface Show the processed part interior surface, by hiding the front faces of the work piece. This option is useful for water jet applications, to inspect the cuts made with a thin tool.
	Transparent machine Display machine components with transparency, for better view of the work piece.
	Hide machine Hide machine components inside the machine view.

3.2.2 Path display



Simulation	The path display can be configured differently for four three simulation types:  The search mode path display options are used to highlight the already machined part, from the beginning of the offline simulation till the selected reentry point.
Width	Defines the line thickness used for the path display. By default the offline path is displayed with 1 pixel line thickness, online path with 2 pixels thickness and search path with 3 pixels.
Uni color	This option activates the wire path display with a customizable single color. When the option is deactivated, the path will be displayed with the programmed color which is also used for material removal simulation and the current tool display color.
Always on top	This option activates the wire path display always on top of the work piece. When deactivated, the path which is inside the part will not be visible.
3D points	This option activates the display of 3D path points additionally to the wire path. 3D path points are also displayed if the advanced calculation option *"Enable swept volume"* is deactivated. In this case the wire lines are not displayed.
Timeline	This option activates the wire path display in time line order. If there are two path lines with two different colors that overlap in the 3D view, the color of the second line (in time order) will be displayed. If the option is deactivated, the color of the closest line to the viewer stand point is displayed.
Intermediate points	Display or hide the interpolated points (for example G2 or G3 movements). If this option is off, only the programmed end points will be visible.
Line style – G0	Defines the line style for G0 path from three possible types.
Reset	Restore default values for the selected Simulation Set.

3.2.3 Section



On this page one or two clipping section can be defined. One clipping plane is defined with a 3D point (X, Y, Z) and an orientation with Euler angles (RX, RY, RZ).

There are five presets buttons available for quickly choosing between predefined sections, each of the first three presets enables one clipping section and each of the last two presets enables two clipping sections.

The sections are orthogonal to the specified axes (e.g. preset X defines a section perpendicular to the X axis), except for the first section of the XY2 preset, which has a custom orientation.

3.2.4 Color

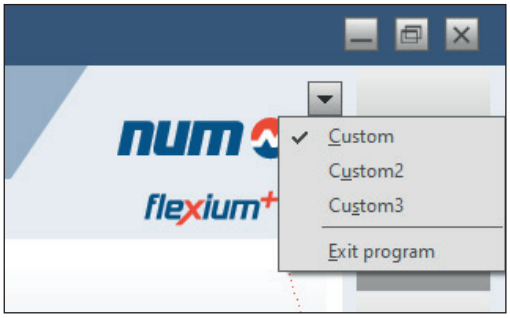
On this page the color palette used for material removal simulation, wire path and tool display is presented and can be customized.

The event for changing the color automatically during simulation could be M6 (tool change) or a subroutine call (G77 for example).

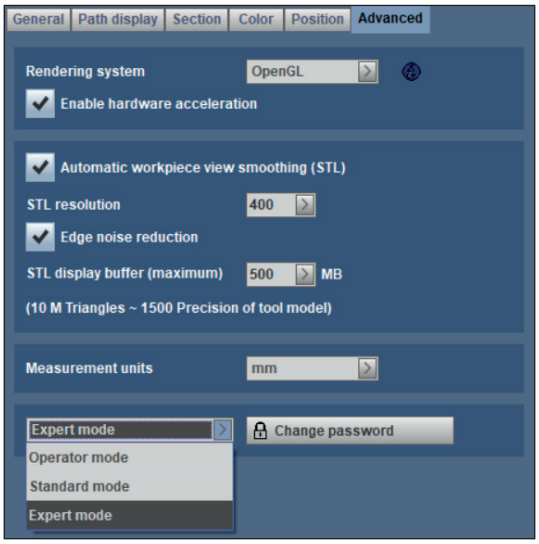
To change a colored view to more realistic metal view with high precision edges the last 2 options are available

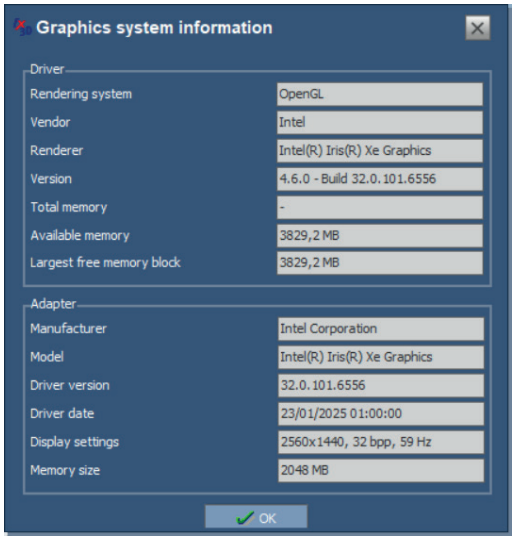
3.2.5 Position

Left	Define left corner position of the main window
Top	Define top corner position of the main window
Width	Define width of the main window
Height	Define height of the main window
Fixed position	Title bar and the frame around the window will be hided; the previous dimensions setup (left, top, width, height) are take into account.
Full screen	Title bar and the frame around the window will be hided; the window will be resized to cover the entire screen (F2 -> Full screen to enable\disable the setup).
Maximize 3D view	The 3D view will cover all the area of the main window: title bar, frame around window, side view bar and bottom command bar will be hided (Esc button or right-click mouse -> Maximize 3D view to revert standard window).
Custom positions	Is possible to save up to 3 custom setup of the main window (just press save icon and follow instructions on the video). To call one of the custom setup, enable *Show shortcut button*, that will show a drop down menu in the top right corner of the main window. Optional Exit program voice could be added (useful when full screen is enable)



3.2.6 Advanced





Rendering system	Select the current rendering system: OpenGL, Direct3D 9 or Direct3D Shader. ***Enable hardware acceleration*** Enable the 3D display hardware acceleration. For better performance, the acceleration should be disabled only in case of display errors. Blue Information icon report grapichs system configuration.
Automatic workpiece view smoothing (STL)	This option activates the work piece view smoothing after the simulation is completed.
STL resolution	Define the maximum model resolution used for STL display mode.
Edge noise reduction	Applies a noise reduction algorithm to the edges (where two colors meet), to improve the quality of the STL display mode.
STL display buffer (maximum)	Defines the maximum video memory to be used for STL display mode.
Measurement units	Defines the unit for all length dimensions from the user interface.
Mode	For the interface there are three operating modes to choose from: • Expert Mode: Access to all configuration dialogs and all Editor Tools. If active, Expert mode is password protected. • Standard Mode: limited access to CNC configuration dialog environment. Also No access to machine editor. If active, Standard mode is password protected. • Operator Mode: simplified interface useful for machine operator and ISO programmer: no access to project setup function, 3D view side control panel hided, dedicated simulation buttons on top bar: - Speed - , Speed + : reduce / increase simulation speed by fly (recommended setup, animation rate Real-time factor X) - Erase path: erase drawn path by fly during simulation - Section: activate\deactivate section visualization of the blank (see chapter 3.2.3)

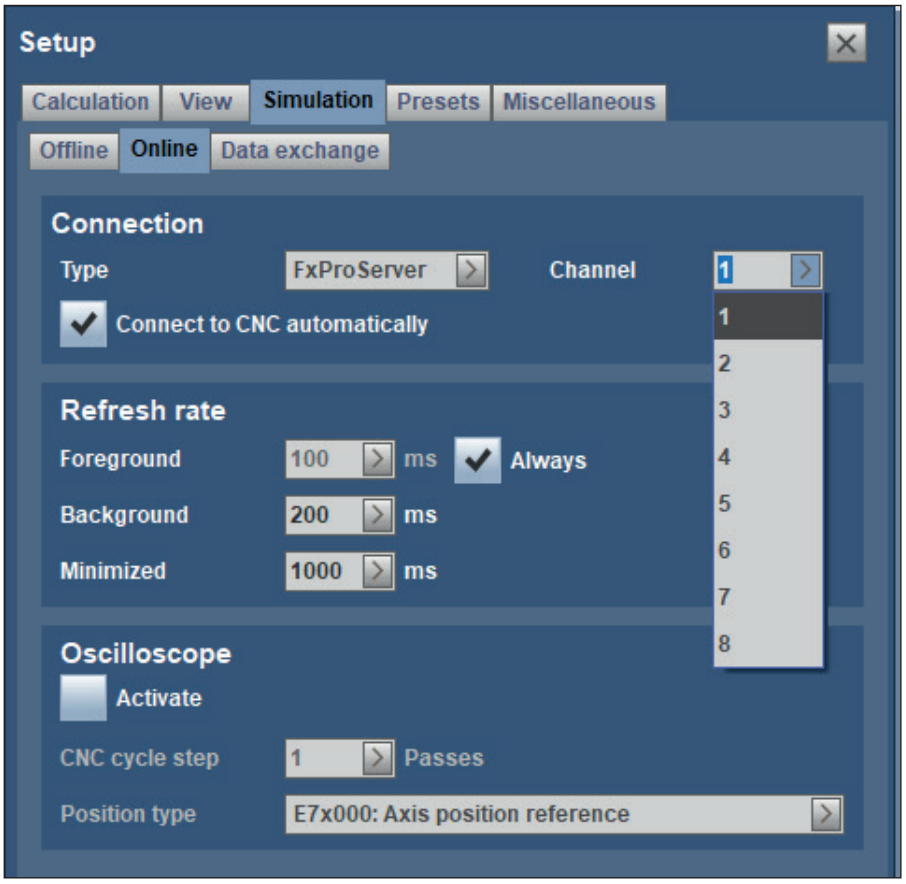
3.3 Simulation

3.3.1 Offline

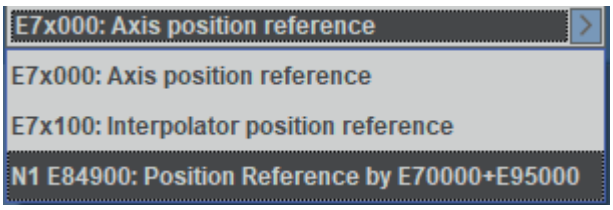
3D model display speedup during simulation	This option activates the display speed up during simulation by displaying the work piece material removal model at a lower resolution. This option can be used to speed up the simulation process when the material removal is simulated at high resolutions and the animation is set to a high frame rate. When the simulation is completed or it is paused, the work piece 3D model will be displayed at full resolution.
Precision of tool model (Maximum)	This value defines a maximum model resolution for the 3D display during the simulation. If the calculated model has a higher resolution than this value, the model will be temporarily displayed at a lower resolution during the simulation.
Endless mode / Delay before restart	This option activates the simulation run in an infinite loop. When the option is activated, the simulation will be automatically restarted after the previous simulation has completed and the defined time delay has elapsed.
E parameter resolution	Define the E parameter resolution for linear and rotary axes according to XPA parameters or to custom values.

Load only selected main program	When the option is on, only the selected main program is loaded to the part program list when the simulation is started from the HMI. If the option is off, then the main program is loaded together with all part programs located in the same folder as the main program.
Reset automatically load from CNC	When the option is activated, all flags that automatically requests resources from the CNC (parameters, offsets, tools, macro zones) are reset to false when the application starts. If the option is off, then the flags are preserved as they were saved inside the CNC configuration *"ini file"* (See chapter 11.3).
Block skip	Enable the execution of NC blocks starting with " / " character.
Enable M01	With option selected, if an M01 occours in a program the simulation will be paused (same behavior as M00).
Save wired path with saved blank	Generally, when you use "Export Model" from options window (F2) to generate from 3D model a STL output file the wired path is saved as well. If you want to prevent this path saving for a partial processed workpiece, uncheck this option. The advantage is that a workpiece, which is processed by different sub-programs, you can save each part processing status depending this partition and use the STL-file as temporary blank for the processing part.

3.3.2 Online



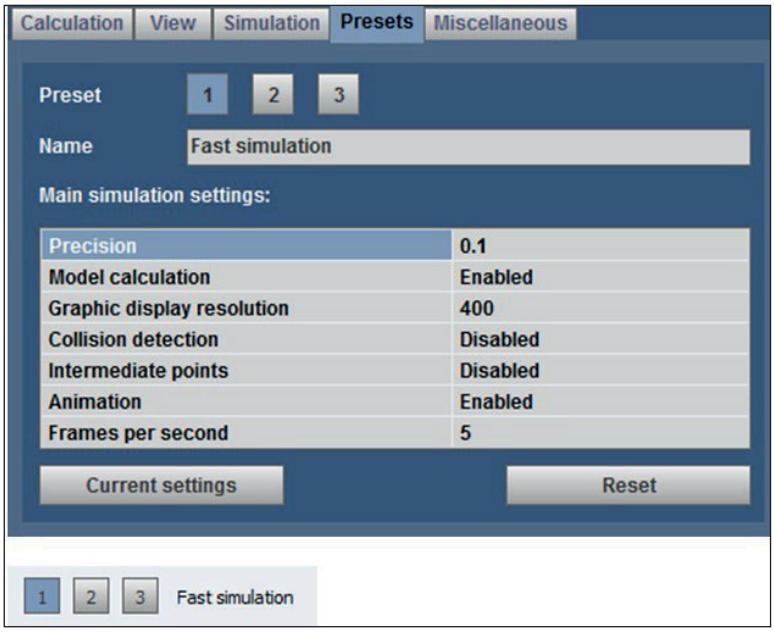
Connection	This option defines the connection type for online simulation. FxServer should be used for Flexium Plus system, and FxProServer for Flexium Pro series.
Channel	Selection of the Axes Group from Oscilloscope (CTRL-left click next to Type combobox if the field Channel is not visible). Linked to channel selected in the machine editor (available from revison 5.0.0.0). Is also possible to select more channels using - or , symbol (eg: 1-3 select 1st, 2nd and 3rd channel; 1,4 select 1st and 4th channel). The limit is always the maximun trace of oscilloscope function.
Connect to CNC automatically	This option activates the automatic connection to the CNC when online simulation mode is selected. If the option is unchecked, the user has to manually connect to the CNC using the F3 button.
Refresh rate	The refresh rate can limited to a defined speed in order to minimize the communication with the CNC during the online simulation. Three different values can be defined for each state of the 3D simulation window: foreground, background and minimized.

Oscilloscope	Activate the oscilloscope function and defines the CNC cycle step at which the axes are transmitted. Cyclestep10 means that every10 cycles the machine axes are transmitted during online simulation. Position type : Different NCK axes positions are available to request and display. E.g.  Can be configured in “Flexium3D_OnlineScopeGeneric.ini” on sub-folder: C:\ProgramData\NUM\Flexium 3D
NCK program header lines	Flexium3D_OnlineScopeGeneric.ini - Editor Datei Bearbeiten Format Ansicht Hilfe [GENERICSCOPENAME] GenericName1=E80000: Position Reference by DynOp program A GenericName2=E81000: Position Reference by DynOp program B GenericName3=E82000: Position Reference by DynOp program C GenericName4=Axes/Position measurement E900xx without CorrectionE952xx .... [GENERICSCOPECONFIG N1] X=E80000 Y=E80001 Z=E80002 A=E80006 B=E80007 C=E80008 [GENERICSCOPECONFIG N2] X=E81000 Y=E81001 Z=E81002 U=E81003 V=E81004 W=E81005 A=E81006 B=E81007 C=E81008 And selected with position type.

3.3.3 Data Exchange

Jump search	Enable the Jump Search function, extension of Graphical Search: after simulation and selection of a point along the drawn path, is possible to search forward or backward a string. The system will give as feedback the number of the block N, and write it on a E-parameter.
Search string	String to search inside part program
Forward\Backward	Direction of search function from the selected point along part program
E-parameter	E-parameter index for result of the search. Usable E-parameters: - from E80000 to E80199 - from E81000 to E81199 - from E82000 to E82999

3.4 Presets



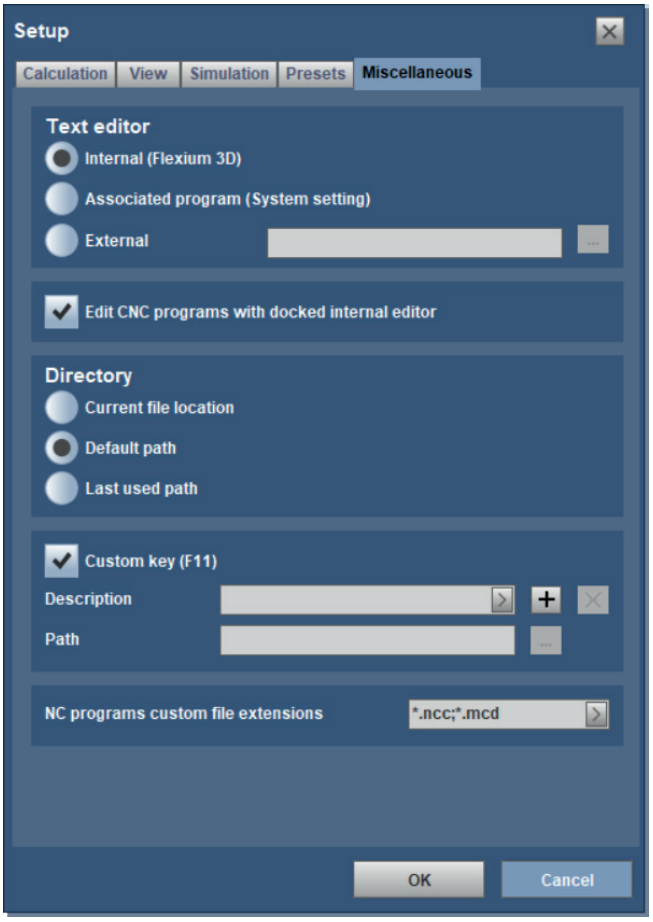
There are three presets for the simulation parameters that determine the simulation speed and accuracy. By default, preset 1 is called
"Fast simulation", preset 2 is *"High accuracy"* and preset 3 *"Path only"*.

The top buttons labeled *"1", "2*" and *"3"* are used to select one preset and the parameters values will be displayed for this preset.

The name of the preset can be edited and the current settings can be saved to the current preset by pressing the *"Current settings"* button. The *"Reset"* button reverts all parameter values to the default value for the current preset.

One set of preset simulation settings can be loaded in the main window by clicking one of the three small panels from the top status panel as indicated in figure.

3.5 Miscellaneous



On this page the editor used for text files inside the application can be configured. Also the starting path used for configuration files browsing can be configured in three different ways.

With option Edit CNC programs with docked internal editor enabled, the icon present on program list toolbar, will open directly Text editor integrated in main window.

The custom F11 button can be configured with 32 items (10 for old versions), where an item can be an external application (exe file) or a CNC configuration (ini file).

Is also possible to add NC program file extensions that will be recognized as standard when a new program is loaded in Flexium3D.

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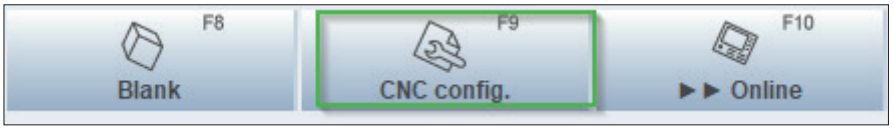
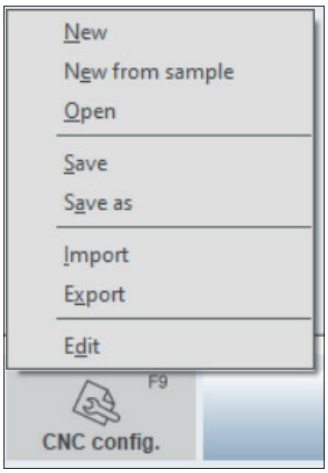
4 CNC configuration dialog

4.1 Window view

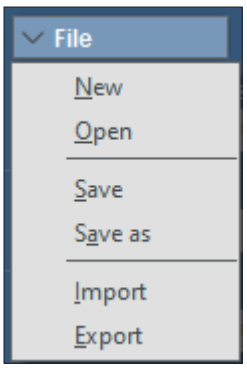
You can open the CNC configuration dialog by pressing F9. A popup menu will be displayed:





New	Creates an empty new configuration.
New from sample	Create a project from sample ones (See chapter 2)
Open	Open/Select an existing configuration file (.ini).
Save / Save as	Save the existing configuration to the current file or a selected ini file.
Import / Export	Load and save complete project environments. (See chapter 13 Import/Export)

Same File handling dialog is also inside the CNC configuration dialog present.



4.2 Work mode PcParser in Flexium 3D

PcParser in general analyses the main/Sub programs for syntax check with error messages, extends the cycle calls, is calculating the Tool length correction (M6 Tx Dy) in tool axis and calculates the 2D tool radius correction (G41/G42) if present.

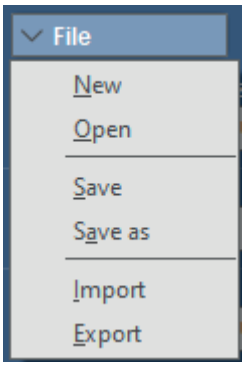
After the CNC configuration dialog is filled up, NC-program tree is present including the selection of main part program the PcParser can be used.

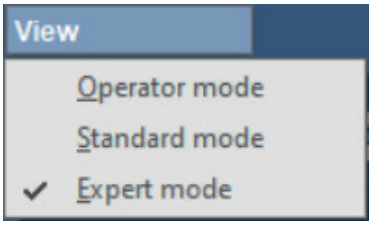
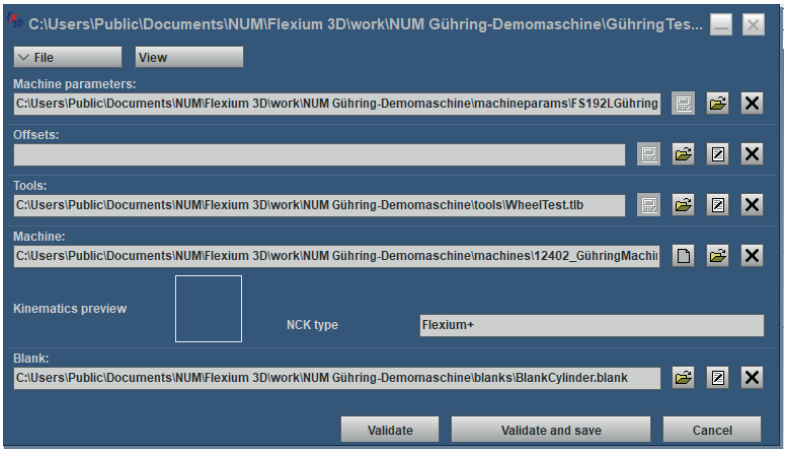
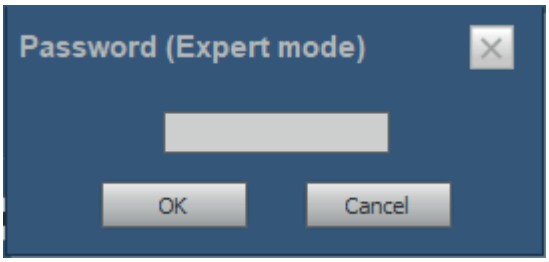
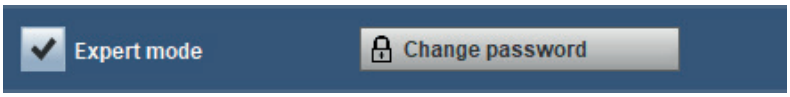
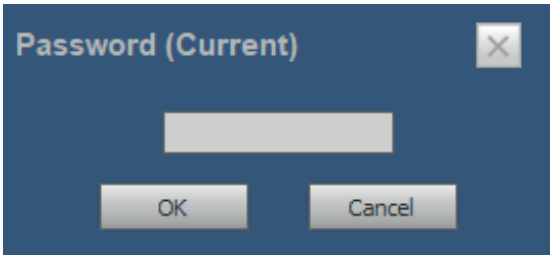
To understand the process or working mode of a PcParser call better in connection with CNC configuration dialog following

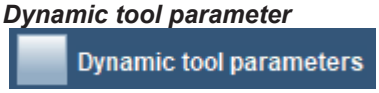
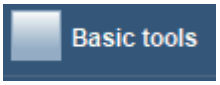
To start parser press green double arrow button or press “Start” button.

- base initial phase for all E-parameter and L-variables;
- all relevant resource files are transferred to PcParser from Flexium 3D-host;
- if %9999 and %11000 are configured fx3d-host transfer both (order is %11000 then %9999) to PcParser;
- PcParser starts parsing always with %11000, if configured, if not starts parsing with %9999, if configured. (this is different to NCK-parser !!!) After this initial-phase PcParser continues with PP main program;
- Each syntactical or semantical error is displayed as language dependent clear text error message with line specification and the possibility to open the editor directly at the error line in the program.

4.3 Configuration window entries

File	
Edit	Edit existing configuration. The CNC configuration input files can be defined inside a dialog window. For part programs it is possible to define initialization files for E-parameter (global or not) and program dependent L-variable as well as macros or subroutines from macro zones.

View	 For the operator there are two operating modes to choose from • Expert Mode: Experienced user only! Access to all configuration dialogs and all Editor Tools. If active Expert mode is password protected • Operator Mode: Expert mode NOT checked. Limited access to CNC configuration dialog environment. Also No access to machine editor. CNC configuration dialog is reduced. Only Machine parameter file, offsets, tools, machine axis calibration and Blank definition is accessible.
	 • Return to Expert mode: Password required  • Password is defined in Setup->View->Advanced  

Edit description	***Machine Parameters*** Machine configuration file: xpa extension for Flexium and Flexium Plus systems, nckProParam extension for Flexium Pro series. This file can be generated in ASCII-format with Flexium Tools.
	Offsets File with part and program origin offsets. Format is ASCII.
	Tools Tool file library; automatically generated from tool editor or IP address of PC where Tool Database software is running.
	Mx Define M-Word for logical tool change operation
	Tool Loading function The tool loading function can be configured to a custom M function that allows the customization of the tool loading sequence (for advanced users only). For each configuration file there is a set of controls buttons.
	Dynamic tool parameter  If active, dynamic tool data change generated by E-parameters inside part programs are used to modify the physical tool data.
	Basic tools  If active, tool table is identical to NCK Tool table (No T-number) and tool change is only dependent on Dx value. The tools will have no physical objects, but only a coordinate for the tool center point, that can be used to trace the wire path.
	Macro Zones Encrypted part programs and macro files organized in macro zones for customer and OEM (machine builder).
	Reset program Special part program that should have a fixed number %11000.
	E-Par, L-Var Predefinition file for other E-parameter and program dependent l-variable. This 2nd special part program have the fixed CNC program number %9999.
	Machine Machine and kinematic description, axes designations and auxiliary functions.
	Collet Description of clamping system.
	Blank Description of blank object. The internal blank editor can be used to define the geometry of the blank.
	Collet Description of clamping system.

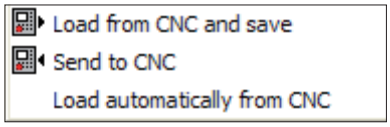
Open and use an existing local file.

Edit the current configuration file; for each type of configuration file there are different editor windows.

Add an existing local file.

Remove file from the current configuration.

Open the CNC options menu. This button is enabled only when a connection to a CNC is detected. A popup menu will be displayed when this button is pressed:



Load from CNC and save: It will request the resource file from the CNC and save a local copy. The communication might take a few seconds to complete;
Send to CNC: It will send a copy of the current configuration file to the CNC (unsaved changes are also sent);
Load automatically from CNC: If this option is activated, then the resource file is requested from the CNC each time you start the simulation.

Configuration
Subset of machine configuration data to define OM calibration, start position and nominal Min/Max travel values.

CNC axis	OM calibration	Start position	Min value	Max value	Maximal feedrate
1	0	0	-90	90	10000
B	0	0	-90	90	10000
C	0	0	0	360	10000
X	0	200	-2000	2000	10000
Y	0	0	0	0	10000
Z	-275	0	-3000	3000	10000

Technology selection (between Milling and Turning) influence the standard view on vertical tool bar.

NCK Type label show the project CNC series.

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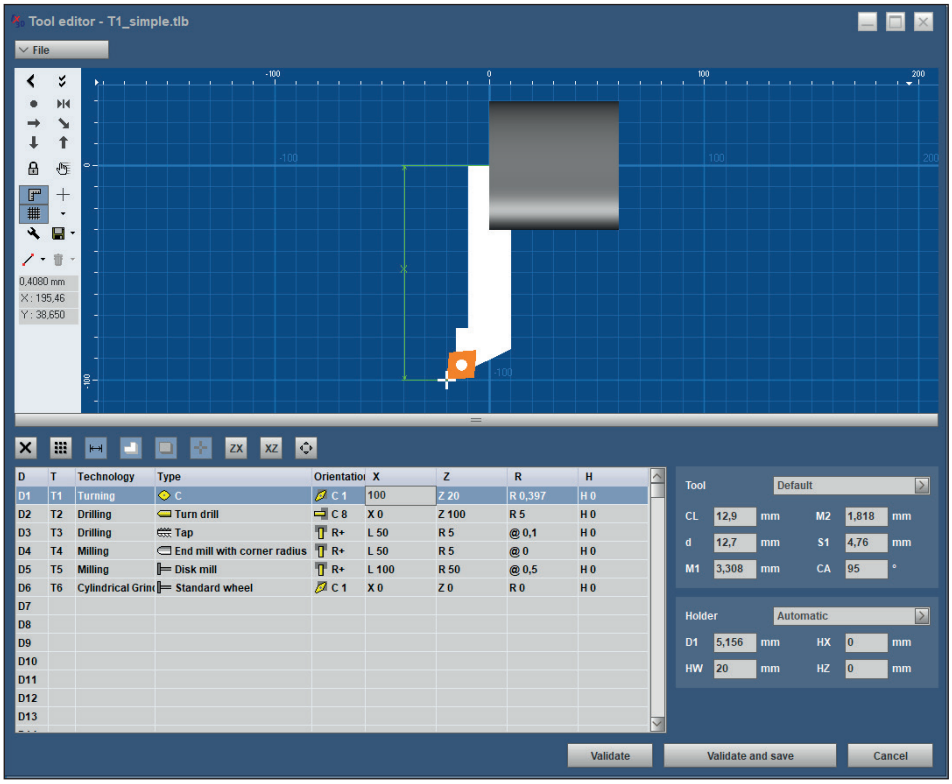
5 Tool editor window

5.1 Window view

5.1.1 Open tool editor window

The F7 button opens the tool editor window

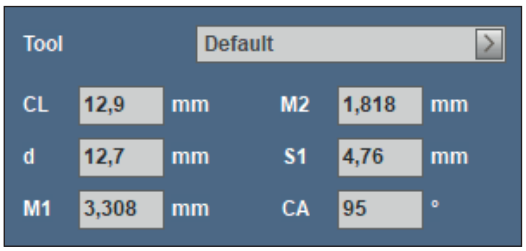
5.1.2 Tool editor page description



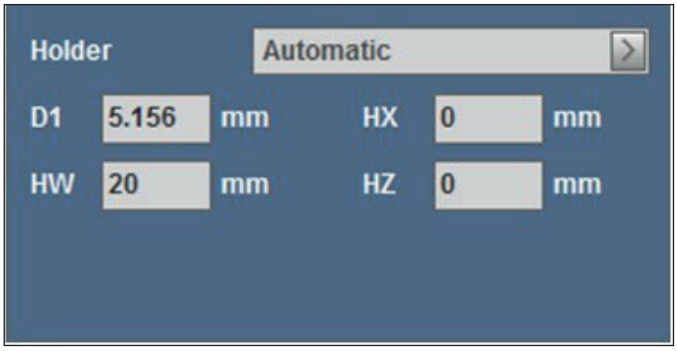
The tool table allows the visualization and modification of the main tool parameters:

- C/O (tip, orientation), L (lenght), R (radius), @(tip radius) for milling and drilling.
- C (tip), X/U (dimension along X axis), Z (dimension along Z axis), R (radius) for turning, thread cutting and grooving.

The other tool parameters depending on the type of the tool that is currently selected in the table can be accessed in a panel located at the right side of the table.
From drop down menù, is possible to select custom, and use a 3do file as tool shape. This feature could be used to import non-standard tool, as hob or worm gear, blade for cutting in turning application, etc. Refer to chapter 7 for more detail about the creation of a 3do file.



Also the tool holder can be configured in the right-side panel.



There are three types of holders that can be selected:
- *Predefined*
- *Automatic*
- *Custom holder*.

When the automatic type is selected, Flexium 3D uses one of the predefined holders depending on the current values of the tool parameters.
A custom holder can be loaded from a file on the disk and can be modified with the integrated 3D object editor.

For some of the predefined tool holder types, the width and the clamping diameter can be adjusted with the integrated 3D object editor (See chapter *"Object editor"*).

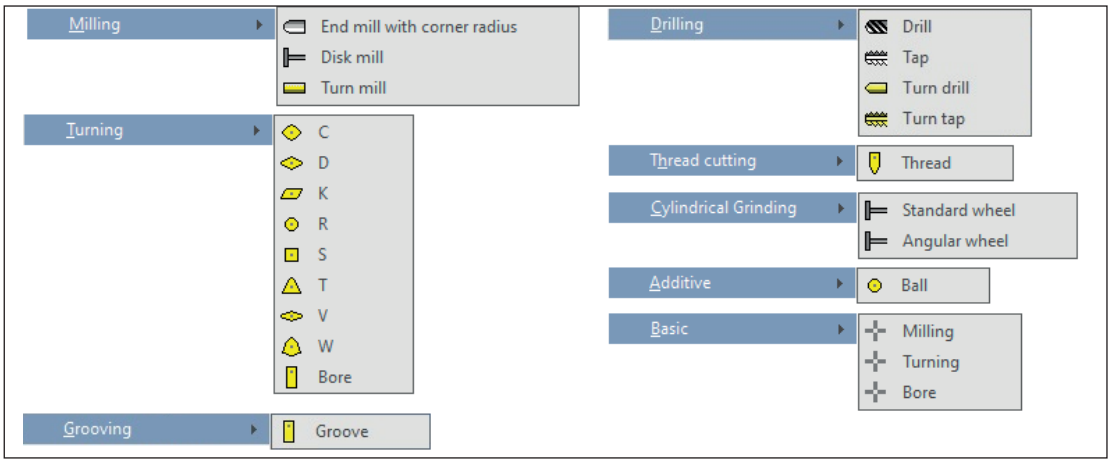
The Tool Table is saved in a file with exension .tlb. The current *"tlb file"* name is indicated on the title bar of the window. Each line in the table corresponds to a D number (D1, D2, ... , D255). For each D number several parameters can be defined on columns:	
T	T (tool) index.
Technology	Milling, Turning, Drilling, Thread cutting and Grooving, New technologies are described in the UI as: Cylindrical Grinding, Additive, Basic.
Type	Depending on the technology you can select different types of tools.
X, Z, R, C, H	For lathe tools.
L, R, @, H	For milling tools.
Other specific parameters depending on the type of tool.	

5.2 Create a new tool

The right mouse click on an empty line in the table will create a new tool.
A popup menu will be displayed with submenus for each technology.

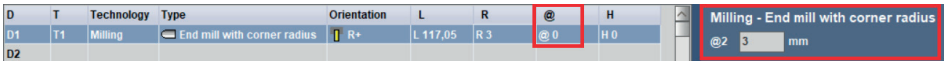
5.2.1 Tool type selection

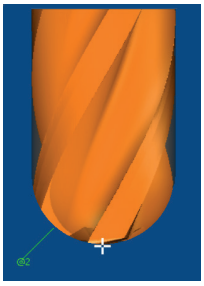
Different tool types are supported for following technologies and modes:



5.2.2 Mill tool: corner radius

For tool type Milling in 5 axes interpolation with RTCP very often the Tool is defined in the tool table





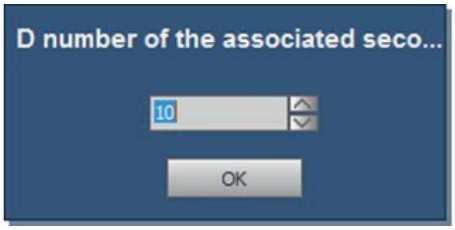
without corner radius (“@” column), because during PP generation of used CAM system, this is already considered.

But for correct material removal visualization, it is mandatory to define end mill corner radius.

In “@2” input cell you define the real physical existing end mill corner radius for offline and online simulation.

5.2.3 Create a new groove tool

When a new groove tool is created, a secondary D index has to be defined:

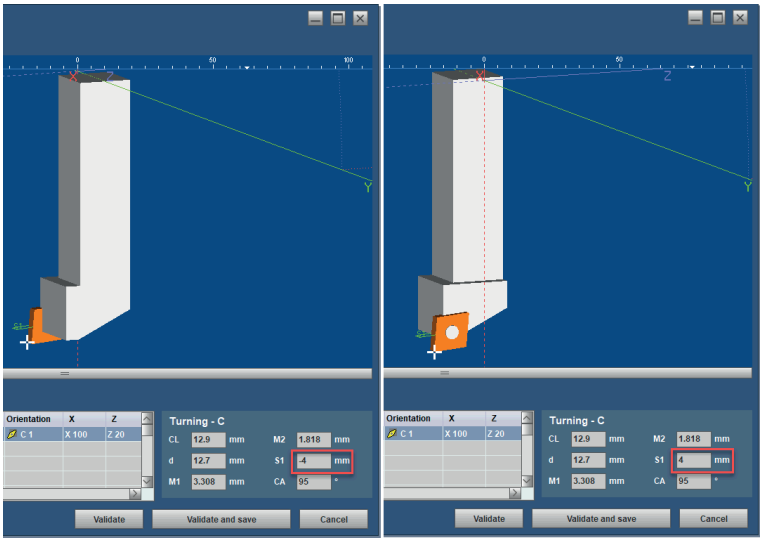


An associated groove tool will be automatically created and when a groove tool parameter is being edited, the associated tool will be automatically updated:

D9 (D10)	T9	Grooving	Groove	C 1	X 100	Z 20	R 0	H 0
D10 (D9)	T9	Grooving	Groove	C 3	X 100	Z 10	R 0	H 0

5.2.4 Turning Tool: Front Turret

For lathe system with Front turret the turning tools cutting edge thickness must be designed along Y plus direction; to do this use negative value of S1 parameter.



5.2.5 Delete an existing tool

To delete an existing tool, select the tool in the tool table and press Del key, key or ✕ button on the tool bar, or select on the technology column the last empty line:

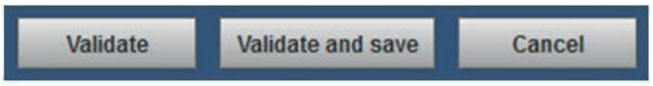
5.2.6 Edit the tool parameters

The tool parameters can be edited by clicking in the table or in the corresponding field from the right-side panel.

Parameters marked in gray color are not editable (they are a result of the other parameters).
The view options toolbar has the following functions:

1	Delete currently selected tool or create a new tool at an empty location.
2	Show all tools options. When activated the complete tool list is displayed in the 3D view.
3	Show / hide quotes in the 3D view. The quote for the current selected parameter is marked in green color.
4	Show / hide tool holder (white object). The tool holder is generated automatically as a support between the turret reference point and the tool center point.
5	Show / hide a generic turret as a gray fixed dimension cylinder.
6	Show / hide the tool center point (TCP).
7	Set the view direction to ZX plane (Turning).
8	Set the view direction to XZ plane (Milling).
9	Reset zoom to fit the current tool to the screen.

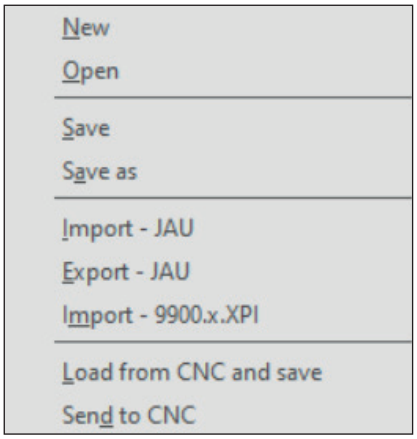
5.2.7 Bottom tool bar



Validate	Validate all changes and use the current data for the simulation. The tlb file is not updated. If there are unsaved changes a save question dialog is prompted before the Flexium 3D application is closed or a new configuration is opened.
Validate and save	Validates the current data for the simulation and saves the changes to the *"tlb"* file.
Cancel	Clear all changes.

5.2.8 File menu

The file menu contains the following options:



New	Create a new empty tool table.
Open	Load an existing tool table (tlb file).
Save / Save as	Save the current tool table.
Import – JAU	Import tools from a JAU file (native CNC tool table extension). New tools will be created using a default type for each technology and default values for additional parameters.
Export – JAU	Export a JAU file with the current tool list.
Load from CNC and save	Request the tool table from the CNC and saves a local tool table file.
Send to CNC	Send the current tool table to the CNC.
Import – 9900.x.xpi	Import the geometry of a grinding wheel.

5.3 Different tool handling methods

The different tool handling modes in Flexium 3D are listed below:

- Basic Simulation Mode (Basic Simulation Mode without T number. Logical Tool referenced only by Dx);
- Flexium 3D - Full Simulation Mode (referenced by DxTy = today Standard);
- Flexium 3D - Extended Simulation with Tool management database.(referenced by Ty).

Today standard is the tool simulation referenced by M06 Tx Dy

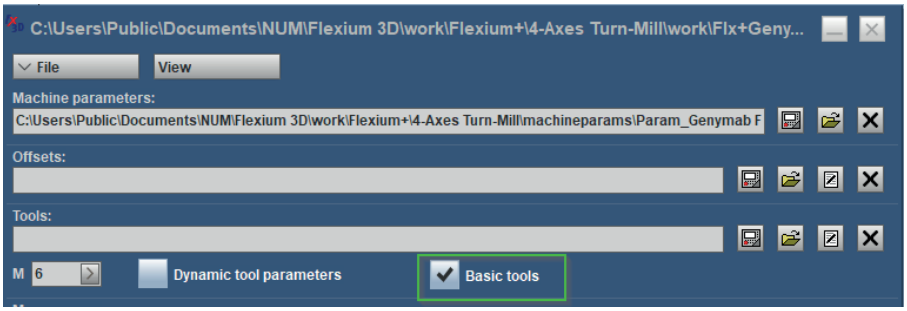
To be much closer to NCK, the Basic Tool mode without T number has been developed. It should be used if only a 2D-TCP visualization in work piece view is needed (eg: turning application).

5.3.1 Basic Tool mode

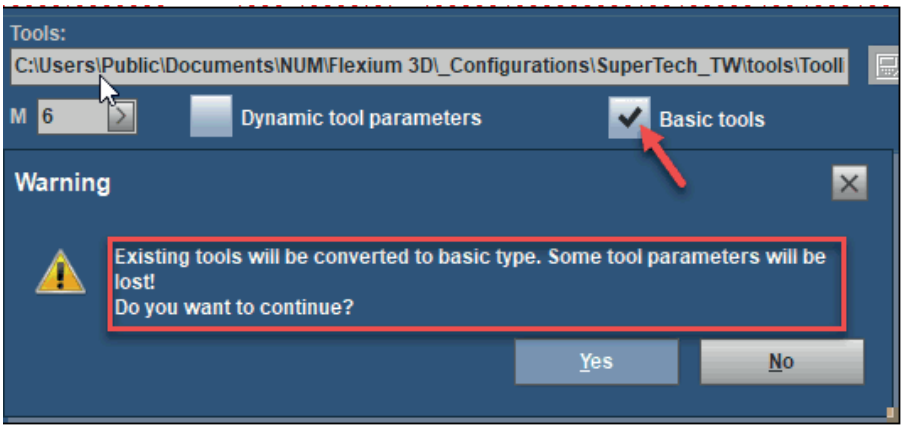
In this so-called Basic Tool mode no physical tool is displayed in Offline/Online simulation.

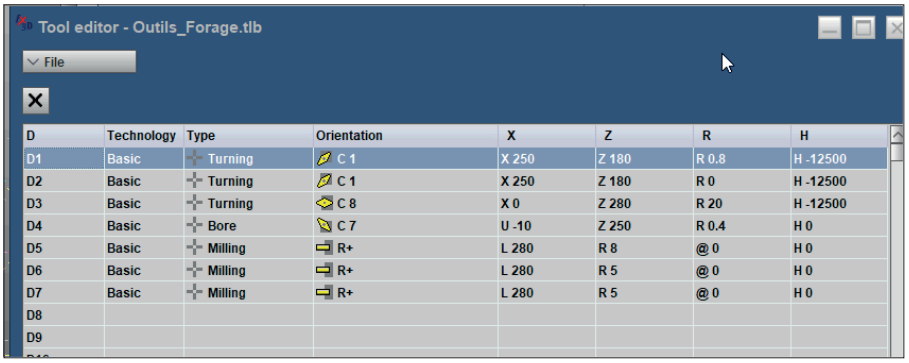
Note: In this mode collision detection or material removal cannot be done, Buttons for machine view (sF5) or sub –buttons are still present but greyed out.

To activate Basic tool mode use this checkbox in CNC-config dialog:



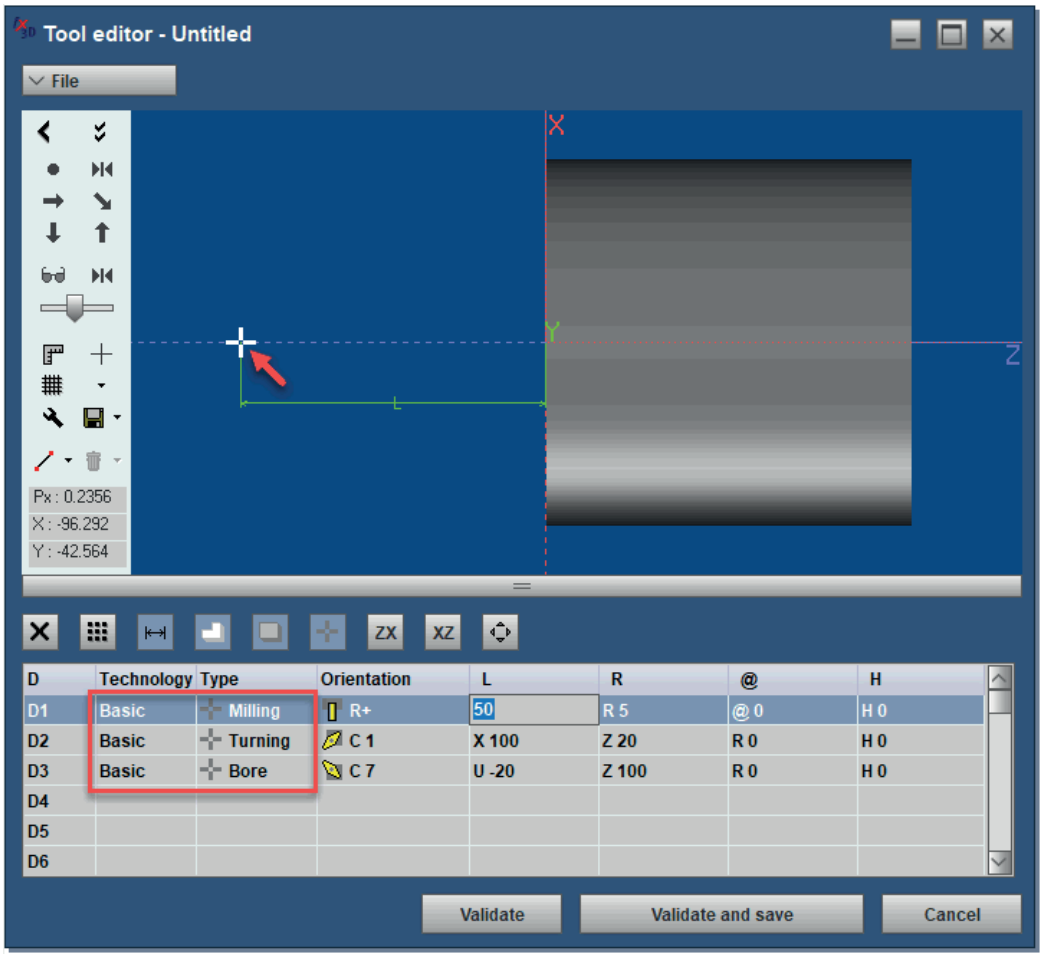
When activating “Basic tools” mode for an existing configuration that already has tools with physical characteristics, the following warning message will be shown to the user:





D	Technology	Type	Orientation	X	Z	R	H
D1	Basic	Turning	C 1	X 250	Z 180	R 0.8	H -12500
D2	Basic	Turning	C 1	X 250	Z 180	R 0	H -12500
D3	Basic	Turning	C 8	X 0	Z 280	R 20	H -12500
D4	Basic	Bore	C 7	U -10	Z 250	R 0.4	H 0
D5	Basic	Milling	R+	L 280	R 8	@ 0	H 0
D6	Basic	Milling	R+	L 280	R 5	@ 0	H 0
D7	Basic	Milling	R+	L 280	R 5	@ 0	H 0
D8							
D9							

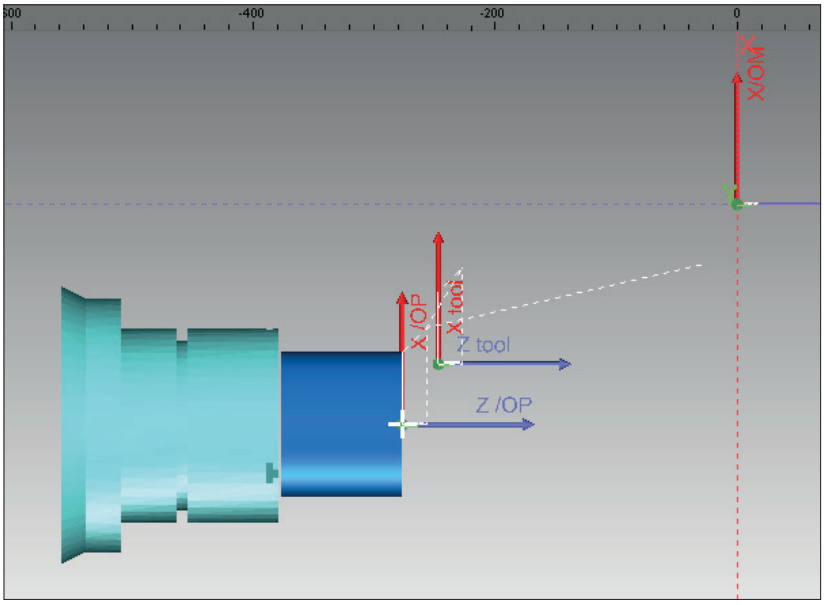
With this activation the view of the tool list will be changed, column with T-number disappear



Please note that in this mode the T column is missing from the table and on the right side of the table there is no panel for additional parameters. Internally all the tools have T0 in their data.

In this restricted mode, only “Basic” tools are allowed. The basic tools have all the correction data coming from a JAU file, but have no additional parameters to describe the geometry of the tool. These tools only define the coordinate of the TCP white cross and there is no physical body for the tools:

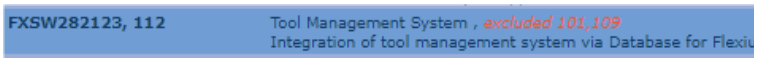
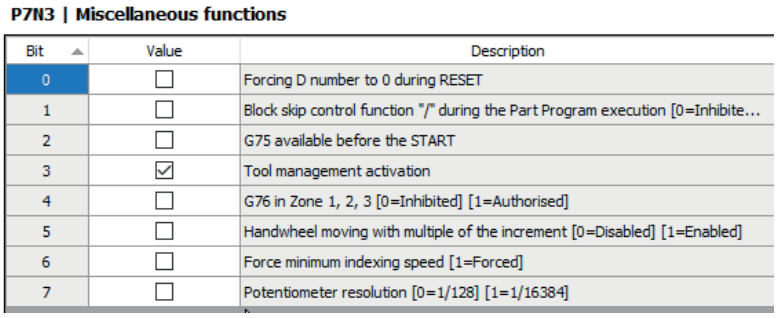
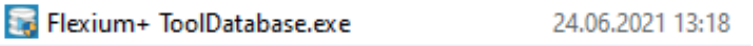
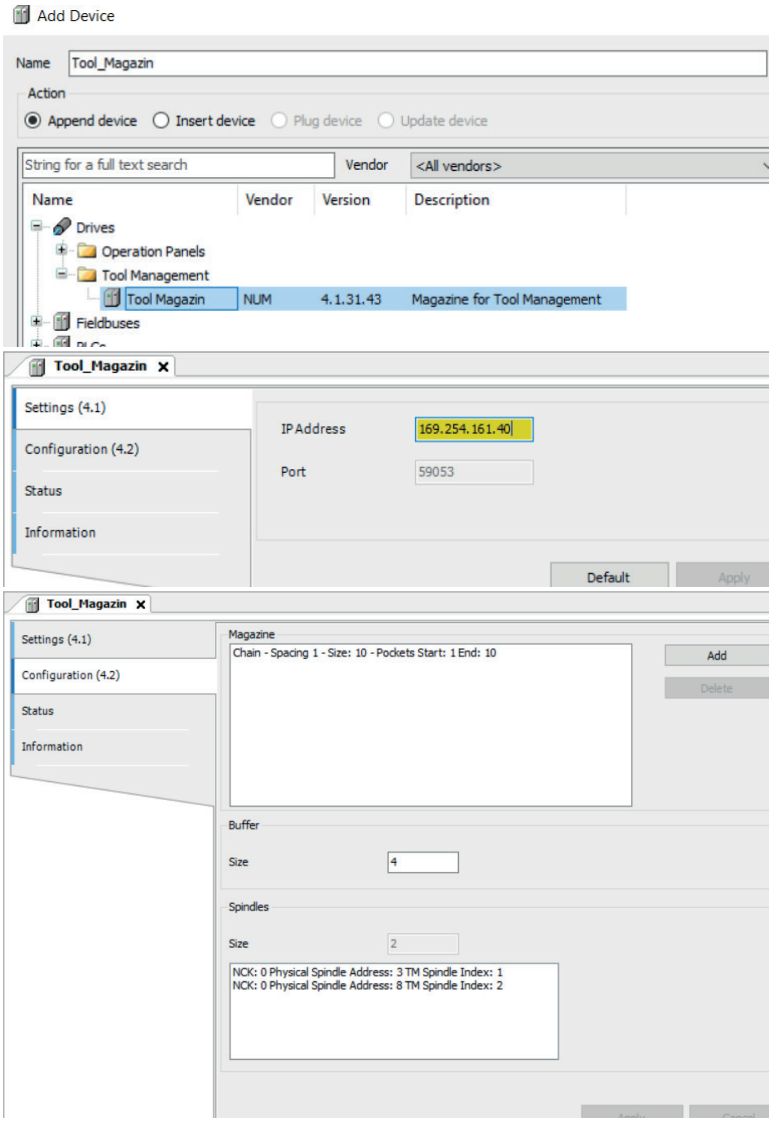
Example: 2D Simulation for a turning application



5.3.2 Tool Management via database

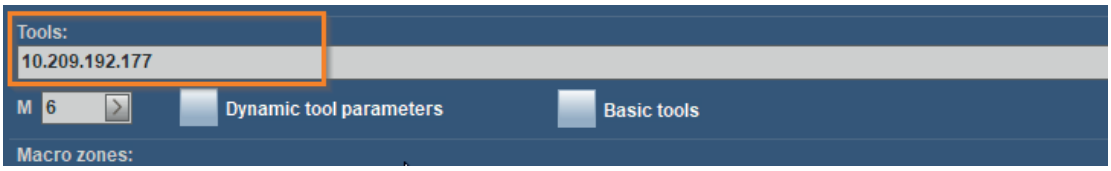
5.3.2.1 General settings, configuration and option

To run correctly Tool management on Flexium 3D the following step are mandatory:

HMI option FXSW282123, 112 installed	
Machine parameter P7 N3 Bit 3 set to TRUE (Tool Management activation)	
Tool database software v4.01.30.00 or above installed and running	
Setup of Flexium Tools application: install Tool Magazin, setup IP address, port and chain configuration inside Tool Magazin tab. A PLC program is necessary to manage the application.	

5.3.2.2 Configuration in Flexium 3D

To get connected to the database configure the IP address of the PC where the Tool database is installed. Instead of local tool file it is necessary to insert IP address in CNC configuration. The relevant IP address is provided with ipconfig command on database PC.



To check the connection status of database a dedicated icon is located at the top of Flexium 3D Main page. Icon has to be green.

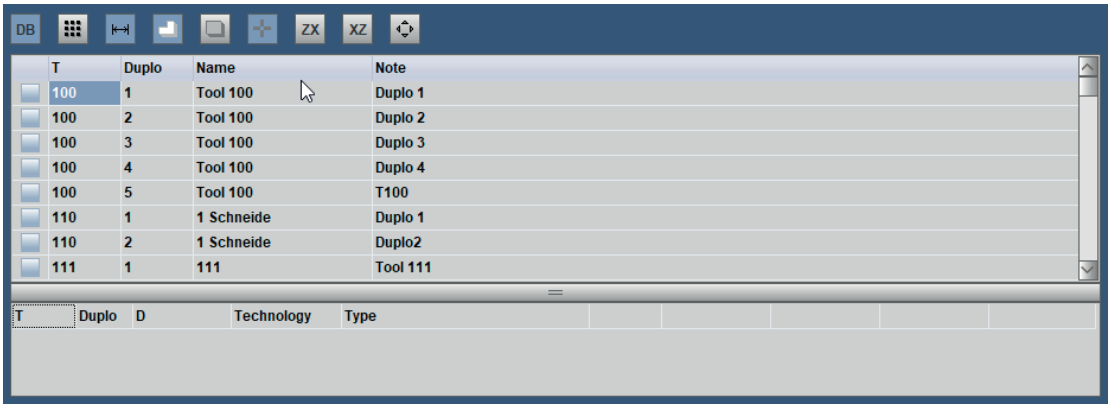
How to prepare a tool database on PC please refer to Flexium+ HMI operation manual (M00039EN) Chapter 12.10

Nominal the tool database is prepared for 3 types of tools: Milling, Turning and Boring. Each tool type has different tool data.

5.3.2.3 Working with Tools from database in Flexium 3D

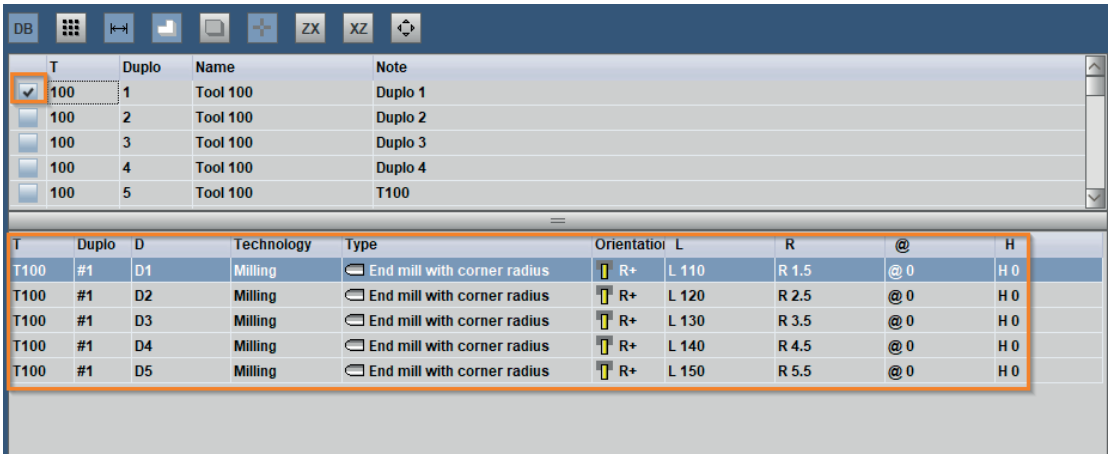
Offline Simulation:

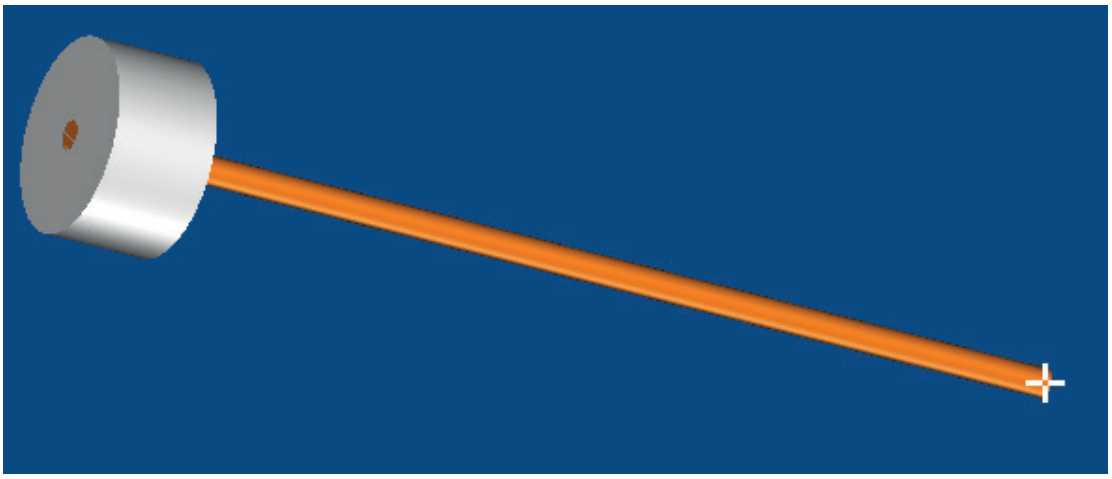
When Flexium 3D in Offline Simulation get connected to the database all tool data are requested, with the limitation that only tool data Txxx from Duplo 1 are transferred to Flexium 3D.



If you select a Duplo 1 tool e.g. T100 you see the corrector data D1,D2 …

The selected tool is displayed in Tool Editor window Offline Simulation





With an adapted part program you can run the Offline simulation with tool change Tool length correction for TCP or tool radius correction.

If needed you can change tool and corrector data and run a different Offline simulation.

With the two buttons “Validate” and “Validate and save” you can overwrite tool data temporarily or send it to the database

Limitation: You cannot create “New tools” in Flexium 3D, because the Tool Database is the master reference

Online Simulation:

In Online Simulation Flexium 3D requested only the tool and corrector data of tools which are logically /physically in the spindle und run the simulation with these data.

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Summary

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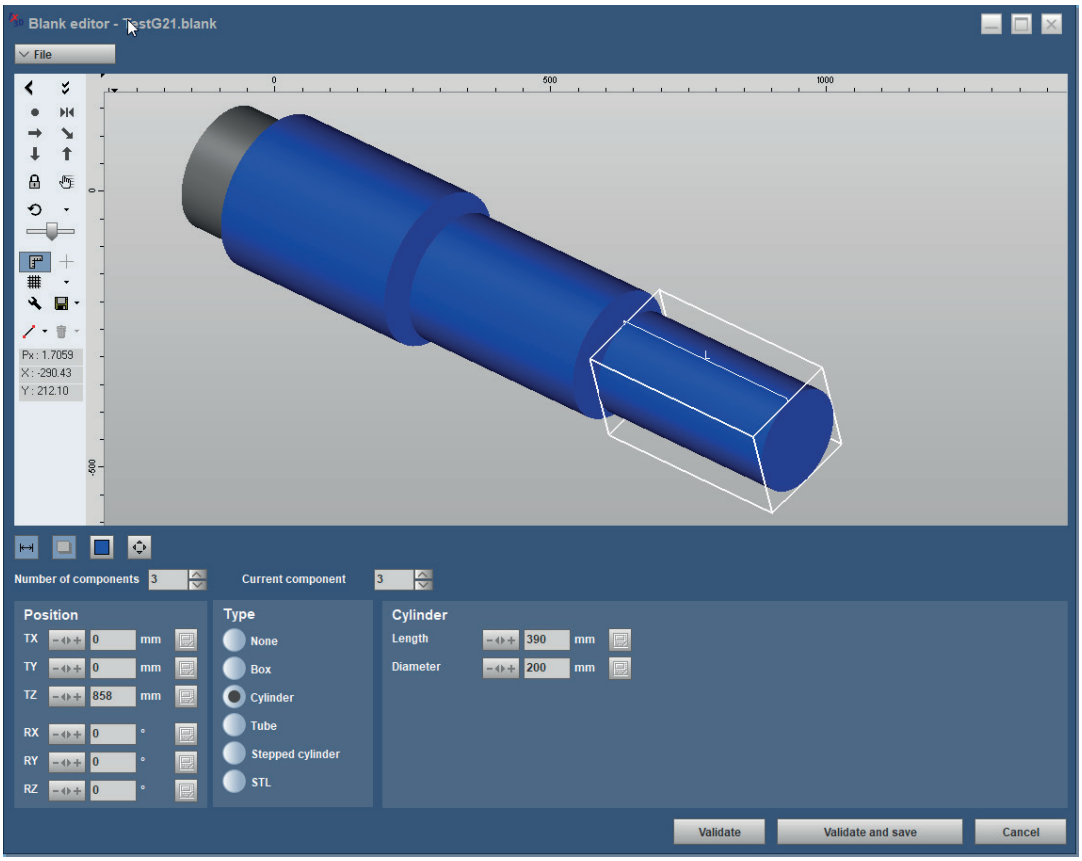
6 Blank editor window

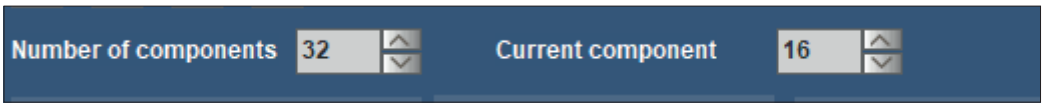
6.1 Open blank editor window

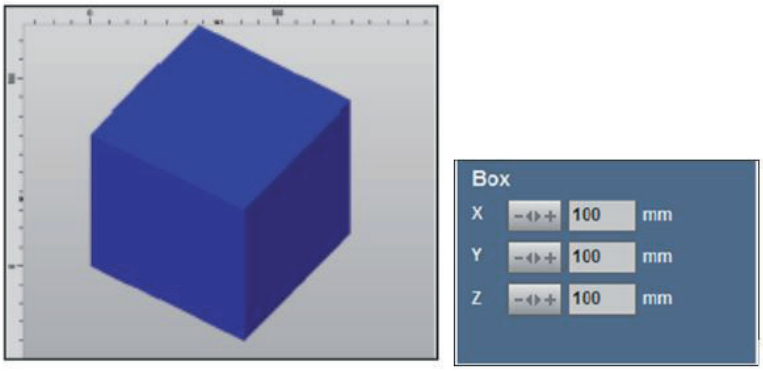
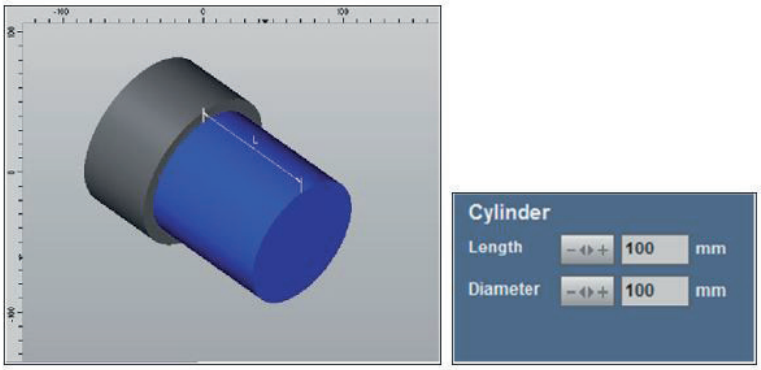
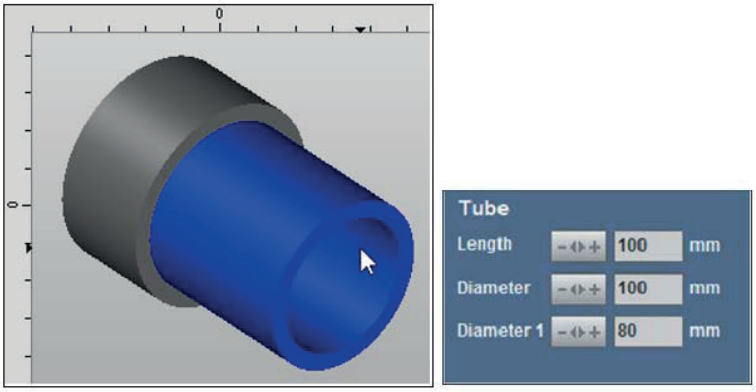
The F8 button opens the blank editor window

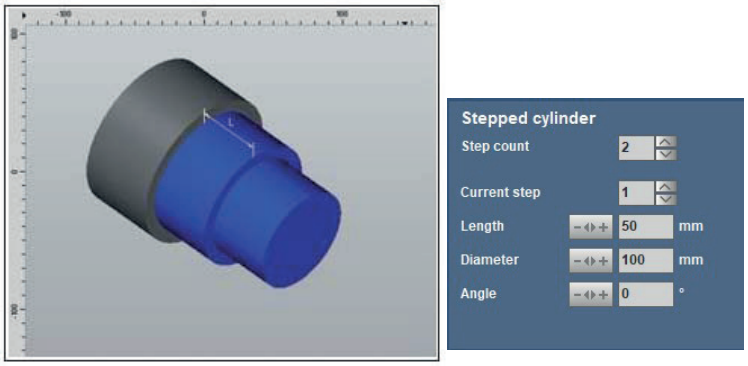
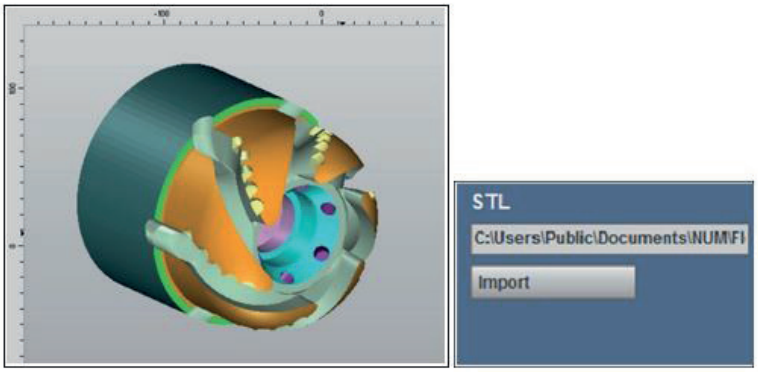
6.1.1 Blank editor page description

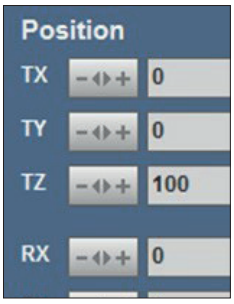
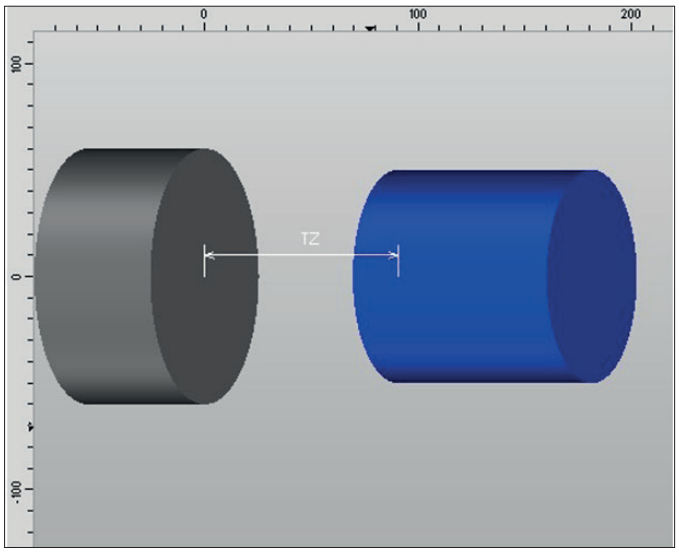
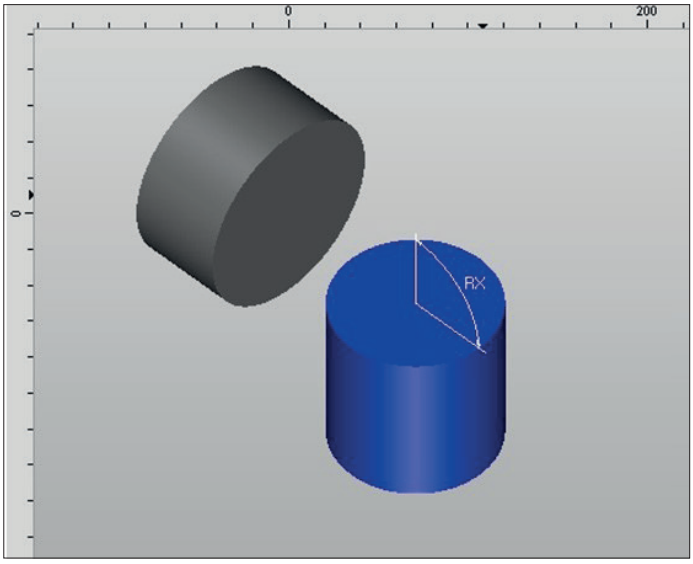


Number of different blank components is 32, so you can combine different types of blank to describe the workpiece.



The geometry of one Outside from the table blank component can be defined as a Box, Cylinder, Tube, Stepped cylinder or STL.	
Box	The box is defined by 3 dimensions in mm (X, Y and Z). The reference point is one corner of the box: 
Cylinder	The cylinder is defined by the length and diameter. The reference point of the cylinder is the center of the rear face (cap). 
Tube	The tube is defined by length, external diameter and internal diameter. The reference point of the cylinder is the center of the rear face: 

Stepped cylinder	Up to ten steps can be defined with different diameters and lengths: 
STL	The blank is defined by the file name of an external STL file. This can be a model saved from a CAD system or a part processed in Flexium 3D and exported to an STL file. 

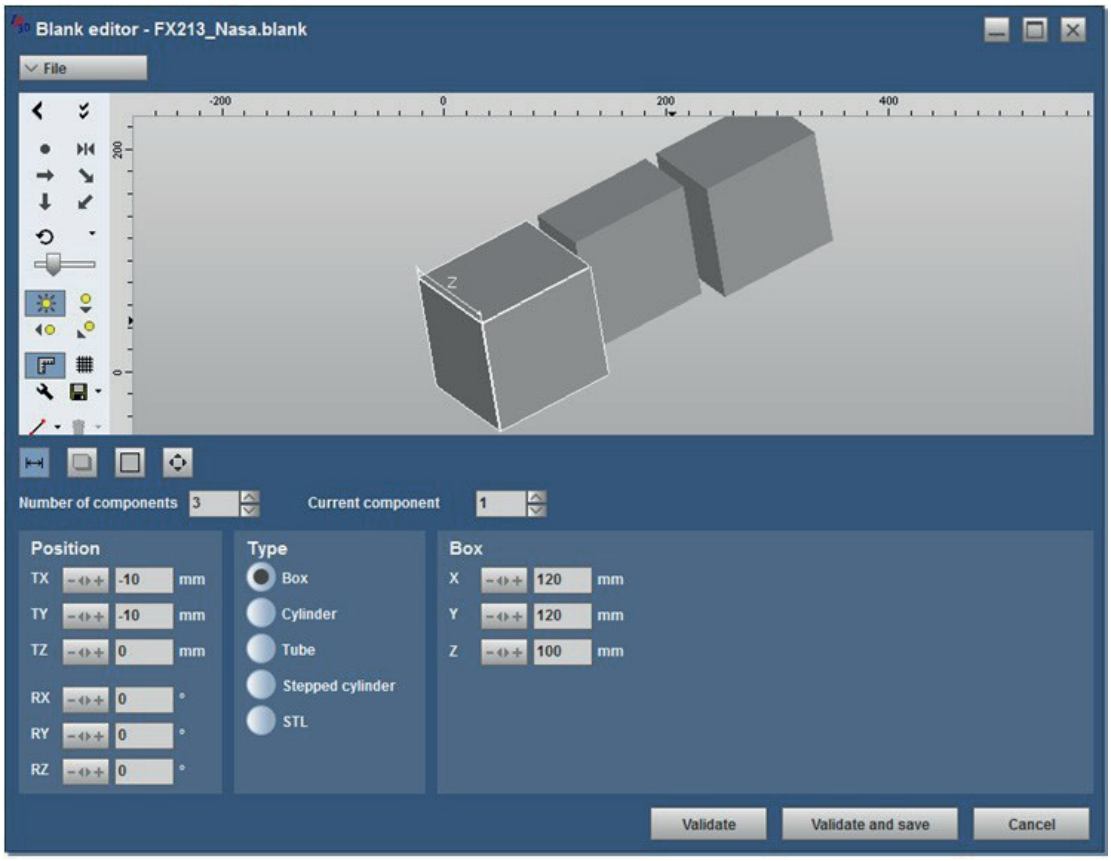
Position	The blank position is the offset from the turret reference point (the mounting reference) to the blank reference point:  In this example (the default position) the blank reference point is shifted 100 mm from the turret reference point: 
	Both linear and angular offsets are permitted. For example RX 90 would rotate the blank 90 degree around the X axis, keeping the reference point fixed: 

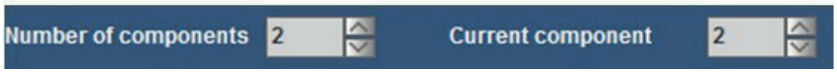
6.1.2 Manually edit values

You can manually edit values (dimension or position) or you can use the *"Slide"* button to decrement / increment the value by 1mm / degree by pressing on the – *or* + signs.

The *"Slide"* button allows you to enter number values without using the keyboard.
If you press the *"Slide"* button in the middle (on the <> sign) and drag to the left / right (while holding the mouse left button down) the value will also change in the corresponding direction.

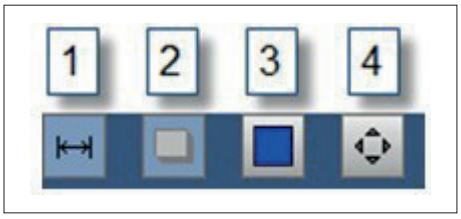
6.1.3 Number of components





Number of Components	Number of work pieces in work area. Max 32
Current component	Define the current component index to allow the input of geometry data. The current component is marked in the 3D view with a white frame.

6.1.4 3D view buttons



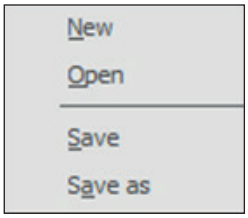
1	Enable the display of the dimension lines (quote) for the parameter currently being edited.
2	Activate the display of a generic turret (gray cylinder or box).
3	Change the color of the blank object.
4	Reset the zoom to fit the blank object to the 3D view.

6.1.5 Bottom tool bar

Validate	Validate all changes and use the current blank for the simulation. If there are unsaved changes a save question dialog is prompted before the Flexium 3D application is closed or a new configuration is opened.
Validate and save	Validate the current data for the simulation and saves the changes to the blank file.
Cancel	Discard all changes.

6.1.6 File menu

The file menu contains the following options:

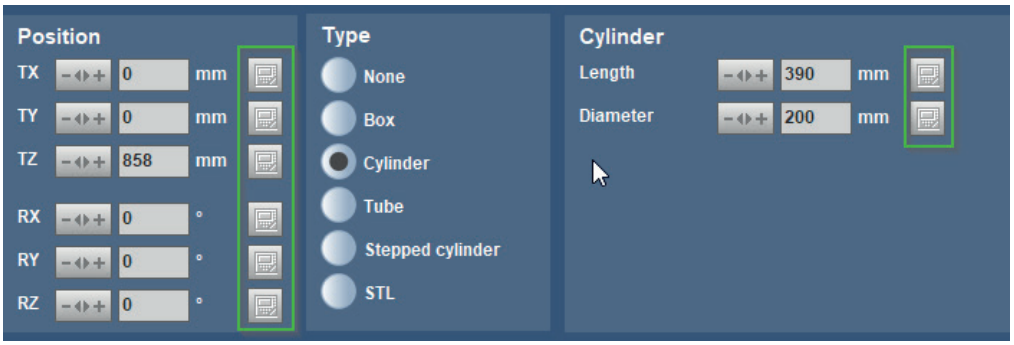


New	Create a new default type blank (cylinder).
Open	Open an existing local file (.blank file type).
Save / Save as	Save the current blank to a local file.

The BLANK file is a text ini file that contains all the information about blank (shape, geometry data, position, color).

6.2 Resize and relocate Blank

Dynamically resizing and relocating blank in Flexium 3D, is possible using 14 E-parameters. Each parameter must be first enable with a dedicated button on the right of the input field.

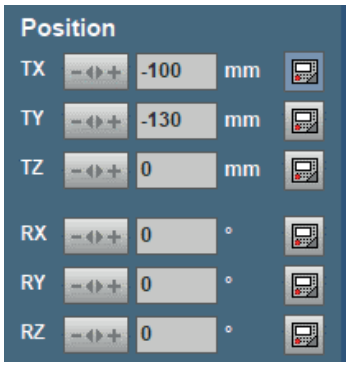


Dedicated E-parameter must be programmed in internal unit; an easy way to program the values in mm/degrees is divide the values by U1/M1 respectively.

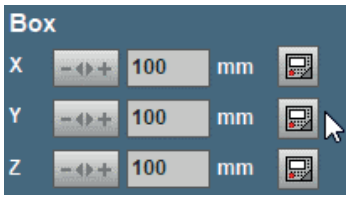
eg:

```
%101.0()()
(Blank box dimension in mm)
E89690=500/U1
E89691=100/U1
E89692=300/U1
(translation and rotation)
E89680=10/U1 (transl.X)
E89681=-50/U1 (transl.Y)
E89682=10/U1 (transl.Z)
E89683=90/M1 (rot.X)
E89684=180/M1 (rot.Y)
E89685=45/M1 (rot.Z)
M02
```

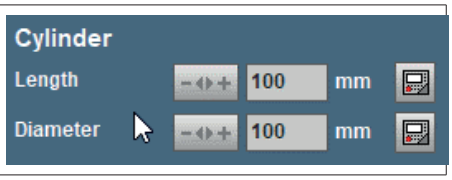
Position:	
TX:	**UE89680**
TY:	**UE89681**
TZ:	**UE89682**
RX:	**ME89683**
RY:	**ME89684**
RZ:	**ME89685**



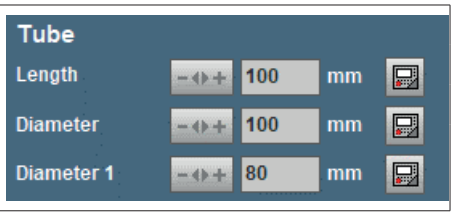
Blank Type:	
Box X:	**UE89690**
Box Y:	**UE89691**
Box Z:	**UE89692**



Cylinder:		
Cylinder:	Length:	**UE89693**
Cylinder:	Diameter:	**UE89694**



Tube:		
Tube:	Length:	**UE89695**
Tube:	Diameter:	**UE89696**
Tube:	Diameter 1:	**UE89697**



Limitation:

- Currently G70 (inch programming) /G71 (metric programming) is NOT supported;
- All supported Blank type can have only 1 component.

6.2.1 Difference between revision

Before version 5.0.0.0, the rotation around Y (RY) follow a different convension respect the other two rotation: positive value apply a counterclockwise offset, negative a clockwise offset.

From revision 5.0.0.0, the rotation arounfd Y (RY) follow the right-hand rule, as the other 2 rotation. Please check the project if you import it from a version older than 5.0.0.0

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Summary

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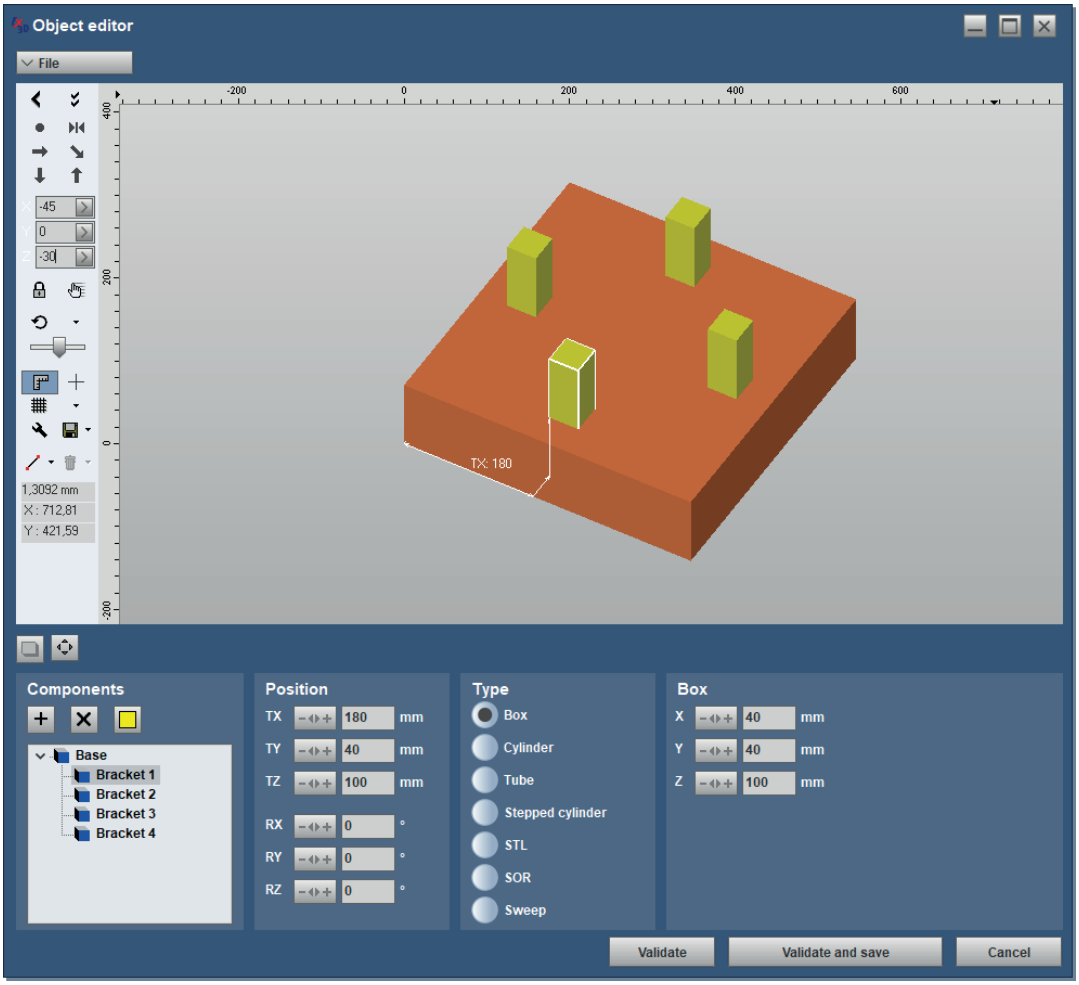
7 Object editor window

7.1 Use of Object editor window

The Object editor window is used to describe simple or complex custom clamping devices for the work pieces (Collet section on CNC configuration window), for custom tool holder devices and for custom tool shape.

7.2 Object editor window



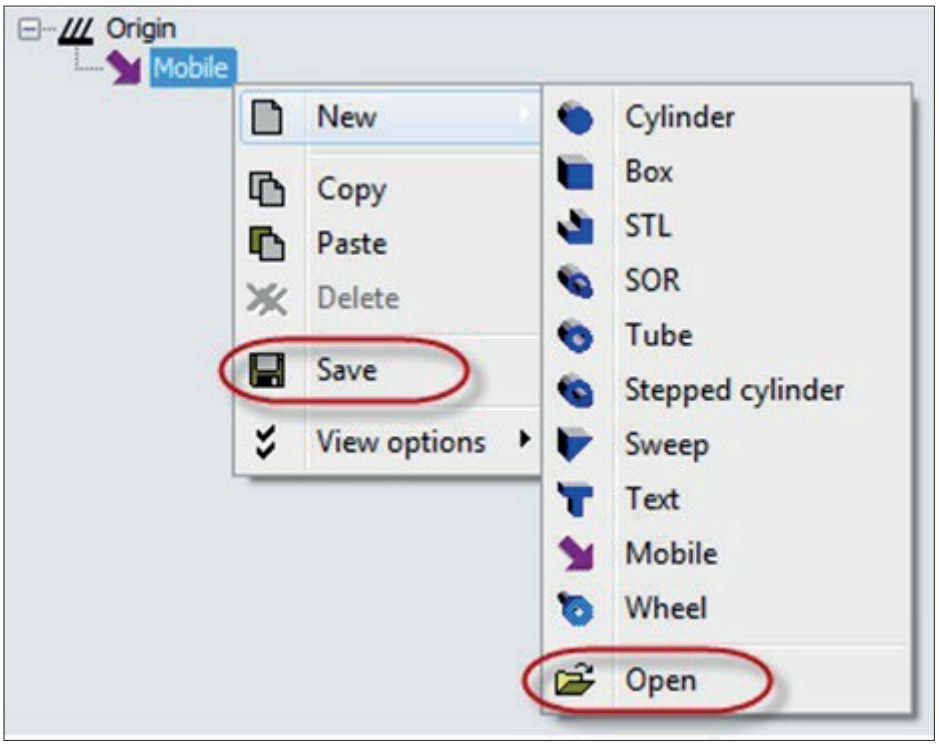
7.2.1 File menu

The file menu contains the following options:

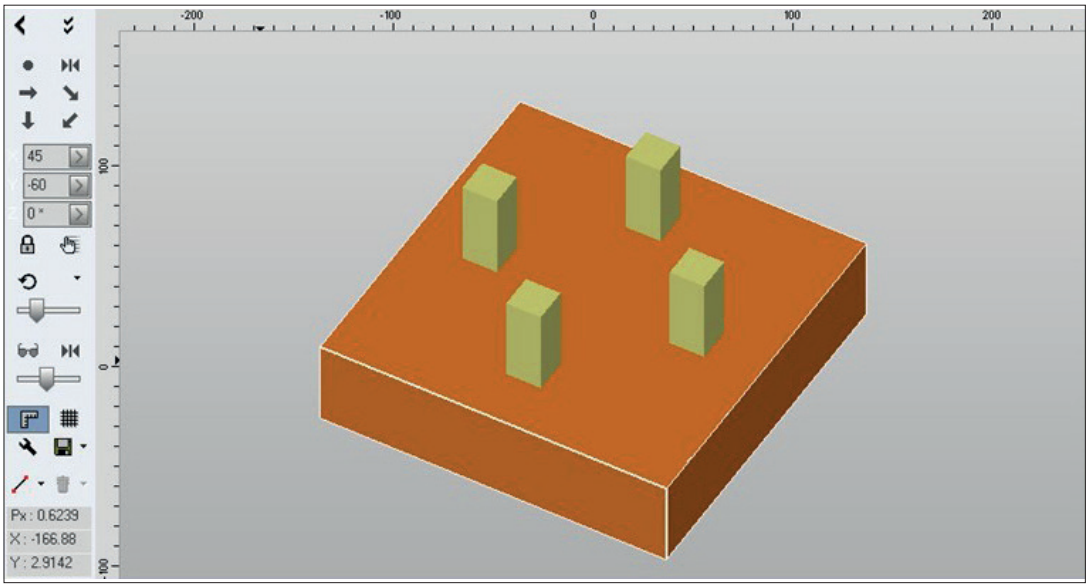
New	Create a new void object.
Open	Opens an existing file (3DO extension).
Save / Save as	Save the current object to a 3DO file.

The 3DO file has a binary format that describes a tree structure of 3D objects. There is one single root object which can have one or several children objects.

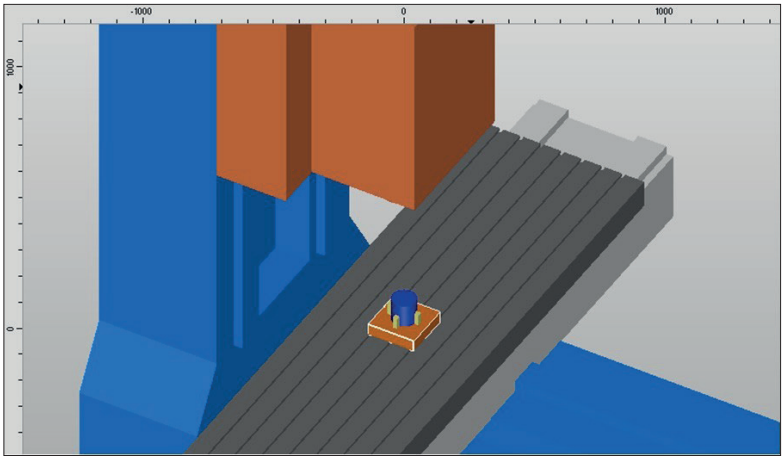
The format is compatible with the 3D objects used for the construction of machine models inside Machine editor window. The 3DO files can be imported and exported by the Machine editor window, using *"Open"* and *"Save"* options from the right click menu of *"Construction"* tree view.



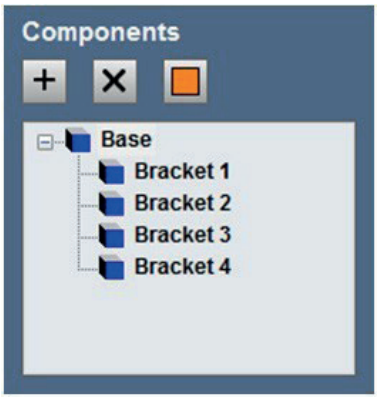
7.2.2 3D view



7.2.3 Custom view options

Show auxiliary object	Display of machine construction together with the current object. This option provides better orientation for the global position of the edited object: 
Fit	Reset the zoom factor for best fit of the current object inside the 3D view.

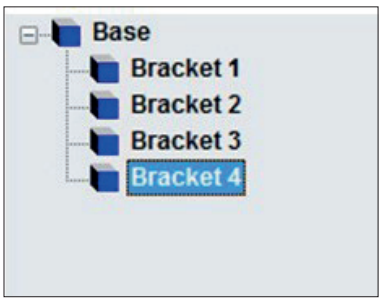
7.2.4 Components panel



The panel is used to add or remove objects from the tree structure.

New	Add new object to the tree structure. The new object will be created as child of currently selected object. If the tree structure is empty, the root object will be created.
Delete	Delete the currently selected object
Select color	Change the color of currently selected object.

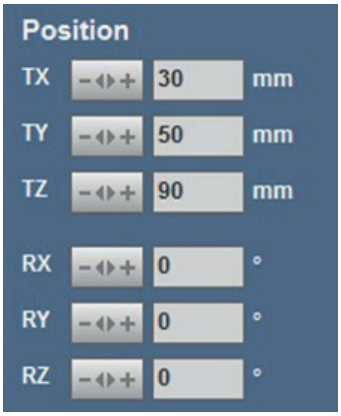
The tree view is used to select the current object.



By drag and drop operation, it is possible to move one selected object to a different parent object (change its location inside the tree structure).

When creating a custom tool, only the first element will be used for material removal.

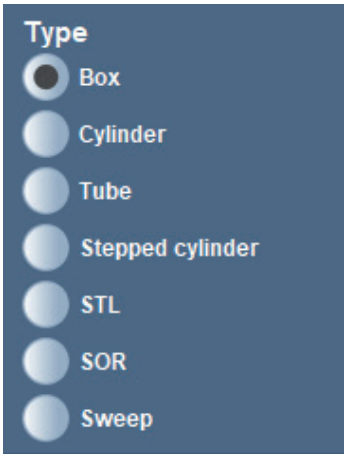
7.2.5 Position panel



This panel is used to change the position of the currently selected object by defining translation offsets (TX, TY, TZ) and angular offsets (RX, RY, RZ).
The position is defined relative to the parent object coordinate system.

If the current object has children objects, the children objects will move together with the current object when the position is changed.

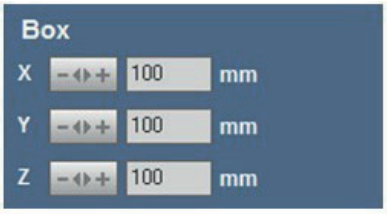
7.2.6 Type panel

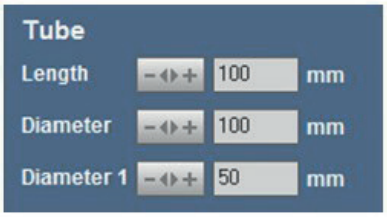


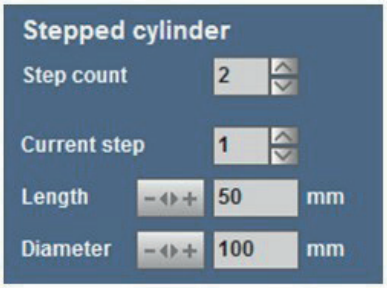
Type panel is used to change the type of the currently selected object. The type can be a standard shape (Box, Cylinder, Tube, Stepped cylinder), free form triangle mesh object (STL), SOR and Sweep (SWP) file format (see Appendix C for more details).

7.2.7 Geometry panel

Depending on the current object type, the geometry panel contains different parameters that describe the geometry of the object:







7.2.8 Button tool bar

Depending on the current object type, the geometry panel contains different parameters that describe the geometry of the object:

Validate	Validate all changes and uses the edited object for the simulation, as clamping device for the blank for example.
Validate and save	Validate the current data and saves the changes to the 3DO file.
Cancel	Discard all changes.

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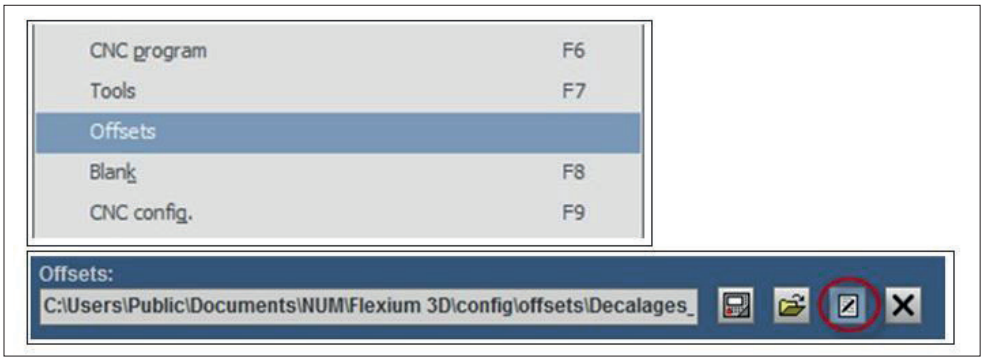
8 Offset editor window

The offsets editor can be used to create or modify a special type of file that is processed during the simulation by Flexium 3D.
An offset file contains three sets of offset values for each of the following NC axes:
X, Y, Z, A, B, C, U, V, W.

8.1 Opening the offsets editor

The editor can be opened from the Options menu (F2) and from the CNC configuration window.

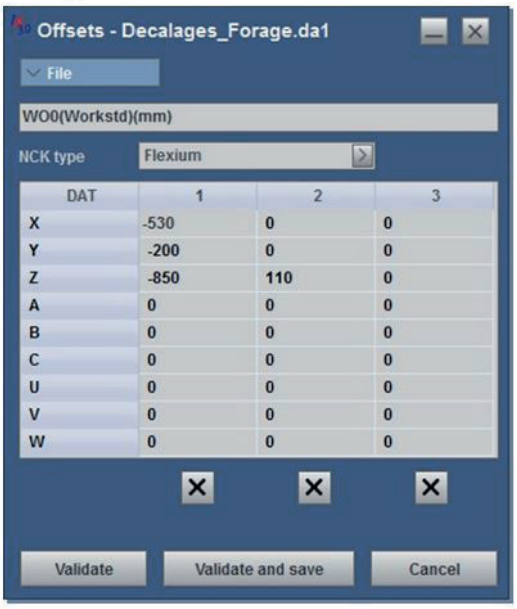


8.2 Window description

The window contains:
- NCK type: Flexium or Flexium+ (valid also for Flexium Pro);
- a file menu;
- an offset table for the NC axes with one axis per row and three columns for each set of offsets;
- three buttons with an X shape icon for resetting each offsets set to zero;
- three buttons to close the window with or without saving the file to disk and storing the changes in memory so that the new values can be used for the next simulation.

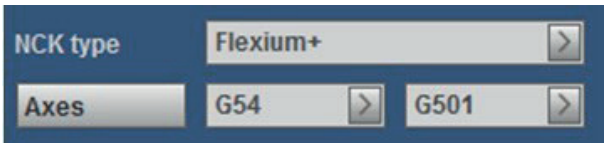
The offset table has a line for each axis name and three offset values for each axis:

NCK type Flexium



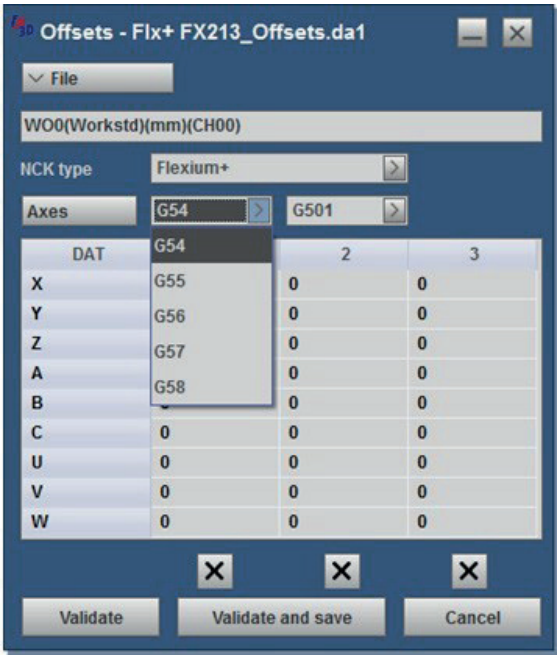
NCK type Flexium+

An offset value can be edited by modifying the value of the corresponding cell in the configuration table. Pressing the return key validates the entered value.

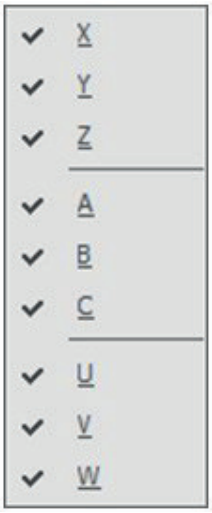


If NCK type is set to Flexium+ (valid also fro Flexium Pro), then 5 DAT1 (part origin) and up to 99 DAT2 offsets (program origin) can be edited. The current DAT1 offset is defined by G54 – G59 and the current DAT2 offset is defined by G501 – G599:

The *"Validate"* button closes the window and stores the changes in memory for the next simulation. In addition, the *"Validate and save"* button also saves the changed file on the disk.
With the *"Cancel"* button, the modifications are discarded and the window is closed.

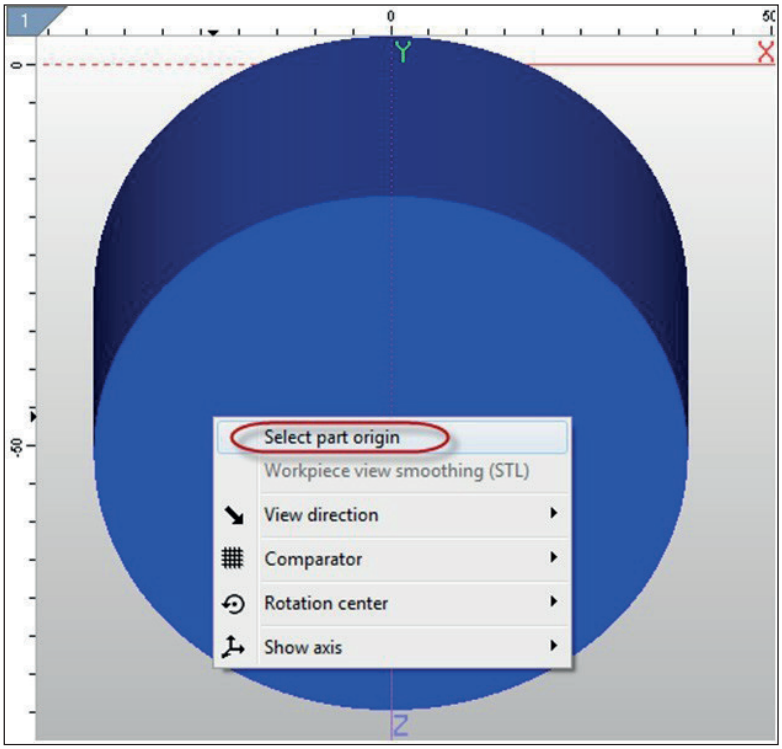


The involved axes can be selected by pressing *"Axes"* button that opens the following menu:

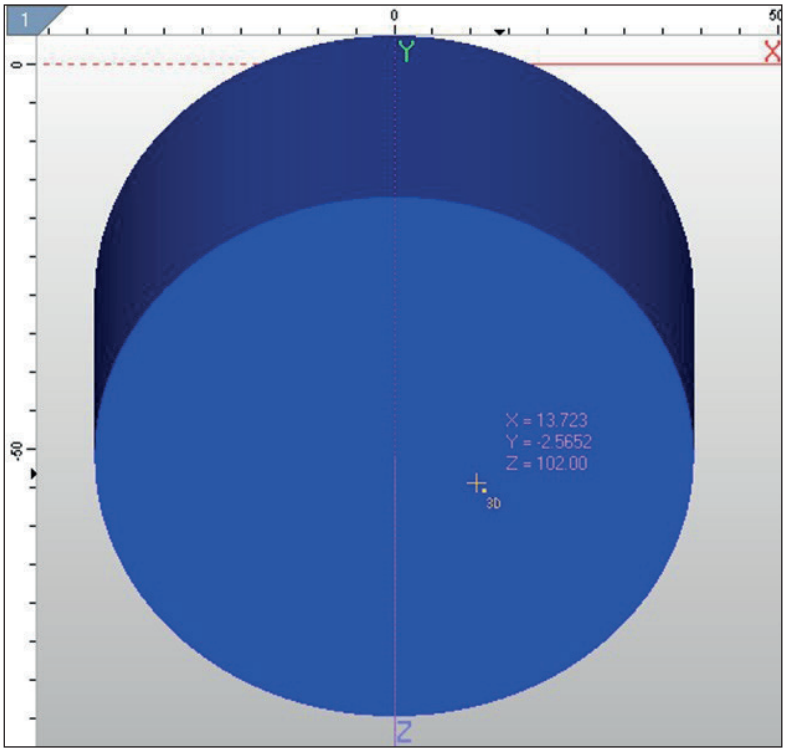


8.4 Select part origin from 3D view

The part origin offset can be defined by selecting a 3D point in the work piece view.
The right mouse click inside the work piece view will open a popup menu:

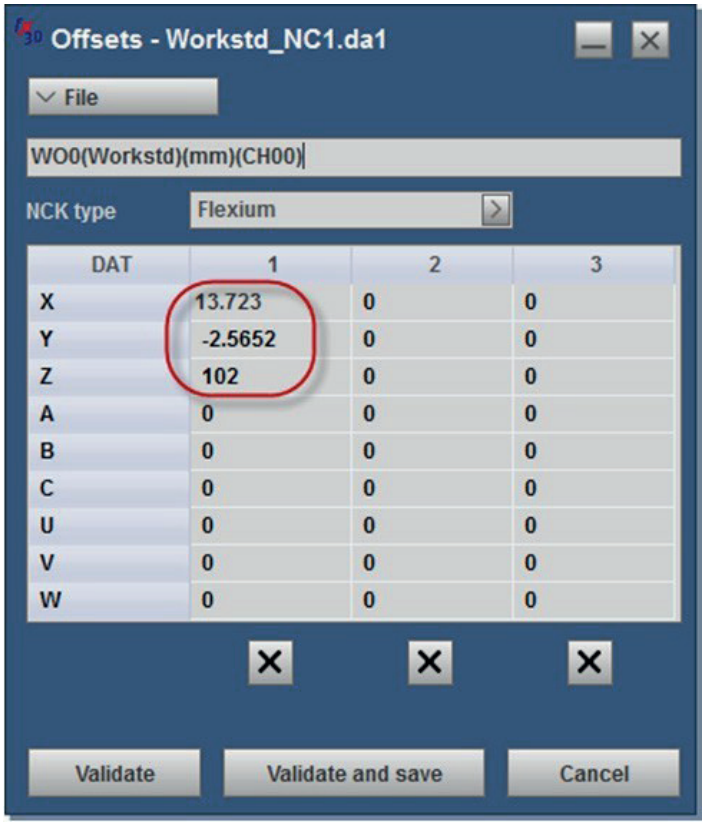


After choosing *"Select part origin"* the work piece 3D view will enter a selection mode where the X, Y and Z coordinates are displayed at the cursor position while the mouse cursor moves over the blank object:



Clicking on a point inside the 3D view will open the Offsets editor window and set the current X, Y and Z values on the first column of the offset table.

If necessary, the values can be edited and then validated:



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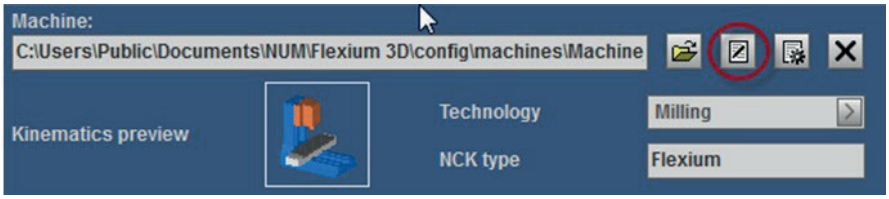
9 Machine editor

Machine editor windows is used to make changes to the machine kinematic by adding or removing axes, change the travel direction or travel limits or change the axes groups. You can also add, remove or move physical objects.

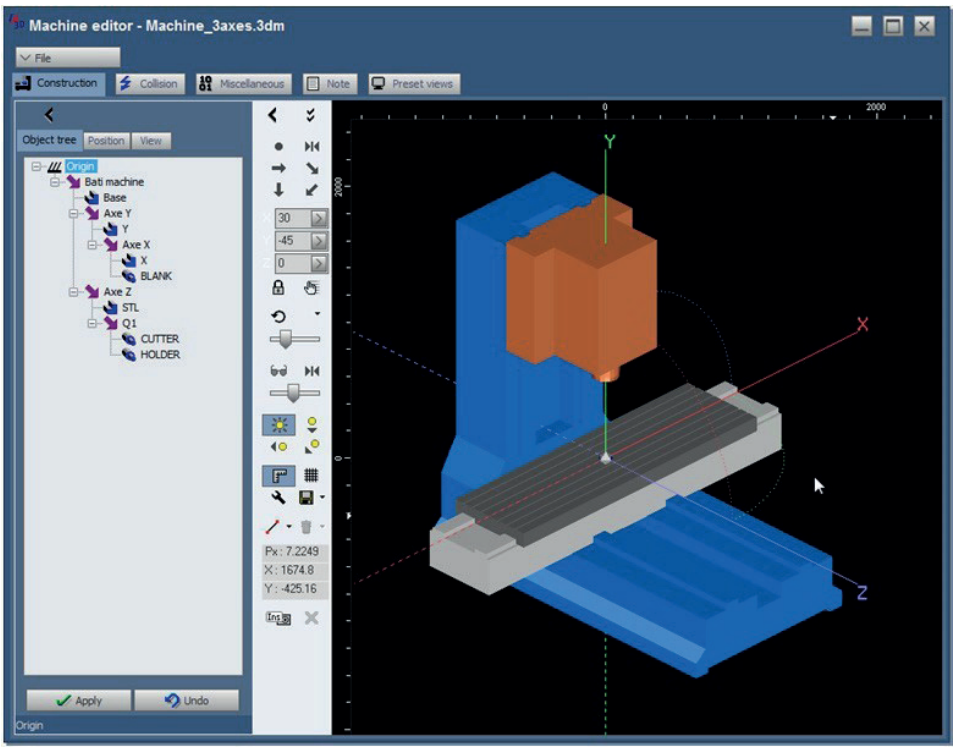
9.1 Open machine editor

To open the machine editor, first open the CNC configuration window then press the edit button for the machine kinematic file:



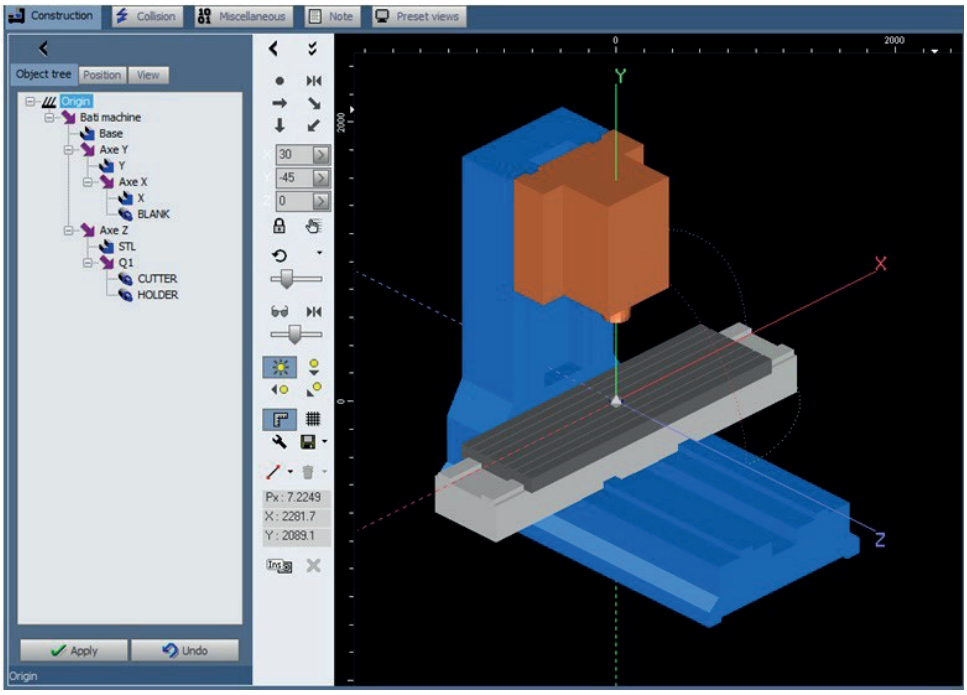
9.2 Page description

There are five pages inside the editor window, signified by tabs:
Construction / Collision / Miscellaneous / Note and Preset views.



9.2.1 Construction

On this page the machine kinematic and physical component are edited.



On the left side there is an object tree view that displays the machine objects and the relationship between them.

There are two main types of objects:
- logical objects: machine axes or group objects
- physical objects: cylinder, box, STL object, ...

Any object can have several child objects. When an object linked to a parent object, it will move together with the parent. In the picture above *"Axe X"* will move together with *"Axe Y"*.

A machine has one logical base object and several axis objects. The axis objects have to be linked to one another to respect the machine kinematic. You can change the parent of of an object by dragging the object with the mouse and dropping it on a new parent object in the object tree view.

The axis objects can have physical components attached to them. In this case the physical objects will move together with the axis.

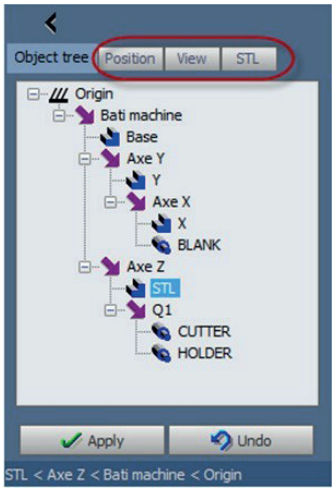
There are some special (and mandatory for 3D simulation) designated names for objects:
- *BLANK*: physical blank object. A machine should always have a blank object that will be replaced with the correct shape blank object for the simulation;
- *Q1, Q2, ... ,Qn*: logical object where tools can be mounted
  (a machine should have at least the Q1 object).

From the file menu, pressing New button the system will generate a complete but basic structure with mandatory and optional parameters. The structure must be modify according to machine kinematics.

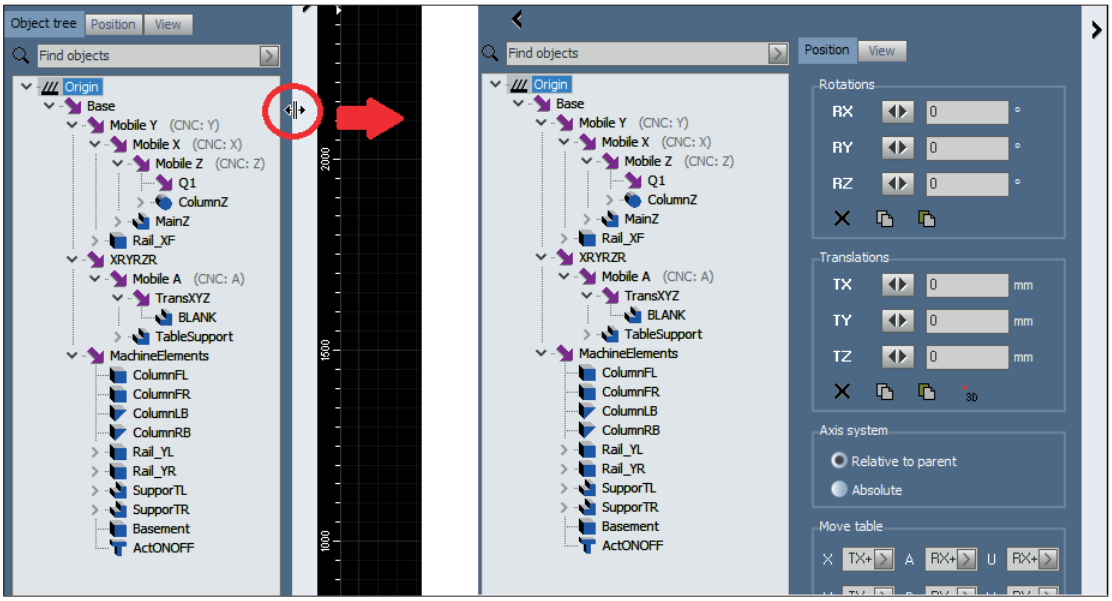
Remember that this New button is different from the New button present in the main CNC Configuration page next to Machine file path: this one will create a very basic structure useful only for TCP path generation, and not for collision check, 3D view and so on.

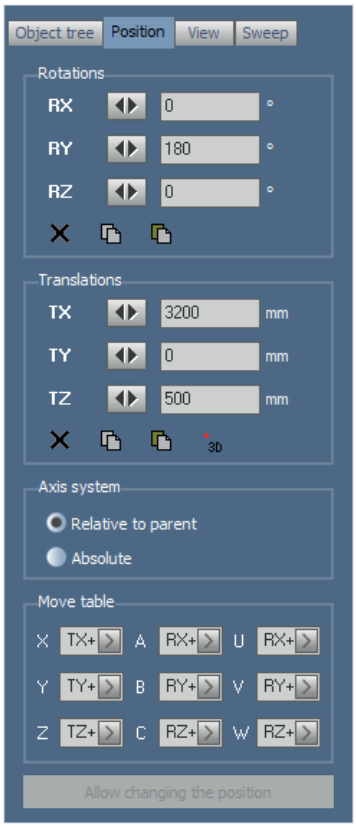
To edit one object it has to be selected in the object tree view. Physical objects can also be selected by double clicking the objects in the 3D view window.

The properties of the current object can be edited on three different pages:
- Position, View, specific attributes.



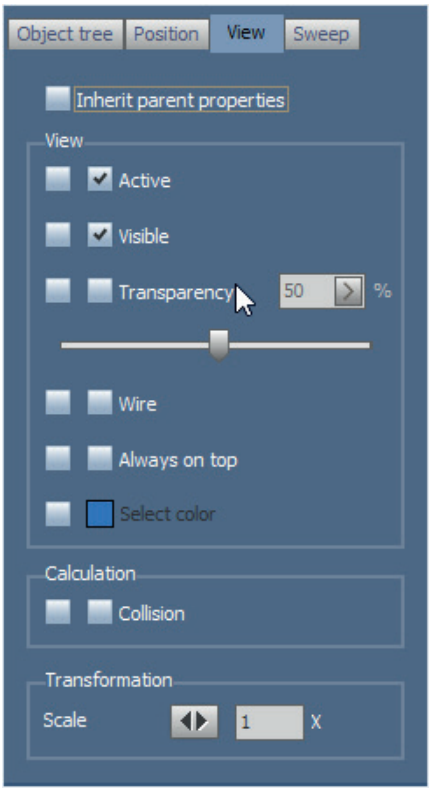
Dragging on the right side the border of Object tree, the property section will be visible side by side. Drag it on the left to restore original visualization.



Position	The Position page allows shifting and rotating the object. All child objects will also be moved. By default the relative position of the current object to its parent is defined. The rotations and translations are made with respect to the parent object coordinate system. Delete, copy, paste and sample function are present, with dedicated buttons. (from rev 5.0.0.0 RY follow right hand rule. to determinate offset sign. Refer to Appendix A6) 

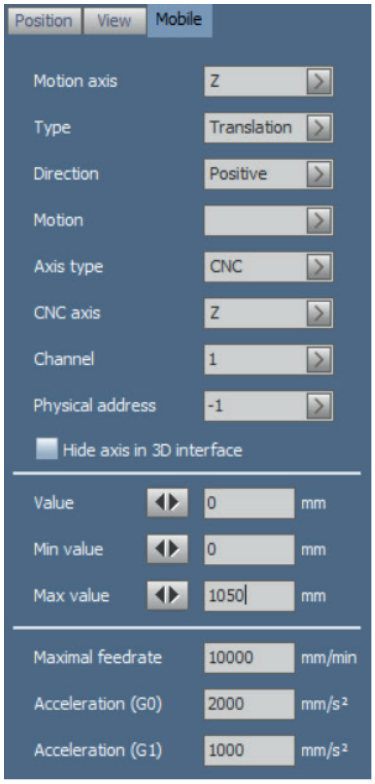
View

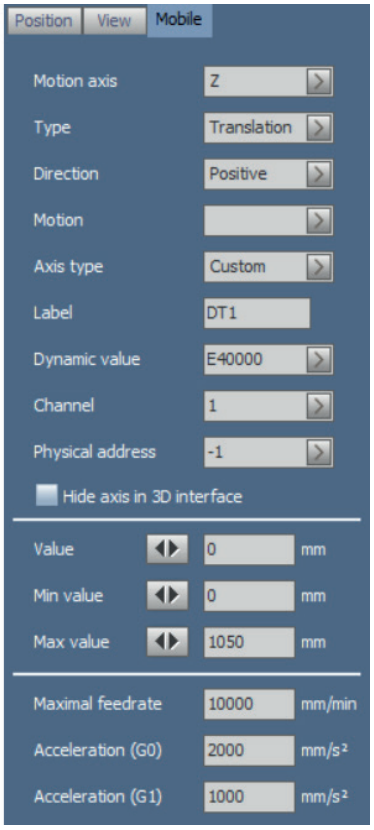
The View page only makes sense for physical objects since axis or group objects don't have a visual representation.



On this page some basic attributes can be set:	
Active:	the object is visible and can generate collisions.
Visible:	the object is visible (an invisible object can still generate collisions).
Transparent:	the object is transparent.
Wire:	the object is displayed with wires.
Select color:	change the object color.
Collision:	the object can generate collisions.
Scale:	scaling factor of the object.

Mobile	For axis objects mobile the following specific attributes can be set:

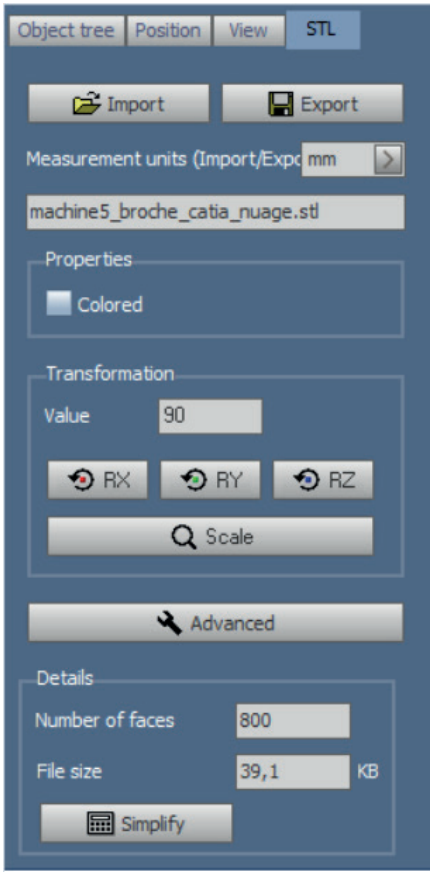


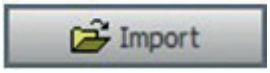


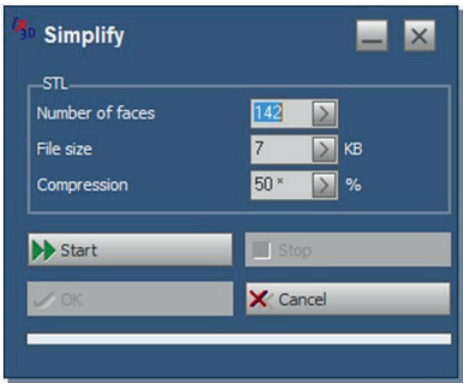
Motion axis:	the logical motion axis (relative to the parent object coordinate system).
Type:	translation, rotation (for swivel axis), or none (for a group object for example).
Direction:	positive or negative. (from rev 5.0.0.0 Direction follow right hand rule. to determinate movement sign. Refer to Appendix A6).
Motion (only for rotation type):	shortest, complete, sign direction, turning spindle and milling spindle.
CNC axis:	the CNC axis or spindle number.
Value:	position of the axis (this is a default park position).
Min value / Max value:	travel limits.
Maximal federate:	used for processing time estimation during G0 movements.

Additional parameters from rev.5.0.0.0	**Channel** select axis channel. **Axis type** Different kind are available: - none: usefull for offset and as folder: not connect to any axes - CNC : link to NCK axis - Spindle: link to NCK spindle - Custom: link to a virtual axes. Define the reference in the field Dynamic value - Dynamic value: reference for custom axes could be an E parameter (eg: E40000 or E80000) or a L variable (eg:L100) fo ONLINE mode (Digital Twin application) or a **[FX3D_HostCmd_...]** variable for OFFLINE simulation (see Appendix B2). - Label: for Custom type, show a name in machine tree. - **Physical address** Use physical address value instead of logical address (useful for ONLINE simulation). Default value -1 disable the function and logical address is used.

For other type of objects there are different specific attributes. For STL objects binary or text STL files can be imported:



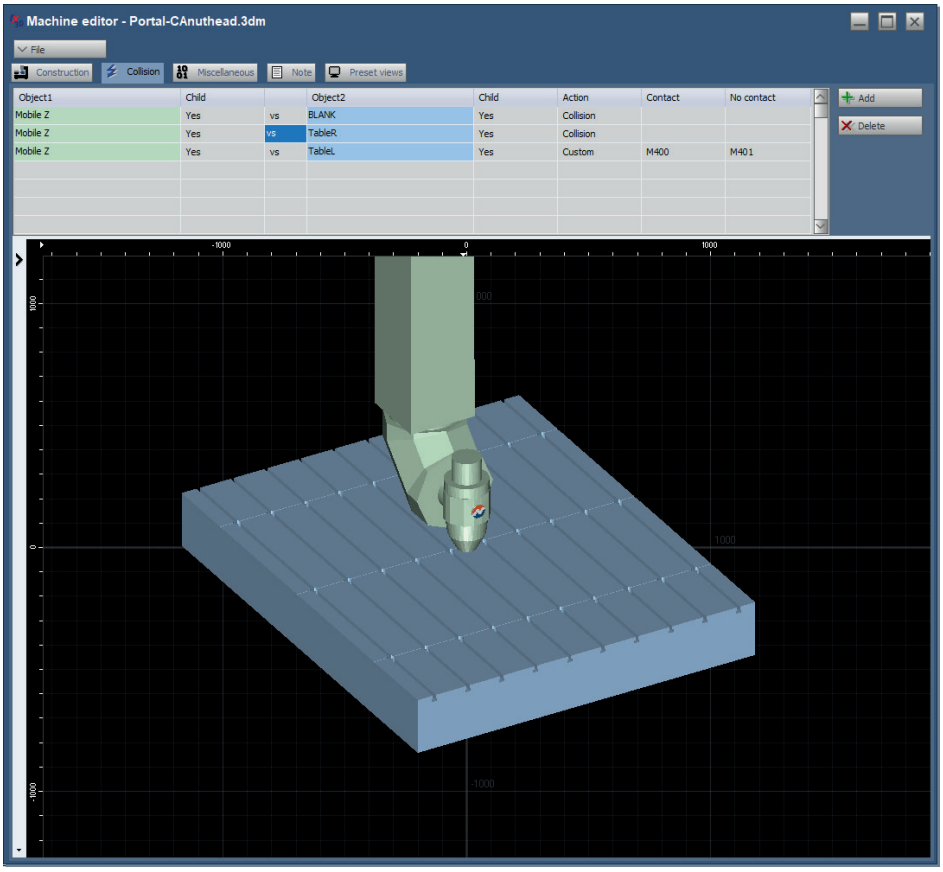
Import	 Import STL file.
Properties	Visualisation coloured or not
Transformation	Transformation settings for: • rotation around Cartesian axes • scaling factor to increase or decrease.

Details	 File information of imported zSTL-file.
Window Simplify	 This window is used to simplify large STL objects. A target number of faces, a target file size or a compression rate can be defined. The *Start* button starts the simplification process.
Advanced Repair STL Surface:	It can be used for STL objects that have wrong illumination or holes in the surface. There are several repair methods that could be applied one after the other, until a correct appearance is reached.
Generate components:	This function is used to divide the object in a series of sub STL objects. Useful, for instance, to hide a part of the machine structure. (check WHE).
Remove triangles intesection:	Remove triangles intersected by children object (Box): can be used to remove a section of an STL object by defining a child 3D box object. After defining the position and size of the box, all triangles that are in contact with this box will be removed. This can be useful for removing some hidden parts (interior components) of complex STL objects.
Color	Instruments to force and manipulate color information in an STL file. The file must be created in origin with color information. Only some 3D file editors are able to save color data in an STL file (STL format don't support color as standard, but have some customizable bit).

For further instructions and examples, see Appendix A.

9.2.2 Collision

On this page a set of pair of objects that should be tested for collision is defined:



The *"Child"* property defines whether all children of the object should be tested for collisions.

For axis objects the child option has to be selected because only physical objects are tested for collision. In the picture above, all child objects of "Mobile Z" should be tested for collision against all child objects of "TableR".

In Action column, select:
- Collision (default): each detection will increase general counter;
- Custom: execute M command on contact and no contact (to define in dedicated column).

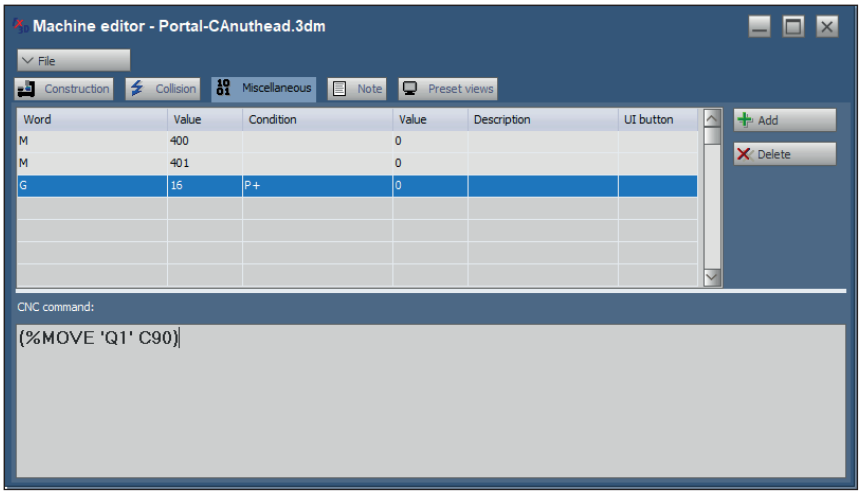
In this way is possible to have a custom signal for each couple of object, useful for Digital Twin simulations (touch probe, sensor simulation, etc...).

The 3D view displays the two object groups.

To check the possible collision with high speed animation (10x, 1fps, 2fps, ...) please be sure to select and setup Intermediate points function (see chapter 3.1.2).

9.2.3 Miscellaneous

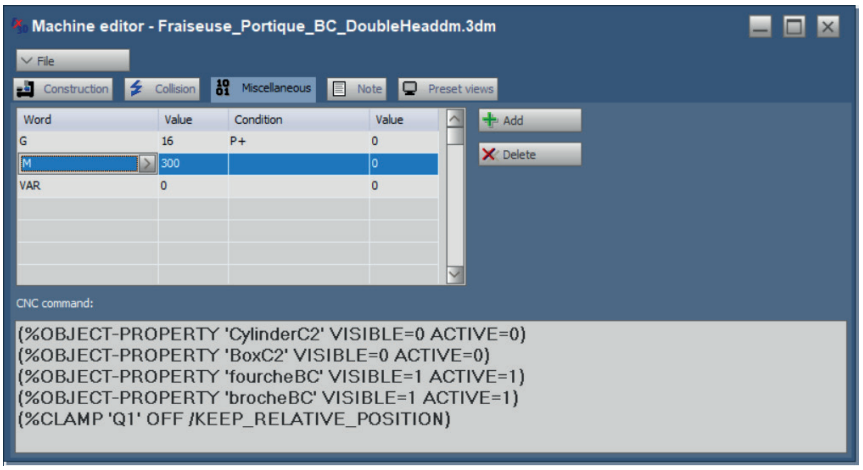
On this page a list of subroutines is defined which are executed for G or M functions:

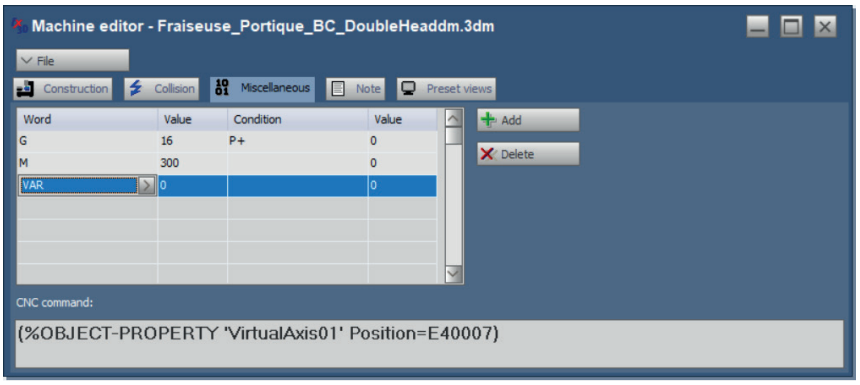


For each G, M or VAR command in combination with an optional condition, is possible to define a small subroutine with CNC commands (only a small subset of commands is supported) or special simulation commands (as round brackets NC comment lines) called Host Command. A list of the most important commands are reported in Appendix B.
Is possible also to add this command (describe with keywoards in description section) in a drop down menù, visible in the header of main window.

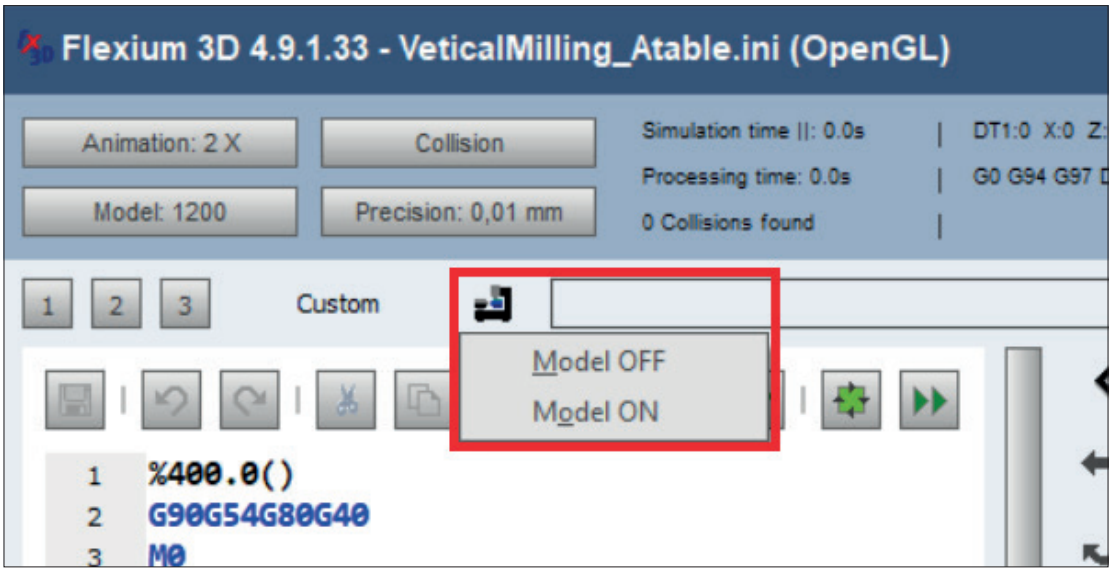
For the example pictured above, when a *"G16 P+"* program line is processed, the (%MOVE 'Q1' C90) is executed, which will rotate object 'Q1' around logical Z axis by plus 90 degree.

Below more example with M and VAR command.



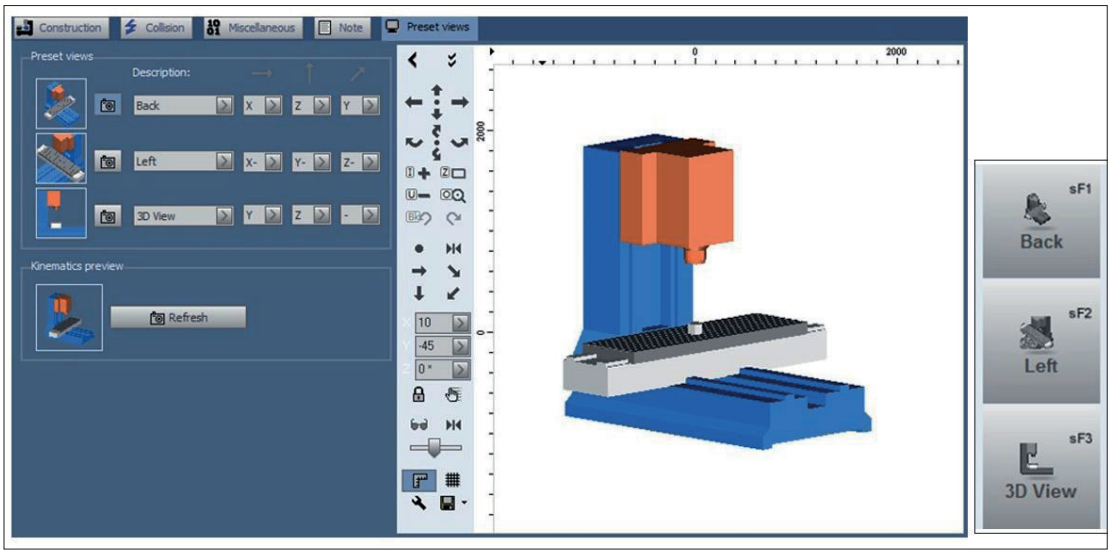


Description and UI button columns, are used to setup a dedicated menù visible on the header of main window, to easy execute command without run a part program.



9.2.4 Preset Views

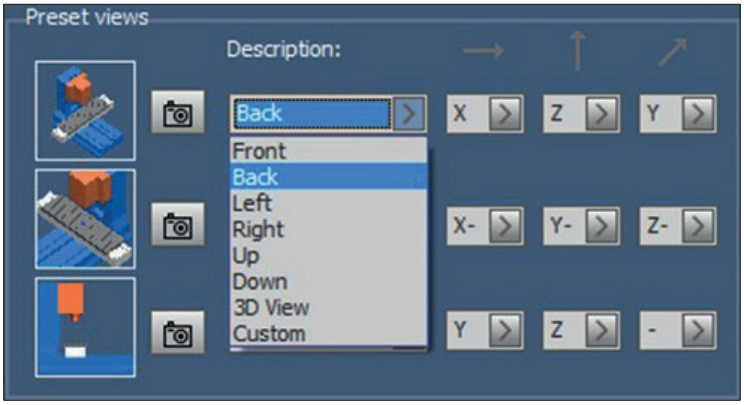
On this page the machine preset views can be customized. The preset views can be selected from the main window right side vertical tool bar when the machine view is selected.



For each preset view button, you can set a 3D view of the machine as the icon and the view description as the tool tip text.
For machine view it is helpful to have predefined views to the kinematics.

Each adjustment of 3D-modell is permitted for *"Preset views"*. A snapshot of the current 3D model view is used as an icon for the machine preset view button.

As tool tip indication for the three preset views, some predefined texts can be selected:



and Refresh	Buttons to take a snapshot of the current 3D model and set it as preset view or kinematics preview.

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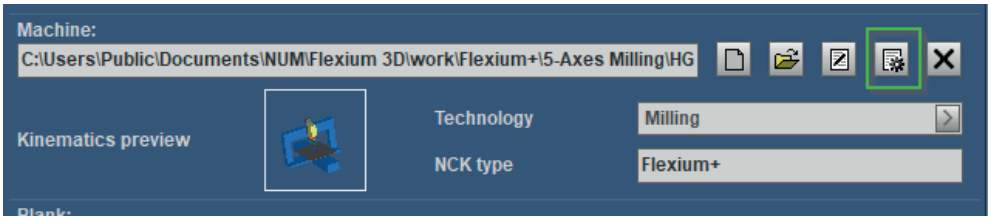
10 Machine axis configuration editor

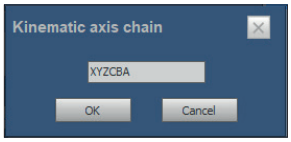
The machine axis configuration window permits the modification of the main NC axis parameters of machines stored in 3dm files. It cannot be used for altering the structure of a machine.

The following parameters of an NC axis can be edited:
- machine origin offset,
- park position value,
- travel limits,
- maximum feed rate.

10.1 Opening the machine axis configuration editor

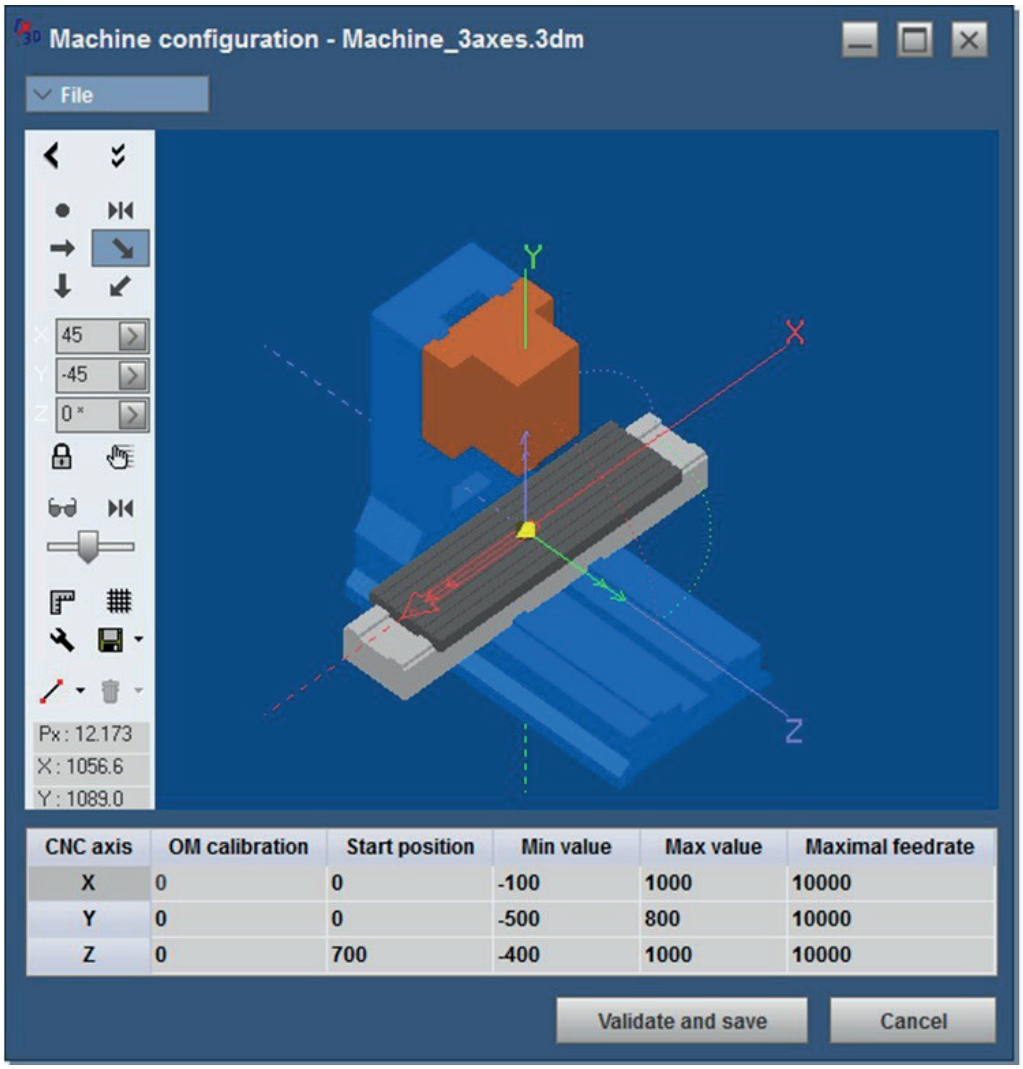
The machine axis configuration window can be opened with a designated button from the Machine section of the CNC configuration window:



	Create new basic kinematic chain: Define simplified kinematic chain from part/blank to the tool. Kinematic has no physical machine components 
	Open the machine axis configuration editor (reference 10.2)

10.2 Window description

The window contains:
- a file menu;
- a 3D view of the machine together with the associated 3D control panel;
- a configuration table for the NC axes with one axis per row and one column for each parameter;
- two buttons to close the window with saving/discarding the changes.



The configuration table has the following columns:
- *NC axis*: axis name
- *OM calibration*: adjusts the machine origin position by changing the position of the NC axis object.
- *Start position*: defines the default position from which the simulation starts. It corresponds to the Value parameter on the Mobile page of the Machine editor window.
- *Minimum value*: lower travel limit
- *Maximum travel value*: upper travel limit
- *Maximal feedrate*: maximal feed rate used for G0 movements.

Actual lower and upper travel limits will be updated with the values read from the machine parameter file (XPA). (At least for first channel).

10.3 Operation

An axis parameter can be edited by modifying the value of the corresponding cell in the configuration table. The axis for which a parameter is being edited is highlighted in the 3D view using an arrow shape.

- Pressing the return key validates the entered value.
- The *"Validate and save"* button records the current file changes, closes the window and sets the edited machine file as part of the CNC configuration.
- With the *"Cancel"* button, the modifications are discarded and the window is closed.

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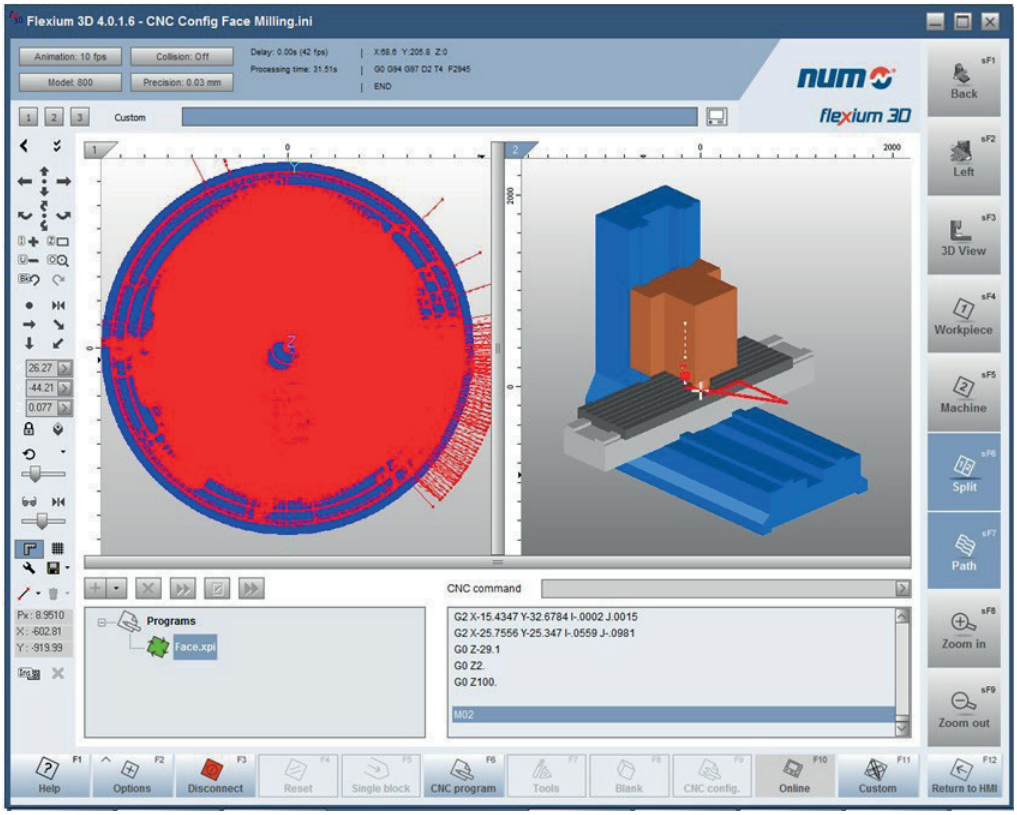
11 Online simulation

11.1 Description

The Online simulation mode is used to visualize real time machine axis movements when a connection to the CNC is established.

During the online simulation the path of the tool center point (TCP) can be visualized, the material removal can be simulated and collision check can be activated.



A CNC icon located at the right of the status message panel signals on-going communication with the CNC by turning the color of the CNC panel display from white to green.

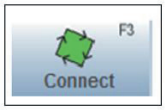
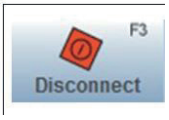
To have an accurate path and material removal simulation, is important to activate the oscilloscope function (see chapter 3.3.2).

11.2 Operation

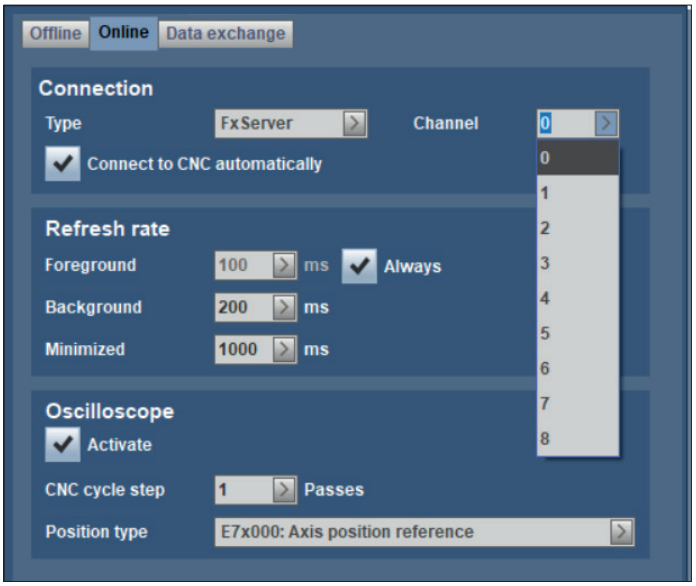
The F10 button shows the current simulation mode and switches from one simulation mode to the other.

	Online mode off
	Online mode currently active

In online simulation mode the F3 button is used to start and stop the simulation:

	Connect to the CNC and start the online simulation
	Disconnect from the CNC and stop the online simulation

By enabling *"Connect to the CNC automatically"* option from the Setup dialog, Simulation Online page, the connection to the CNC will be automatically established when the simulation mode is switched to Online.



All simulation settings can be set for online simulation similar to offline mode: animation, material removal model calculation, collision detection, precision, reset model at start and others.

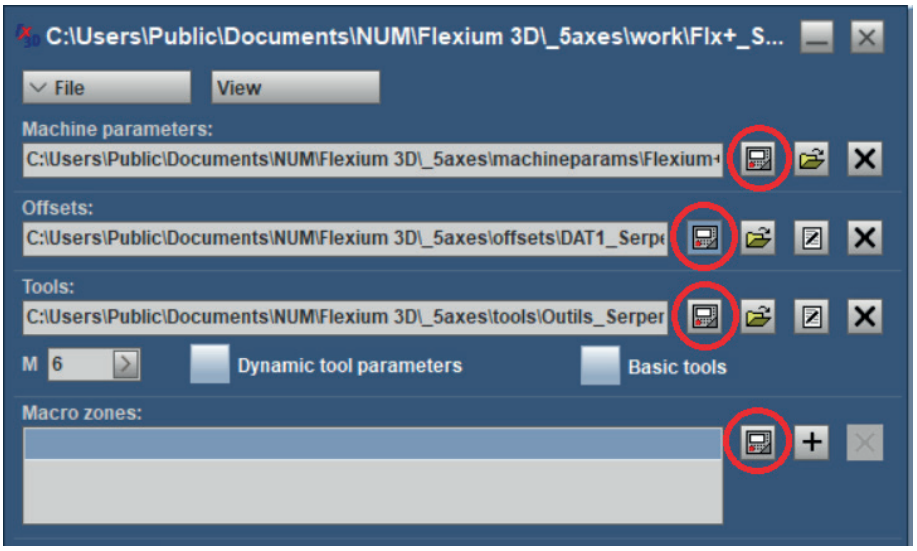
The settings for most simulation parameters are stored separately for offline and online simulation mode.

The online simulation path can be verified against the offline simulation path by following these steps:

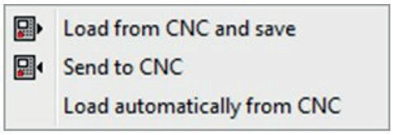
1. Activate the wire path display (s+ F8).
2. Run the simulation in offline mode (F3).
3. Open the setup dialog and disable *"Connect to the CNC automatically"* from the Simulation, Online page.
4. Activate *"Always on top"* and *"Time line"* options from the View, Path display page.
5. Switch to online simulation mode (F10).
6. Deactivate *"Reset model at start"* from the Options menu (F2) or activate *"Reset model at start"* and select *"Save current machined part as temporary blank"* from the same Options menu.
7. Connect to the CNC and start the online simulation (F3).

11.3 Synchronizing CNC data

The options to exchange data between the simulation environment and the CNC are available on the CNC configuration dialog for machine parameters, options, offsets, tools and macro zones:



Pressing the CNC options button for one resource file will open a popup menu:



The option *"Load from CNC and save"* can be used to synchronize the resource files used for the simulation with the current data from the CNC.

The option *"Send to CNC"* will send the currently used data for the simulation to the CNC.
If the *"Load automatically from CNC"* option is checked, then the resource file will be requested from the CNC every time the simulation starts.

11.4 Online Collision Detection (OCD) in manual mode

To use Online collision detection NCK option FXSO000521 is mandatory

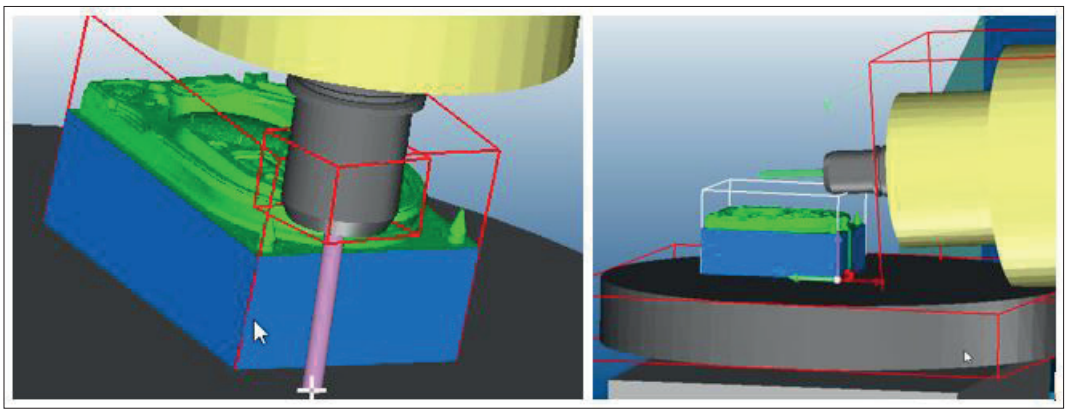
Online Collision detection Manual Moves

11.4.1 View back to Offline Collision detection behavior

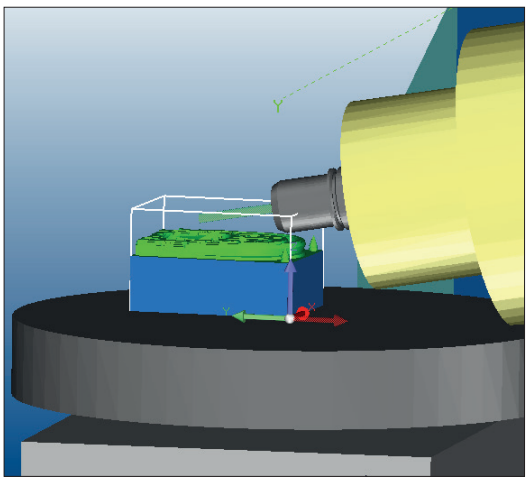
Since the beginning of Flexium 3D, and this is more than 10 years ago, the Offline collision detection was integral part in NUMs Simulation software. The same collision check analysis, started from the main program of the application, is done before the part programs are transferred to the NCK using the same Tool list, Offset table, kinematic description and transformation properties as it is on target machine. Only in case on collision free analysis the part programs are transferred/downloaded to NCK.

One big advantage of NUM's collision detection, also valid as competitive differentiator, is the consideration of the actual blank dimension and the removed material during collision calculation. For sure Flexium 3D is not only considering the tool path. Rather, the tool dimension and orientation, holder and all other machine components are considered simultaneously during collision calculation.

Visualization of collisions in Flexium 3D in form of red marked components shown with some examples.



There will be no collision detected if tool holder or other spindle/machine components are moving into the area of already removed material. White box is original blank size. Tool holder (grey cylinder) is located inside the already removed material, but no collision.



11.4.2 Enhancement to Online collision detection using Flexium 3D

Why there is a need for online collision detection in manual mode?

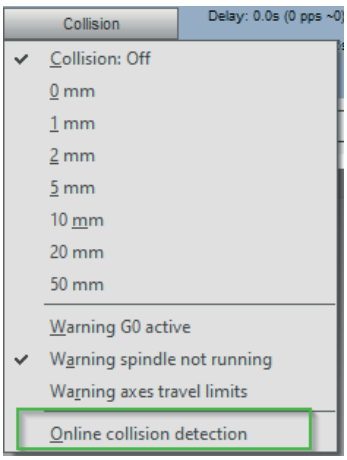
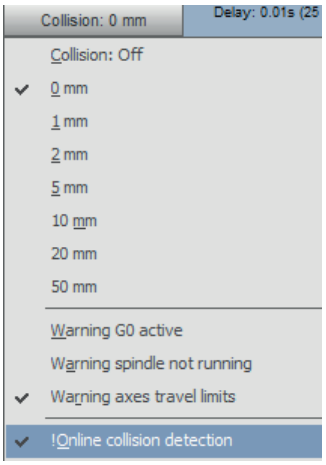
As machining processes become more and more sophisticated, the risk of collisions between the tool, workpiece and machine components increases. Collisions mean costly downtime and cause expensive damage to the workpiece, spindles and sometimes even the whole machine.

Flexium 3D features together with NCK extensions form new functions which allow monitoring and preventing online collisions during manual/jog movements, moving axes with handwheel or in Search-mode (part program resume) and in commissioning phase. In case of 5-axes machining with complex blanks shape, manual movements can be very critical, but online collision detection makes it simpler and safer!

Therefore NUM's Flexium+ System is a high performance CNC controller for milling and turning applications as well as grinding, water jet cutting and gear wheel manufacturing which enables with Online collision avoidance software enhancements a real time verification and collision detection dedicated in critical manual modes.

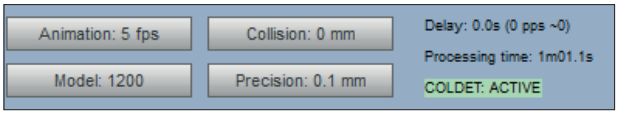
To activate online collision detection, change Flexium 3D mode to Online (F10) and press collision button at the top.

Wait for till communication to NCK via DNCserver /FxServer is established

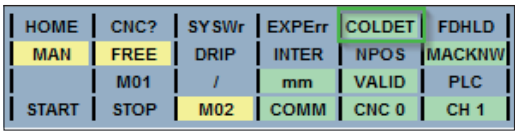
Wait Communication to NCK	If NCK confirmed the request:
	

There exists the possibility to configure safe distances for different machine components in Flexium 3D configuration.

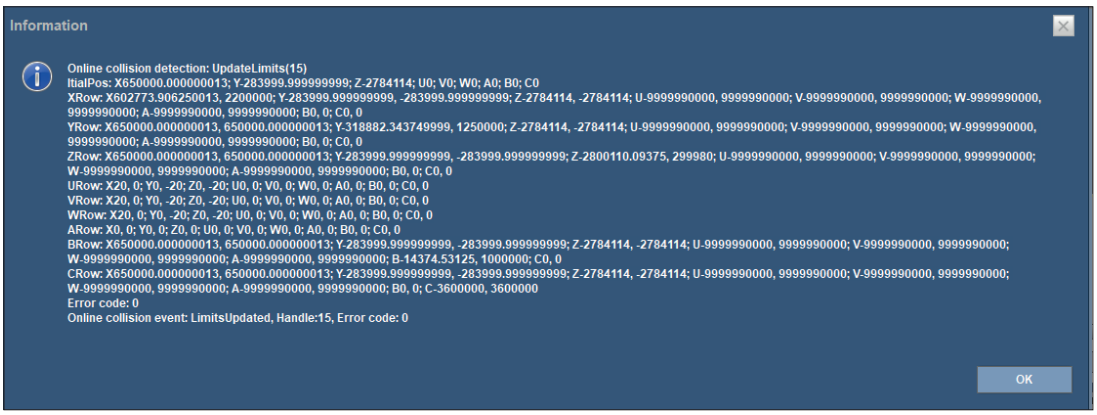
Flexium 3D calculates for all axes the limits, the limit information to NCK. If the limits are transferred and complete the OCD Status on Flexium 3D will change to **COLDET: ACTIVE**



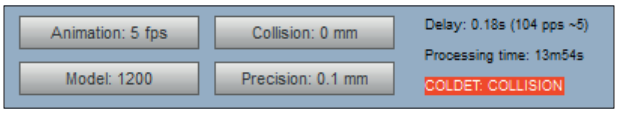
Same status COLDET the operator will have on Flexium+ HMI



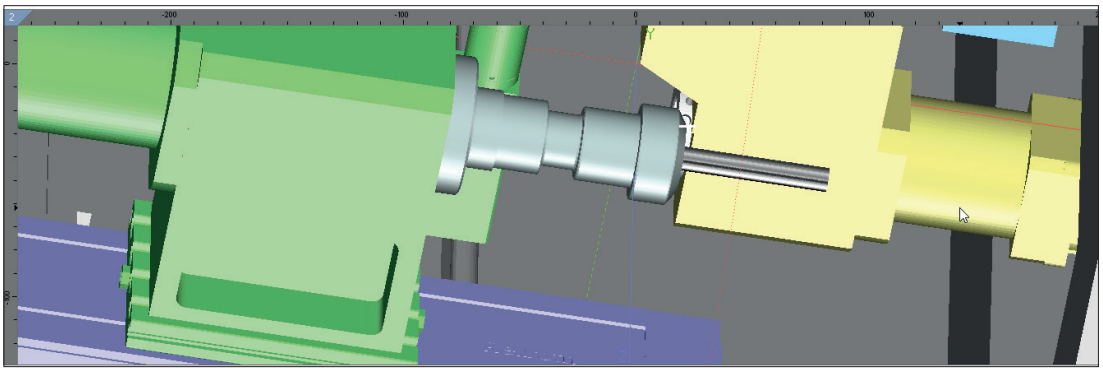
As control dialog, if you press green status button, you have a view of calculated axis limits



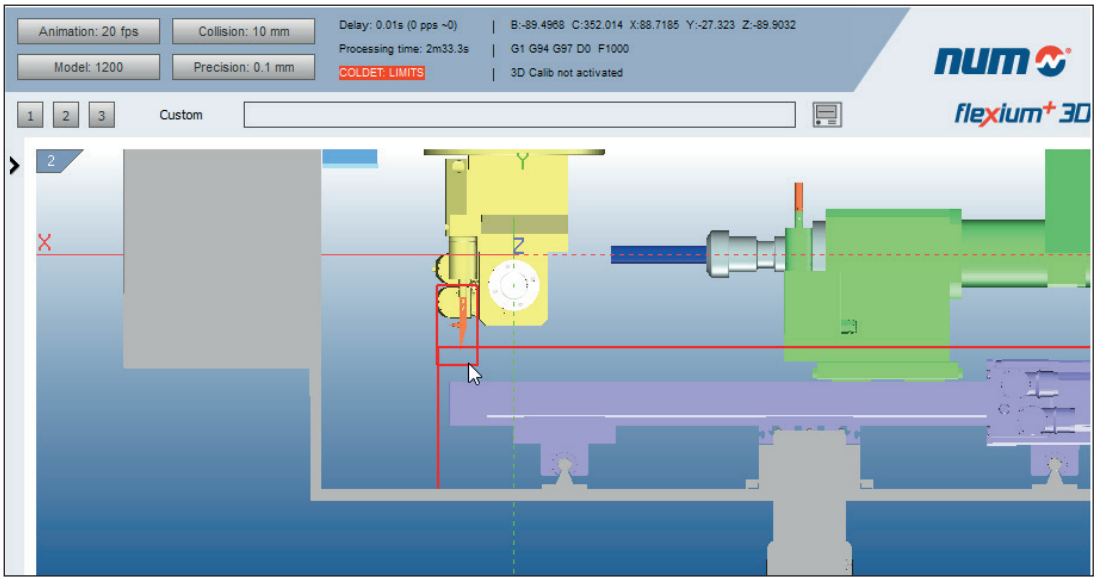
If there is a collision, try to move outside the limits, means one machine object will have a collision, the status will change to.



And the axes in NCK are free until a collision free position is moved.



Example for Collision message with 10mm safety distance.



If Flexium 3D is in Online mode, means connected to NCK, and “!Online collision detection” machine frame into account, detailled machine is activated, (displayed as green status) Flexium 3D takes the real axis positions in machine frame, detailed machine kinematic and calculates implicit for the physical present servo axes the permitted ranges for single and multi-axes moves (also in case of RTCP and/or inclined plane transformation). These permitted ranges contain also the consideration of work piece position with material removal, tool dimensions as well as machine components, like jaws, clamps and fixtures.

The NCK works with these permitted ranges during manual axes moves and controlled their Looka-head and ramp algorithm in a way that limits are never exceeded. If an axis is at a collision limit and the next move in critical direction would create a collision, the axes move or machining process is blocked and a dedicated operator warning is displayed.

For AUTO mode, the mentioned only collision free tested part programs should be executed. It is clear that a mandatory pre-condition is that the kinematic description contains all relevant machine components which are critical for the collision check.

Always be sure that the following data are correctly setup:

- use same parameter file of the real machine: a mismatch between parameter (eg: number of decimal digit) could lead to invalid data exchange;
- the feature use rising and falling edges of jog signals from plc to read and write data for collision control: be sure to correctly manage these signals from plc (no “always on” signals could be used for jog or handle);
- be sure that in machine editor exist all the axes of the parameters files: if for instance a W axes is define for tool change, add a Mobile element assign to CNC W axes also in machine editor.

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12 Text editor

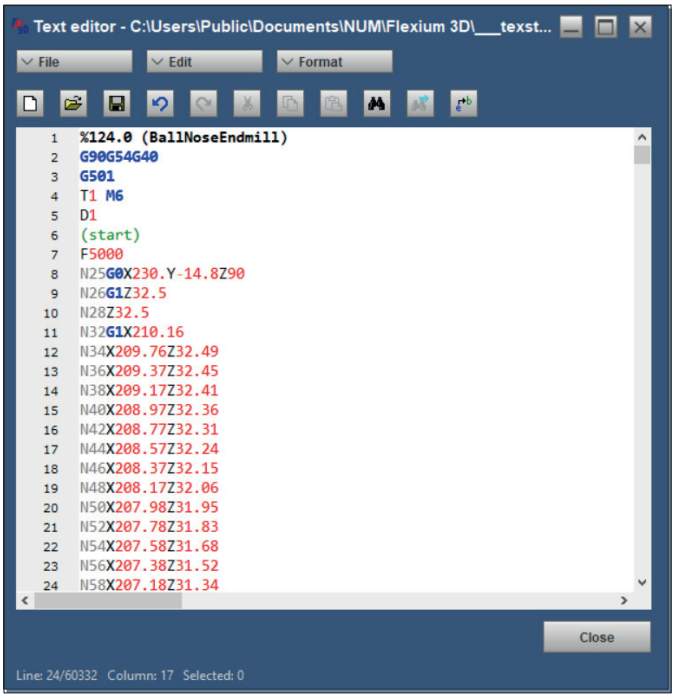
12.1 Text editor view

The purpose of the internal text editor is to allow making quick modifications to various text files used by the application.

The general icon is .

12.2 Page description

You can reach the editor from part program list view or CNC configuration window.



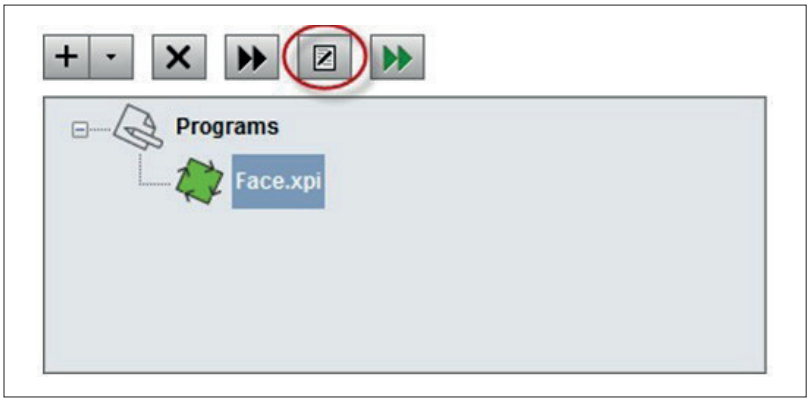
It supports the following operations, available in its toolbar and menu:

- Create new file
- Open file
- Save current file
- Undo, redo (one step only)
- Cut, copy, paste
- Find, find next
- Go to line number (available only through keyboard shortcut Ctrl + G)
- Change display font.

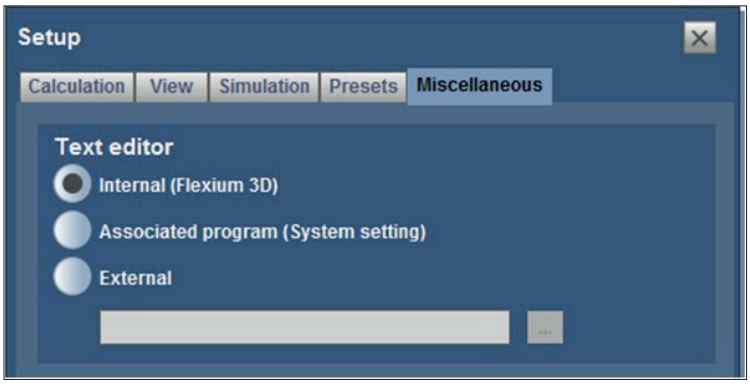
It has vertical and horizontal scrollbars and the word wrapping for longer lines is disabled. There is also a status bar displaying the total number of lines and the line and column coordinates of the cursor.

By design it has tolerable loading/saving times for files smaller than 10MB. There is a file size limit of 64KB when running in Windows 95/98/Me.

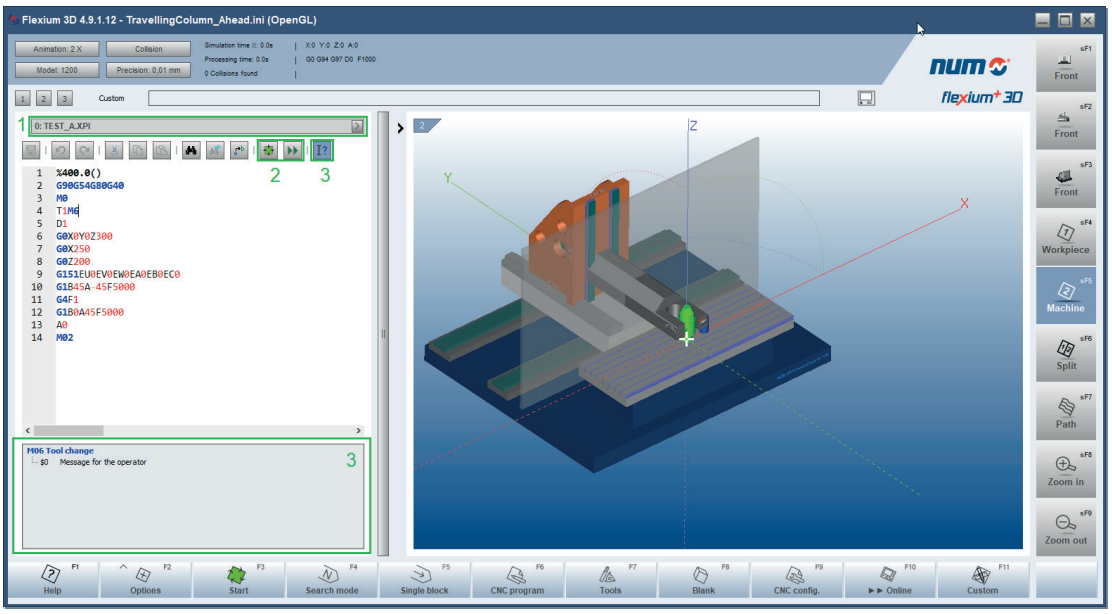
In Flexium 3D is possible to use the text editor to edit xpi, da1, mcd, jau, sor, ... files.
For example, the current part program can be edited by clicking the edit button:



Alternatively, external editors can also be used for text files. The setup for an external editor is made on the *Miscellaneous* page of the *Setup dialog*:



12.3 Text editor integrate in main window



From F2-Option menu, is possible to enable Text editor, that will open side by side with machine view, workpiece view or split view. The user could operate on a part program, without open and close it in a separate window.

Additional features are available respect standard editor, and in particular:

1. A drop down menu with the list of part program is available on the top part of the editor (if F6-CNC Program menu is closed): selecting a part program from the drop down menu will set it also as main program.
   If F6-CNC Program is open, select part program with dedicated tree view;

2. Additional cycle start and program parsing operations are available: executing a simulation with them will not require program saving, allowing the user to test a temporary part program modification;

3. Help online feature could be show with a dedicated icon (the help area will appear on the bottom part of the editor): click on the line beside the G code, E parameter, etc … will show the corresponding description.

The file that contain the description of the ISO function must be define as syntax_en.xml and must be add manually in the folder *C:\Program Files (x86)\NUM\Flexium 3D\Bin\help*.
If help folder does not exist create it.

All language are supported, changing the last 2 letter of file name: gr-German, en-English, fr-French, sw-Swedish, po-Portuguese, cs-Czech, it-Italian, tu-Turkish, fi-Finnish, hu-Hungarian, nl-Dutch, sp-Spanish, pl-Polish, ch-Chinese (simplified), hr-Croatian, ro-Romanian, ru-Russian, jp-Japanese, cht-Chinese (traditional), th-Thai.

A sample file could be download from ftp area (section Home – Customer_support – Internal_Common – Flexium3D)

Attention!
The file provided on FTP area is only a sample file and only in English language.

If provided to a customer, a check with official manual must be execute as well as translation to customer language. The file is customizable, to allow special G code function or dedicated E parameter assignment.

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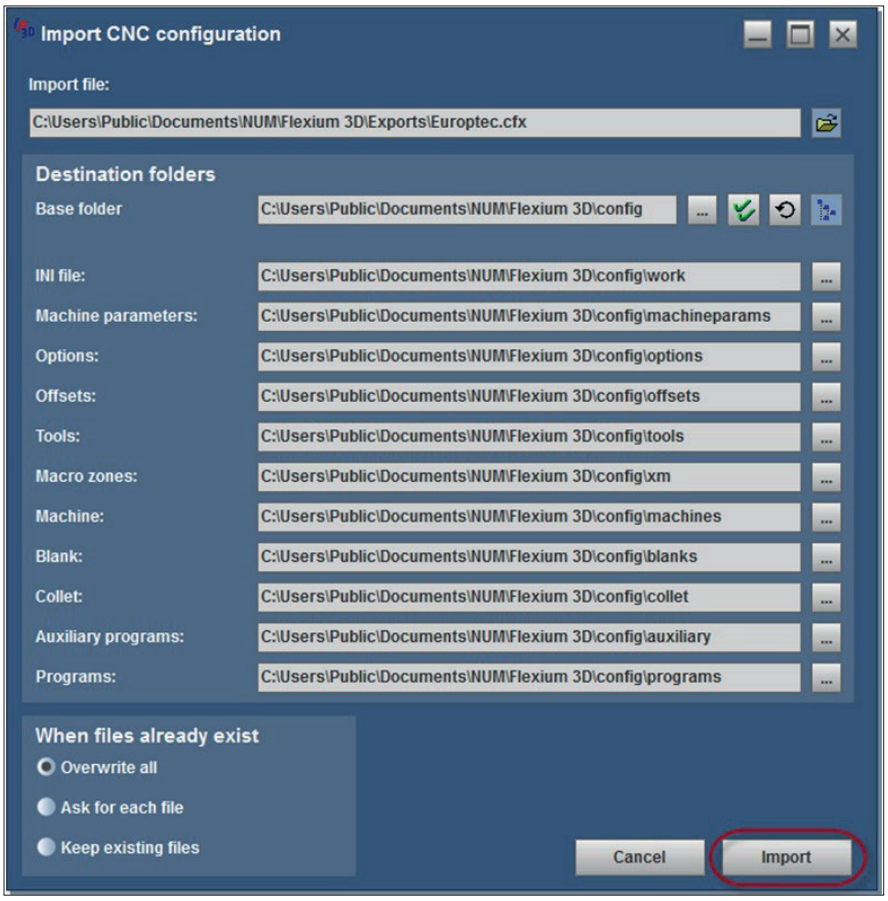
13 Import / Export CNC configuration

13.1 Import a CNC configuration

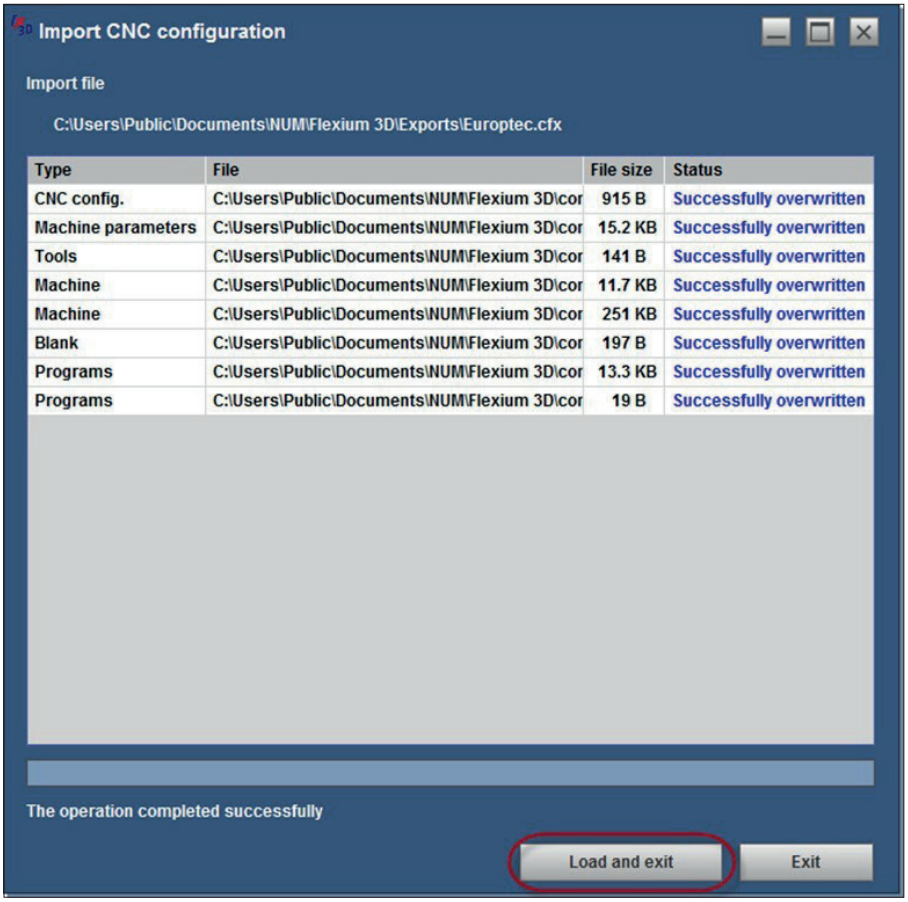
13.1.1 Command description

Unpack an exported file (cfx extension) into specified destination folders.

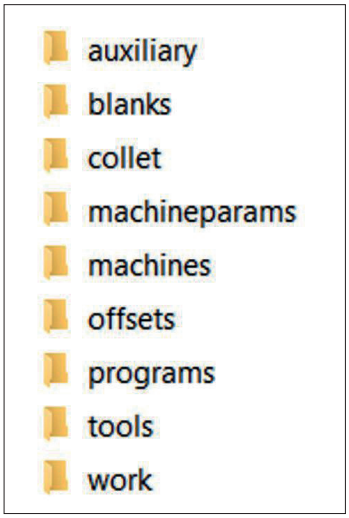
Each type of configuration file can be extracted into a separate folder and the last recently used paths are stored for future use.



While the import file is being unpacked, a progress window is displayed.



The picture below show the name of standard folders created; inside the imported files.



13.1.2 Operating steps

1	Select the archive file containing the CNC configuration to be imported (*Import file*).
2	Verify/Select the destination folder for each kind of configuration file*.
3	Select the action to be taken in case a file with the same name as the one to be extracted already exists in the destination folder.
4	Click the import button to start the extraction operation*.
5	The window is closed and another window is shown for displaying the progress of the extraction operation.
6	If the destination folders do not exist on disk, the application attempts to create them before the extraction begins.
7	Each file in the archive is extracted and placed on the disk in the folder corresponding to its type. The status line at the bottom of the window shows the number of the currently processed file.
8	The extraction status of each file is displayed in the *“Status” column**.
9	The user is prompted when a file with the same name as the currently processed file already exists in the destination folder (in case the *“Ask for each file”* option was selected).
10	When the operation is finished, the status line displays the success state.
11	If the import file contained a configuration ini file, the configuration can be loaded when closing the window with *“Load and exit” button**.

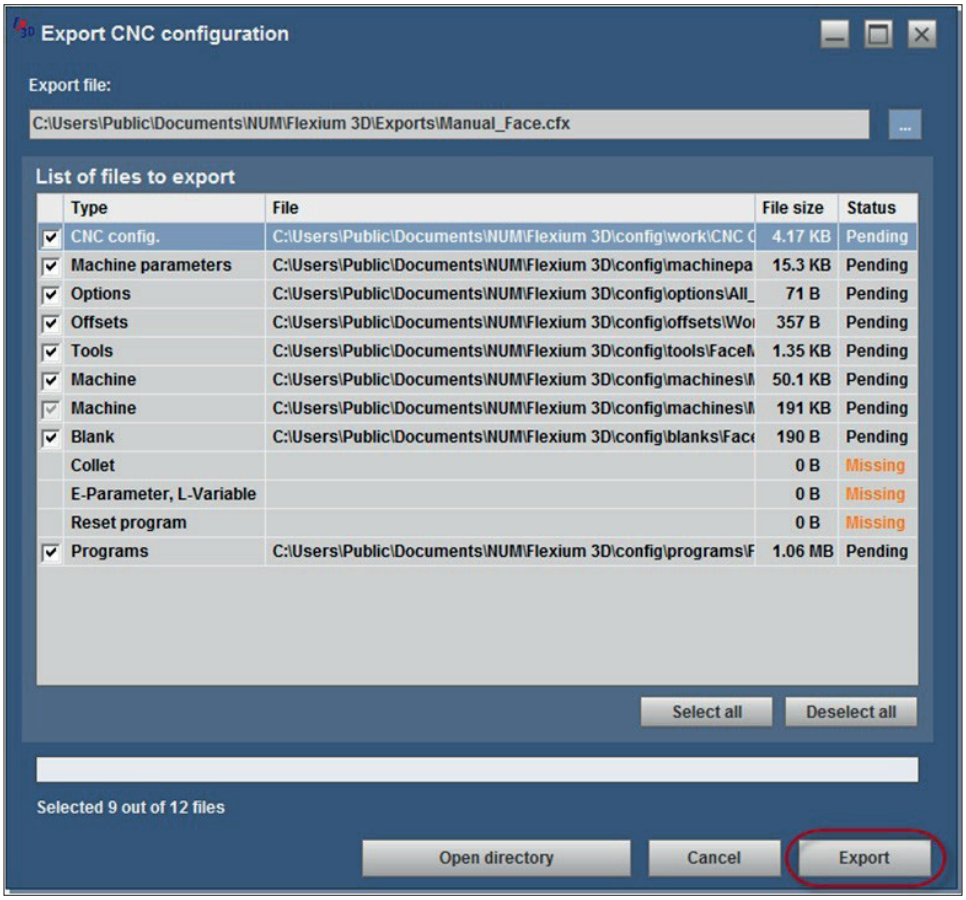
13.1.3 Important notes

Browse from base folder	When the *"Browse from base folder"* option is checked, the destination folders can only be selected from the subfolders of the base folder. If a destination folder is not a subfolder of the base folder when the option is re-checked, then the path is shown in a different color.
Import button	The import button is enabled only after an import file has been chosen.
Status column	Description of the status column: ***Error creating folder*** The destination folder could not be created. ***Overwriting*** The current file is being extracted and it overwrites an existing file. ***Extracting*** The current file is being extracted. ***Skipping*** The current file is being skipped (based on the choice of the user). ***Successfully extracted*** The file has been extracted. ***Successfully overwritten*** The file has been extracted overwriting an existing file. ***Error writing file contents*** Problems with the medium where the file is being written (possibly no enough disk space). ***Error! Cannot open file for writing*** There is a file with the same name in the destination folder, which is read-only or locked by another program. ***Error*** Other errors. ***Aborted*** The user aborted the import operation while the file was being extracted.
Configuration ini file	The configuration ini file: • A warning message appears at the end of the operation if the archive contained no configuration ini file or the file has been skipped.

13.2 Export a CNC configuration

13.2.1 Command description

To archive the current CNC configuration files into a single compressed file allows selecting the files to be archived and the name as well as the path of the compressed file.



13.2.2 Operating steps

1	Select destination path and name for the compressed file (*Export file*).
2	Select one or multiple configuration files to be exported. Result of selection can be seen in the *"Status"* column.
3	Click the export button to start the archival operation.
4	The status of each file is updated during the archival operation. The status line at the bottom of the window shows the number of the currently processed file.
5	When the operation is finished, the status line displays the success state.

13.2.3 Important notes

Export button	The export button is enabled only when an export file has been chosen and at least one configuration file is selected.
Status column	Description of the status column: • Before beginning the archiving operation: ***Pending*** Selected for archiving. ***Skipped*** Not selected. ***Missing*** File not available (empty entry in the ini file, or file not present on disk). • During the archiving operation: ***Successfully archived*** No problems encountered while archiving the file. ***Skipped (not selected)*** File has not been processed because it was not selected for archiving. ***Error! Cannot open file*** File has been renamed/moved/deleted or may be locked by another program. ***Error reading file contents*** Problems with the medium where the file resides. ***Error*** Other errors.

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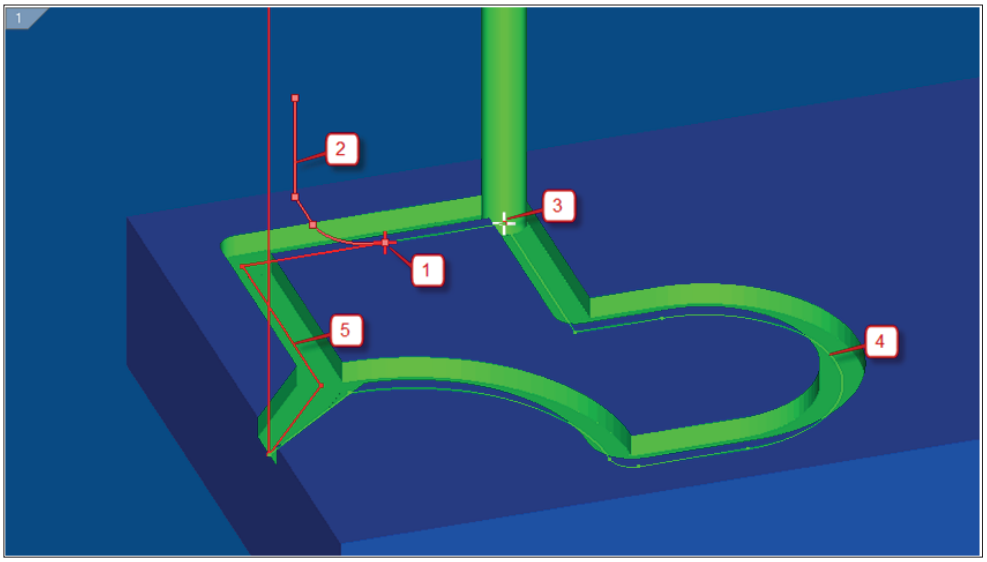
14 Graphical supported “Search mode”

14.1 Description

Search mode allows the user to visually select a restart point for the program execution, which can be used in case the program was aborted for various reasons (tool failure for example).

The user can also customize and visualize the approach path to the selected point.



1	Resume point selected by the user.
2	Approach path.
3	End point of selected line.
4	Offline simulation path.
5	Already processed path.

14.2 Operation

Before starting the Search mode, the user has to perform an offline simulation, using the same CNC configuration as on the real machine: part programs, tools, offsets, etc.

The Search mode window can be started by selecting the *"Search mode"* option from the F2 menu.

14.2.1 Selection of resume point

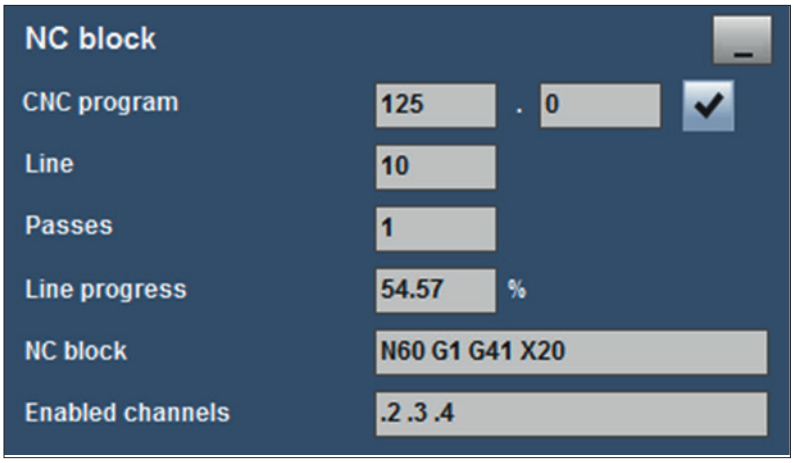
While the Search mode window is open, the user can select a resume point by clicking with the mouse on the simulated path inside a 3D view.

The resume point can also be defined by the path distance to the point. This can be entered as a value inside the *“Position”* field or it can be moved around using the track bar.

Buttons description:

–	Minimize the current group to a single line.
+	Maximize the group.
🔒	Locks the position of resume point. When pressed, the resume point cannot be moved by clicking inside a 3D view.
G0	Show or hide G0 path. When G0 path is hidden, it is not possible to select a resume point on a G0 line.

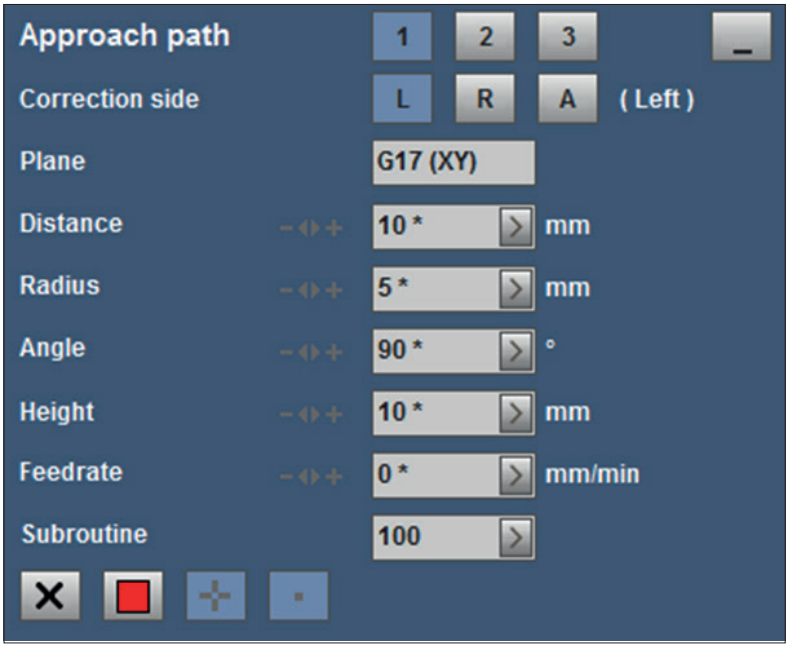
When a resume point is selected, the NC block information is displayed, indicating the program, channel, line number, number of passes (case of subroutine calls) and at what line progress the point is created.



For multi-channel applications the user must select the check box next to the channel number and also indicate a list of additional enabled channels.

14.2.2 Approach path

The approach path parameters define a trajectory that reaches the selected point with a direction tangent to the programmed path.

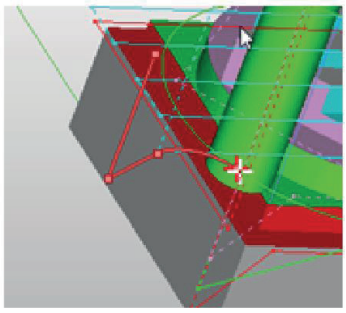
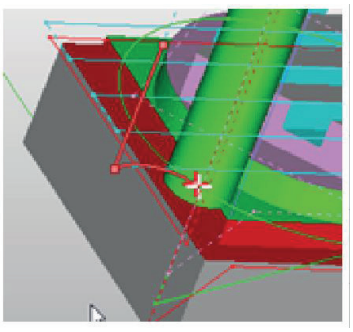
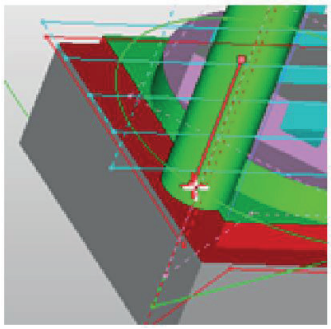


User interface description:

Approach path	The user can select one of three predefined approach paths: The approach path parameters for all three presets are saved to the application "*ini file*".
Correction side	The approach side can be set to Left, Right or Automatic: When automatic side is selected, the actual side will be determined by the G41 / G42 function which is active at the selected path point.
Plane	Shows the interpolation plane that is active at the selected point.
Distance	Defines the length of the approach point.
Radius	Defines the arc radius for the tangential approach.
Angle	Angle of approach path relative to path direction.
Height	Height of approach path.
Feedrate	Optional parameter to define the feed rate of approach path. If the value is set to 0, no feed rate value is transmitted to the NCK. In this case the approach path will have the same feed rate as the programmed path.
Subroutine	Optional parameter to define a custom subroutine with optional channel number to be executed before starting the approach path (a drilling cycle for example).
	Reset the current approach path to default parameters.

	Select color for approach path display.
	Mark the selected point with cross sigh.
	Display the approach path transition points.

Definition of possible approach contours (red) to TCP trajectory

Linear + circular tangential approach	Circular tangential approach only	Direct vertical TCP approach
		

14.2.3 Send Search command to NCK

The selected resume point and the approach path parameters will generate the Search command, which is visible on the bottom section of Search mode window:

```
:10 .0 H125 ;1 EP54 G3 X-5 Y5 EX-5 EY10 I0 J5 EO100 ER10 / .2 .3 .4
```

Inside the search command, the approach path points are described in a local coordinate system where the abscissa is the path direction and the ordinate is orthogonal to the path direction.

If needed, the command can be manually edited before sending to NCK.

When the [Validate] button is pressed, the Online simulation is automatically started and the Search command is transferred to the NCK.

The “F” button enable fast mode search, to reduce computation time.

14.2.4 NCK behavior with graphical SEARCH mode

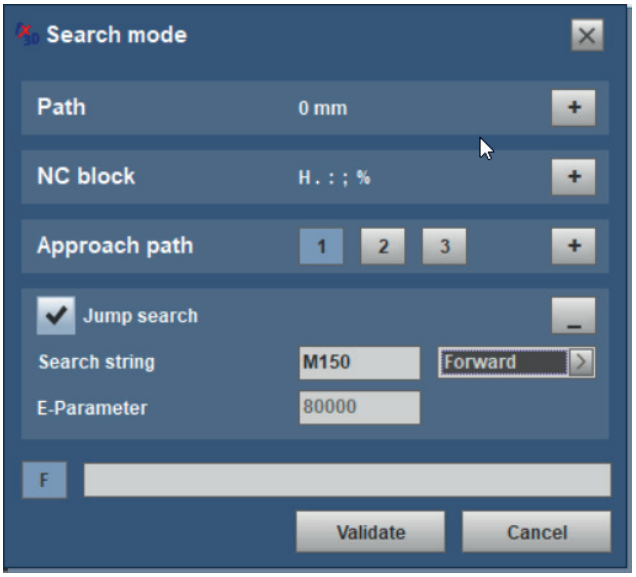
To describe the interaction with NCK using the graphical Search Mode please follow the written procedure below.

- SEARCH sub mode is mandatory.
- Define re-engage position with approach contour.
- Validate / transmit SEARCH mode command string for NCK.
- NC Start:
  - ➢ Internal block search
  - ➢ Calculation of resume position
  - ➢ Preparation of axes movements to begin of approach contour.

- Manually execution of prepared axes movements.
- Change sub mode to SINGLE or AUTO .
- NC Start: continuous execution of subroutine, approach contour and connection with TCP trajectory.
- Proceed part program execution.

Graphical Search mode is only released for Flexium+ system environment.

14.2.5 Jump search function



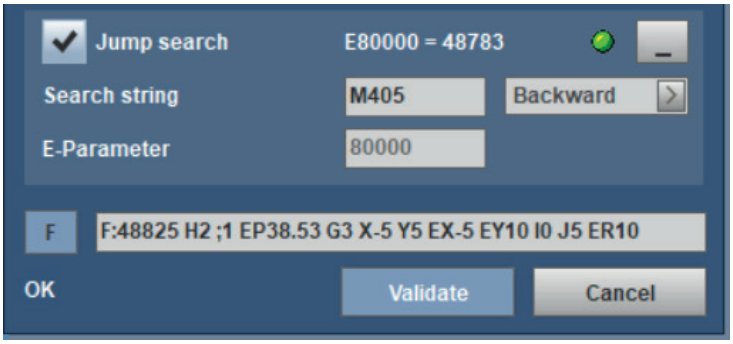
Jump Search function is an extension of Graphical Search: after simulation of a part program and selection of a point along the drawn path, is possible to search forward or backward a string.

The system will give as feedback the number of the block N, and write it on a E-parameter when the Validate button is pressed.

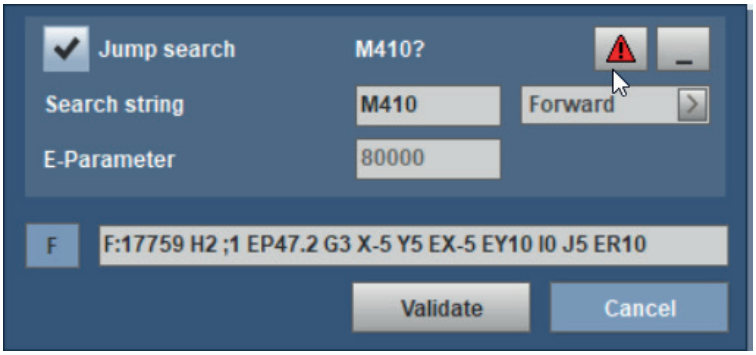
Attention! Flexium 3D need to be able to connect to NCK, to provide the data exchange function. Check the connection!

Enabling the function	The function need to be enable in the menu F2-Option \ Setup, Data Exchange function under Simulation Tab.
Search string	String to search inside part program
Forward \ Backward	Direction of search function from the selected point along part program
E-parameter	E-parameter index for result of the search. Usable E-parameters: • from E80000 to E80199 • from E81000 to E81199 • from E82000 to E82999

If the procedure is successfully completed, a green icon will be display next to Jump search check box; the E parameter value read out from NCK is visualized.



If the procedure is not completed correctly, a red alarm icon will be dispay; click on the icon will provide the type of alarm.



14.2.6 Current NCK limitations using Search-Mode

The graphical Search Mode can be used for NCK configuration in:

- NC mono-channel.
- NC multi-channel for one NC channel among all (the other NC channels do not execute a search of block but some of them can be enable to start their program at beginning when graphical search mode is completed).

It is not available for NC channels in independent mode. It is not available for auxiliary channels.

The standard search mode and the graphical search mode are exclusive: only one type can be executed at a time.

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Summary

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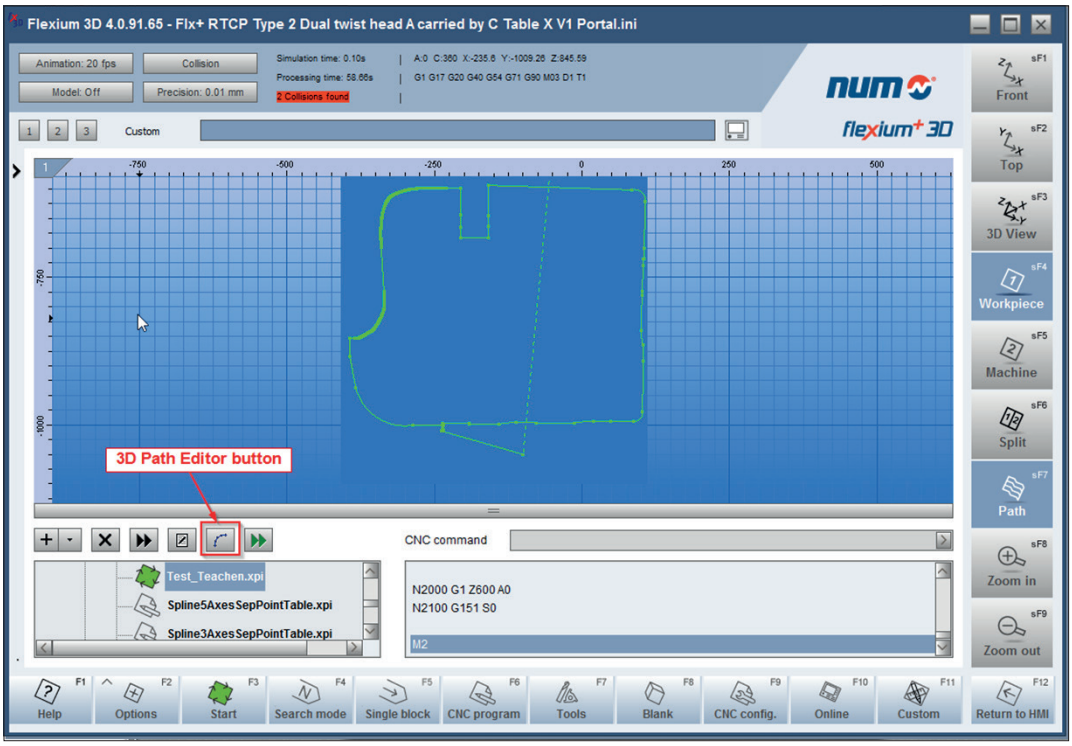
15 3D Path Editor

15.1 Main purpose and target

3D Path Editor allow the operator to visualize a present and selected part program in different plane views for modifying or reworking the trajectory, with the target to see the modifications directly in the 3D Path View.

Target of using 3D Path editor is to integrate geometrical corrections of the path as well tool orientation adjustments depending measurement results or optical check without re-generating the whole part program.

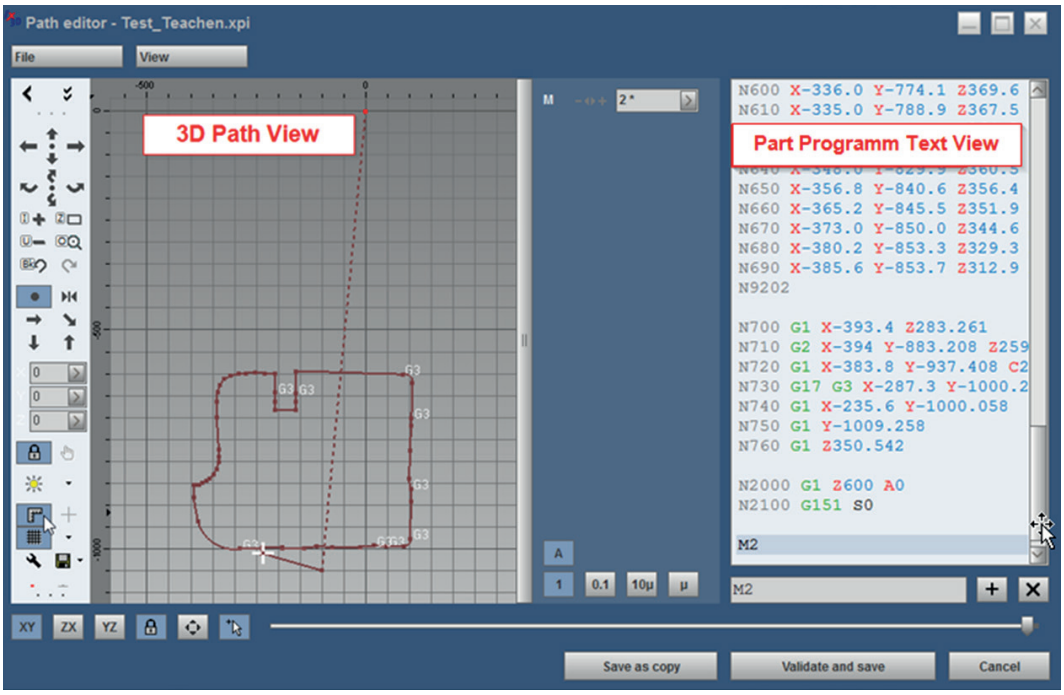
The operator can start the 3D Path Editor from Flexium 3D main window by pressing "*3D Path Editor*" button with a selected part program (below shorten as PP).



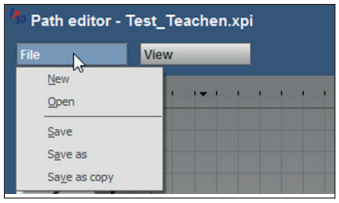
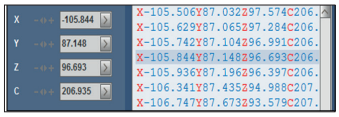
15.2 Start Window in 3D Path Editor

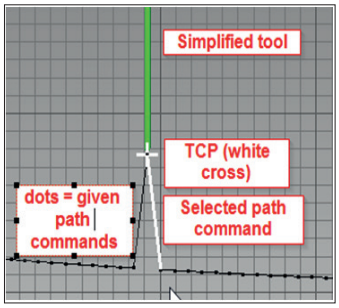
3D path editor main screen is divided into two main windows:

- 3D Path View: Visualization of complete trajectory of the part program in default plane view (G17). White cross pointed out current PP line.
- Part Program Text View: Text window of selected part program with syntax high lightning, dynamic edit command box, command line editor and visualization of current PP line.



15.3 Standard Operational Controls

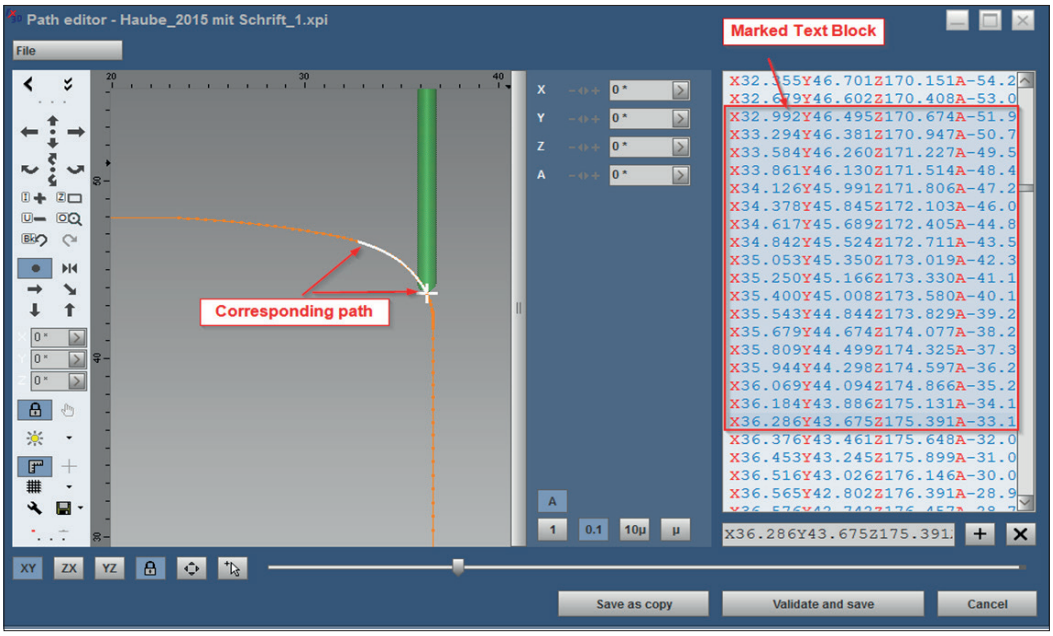
Control	Description
	**File operations**: • Display of currently loaded Part Program. • Complete File handling options.
	View options: • Color definition for selected G&M functions (e.g. cycles execution etc.). • Configuration of visible G&M functions in 3D path view window. • Operational buttons for Add, Delete and Activate.
	• Select PP line (dark grey) via mouse click or curser operation. • Dynamic edit command box (position slider and numerical edit).
	• +/- Slider operation. • Original value is marked with * to return to start value.
 	• Intermediate points: Scaling Pre-section for position modification with slider. • Value increment: graphical modification automatic selected (depending On Zoom setting).
	• Single Edit command line. • Add or delete axes positions. • Add or delete lines.
	• View directions (G17/18 /19). • Default is G17. • Current TCP position will be kept at plane change.
	• Prevent 3D rotation with mouse. • Fit 3D Path view to window. • Enable to select point on 3D path with the mouse (Blue=released/grey=blocked).

	**3D Path view color explanation** Tool orientation (green) will be visualized as simplified tool with fix diameter (not changeable in 3D Path View) at current TCP position (white cross) and current movement command (white path element).
	• Part program progress bar (with direct reference to text and path view).
	• Direct file handling controls “Save as Copy” with automatic file name numbering.

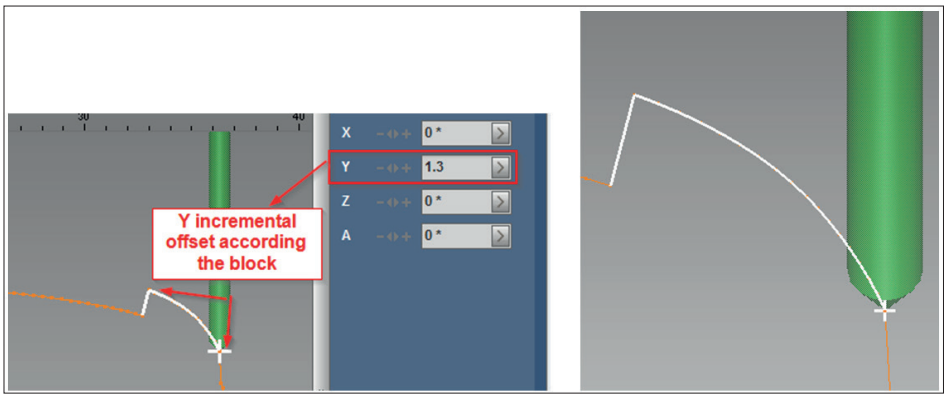
15.4 Complex Operational Control

As complex operational control *"Multi Line Edit"* has to be mentioned.
With mouse or keyboard operation you select the considered block (red frame) by holding Shift key while selecting program lines.

The complete block is visualized as *"white"* marked trajectory as corresponding path.



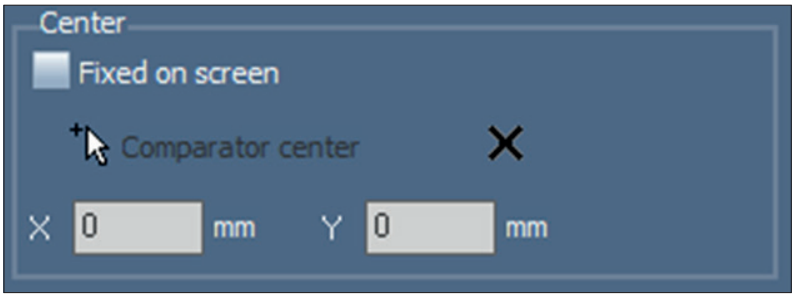
Axis dependent, incremental offset for complete marked block with simultaneous path correction. Offsets are set to 0 for incremental move.



15.5 Specific Operational Controls

This is a list of specific controls and their effect:

- Offset table in 3D Path view NOT used.
- Graphical supported point search or select a PP line in text view.
- Graphical selecting a path condition (When clicking on a path line between two programmed points, the start point or the end point will be selected, whichever is closer to the selected point).
- 3D Edit view: TCP is end point of selected line.
- Double click on plane selection: complete PP fit view.
- Single click on plane selection: plane view with steady TCP.
- Take care that *"Fixed on screen"* checkbox is deactivated to have direct reference to program frame in 3D path editor (reachable via view options).

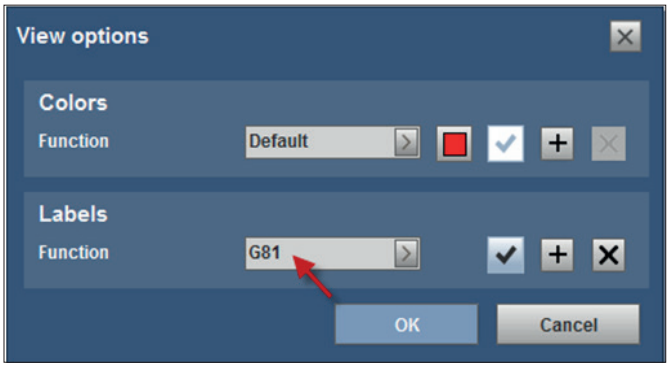


- ED+/- : Rotation (No direct 3D path reaction)
- EB+/- : Fillet and chamfer (No direct 3D path reaction).

15.6 Current limitations

- **Currently NOT supported**
  - Cycle definition & execution:
    - G81-G89: drilling, tapping and boring cycles
    - G31: Thread chasing cycle
    - G45: Pocket cycle
    - G31-G38: Turning cycles
    - G63-G66: Roughing and Grooving cycles.

- The cycles can be visualized as a label marker along the 3D path, by adding the specific G function to the View -> View Options -> Labels list:



- **NOT supported in simplified Parser**
  - Tool radius compensation G40/G41/G42 (visualize original path).
  - Tool length compensation is not supported in 3D path editor

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Summary

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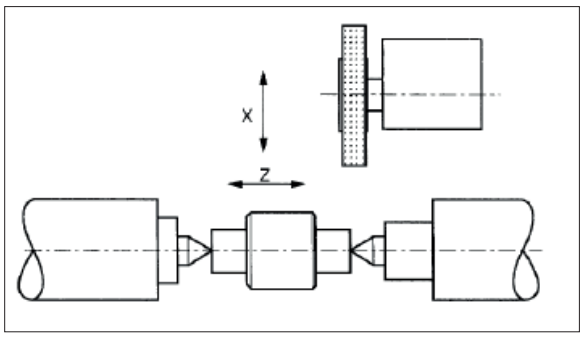
16 Cylindrical and Non-circular Grinding Simulation

16.1 General aspects

16.1.1 General description of Cylindrical grinding applications

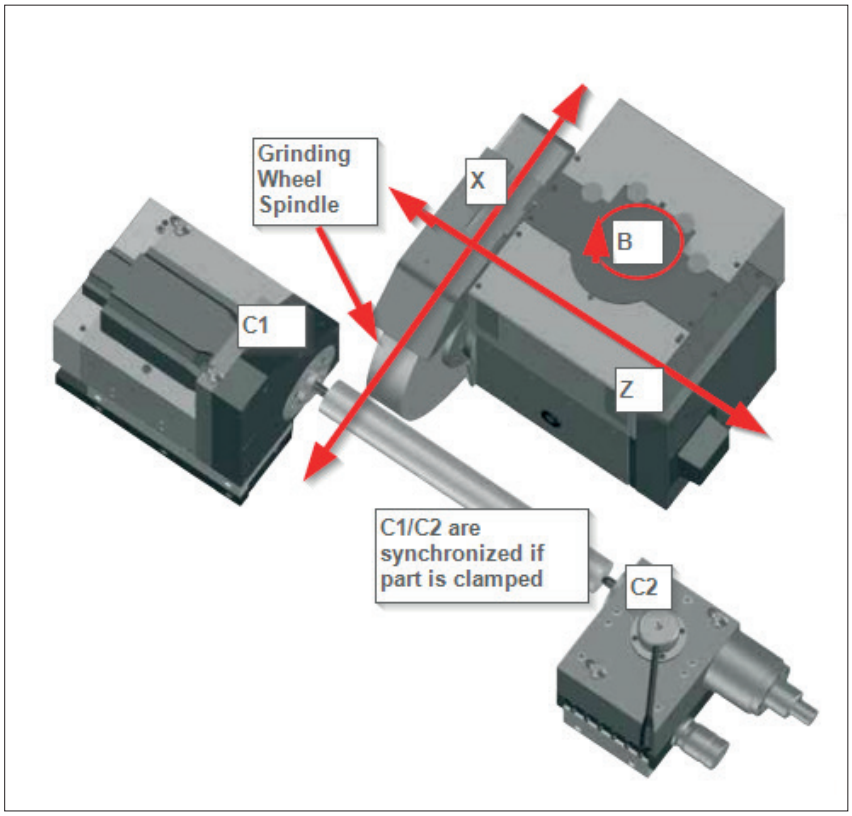
Physical axes:
Linear:

- X (Moves Spindle Work head towards part)
- Z ((Moves Spindle Work head axial along the part)
- Schematic:



Rotary:

- C (C1/C2 synchronized or single C with tailstock
- B (Rotate Spindle Work head)

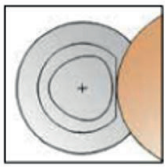


Interpolation types

C-axis The option of interpolating the X- and C axes makes it possible to use the cylindrical grinding machine also for Non-circular shapes such as polygons, free contours and eccentric forms. The rotary encoder with a resolution of 0.0001° is installed directly on the work head spindle. The non-circular movement is superimposed on the grinding movements so that the grinding machine can use all the grinding cycles on Non-circular grinding too, including the handwheel release for the X-axis.

X/C Interpolation

- Non- circular workpieces
- Thread grinding
- Jig grinding
- Groove grinding



X/Z Interpolation

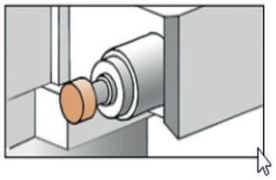
- Tapper grinding
- Profile grinding
- Dressing

X/Z/B Interpolation

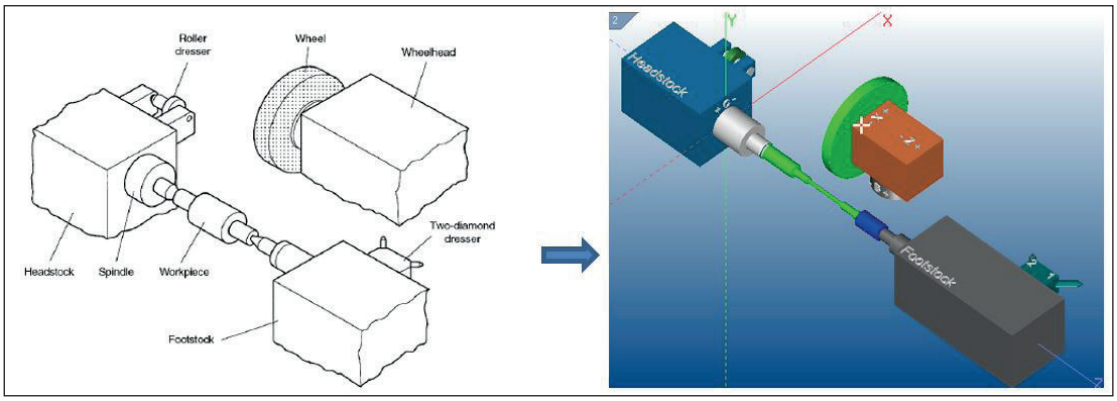
- Contour B+
- Profile grinding with controlled grinding wheel

Internal grinding attachment

- High-frequency internal grinding spindle



16.1.2 Genius Kinematics for a Circular (OD/ID)- and NON-circular grinding machine.

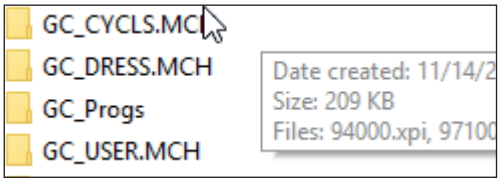


16.2 Generating Grinding process

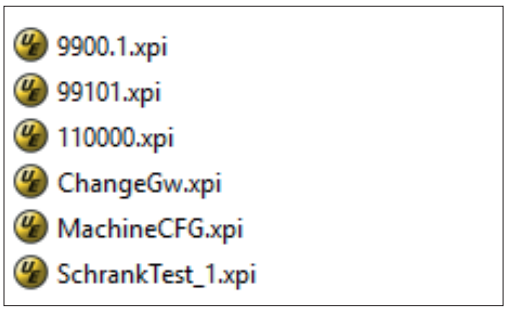
Used PPs , Cycles, and other macro Files

With FXCAM Technology HMI NUMgrind the user creates the part program with adjusted cycles, sequence of operations, Tools and profile.

General:



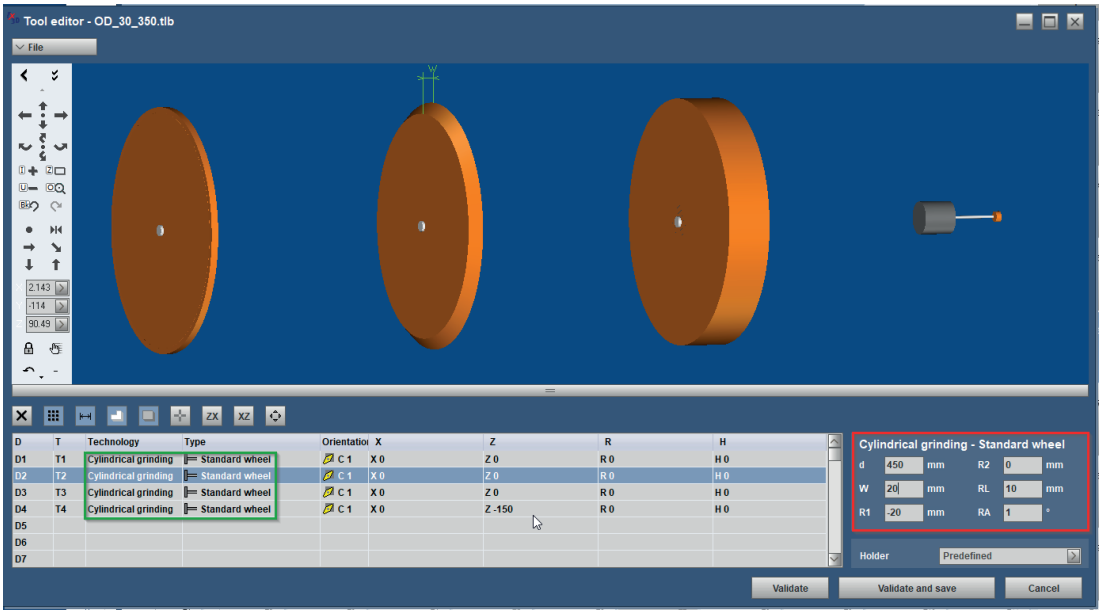
Processing PPs from Folder GC-Progs:



16.3 Grinding simulation in Flexium 3D

In conjunction with NUMgrind, Flexium 3D (NUM's graphical simulation software) provides the operator with another very useful tool to further enhance productivity. Once a part program is generated through NUMgrind technology, the operator can simulate the grinding program through Flexium 3D based on the customized machine kinematic. The software can be configured in a variety of different ways to match many standard cylindrical grinding machines. It can be used directly at the machine or as a standalone program for grinding process verification. This software gives the operator more power to ensure that the generated programs match their desired outcome before grinding process is executed on the machine.

To achieve a correct grinding simulation the wheel types and data for a Cylindrical Grinding application (standard, angular and special with green color) can be imported from wheel files in Flexium 3D. A set of additional tool parameter describe the pattern of the wheel itself like radius, peripheral face and side (red color). The picture below shows a typical set of wheels to process OD/ID grinding applications of shafts and/or tubes



With the new tool types following wheel forms can be described:

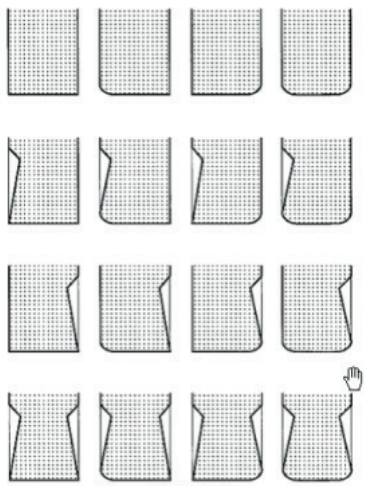


Figure 2: **Wheel Forms**

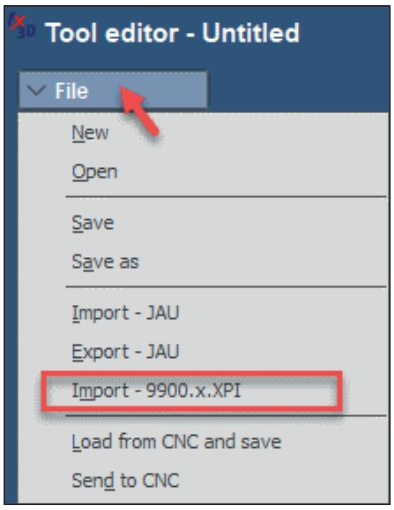
A wheel form is defined with:

- Wheel Width (W)
- Radius (+) / Chamfer (-) Left (R1)
- Radius (+) / Chamfer (-) Right (R2)
- Relief Length (RL)
- Relief Angle (RA)

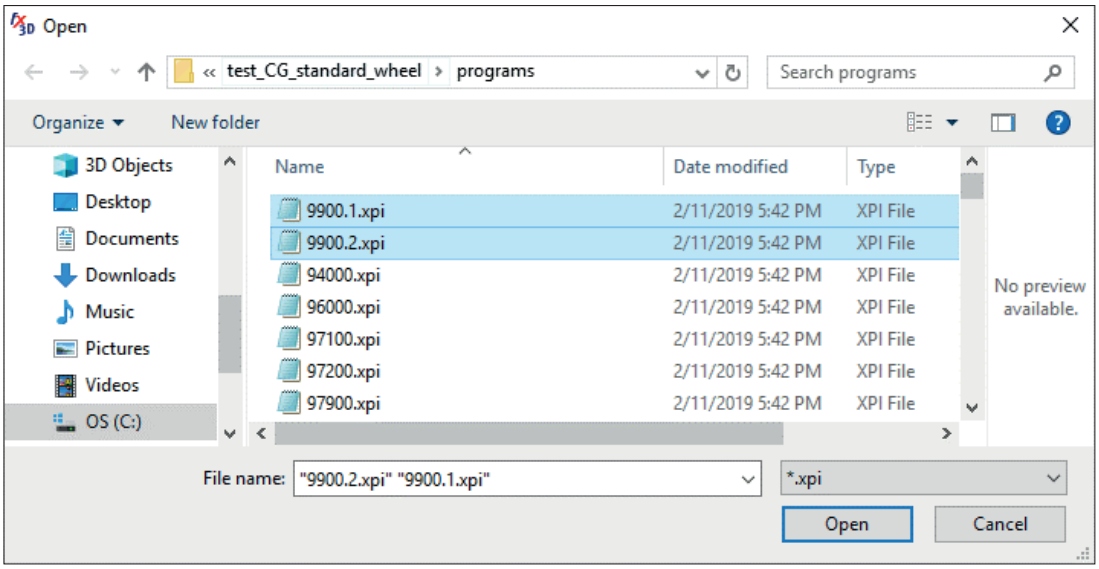
Abbreviations in brackets are referencing to the wheel table above

The defined wheel geometry data are placed in 9900.n.xpi (n=1,2,..9)

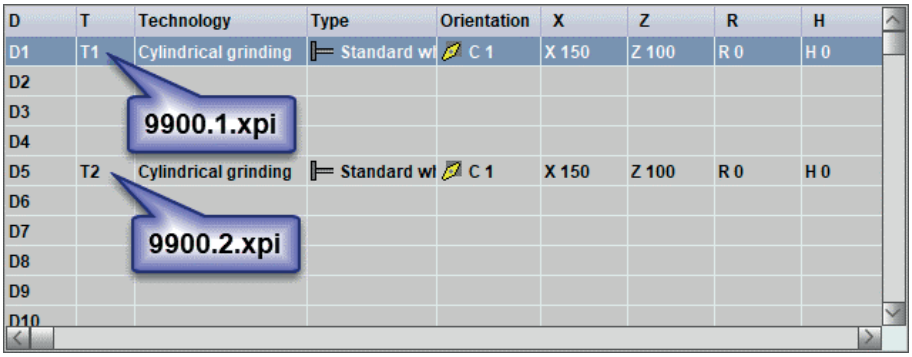
To import such a a wheel geometry This function is added to the File menu inside Tool editor window:



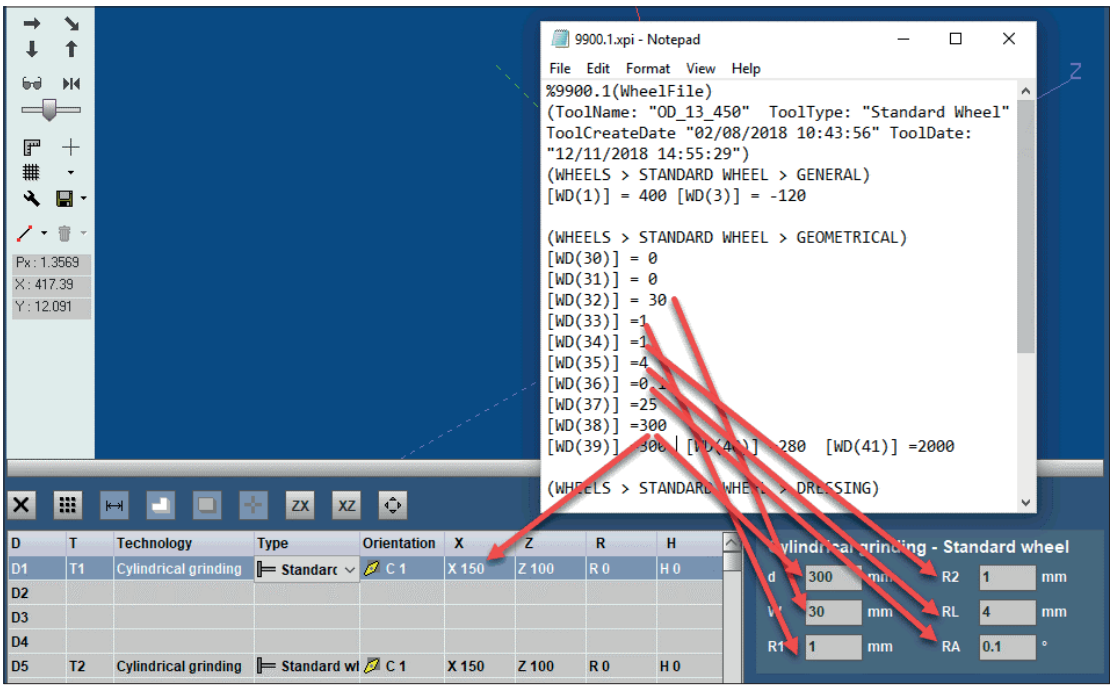
When selecting this menu option a open file dialog will be shown. It is possible to select multiple files for import:



For each XPI file a new tool (Standard wheel type) will be created:



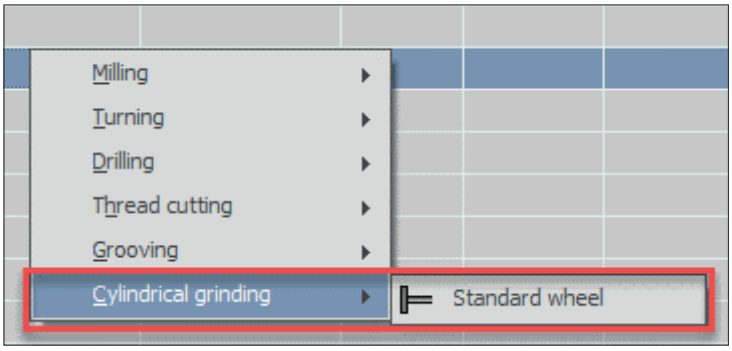
Some parameters are hard coded (D1/5/9..., T1/2/3..., C1, Z100) and some are coming from the XPI file:



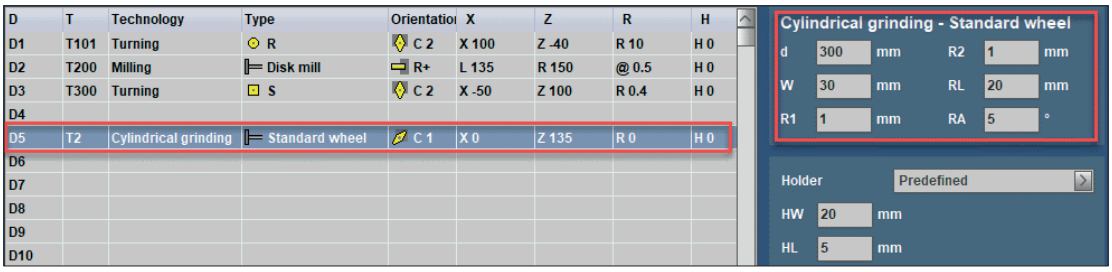
Remark: X will be automatically calculated as Diameter ([WD(38)]) / 2.

“Standard wheel” tool type for cylindrical grinding.

The new tool type can be created here:



Complete Standard tool data

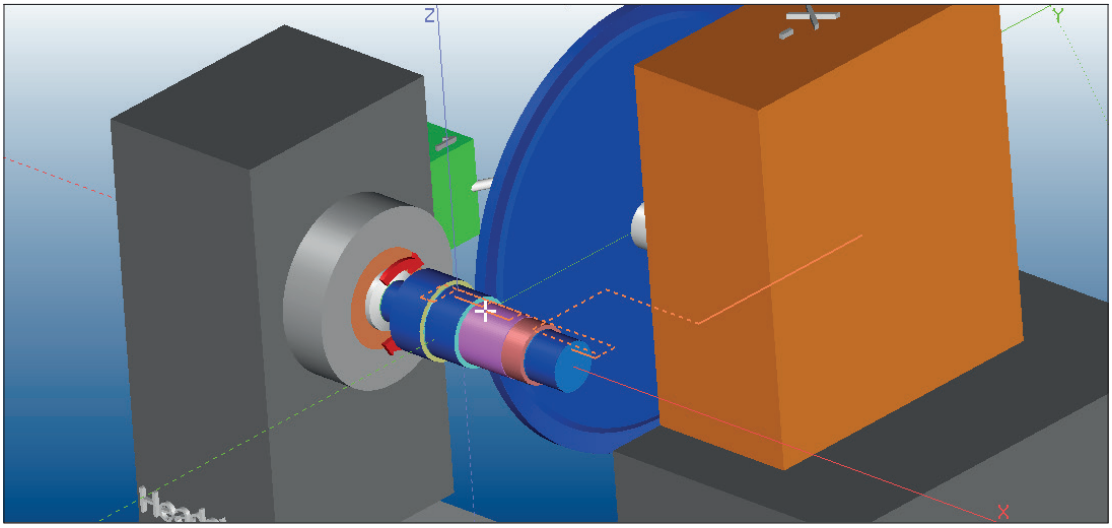


Of course wheel changes are supported from NUMgrind and in both simulation modes. A wheel package has to be configured as single wheels in the tool editor

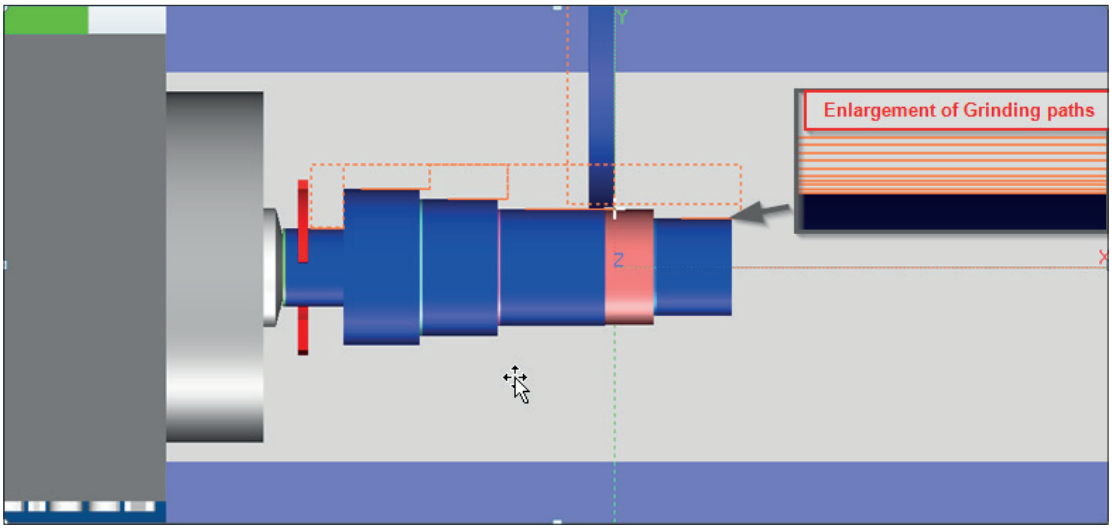
16.4 Offline Grinding Simulation

Offline grinding simulation can be used as an advanced process to verify and evaluate the grinding program either direct on the machine during part processing of another part or as standalone in production planning department. This pre-evaluation of the grinding project prevents downtime of the machine in case of programming errors

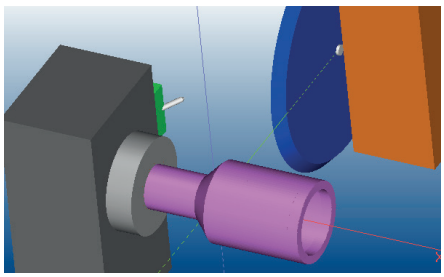
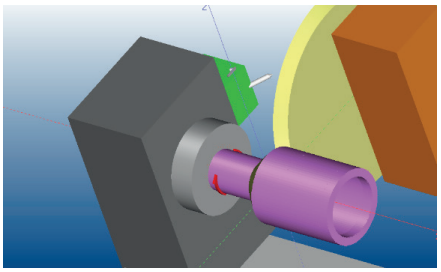
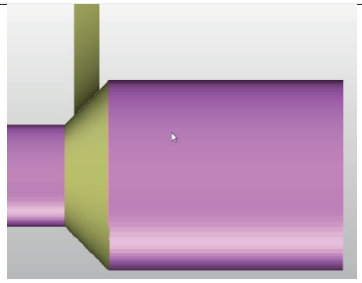
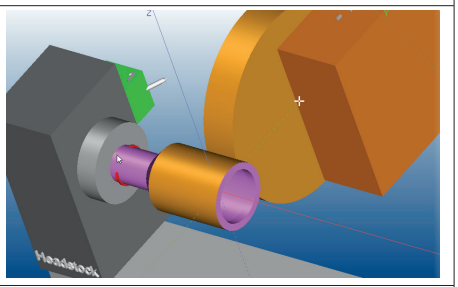
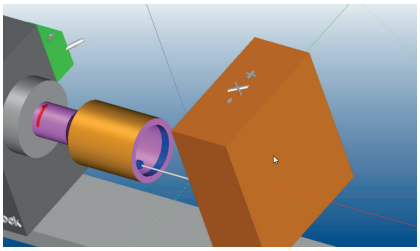
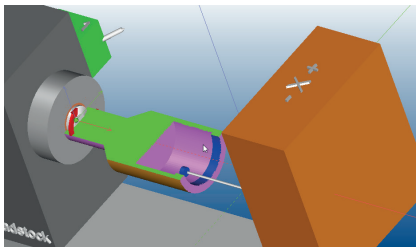
Following 3D picture shows the grinding process of running part Wheel spindle. Blue color associates the already machined area of the shaft.



This view in Z-X plane enables an evaluation of the transitions on the shaft and the tracking of the grinding paths in detail. Features for measuring diameters and shaft areas round off the simulation as an upstream process.

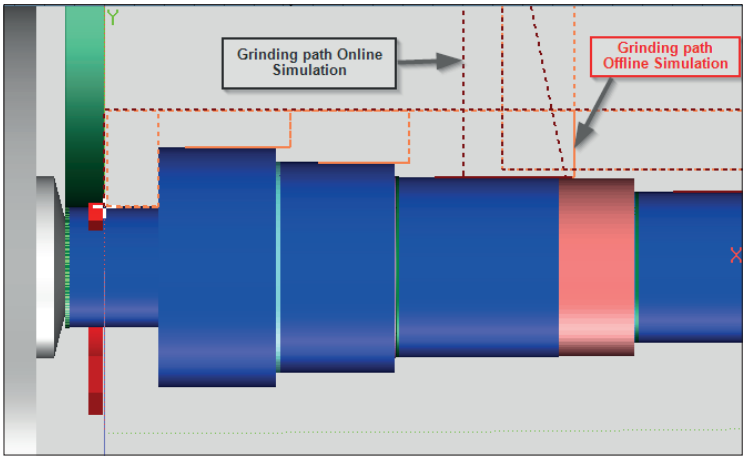


Next pictures show the inner and outer diameter processing of a more complex blank, with the use of three different wheels to evaluate generated part program with used cycles and specific wheel definitions.

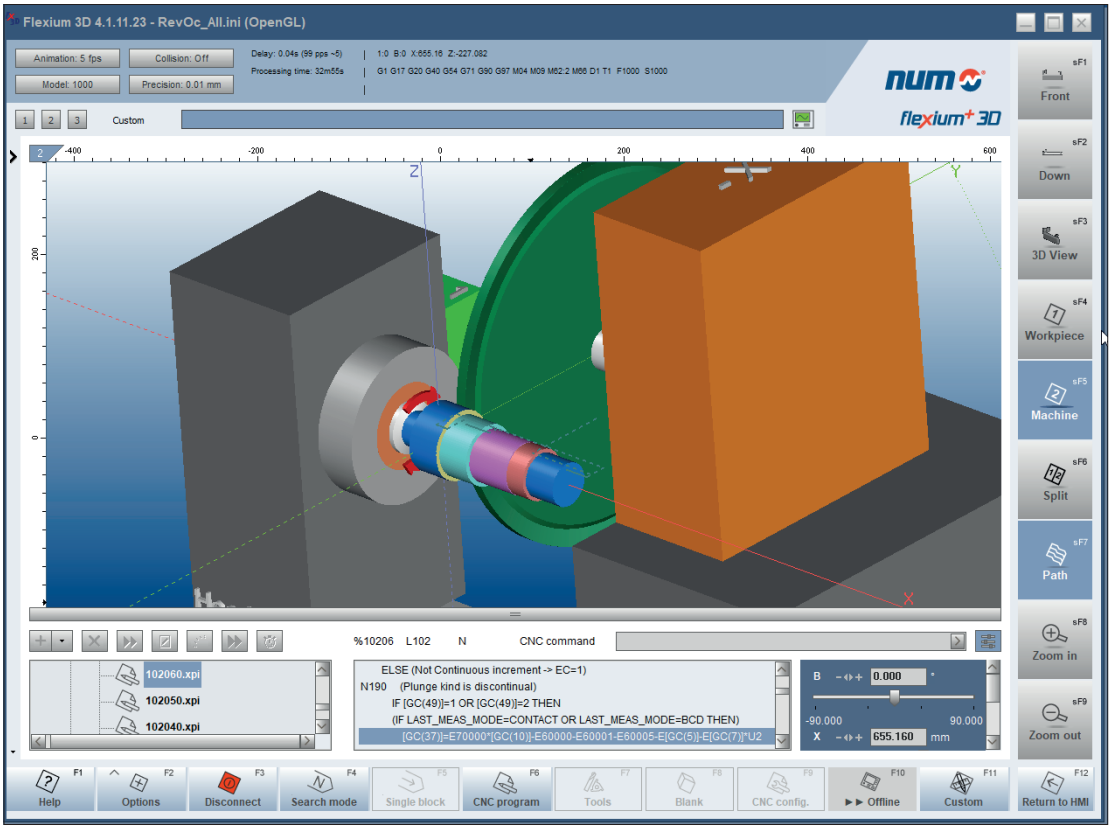
Blank formed as shaft and tube	
Processing start of OD conical traverse cycle G210 at inclined shaft part. (Machine view)	
Processing start of OD conical traverse cycle G210 at inclined shaft part. (Part view)	
Wheel change and OD traverse cycle G206 at surface of the tube part.	
Wheel change and ID traverse cycle G207 at inside of the tube part.	
Visualization of ID traverse cycle G207 at inside of the tube part. Section of the work piece illustrates the real material removing during grinding process.	

16.5 Online Grinding Simulation

It is also possible to directly visualize the axis positions in real time in the so-called online simulation window simultaneously during the grinding process. A superposition of grinding tracks to the upstream evaluation process is also possible, as the grinding trajectories are displayed in different colors.



Complete view of online simulation with tracking NC block display, material removal.



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Summary

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A.1 Guidelines for machine's kinematic configuration

A1.1 Introduction

This appendix provide a general description of the Flexium3D projects that reproduce the 24 5-axes kinematics supported by Flexium+ System.

In the following pages will be describe how to import and use the sample projects.

A1.2 Project import

All project attached, have been developed with version 4.2.00.00; the export file (cfx extension) contain:

Machine parameters file (xpa extension)

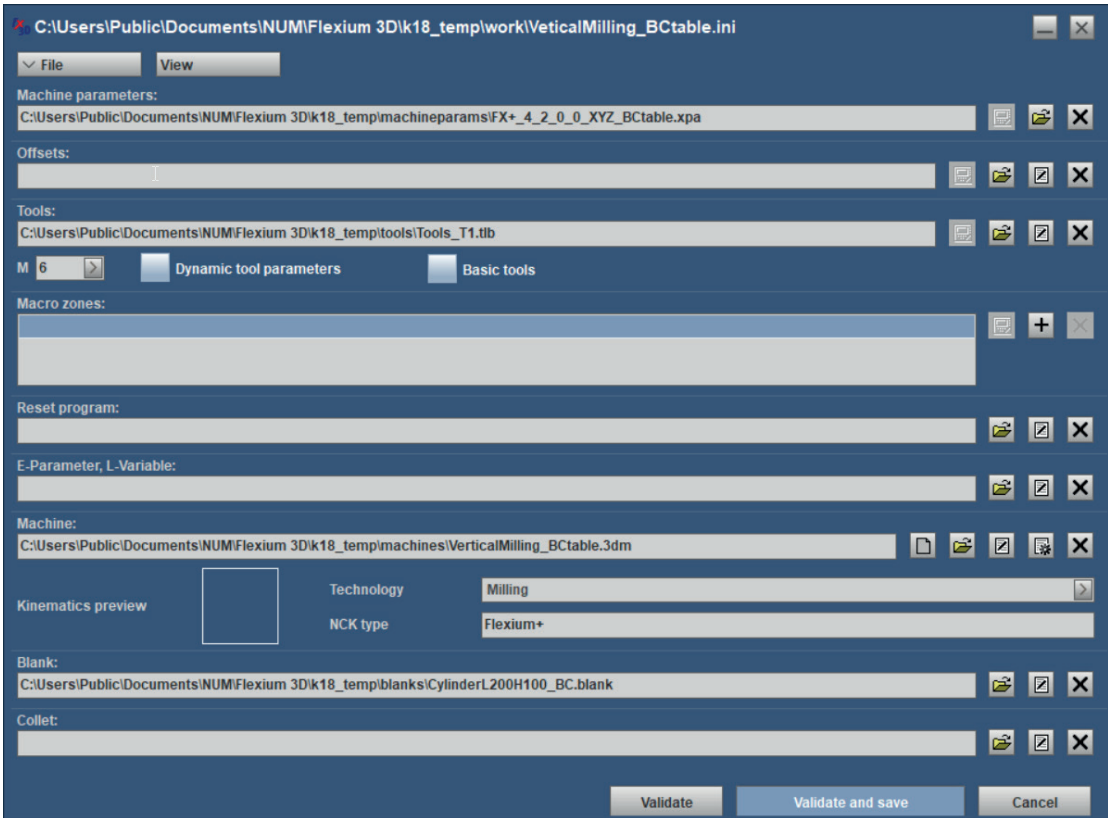
Tools file, with a single milling tool (tlb extension, create with tool editor)

Machine model file (3dm extension, create with machine editor)

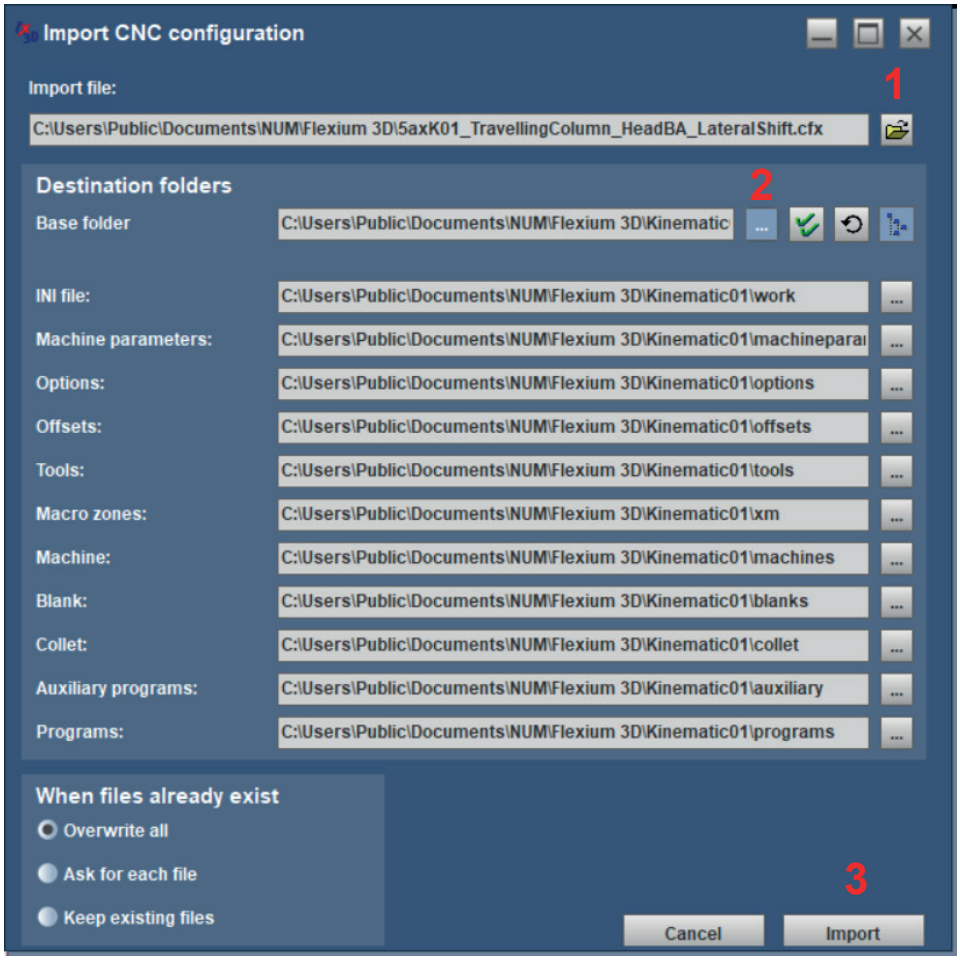
Blank file (blank extension, create with blank editor)

RTCP and iPLANE macro program

Sample program to test RTCP function.



Follow the standard procedure to import a cfx file (see manual M00029-EN chapter 13); for an easy use of the system is recommended to create a dedicated folder (eg: C:\Users\Public\Documents\NUM\Flexium 3D\Kinematic01). The main folder will contain sub-folders for each kind of files.



A1.3 Machine model file

The machine model is created using the Machine editor of Flexium 3D; refers to manual M00029-EN for more details.

A1.3.1 Construction Tab: main structure

The structure of the machine has been created to easy understand the kinematic tree.

Under the Base object (reference for the machine origin system) you will find:

- *Base_to_TCP (Q1)* kinematic chain;
- *Base_to_Blank* kinematic chain (or only a BLANK element, if the work piece is fixed on a table, and no axis move it);
- A *Machine Elements* branch that contain all the object of the machine not connected to an axis:

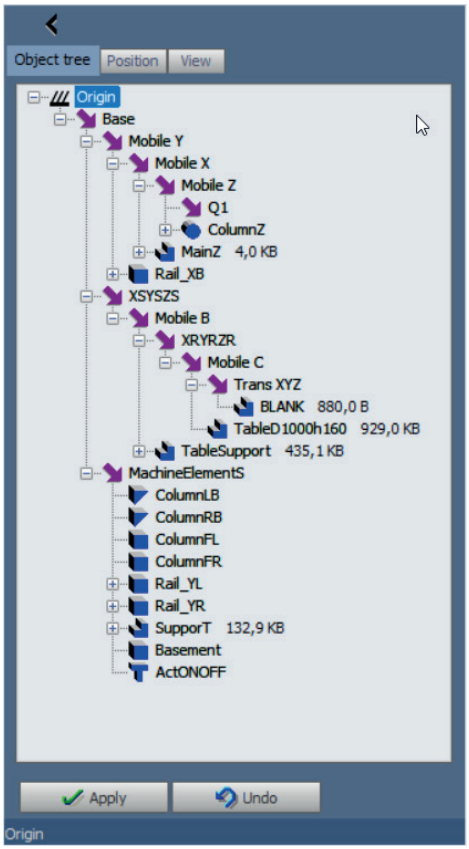


Figure 1: kinematic n°18

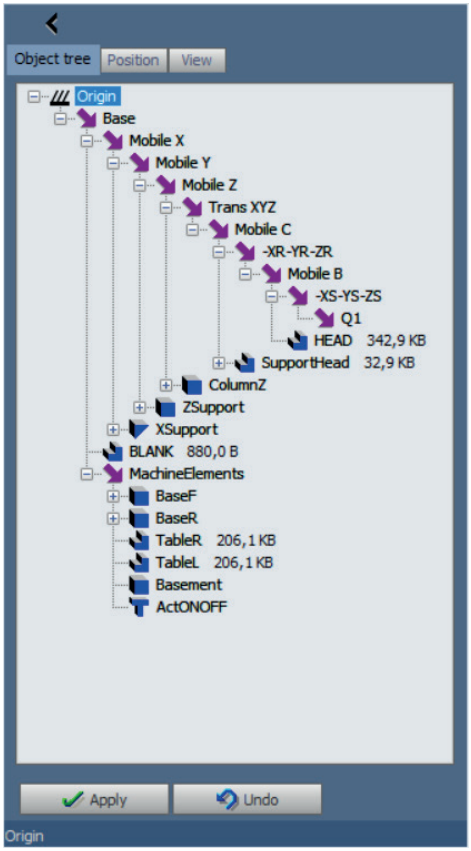
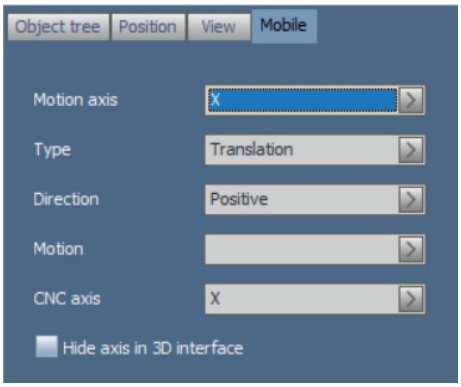
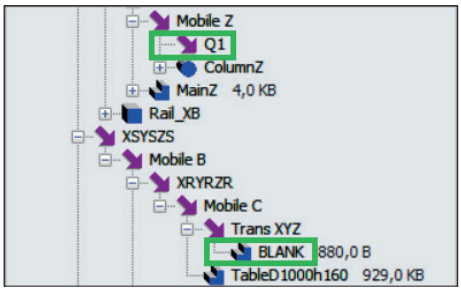
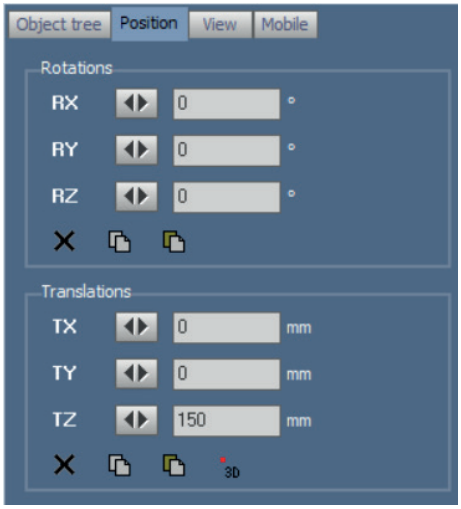


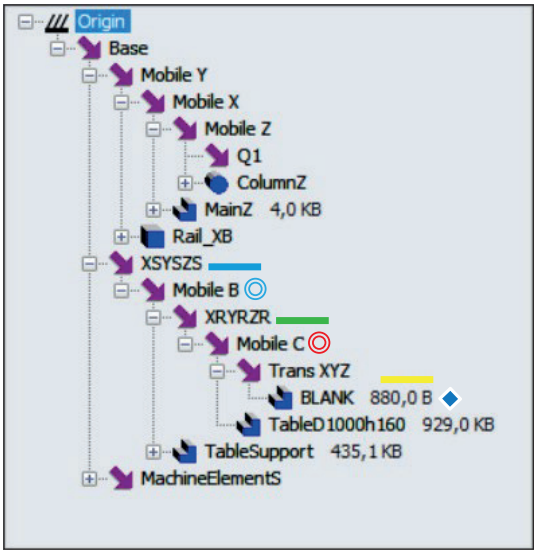
Figure 2: kinematic n°4

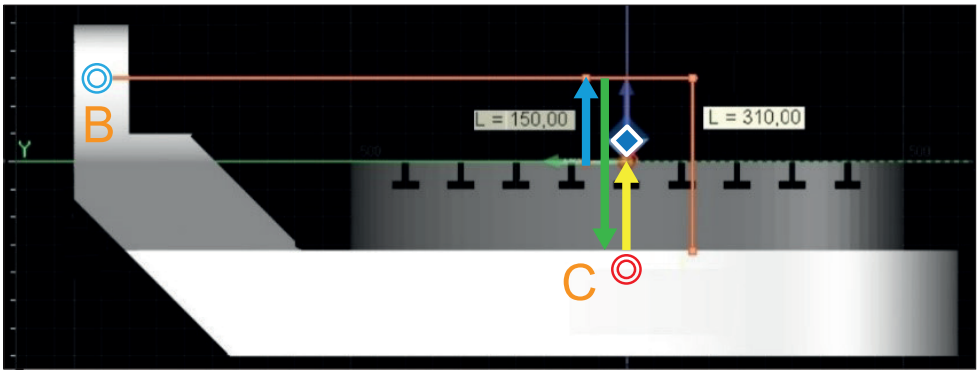
To be aligned with Flexium Tools software, (Flexiump NCK device, Transformation Tab 1.6, Kinematics setup page) some standard names have been used.

	CNC Axis (Mobile Tab compiled, Position Tab empty): - **Mobile X, Y, Z, A, B, C** identify a mobile element connect to the corresponding CNC axis.
	Mandatory Elements (Mobile and Position Tab empty): - **Q1**: mobile element that identify the TCP - **BLANK**: object (or Mobile element) to identify the attach position of the blank (work piece) that will be used
	Offset elements (Mobile Tab empty, Position Tab compiled): - **XSYSZS**: starting from the base and looking toward the blank, offset from machine origin to first rotary\swiveling axis (same description of RTCP and iPLANE macro in FlexiumTools); - **XRYRZR**: starting from the base and looking toward the blank, offset from the first to the second rotary\swiveling axis (same description of RTCP and iPLANE macro in FlexiumTools); - **-XR-YR-ZR**: starting from the machine and looking toward the tcp, offset from the first to the second rotary\swiveling axis. Inside FlexiumTools software the offsets must be insert with inverted sign, because the RTCP and iPLANE macro need offset from second to first swivel\rotary axis; - **-XS-YS-ZS**: starting from the machine and looking toward the tcp, offset from the second rotary\swiveling axis to TCP element (Q1). Inside FlexiumTools software the offsets must be insert with inverted sign, because the RTCP and iPLANE macro need offset from tcp to second swivel\rotary axis;

	- **TransXYZ**: additional translation offset to complete the two chains. To describe correctly Base to TCP chain could be necessary add an offset from Machine origin to the first rotary\swivel axis. To describe correctly Base to Blank chain could be necessary add an offset from the last rotary\swivel axis to the blank element. - **Trans A, Trans B and Trans C**: some kinematics (nutating head in particular) need to apply a rotation to the frame. These elements contain respectively the rotation offset around X, Y and Z axis.

Sample 1: tilting turn table kinematic 18



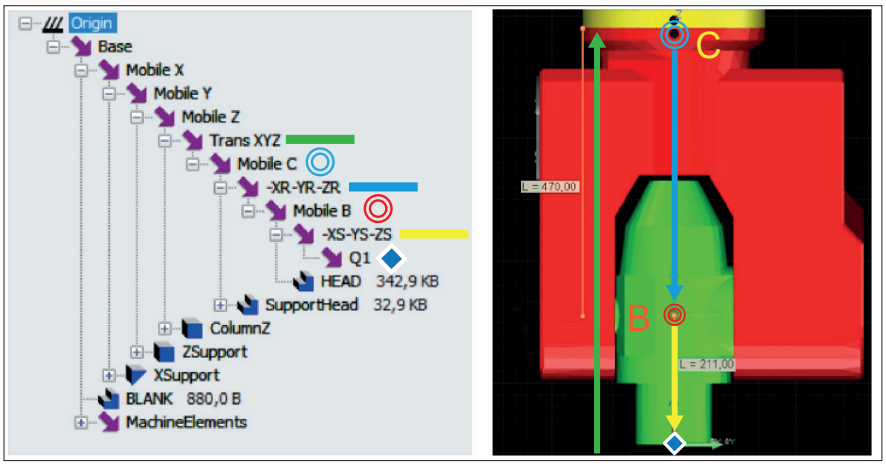


In this project the kinematic is configured to have machine origin at table surface and spindle nose at table surface programming X0 Y0 Z0 B0 C0. Blank is exactly at machine origin (X0 Y0 Z0).

The first kinematic chain is simply define by the three linear axis (in order Mobile Y, X and Z) connected to spindle nose (Q1).

The second chain start with translation from machine origin to first swivel axis (XSYSZS), followed by the corresponding CNC axis (Mobile B). Then continue with translation from swivel axis to rotary axis (XRYRZR) followed by CNC axis (Mobile C). Finally, an additional translation (Trans XYZ) is applied to have the Blank at machine origin.

Sample 2: dual twist head kinematic 4



In this project the kinematic is configured to have machine origin at table surface and spindle nose at table surface programming X0 Y0 Z0 B0 C0. Blank is exactly at machine origin (X0 Y0 Z0).

The first kinematic chain start with the three linear axis (in order Mobile X, Y and Z) at machine origin.

Then is necessary to compensate the distance from machine origin to machine head (with a Trans XYZ) and insert the first rotary axis (Mobile C).

The chain continue with translation from first rotary axis to the second (-XR-YR-ZR) followed by CNC axis (Mobile B).

The last step is adding the offset from the second rotary axis to TCP (-XS-YS-ZS), connecting the spindle nose (Q1).

The second chain is composed by only one element: the blank.

A1.3.2 Construction Tab: rotary axis

When a rotary axis is used, different kind of configuration could be selected in *motion* option:

- *Modulo*: same behavior as *rotary* on parameter file (P1N0)
- *Limited deflection*: same behavior as *rotary limited excursion* on parameter file (P1N1)
- *Shortest*: dedicated setup of Flexium3D (similar to modulo setup)

To reproduce exactly the same movements as real NCK, could be necessary modify some E-parameter; in particular, for rotary (modulo) axes could be selected shortest path (E11010=0) and inhibit movements when same position is programmed (E11009=0). See paragraph 4 in this document for more details or manual M00029EN.

A1.3.3 Collision Tab

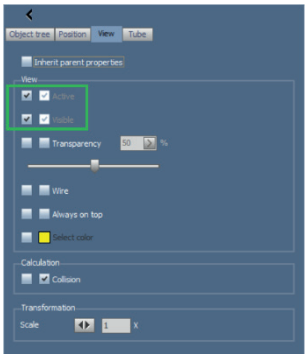
Simple collision between main axis (with child option on YES) that move Q1 and main axis (with child option on YES) that move Blank is setup; if no axis move the blank, some collision check control collision between main axis versus blank element, and main axis versus fixed elements (like table, support table and so on) are setup.

A1.3.4 Miscellaneous Tab

Two M code (M400 and M401) are prepared to hide\show all the structure of the machine except Head and Table elements; this could be useful in machine view to control head and table movements during machining.

To do this, some dedicated Flexium 3D command are used (between brackets, as standard comments).

Furthermore all the children of main branches that need to be hide (like MachineElements or ColumnZ) must have Inherit parent properties check box selected, on Active and Visible property (on View Tab).



Eg: M400

(%ACTIVE OFF 'MachineElements')

(%ACTIVE OFF 'ColumnZ')

(%ACTIVE OFF 'MainZ')

(%ACTIVE OFF 'Rail_XB')

Adapt or erase this two M code if necessary or overlap with existing functions.

A1.3.5 Note Tab

Note tab contain the release history of machine sample and the offset useful to build kinematic macro.

A1.3.6 Reset program, E-parameter L-variable

In OFFLINE mode, to be able to reproduce the real NCK behavior as much as possible in the easiest way, two special part program are supported:

- %11000: (Reset program input field) same characteristics of NCK reset program
- %9999: (E-parameter L-variable input field) part program similar to reset program, but able to read and write E parameters and L variables. Useful to reproduce the initial state of the machine.

The two part programs are execute (during OFFLINE simulation):

- At the beginning of the part program
- With a define sequences: first %11000 and then %9999

Example 1: setup shortest way on rotary axes and no movements if same position programmed

%11000()

E11010=0

E11009=0

Example 2: modify axis travel limits of @0 axis

%9999()

E97600=-450000 (-450mm if internal unit is micron)

E97700=500000 (500mm if internal unit is micron)

A1.4 Previous version (before 4.2.00.00)

To be able to use sample projects with old version of Flexium 3D (eg: 4.1.30.30) some changes are needed:

- Use a parameter file compatible with used version (eg: 4.1.30.30)
- 3dm machine editor file created with version 4.2.00.00 is not compatible back with old version (mainly to changes into cylinder and tube elements); use the dedicated function *Save compatible version* or remove cylinder and tube elements from construction tab.

On https://ftp.num.com/ sample projects version 4.1.30.30 are also available.

(section Home - Customer_Support - Internal_Common - Flexium3D - Kinematic Samples).

A1.5 Know issues and resolutions

September 2023 :

Version 4.2.10.00 and before: in the machine editor, a Mobile object connect to B axis need to be configured in Mobile Tab in a different way respect other axis: direction negative if insert in Base_to_TCP chain, direction positive if insert in Base_to_Blank chain. **This behavior is correct with revision 5.0.0.0.**

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Summary

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B.1 Host Command

B1.1 List of command

Command	Description	Examples / Remarks
Tn M6 Qn	Load a tool on a spindle	T1 M6 Q2
Tn M6 QWn	Load a tool on a wheel changer position	T1 M6 QW1
(%ACTIVE ON/OFF 'name')	Set one object to active or inactive in the machine model	(%ACTIVE OFF 'steady_rest') (%ACTIVE ON 'tool_probe')
(%ANIMATION fFPS/f%/fP/fX/fT/0)	Overrides user defined animation speed during simulation. The user animation settings are restored when the simulation has completed. If animation speed is programmed, the animation is automatically turned on. (%ANIMATION 0) will turn off the animation.	(%ANIMATION 10FPS) sets the animation in frames/second. Recommended values: 1, 5, 10, 20. (%ANIMATION 100%) sets the animation to all frames or a fraction of frames. It is allowed to use 'P' char instead of '%'. For example 10P is equivalent to 10%. (%ANIMATION 1X) sets the animation to real-time or accelerated real-time. (%ANIMATION 10T) sets the animation to percentage of time.
(%CLAMP 'name1' ON 'name2' [/KEEP_RELATIVE_POSITION]) (%CLAMP 'name' OFF [/KEEP_RELATIVE_POSITION])	Clamp on: links name1 objet as child of name2 object; Keep relative position: will keep the relative position of the child object relative to the parent; If this is not set, the absolute position will be preserved. Clamp off: links an object to the initial parent object	(%CLAMP 'Q1' ON 'Axis X' /KEEP_RELATIVE_POSITION) (%CLAMP 'Q1' OFF)
(%CUTCOLOR R G B)	Set the current cutter color	(%CUTCOLOR 255 165 0)
(%MOVE/MOVE-R/MOVE-INC/ MOVE-R-INC 'name' Xf Yf Zf Af Bf Cf Uf Vf Wf)	Move one object relative to his parent MOVE: override current position MOVE-R: add offset values to the current relative position MOVE-INC: apply offset transformations one by one; in this case the order in which the axes values are defined is important MOVE-R-INC: apply offset transformations one by one starting from the current relative position. B behaviour different respect A and C: B positive counterclockwise offset, B negative clockwise offset.	(%MOVE 'Q1' X100 Z-10) (%MOVE 'OBJ1' A90) rotates OBJ1 around X axis (the A axis can be assigned to a different rotation axis) (%MOVE-R 'Q1' X100 (%MOVE-R 'Q1' X200) (final position would be X300) (%MOVE-INC 'Q1' X100 B90) is not equivalent to (%MOVE-INC Q1 B90 X100)

(%OBJECT-PROPERTY 'ObjectName'
<Property>=<Value>
<Property>=<Value> ...)
	Change display / calculation properties of an object. Supported properties:
	ACTIVE=0/1
	VISIBLE=0/1
	TRANSPARENT=0/1
	WIRE=0/1
	COLLISION=0/1
	COLOR=RGB(255, 255,255)
	SIZE-X=f //box size X
	SIZE-Y=f //box size Y
	SIZE-Z=f //box size Z RADIUS=f //cylinder or tube exterior radius
	DIAMETER=f //cylinder or tupe exterior diameter DIAMETER-INT=f //tube interior diameter
	LENGTH=f //cylinder or tube length
	POSITION=f //position of a mobile object (virtual axis)
		(%OBJECT-PROPERTY 'Base' VISIBLE=1 TRANSPARENT=1 WIRE=0)
		(%OBJECT-PROPERTY 'Base' COLOR=RGB(255,128,0) TRANSPARENT=1)
		(%OBJECT-PROPERTY 'Box' SIZE-X=UE60000)
		
		(%OBJECT-PROPERTY 'Base' VISIBLE=E60000>100)

Command	Description	Examples / Remarks
(%ODGRINDING ON/OFF) (%ODTURNING ON/OFF)	Activate or deactivate the odgrinding (turning) mode ODTURNING is equivalent to ODGRINDING	(%ODGRINDING ON) (%ODGRINDING OFF)
(%PATH ON / OFF)	Turn on / off wire path display	(%PATH OFF) (%PATH ON)
(%PATH-LABEL 'label')	Define a label marker to be displayed with the wire path	(%PATH-LABEL 'CYCLE1')

Command	Description	Examples / Remarks
(%AXIS-GROUP n) (%AXIS-GROUP <FE>)	Change current axis group index, which is used to address axes in part programs. Default value is 0.	(%AXIS-GROUP 1) X100 (X axis from axis-group 1 / channel 2 will be moved)
(%COLLISION ON/OFF)	Temporary disable collision calculation.	(%COLLISION OFF) X100 (no collision detection) (%COLLISION ON)
(%MATERIAL-REMOVAL ON/ OFF ‘nameBlank’ ‘nameCutter’ [/<flag1>…]	Activates / deactivates custom material removal on first object as blank and second object as cutter Object name can be special object: %CUTTER% means current active tool object	(%MATERIAL-REMOVAL ON %CUTTER% WD1 /OD) Remark: flag /OD activates cylindrical processing. (%MATERIAL-REMOVAL OFF %CUTTER% WD1)
(%MODEL ON/OFF)	Temporary disable material removal calculation	(%MODEL OFF) X100 (no material removal) (%MODEL ON)
(%MOVE/MOVE-R/MOVE-INC/ MOVE-R-INC ‘name’ Xf Yf Zf Af Bf Cf Uf Vf Wf)	Move one object relative to his parent MOVE: override current position MOVE-R: add offset values to the current relative position MOVE-INC: apply offset transformations one by one; in this case the order in which the axes values are defined is important MOVE-R-INC: apply offset transformations one by one starting from the current relative position	(%MOVE Q1 X100 Z-10) (%MOVE ‘OBJ1’ A90) rotates OBJ1 around X axis (the A axis can be assigned to a different rotation axis) (%MOVE-R Q1 X100) (%MOVE-R Q1 X200) (final position would be X300) (%MOVE-INC Q1 X100 B90) is not equivalent to (%MOVE-INC Q1 B90 X100)
(%OBJECT-COMMAND ‘Object-Name’ <Command1> <Command2> …)	Executes commands on object identified by name: UPDATE-GEOMETRY-FROM-MODEL //only implemented for SOR objects for wheel dressing simulation DELETE-MODEL //deletes the material removal model DELETE-WIRE-PATH //deletes the wire path model	(%OBJECT-COMMAND %CUTTER% UPDATE-GEOMETRY-FROM-MODEL DELETE-MODEL DELETE-WIRE-PATH) //Object name %CUTTER% refers to current active tool (%OBJECT-COMMAND ‘Obj1’ DELETE-MODEL)
(%PARKPOS Xf Yf Zf Af Bf Cf Uf Vf Wf)	Set the machine park position	(%PARKPOS A0 C0 X400 Y150 Z150)
(%RESET-MODEL)	Delete the 3D tool model	

(%SIM <command>)	Execute <command> as regular NC command during simulation only. Remark: this command will be ignored if executed on real NCK	(#SIM Z100) (#SIM L100=100) (#SIM $Message only for simulation) (#SIM (#OBJECT-PROPERTY COLLET COLOR=RGB(255,255,0)))
(%WRITE-VARIABLE-ONLINE <variable name>=<value>)	Write a variable value during online simulation. Variable name can be an E parameter, L variable, symbolic variable or PLC variable.	(%WRITE-VARIABLE-ONLINE E84900=200) (%WRITE-VARIABLE-ONLINE Application.OEM_END.Collision_FX3D=1)
(%OBJECT-PROPERTY ‘Object-Name’ TX TY TZ RX RY RZ)	Move one object relative to his parent (all rotation follow right-hand rule)	(%OBJECT-PROPERTY ‘BLANK’ TX20 RY45)

B.2 Variable usage inside Host Command

The Host Command usage could be improve using, as value of a parameter, a variable instead of numeric constant.

To do that is necessary to define a variable using the standard way, inside part program with the specific prefix **FX3D_HostCmd_** .

Example:

```
%1()
VAR
        [FX3D_HostCmd_positionX]
ENDV
(START OF THE PROGRAM)
[FX3D_HostCmd_positionX]=120.56
G90G40G54
G501
T1M6
D1
(%MOVE 'Q1' X[FX3D_HostCmd_positionX] Z-10)
G1 F500 X100 Z200
…
…
```

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Summary

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C1.1 SOR extension (solid of rotation) 221

C1.2 SWP extension (or SWEEP) 223

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C.1 Proprietary file format

To be able to define a custom 3D object, Flexium 3D could use a custom txt file, with proprietary extensions.

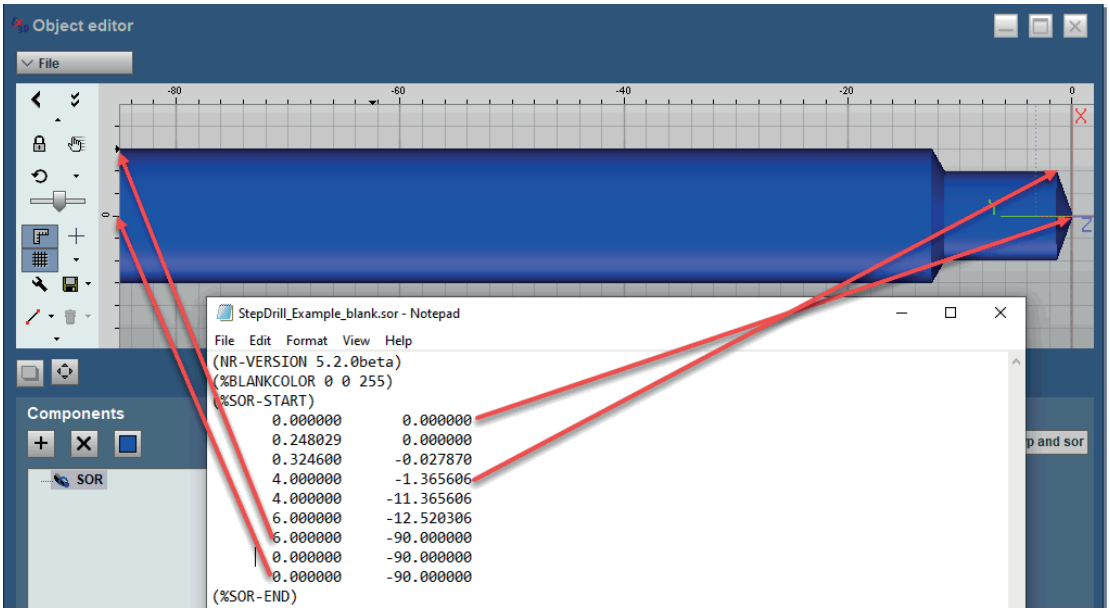
Create a simple text file and rename the extension as sor or swp.

C1.1 SOR extension (solid of rotation)

A SOR file describe a solid with a 2D profile that will be rotate along main axes of the tool\ object. Could be define with two different method:

1. profile described only with points: radius and length coordinates):

Example:

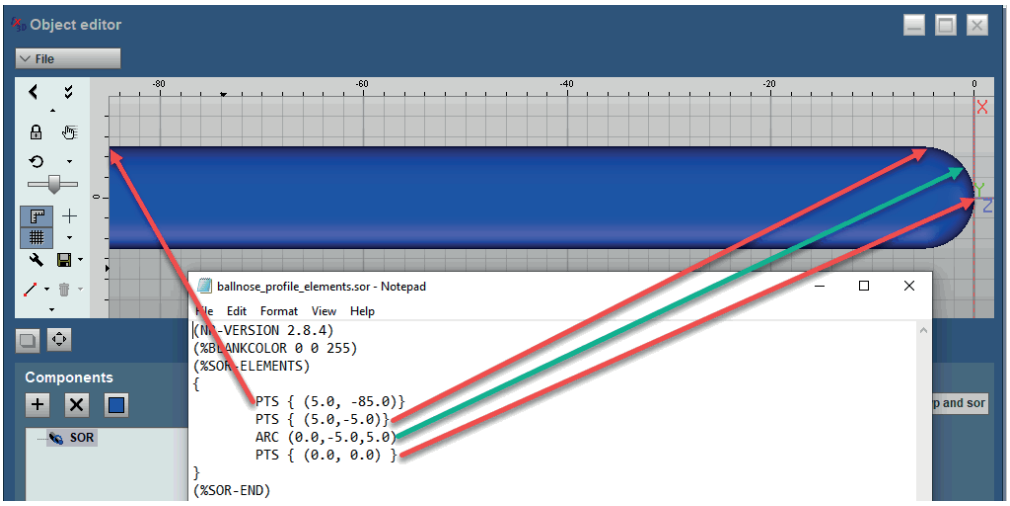



```
(%SOR-START)
        0.000000      0.000000
        0.248029      0.000000
        0.324600     -0.027870
        4.000000     -1.365606
        4.000000    -11.365606
        6.000000    -12.520306
        6.000000    -90.000000
        0.000000    -90.000000
        0.000000    -90.000000
(%SOR-END)
```

2. Profile describe with points (radius and length coordinates) and arcs (described with center point radius/length and arc radius).

Arc elements need a point before the arc and one point after the arc (points must lay on the circle)

Current limitation: SOR profiles only support clockwise arc orientation.

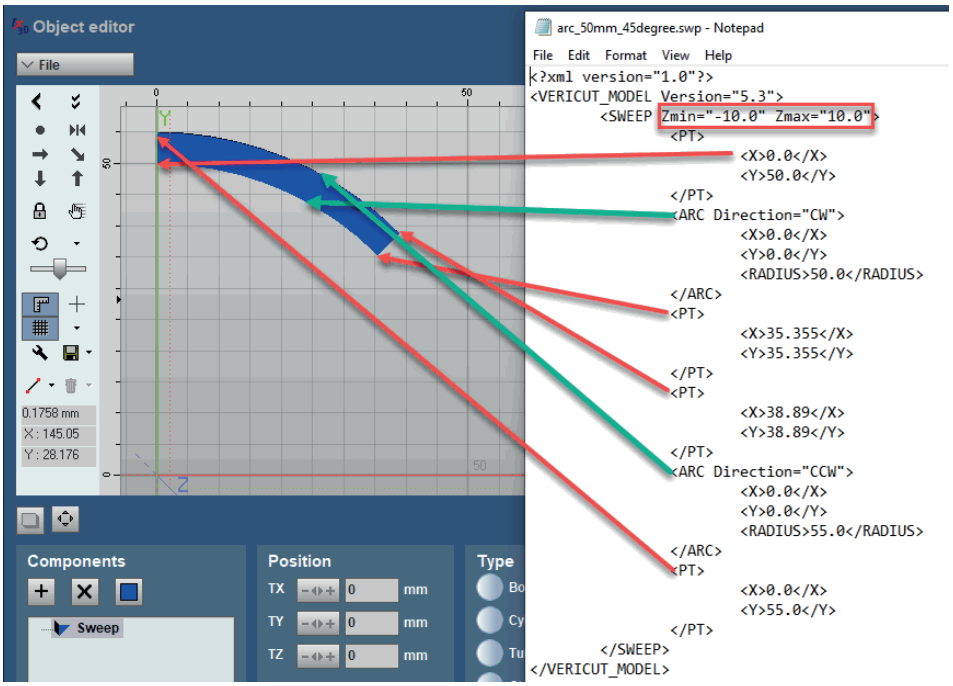



```
(%SOR-ELEMENTS)
{
        PTS { (5.0, -85.0)}
        PTS { (5.0,-5.0)}
        ARC (0.0,-5.0,5.0)
        PTS { (0.0, 0.0) }
}
(%SOR-END)
```

C1.2 SWP extension (or SWEEP)

SWP has a XML file format that describes a closed polygon (X, Y) with points and arcs (both CW and CCW direction are supported).

The thickness of the object is defined by two parameters: Zmin and Zmax.




```
<?xml version="1.0"?>
<VERICUT_MODEL Version="5.3">
        <SWEEP Zmin="-10.0" Zmax="10.0">
                <PT>
                        <X>0.0</X>
                        <Y>50.0</Y>
                </PT>
                <ARC Direction="CW">
                        <X>0.0</X>
                        <Y>0.0</Y>
                        <RADIUS>50.0</RADIUS>
                </ARC>
                <PT>
                        <X>35.355</X>
                        <Y>35.355</Y>
                </PT>
                <PT>
                        <X>38.89</X>
                        <Y>38.89</Y>
                </PT>
                <ARC Direction="CCW">
                        <X>0.0</X>
                        <Y>0.0</Y>
                        <RADIUS>55.0</RADIUS>
                </ARC>
                <PT>
                        <X>0.0</X>
                        <Y>55.0</Y>
                </PT>
        </SWEEP>
</VERICUT_MODEL>
```

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