

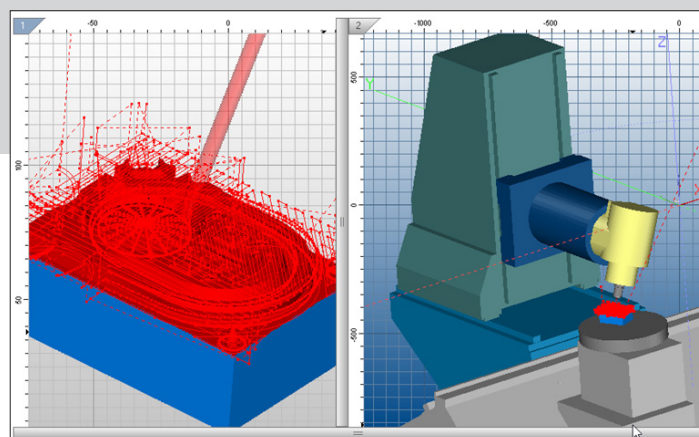
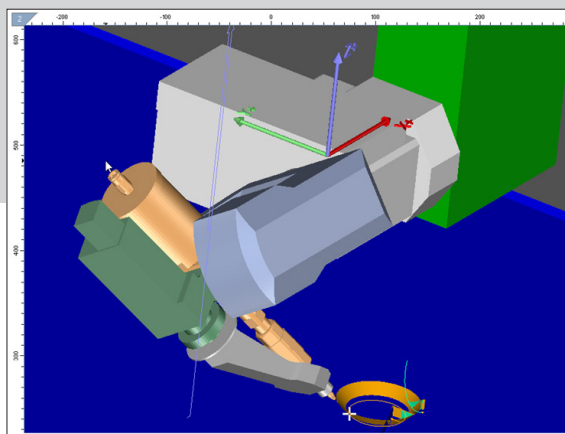
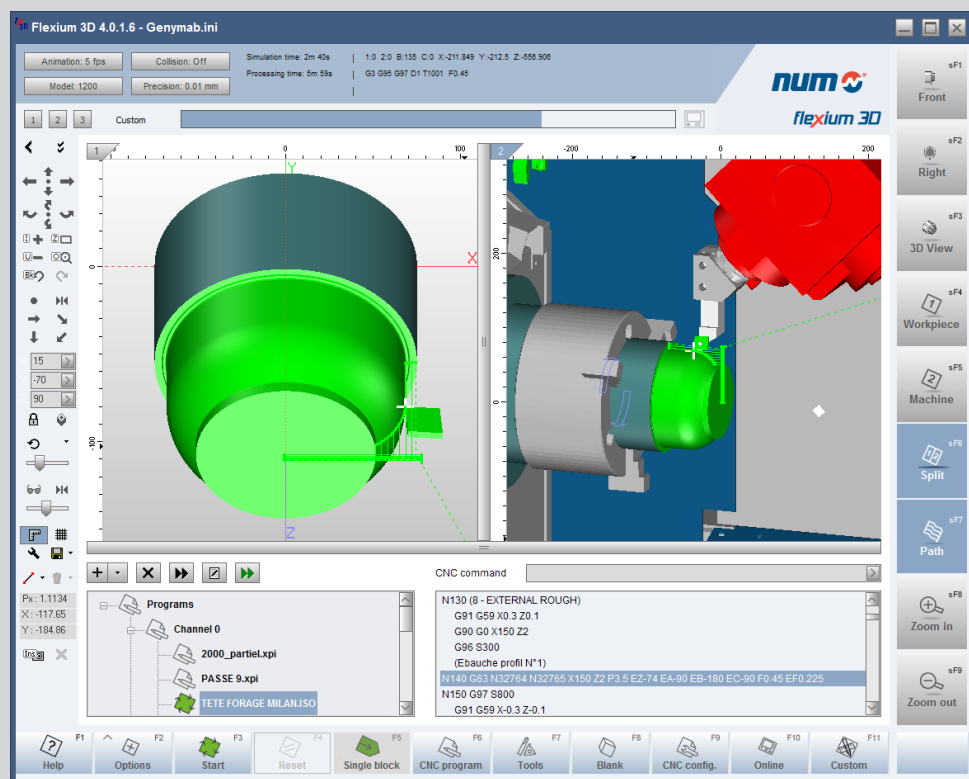
Flexium 3D

Operating Manual

(for Flexium release 4.0.20.00)

M00029EN-03

Edition 02.2016



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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.
	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

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

1.5 NUM Agencies

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

1.6 Index

The general index at the end of the manual facilitates access to information by keywords.
Please click here [index](#)

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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).

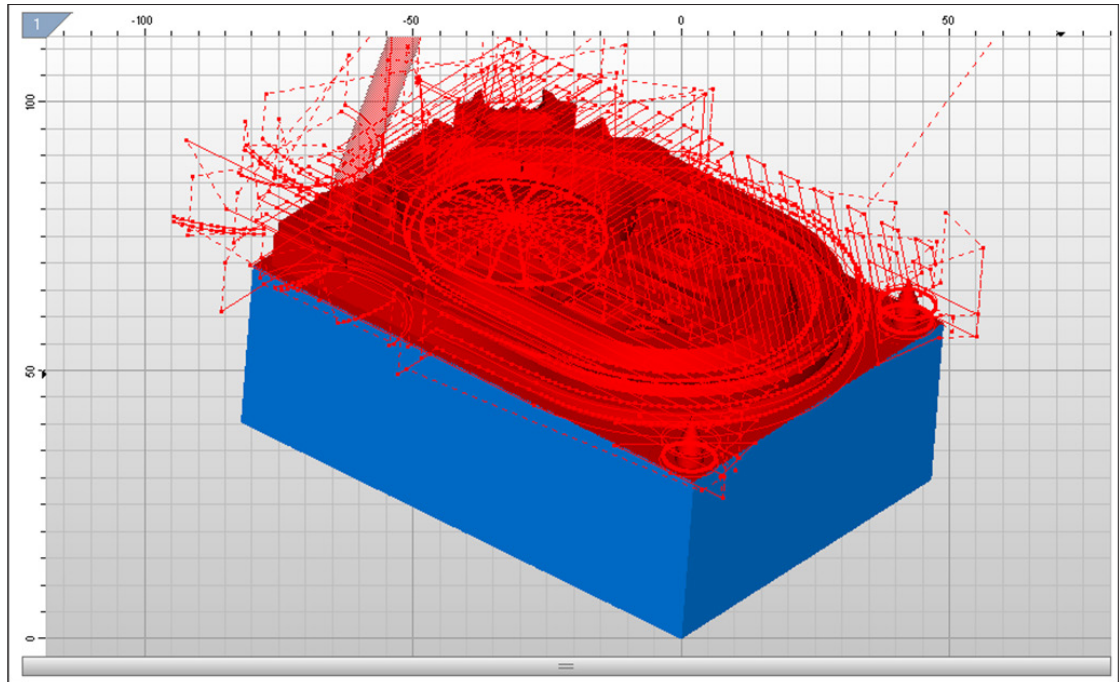
An ISO-parser is integrated inside the host application (simulation), which analyses the part program, makes all tool length and tool radius compensations as well as offset transitions and rotations and substitutes mill-, drill- and turning cycle definitions with real movements.

2.2 Main window

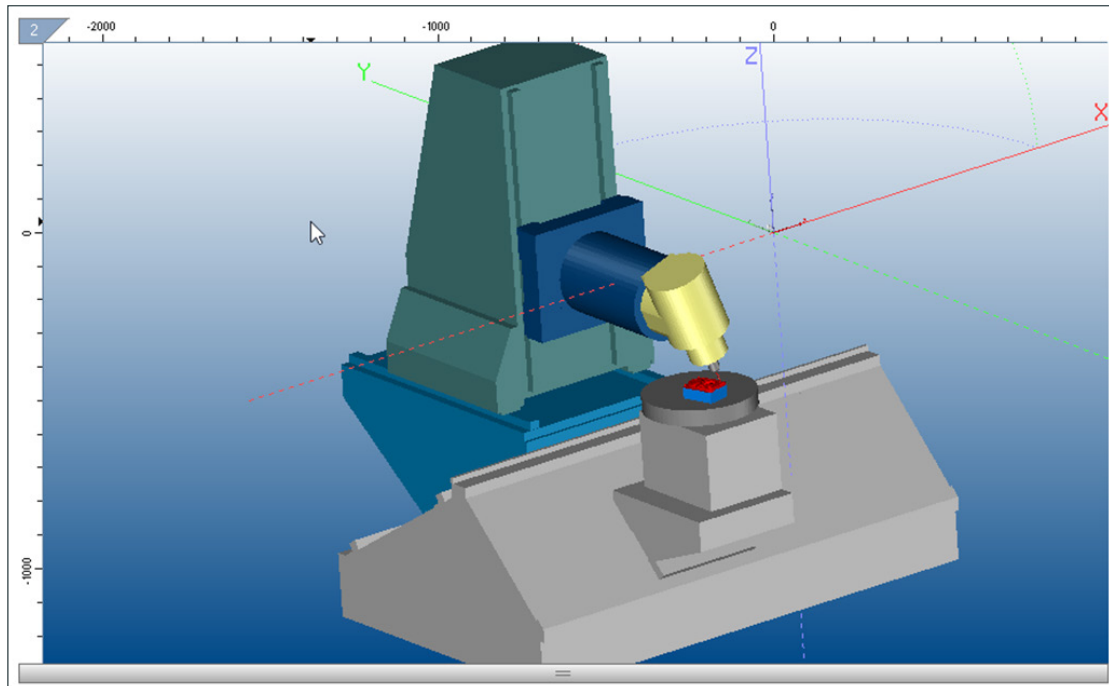
The Flexium 3D main window layout is intended to look similar to Flexium HMI context. Depending on the configuration of Flexium 3D three layout views are possible:

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

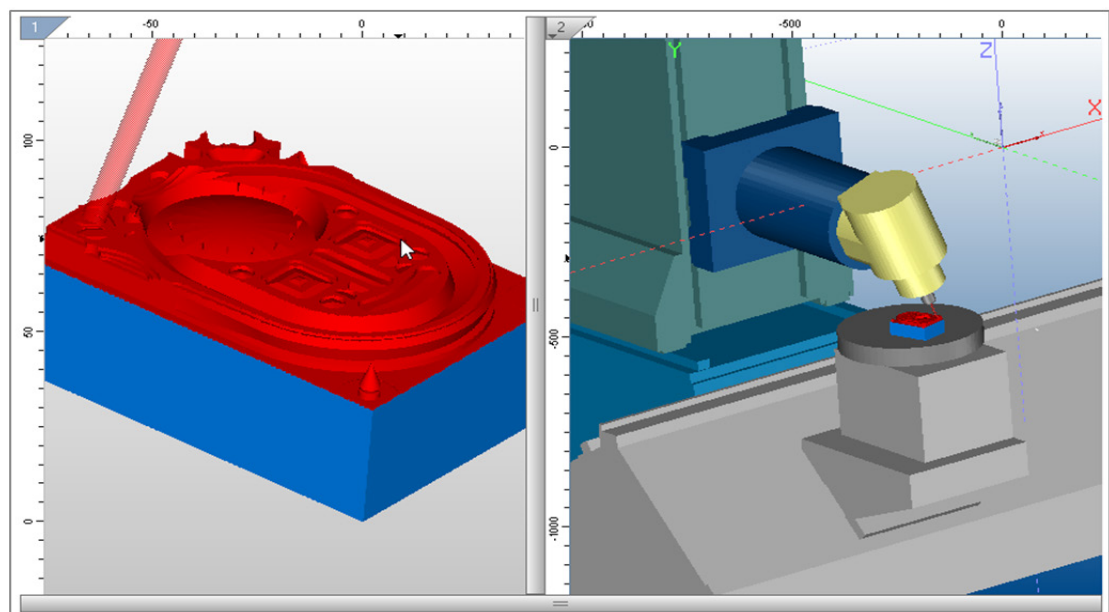
2.2.1 Flexium 3D Main Window: Part View



2.2.2 Flexium 3D: Machine View

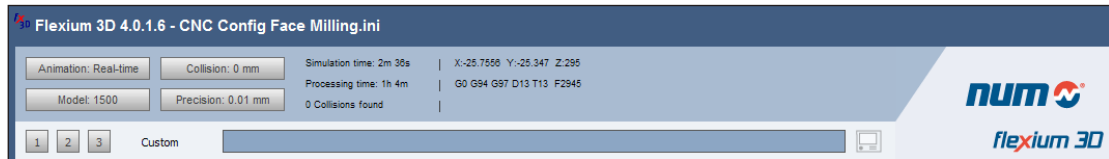


2.2.3 Flexium 3D: Split Window View

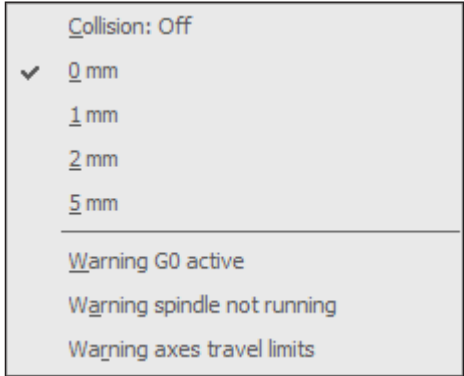


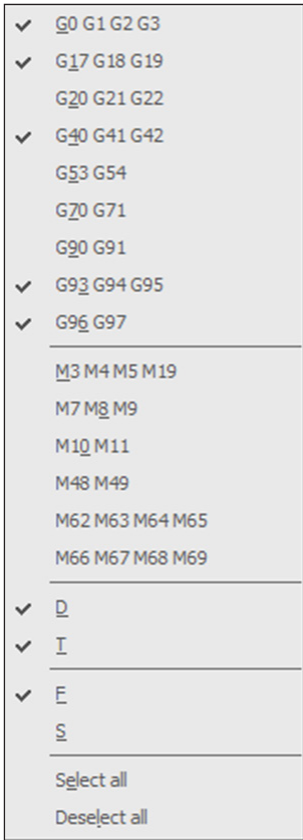
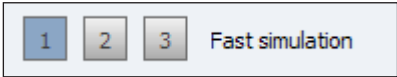
2.2.4 Status panels

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



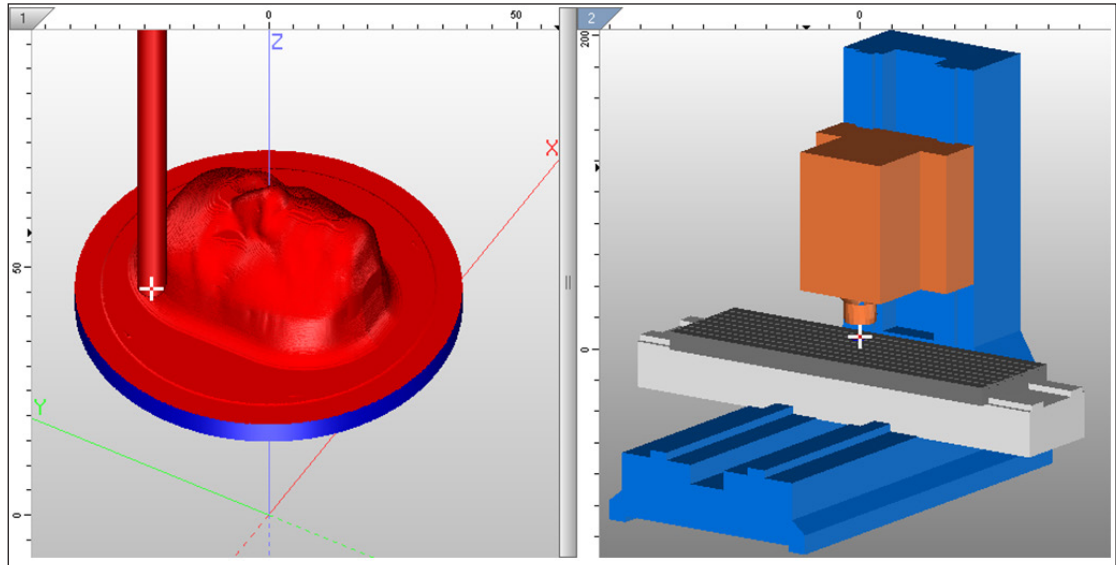
Panels description	
Window title	Version number of Flexium 3D/Flexium 3D and currently loaded CNC configuration file. This file (.ini) contains a set of resource files like machine description and machine parameter, tool table, blank definition, macro zone files, parameter and variable files and part 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, (%), and Real-time factor (X). For example 50% will display every other frame (odd frames), "All frames" will display every machining position, including interpolated positions. 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, where 1000 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.
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: • 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.
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 messages 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”.
Display of axis positions	All simulated axes are displayed as positions in machine coordinate system.
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.

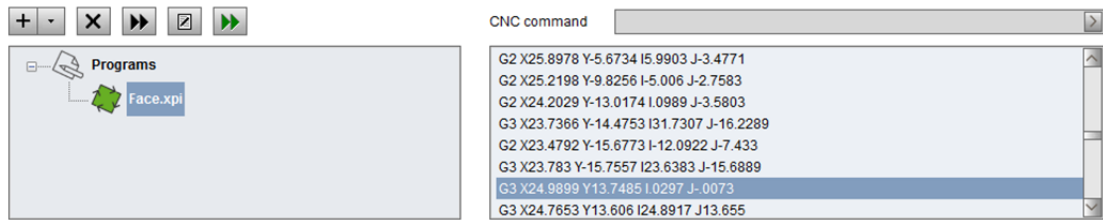
2.2.5 3D window view

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.

2.2.6 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 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;
- 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).

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).

2.2.7 3D view side control panel

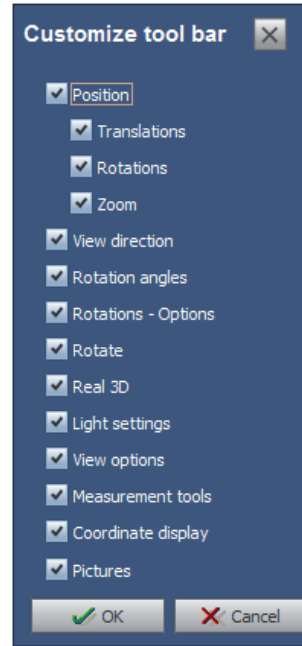
The vertical panel on the left side of the main window controls the current 3D window display (work piece or machine view) and allows changing of view position, rotate, zoom in and out, turn on and off some display options (like comparator and grid), make measurements, save pictures, and so on.

Each control button has tool tip information.

Control panel



Customizing the tool bar



2.2.8 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:



F2

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

Start	F3
Stop	F3
Reset	F4
Single block	F5
✓ Use the 'E-Par, L-Var' file	
Block skip	
Online simulation	F10
Search mode	
CNC program	F6
Tools	F7
Offsets	
Blank	F8
CNC config.	F9
✓ Workpiece view	Shift+F4
Machine view	Shift+F5
Split window view	Shift+F6
Section	
Full screen	Ctrl+F11
High resolution workpiece display	
Workpiece view smoothing (STL)	
Export model	
✓ Animation	
✓ Path display	
✓ Enable model calculation	
Reset model	
✓ Reset model at start	
Save current machined part as temporary blank	
Delete saved blank and use original blank instead	
Setup	
Language	▶
Return to HMI	Ctrl+F7
Exit program & Return to HMI	Shift+Ctrl+F7

All buttons from bottom bar [F3] to [F9] are selectable.

In addition to this you can select 3D-view to:

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.

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.

F2

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.

Animation

Turn on/off the animation.

Path display

Turn on/off the wire path display.

Enable model calculation

Turn on/off the material removal simulation on the work piece.

Reset model

Reset the processed part to blank object.

Reset model at start

Trigger a model reset when the simulation is started.

Save current machined part as temporary blank

Save the current processed part as blank for future simulations.

Delete saved blank and use original blank instead

Clear the saved blank from memory and restore the original blank object.

Setup

Open the Setup dialog (see chapter 3).

Language

13 languages are supported by Flexium 3D and are online changeable.

**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

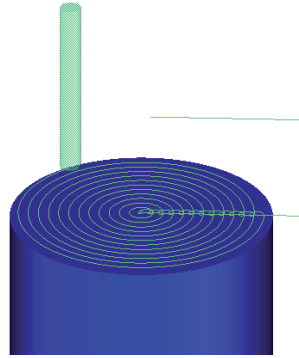
Animation during the simulation.

F2

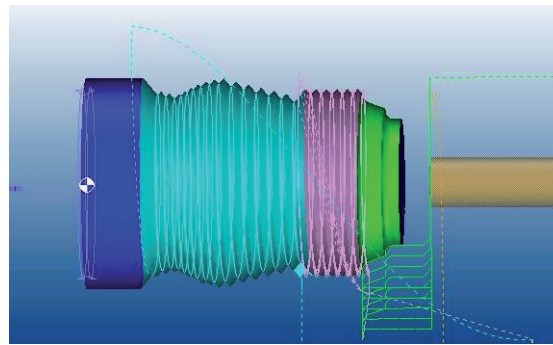
Path display

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.

Colours are selectable; automatic new color optional at Tool change M06 or subroutine call G77.



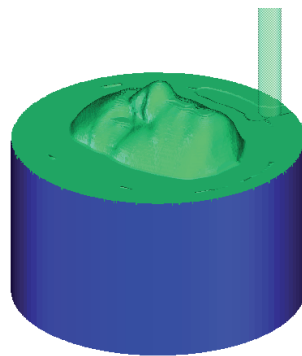
Example for milling



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: 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 5 languages are supported by Flexium 3D and are online changeable.
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.

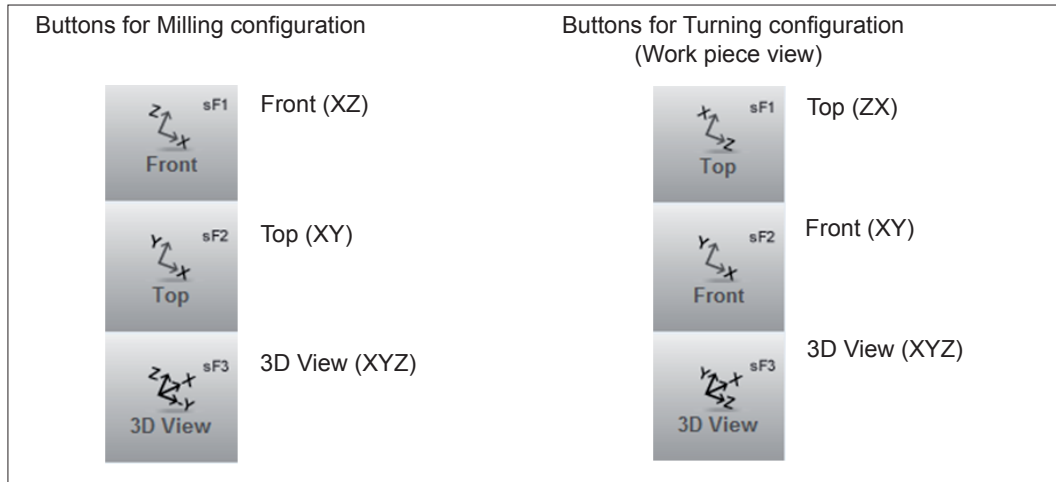
F4	Reset terminates the simulation
F5	Activates/deactivates the “Single block” mode If this mode is active then the simulation is made line by line by pressing Start (F2) after each line.
F6	NC program window Show or hide the NC program.
F7	Open Tool Editor (See chapter 5).
F8	Open Blank Editor (See chapter 6).
F9	Open the “CNC configuration” dialog
F10	Switches between the offline and online simulation modes
F11	Update table text: F11 Custom Key to start up to 3 applications (e.g. different FX3D configurations or 3rd party programs). Configuration can be made in <i>Setup -> Miscellaneous -> Custom Key</i>
F12	Reserved

2.2.9 Vertical toolbar shortcut buttons

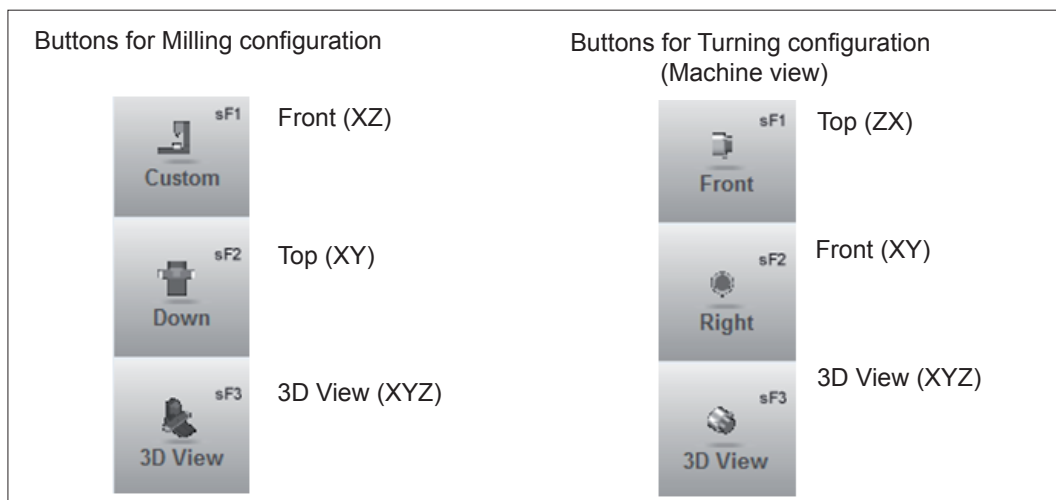
On the right side of the main window there are several shortcut buttons also accessible with keyboard combination  + F key.

2.2.10 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:

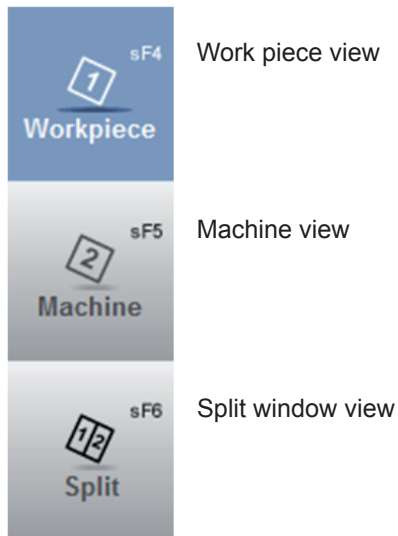


Custom functions button to launch external applications or open preset CNC configurations.



2.2.11 3D view layout buttons

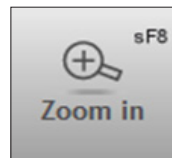
The next three buttons are used to change the 3D views layout.



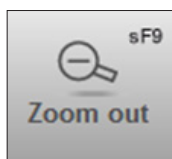
2.2.12 Additional buttons



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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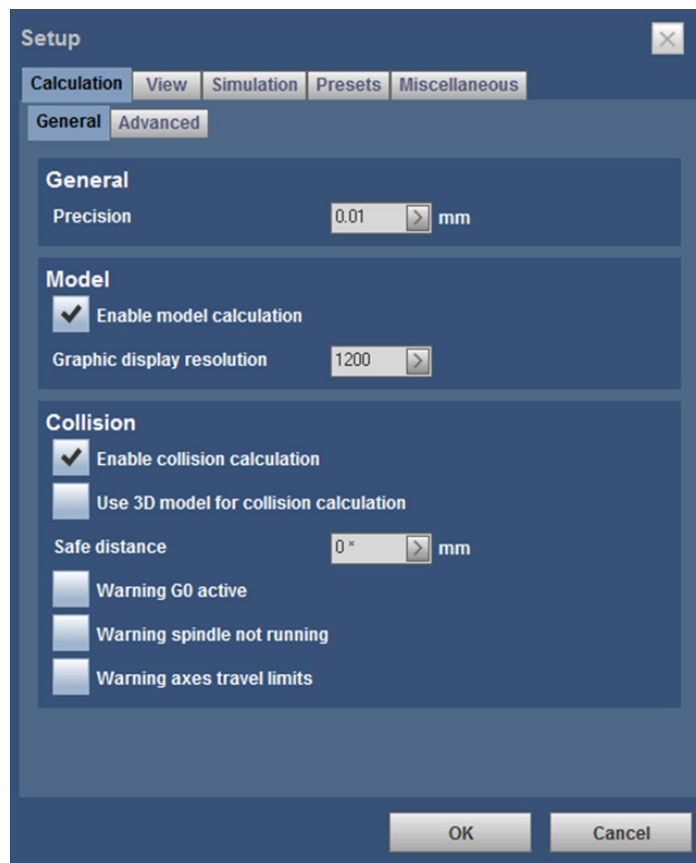
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3 Setup Dialog

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

The screenshot shows the 'General' settings tab. It is divided into three main sections: General, Model, and Collision. In the General section, the Precision is set to 0.01 mm. In the Model section, 'Enable model calculation' is checked, and the Graphic display resolution is set to 1200. In the Collision section, 'Enable collision calculation' is checked, 'Use 3D model for collision calculation' is unchecked, and the Safe distance is set to 0 mm. There are also three unchecked checkboxes: 'Warning G0 active', 'Warning spindle not running', and 'Warning axes travel limits'.

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 “<i>Safe distance</i>”. 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 “<i>Use 3D model for collision calculation</i>” 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 “<i>false</i>” 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.

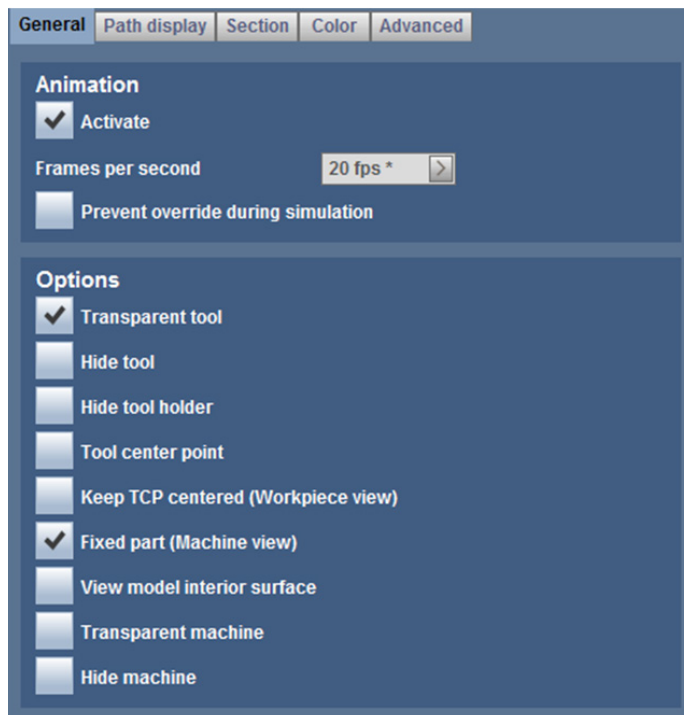
3.1.2 Advanced



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.
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 “ <i>High resolution workpiece display</i> ” 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).

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.
Options	<i>Transparent tool</i> Display tool with transparency.
	<i>Hide tool</i> Hide the tool object for better view on the workpiece.
	<i>Hide tool holder</i> Hide the tool holder object.
	<i>Tool center point</i> Mark the tool center point inside the 3D views with a white cross.
	<i>Keep TCP centered (Work piece view)</i> 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.
	<i>Fixed part (Machine view)</i> Keep the work piece fixed on screen inside the machine 3D view. The machine components move around the work piece.
	<i>View model interior surface</i> 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.
	<i>Transparent machine</i> Display machine components with transparency, for better view of the work piece.
	<i>Hide machine</i> Hide machine components inside the machine view.

Fixed part	When this option is activated, the part (work piece) will be kept fixed on screen during the simulation in the machine 3D view. The machine components will move around the part. If the option is off, then the machine origin will be kept fixed on screen during the simulation and the part object will move.
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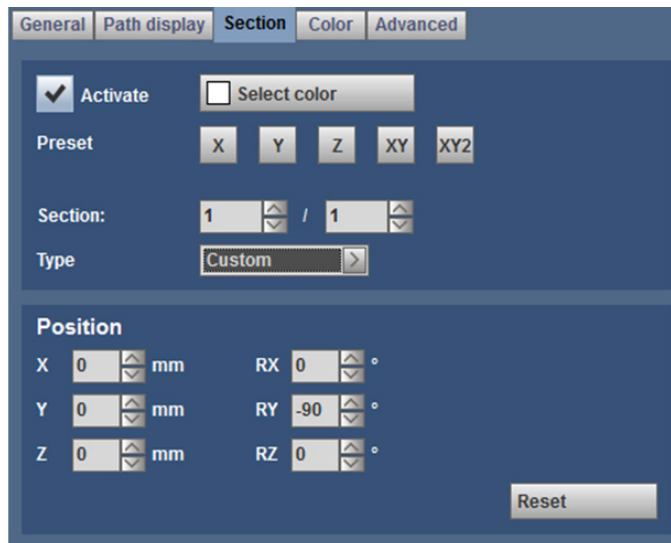
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3.2.2 Path display



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.
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.
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 " <i>Enable swept volume</i> " is deactivated. In this case the wire lines are not displayed.
Simulation	The path display can be configured differently for three simulation types: A vertical list of three options: 'Offline' (highlighted in dark grey), 'Search mode' (in light grey), and 'Online' (in light grey). 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.
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. A vertical list of three options: 'Continuous' (in light grey), 'Dashed' (highlighted in dark grey), and 'Remove' (in light grey).

3.2.3 Section

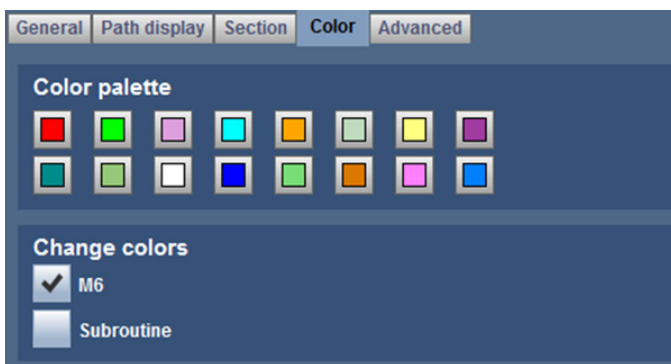


On this page one or two clipping section can be defined. One clipping plane is defined with a 3D point (X, Y, Z) and an orientation with Euler angles (RX, RY, RZ).

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.

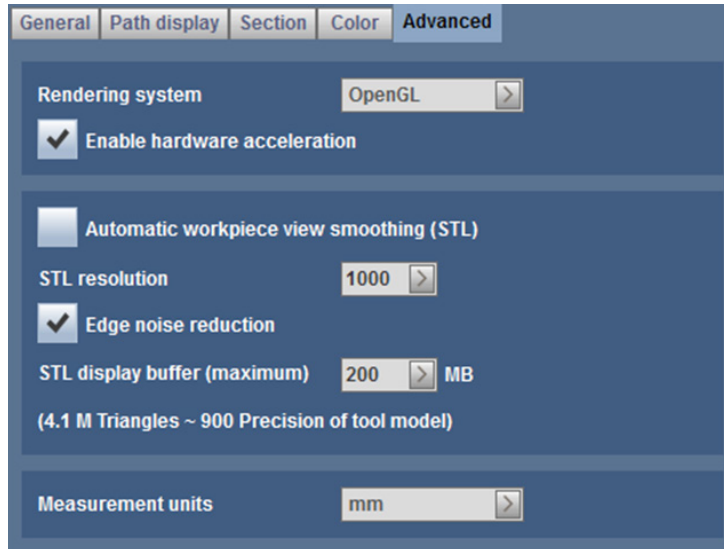
3.2.4 Color



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).

3.2.5 Advanced



Rendering system	Select the current rendering system: OpenGL, Direct3D 9 or Direct3D Shader. <i>Enable hardware acceleration</i> Enable the 3D display hardware acceleration. For better performance, the acceleration should be disabled only in case of display errors. <i>Automatic work piece view smoothing (STL)</i> Activate the work piece view smoothing after the simulation is completed.
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. Improves the edges quality for the 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.

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.

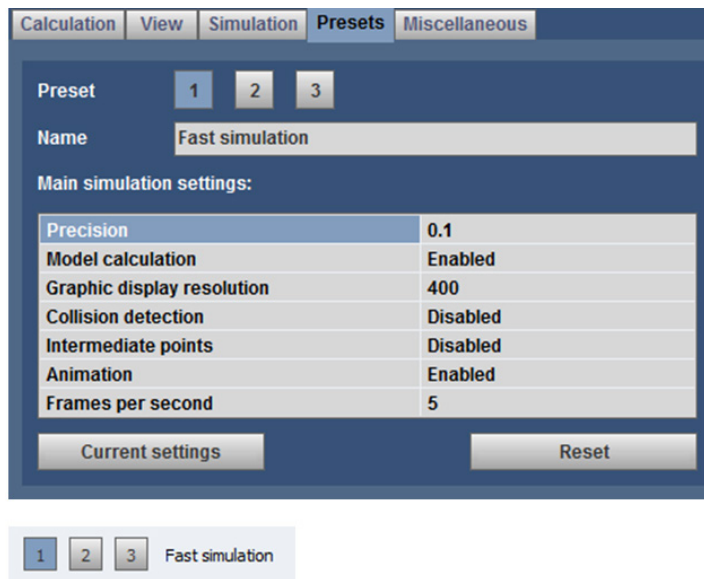
Reset automatically load from CNC	When the option is activated, all flags that automatically requests resources from the CNC (parameters, offsets, tools, macro zones) are reset to false when the application starts. If the option is off, then the flags are preserved as they were saved inside the CNC configuration <i>"ini file"</i> (See chapter 11.3).
Block skip	Enable the execution of NC blocks starting with " / " character.
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

The screenshot shows the 'Online' setup dialog. At the top, there are two tabs: 'Offline' and 'Online', with 'Online' being the active tab. Below the tabs, the dialog is organized into several sections. The 'Connection' section includes a 'Type' dropdown menu set to 'FxServer' and a checked checkbox for 'Connect to CNC automatically'. The 'Refresh rate' section has three input fields for 'Foreground' (100), 'Background' (200), and 'Minimized' (1000), each followed by a unit of 'ms'. There is also a checked checkbox for 'Always'. The 'Oscilloscope' section features an 'Activate' button and a 'CNC cycle step' input field set to '1' with a unit of 'cycle(s)'. At the bottom, the 'NCK program header lines' section has an input field set to '1'.

Connection	This option defines the connection type for online simulation. FxServer should be used for Flexium CNC.
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. Cycle step 10 means that every 10 cycles the machine axes are transmitted during online simulation.
NCK program header lines	Defines the number of additional header lines that are added to part programs when uploaded to NCK. This value is used to display the correct current program line during online simulation and also to reach the correct restart line during Visual Search mode.

3.4 Presets



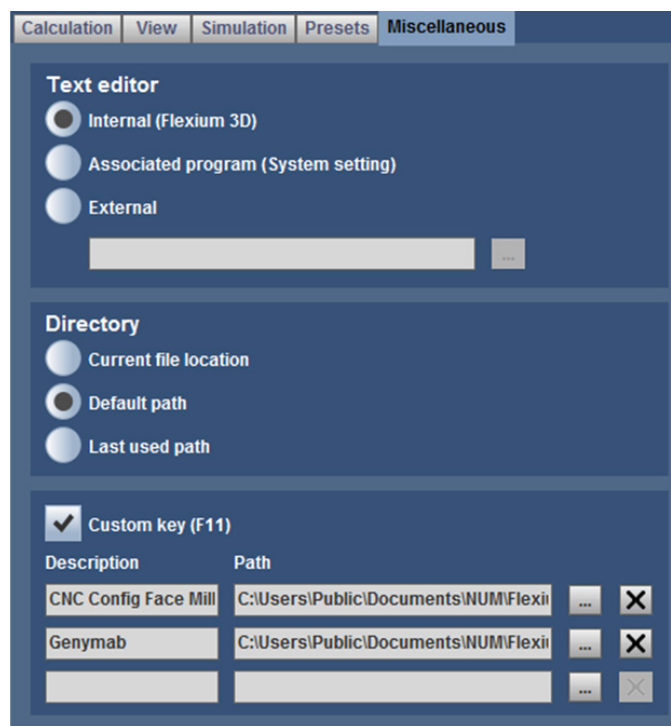
There are three presets for the simulation parameters that determine the simulation speed and accuracy. By default, preset 1 is called *“Fast simulation”*, preset 2 is *“High accuracy”* and preset 3 *“Path only”*.

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.

The custom *F11 button* can be configured with three items, where an item can be an external application (exe file) or a CNC configuration (ini file).

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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.
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)

4.2 Configuration window entries

Edit

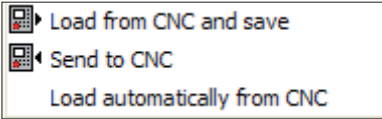
Edit existing configuration.
The CNC configuration input files can be defined inside a dialog window.

For part programs it is possible to define initialization files for E-parameter (global or not) and program dependant L-variable as well as macros or subroutines from macro zones.

The screenshot shows a configuration dialog window titled "C:\Users\Public\Documents\NUM\Flexium 3D\config\work\CNC C...". The window contains several sections for configuring CNC parameters:

- File:** A dropdown menu.
- Machine parameters:** A text field containing "C:\Users\Public\Documents\NUM\Flexium 3D\config\machineparams\PARAM_".
- Options:** A text field containing "C:\Users\Public\Documents\NUM\Flexium 3D\config\options\All_Option_NC1.o".
- Offsets:** A text field containing "C:\Users\Public\Documents\NUM\Flexium 3D\config\offsets\Workstd_N".
- Tools:** A text field containing "C:\Users\Public\Documents\NUM\Flexium 3D\config\tools\FaceMilling.ttl".
- Tool loading function:** A dropdown menu showing "M 6 *".
- Macro zones:** A list box with a "+" button to add new zones.
- Reset program:** A text field.
- E-Parameter, L-Variable:** A text field.
- Machine:** A text field containing "C:\Users\Public\Documents\NUM\Flexium 3D\config\machines\Machine".
- Kinematics preview:** A 3D model of a machine tool.
- Technology:** A dropdown menu showing "Milling".
- NCK type:** A dropdown menu showing "Flexium".
- Blank:** A text field containing "C:\Users\Public\Documents\NUM\Flexium 3D\config\blanks\FaceCylinder.blanl".
- Collet:** A text field.

At the bottom of the window are three buttons: "Validate", "Validate and save", and "Cancel".

Edit description	Machine Parameter Machine configuration file (.xpa). This file can be generated in ASCII-format with Flexium Tools.
	Options License file for the CNC (.opt). Format is ASCII.
	Offsets File with part and program origin offsets. Format is ASCII.
	Tools Tool file library; automatically generated from tool editor.
	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 dependant I-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.
	Tools 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:
	 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  Send to CNC Load automatically from CNC 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.

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5 Tool editor window

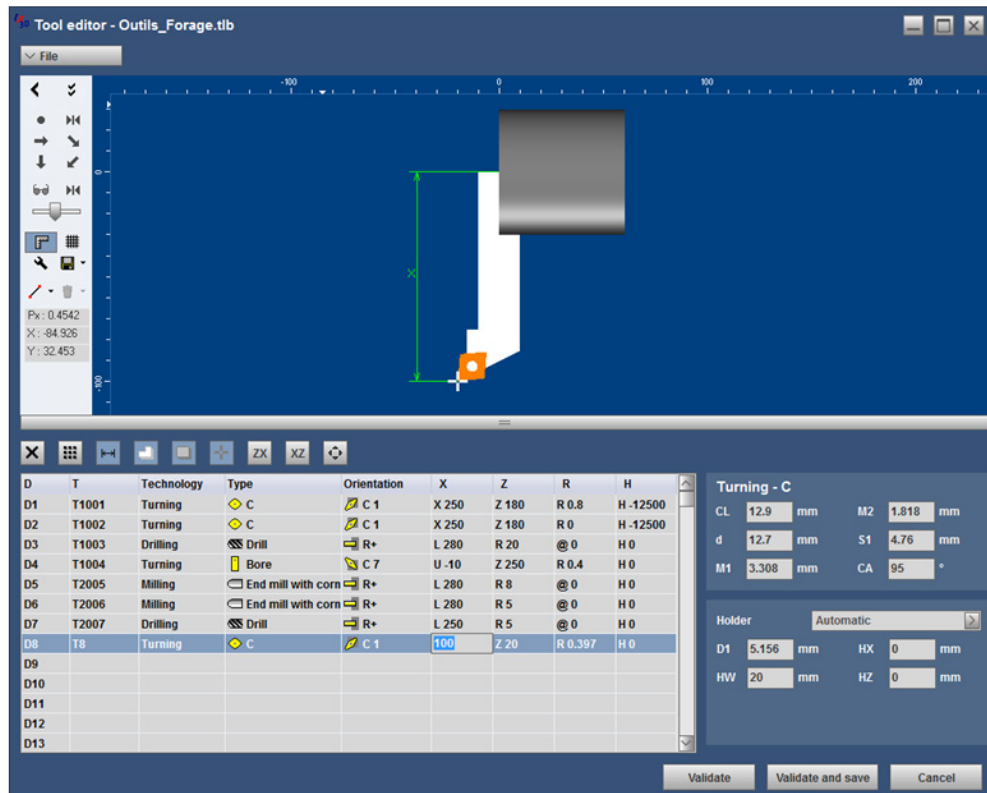
5.1 Window view

5.1.1 Open tool editor window

The **F7** button opens the tool editor window



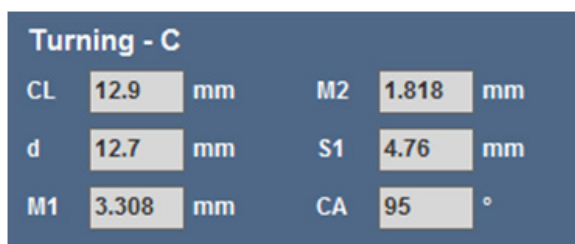
5.1.2 Tool editor page description



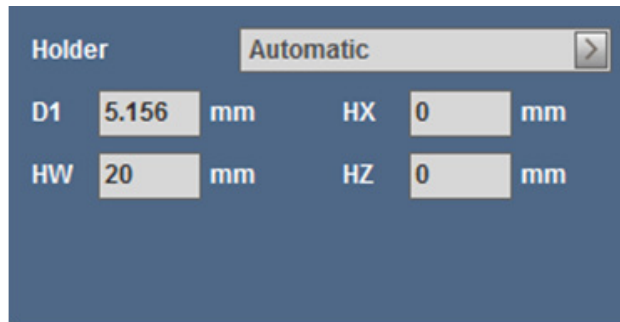
The tool table allows the visualization and modification of the main tool parameters:

- C/O, L, R, @ for milling and drilling
- C, X/U, Z, R 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.



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 current “ <i>tlb file</i> ” 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.
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.	

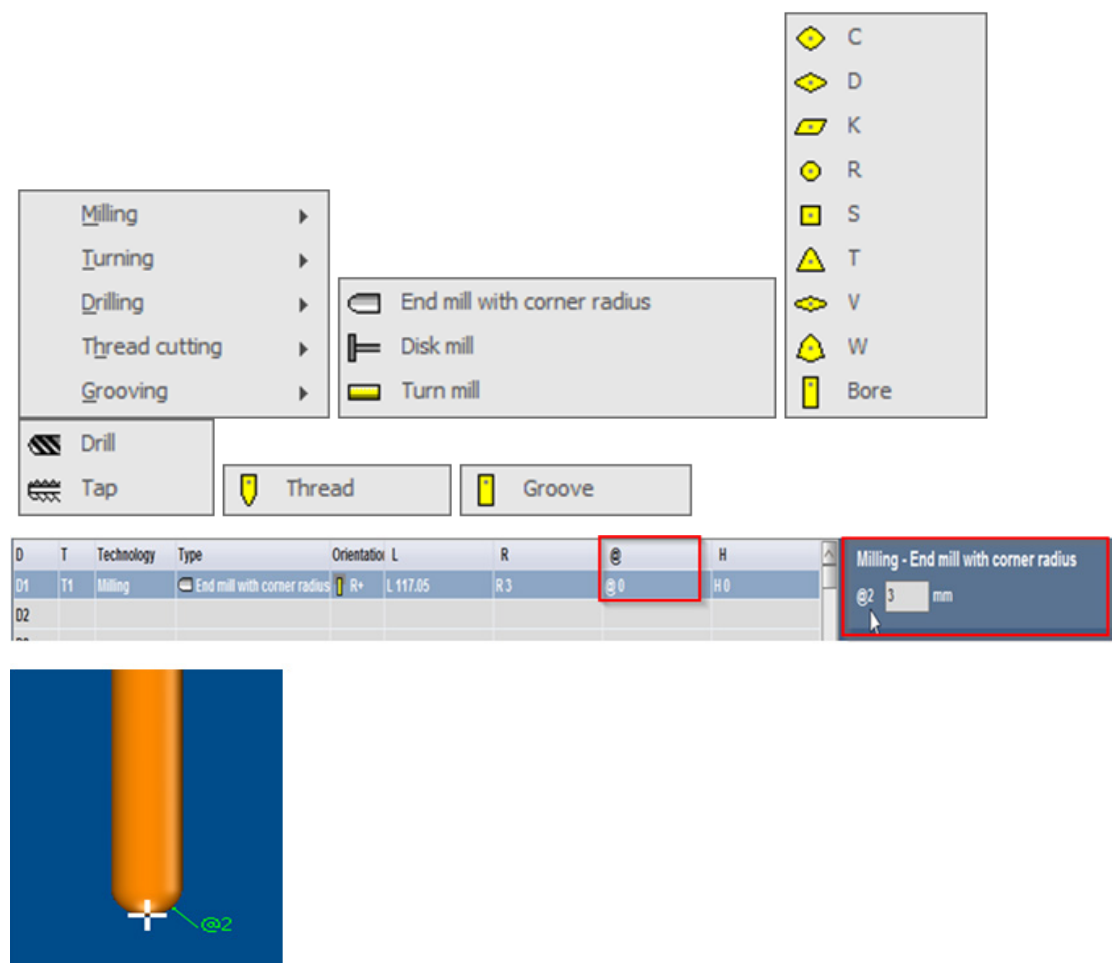
5.2 Create a new tool

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 presented for each technology:



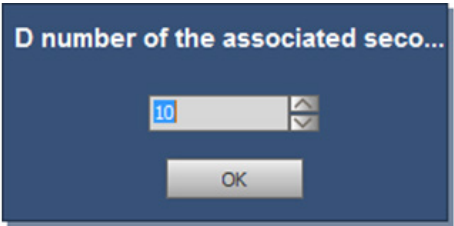
For tool type Milling in 5 axes interpolation with RTCP very often the Tool is defined in the tool table without corner radius ("@" column), because during PP generation of used CAM system, this is already considered.

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.

5.2.2 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

5.2.3 Delete an existing tool

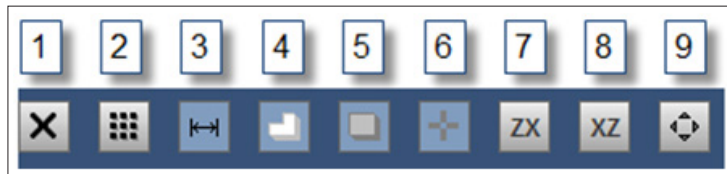
To delete an existing tool, select the tool in the tool table and press **Del** key, key or  button on the tool bar, or select on the technology column the last empty line:



5.2.4 Edit the tool parameters

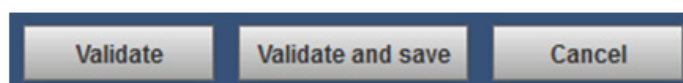
The tool parameters can be edited by clicking in the table or in the corresponding field from the right-side panel.

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 quota in the 3D view. The quota 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.

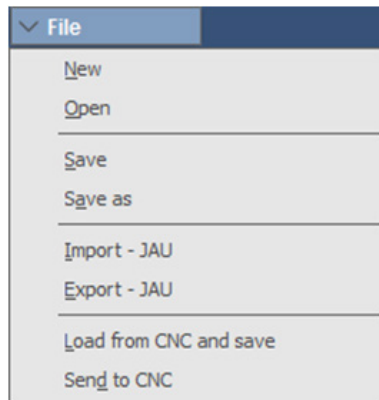
5.2.5 Bottom tool bar



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.

5.2.6 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. 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.

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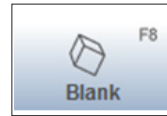


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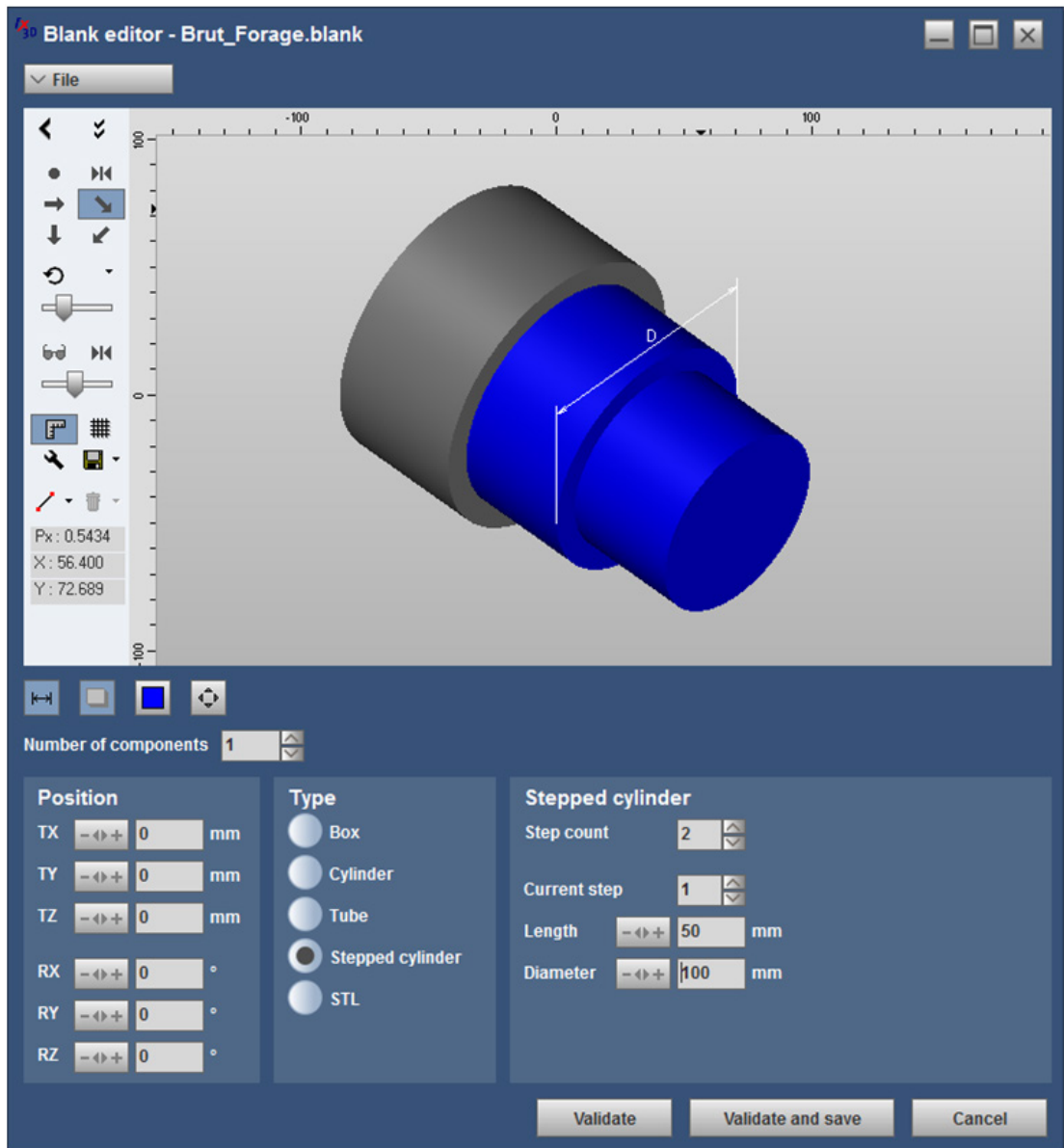
6 Blank editor window

6.1 Open blank editor window

The **F8** button opens the blank editor window



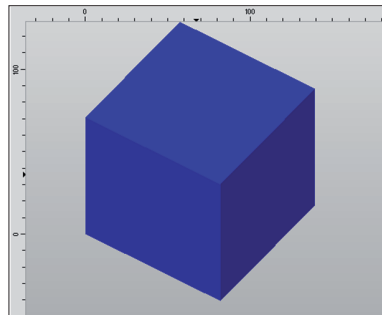
6.1.1 Blank editor page description



The geometry of one blank component can be defined as a Box, Cylinder, Tube, Stepped cylinder or STL.

Box

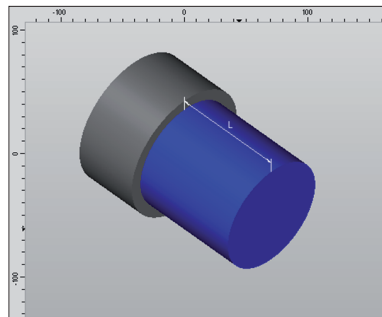
The box is defined by 3 dimensions in mm (X, Y and Z). The reference point is one corner of the box:



Box		
X	100	mm
Y	100	mm
Z	100	mm

Cylinder

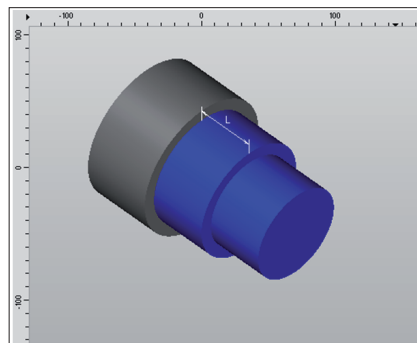
The cylinder is defined by the length and diameter. The reference point of the cylinder is the center of the rear face (cap).



Cylinder		
Length	100	mm
Diameter	100	mm

Tube

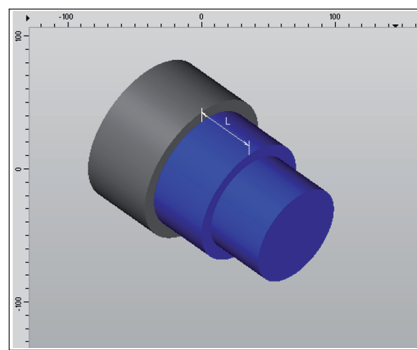
The tube is defined by length, external diameter and internal diameter. The reference point of the cylinder is the center of the rear face:



Tube		
Length	100	mm
Diameter	100	mm
Diameter 1	80	mm

Stepped cylinder

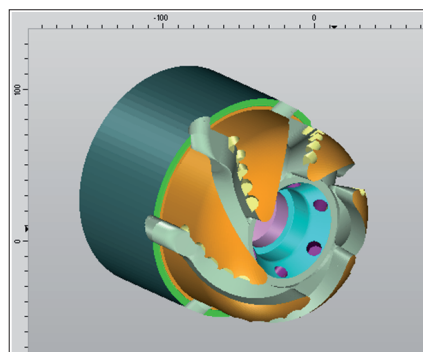
Up to four steps can be defined with different diameters and lengths:



Tube		
Length	100	mm
Diameter	100	mm
Diameter 1	80	mm

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.



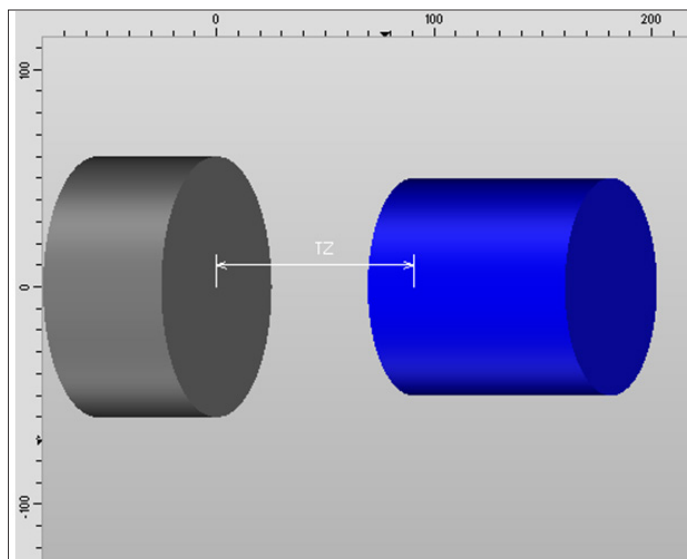
STL	
C:\Users\Public\Documents\NUM\FI	
Import	

Position

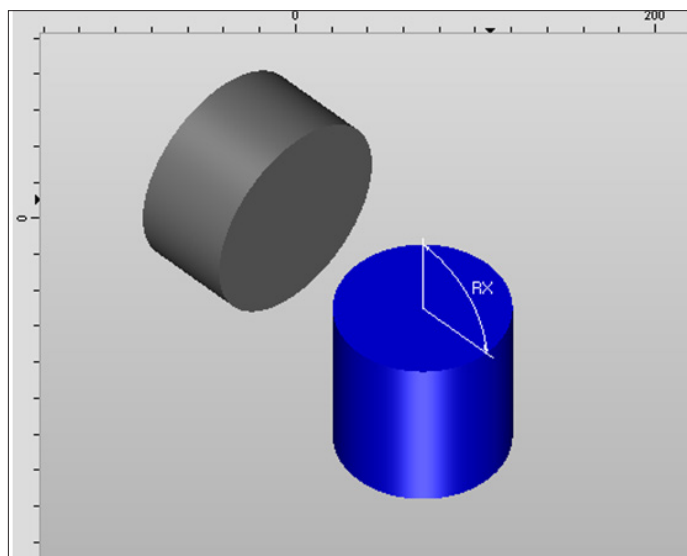
The blank position is the offset from the turret reference point (the mounting reference) to the blank reference point:

Position		
TX	- < > +	0 mm
TY	- < > +	0 mm
TZ	- < > +	100 mm
RX	- < > +	0 °
RY	- < > +	0 °
RZ	- < > +	0 °

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:



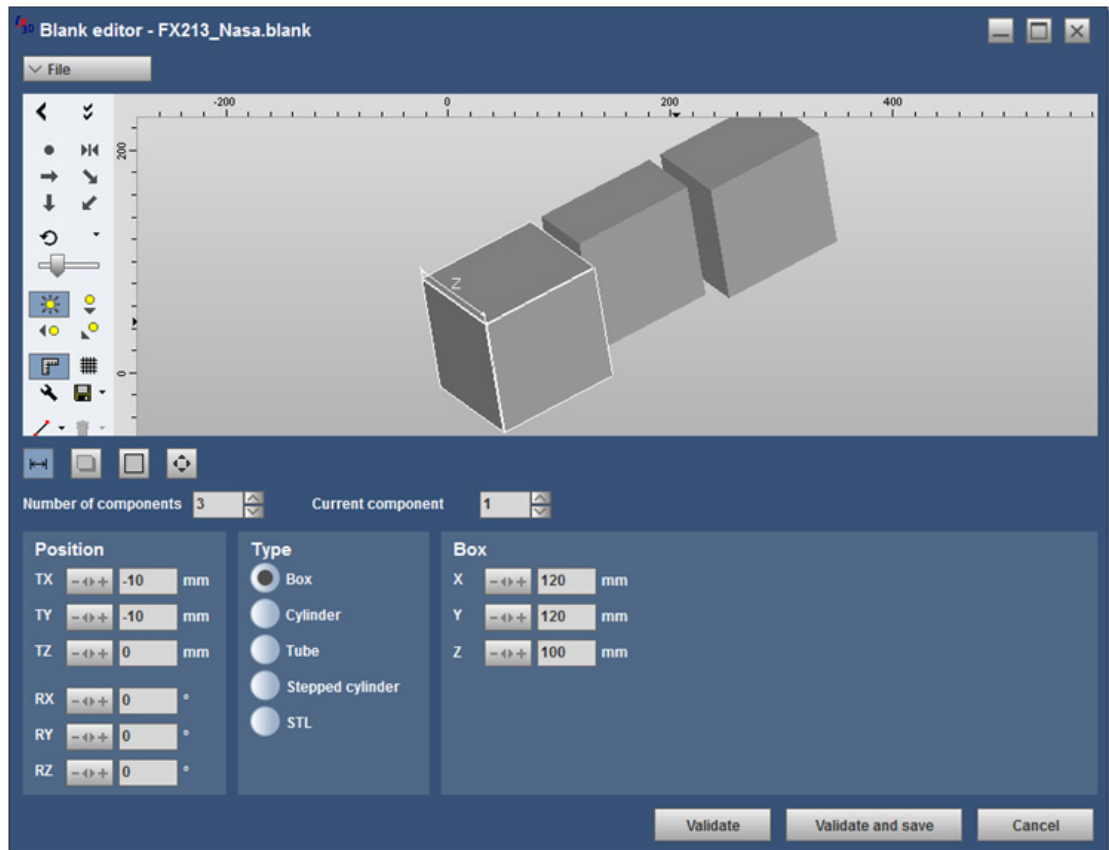
6.1.2 Manually edit values

You can manually edit values (dimension or position) or you can use the “Slide” button  to increment / decrement 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



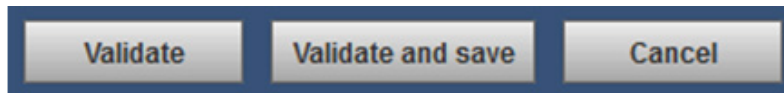
Number of Components	Number of work pieces in work area.
Current component	Define the current component index to allow the input of geometry data. The current component is marked in the 3D view with a white frame.

6.1.4 3D view buttons



1	Enable the display of the dimension lines (quota) 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).



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Summary

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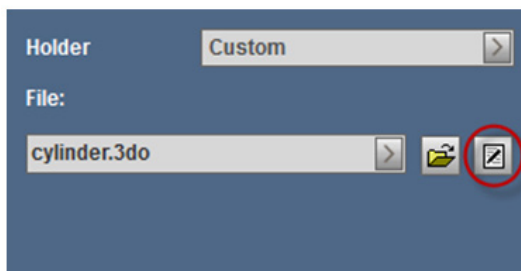
14

Page deliberately empty.

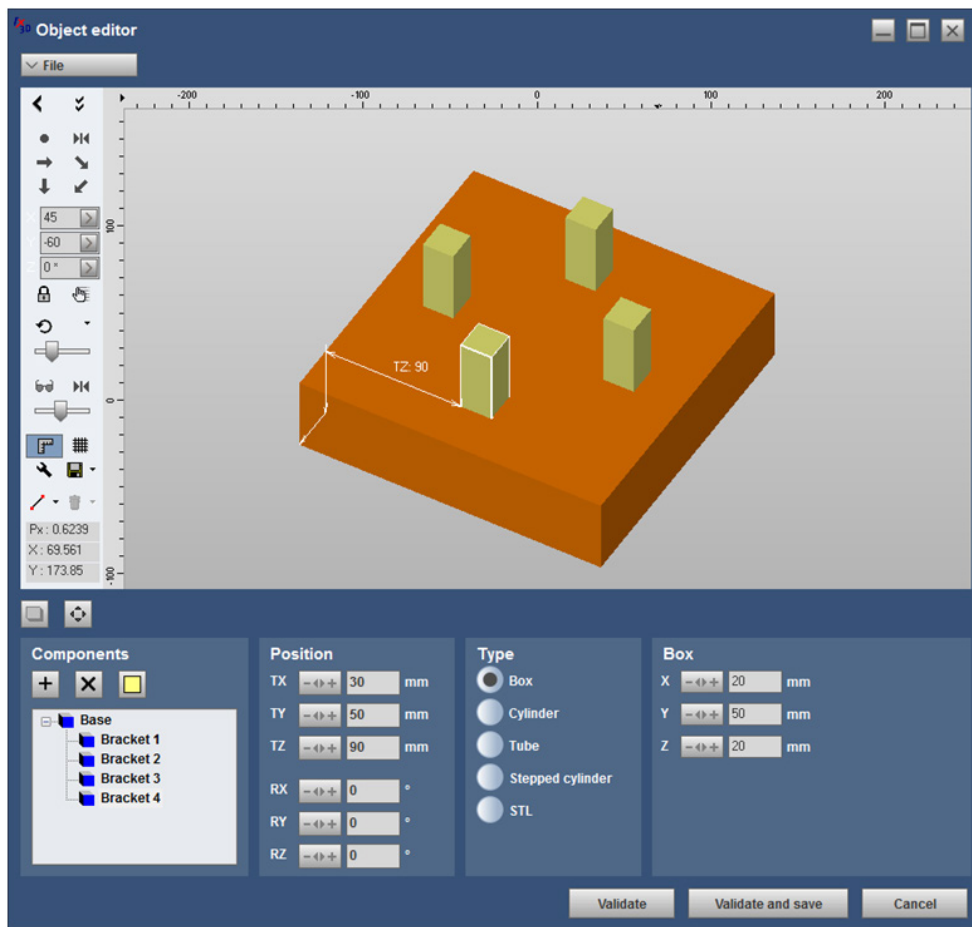
7 Object editor window

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) and also for custom tool holder devices (Holder section on Tool editor window):



7.2 Object editor window



7.2.1 File menu

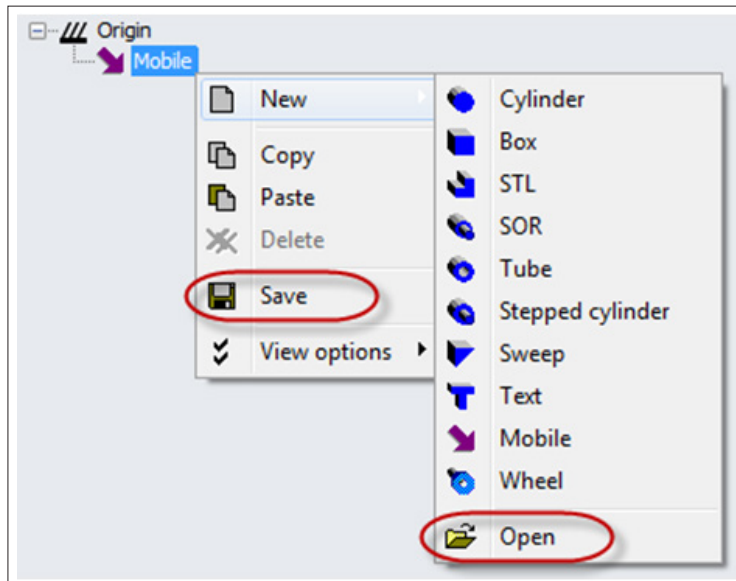
The file menu contains the following options:



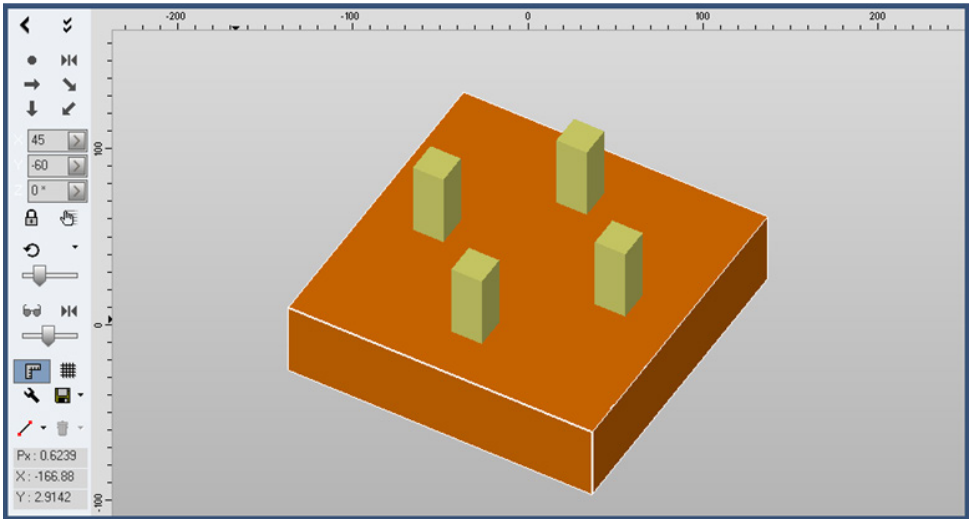
New	Create a new void object.
Open	Opens an existing file (3DO extension).
Save / Save as	Save the current object to a 3DO file.

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.

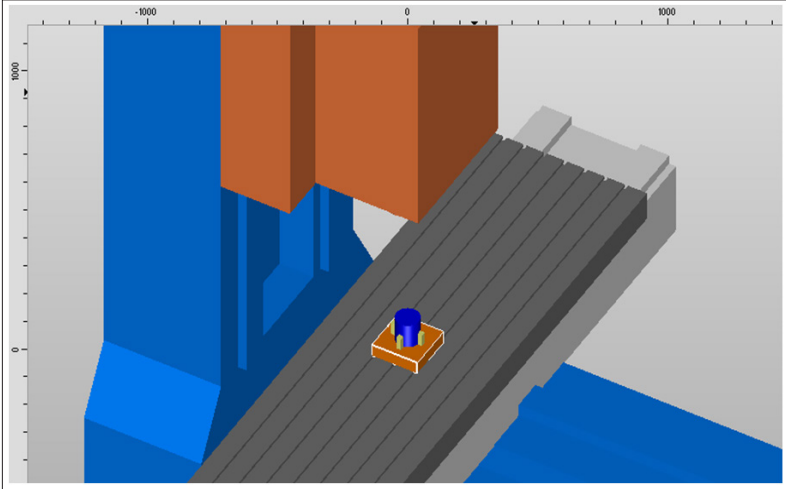
The format is compatible with the 3D objects used for the construction of machine models inside Machine editor window. The 3DO files can be imported and exported by the Machine editor window, using “Open” and “Save” options from the right click menu of “Construction” tree view.



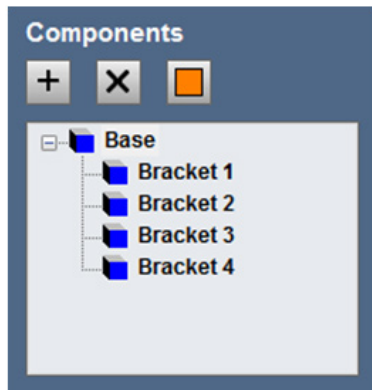
7.2.2 3D view



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.

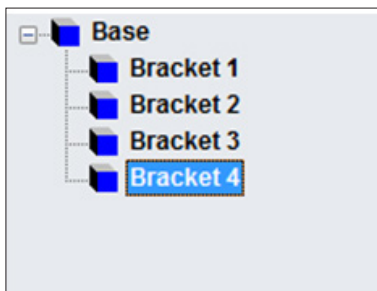
7.2.4 Components panel



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

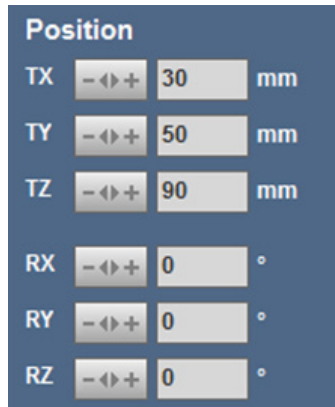
New 	Add new object to the tree structure. The new object will be created as child of currently selected object. If the tree structure is empty, the root object will be created.
Delete 	Delete the currently selected object
Select color 	Change the color of currently selected object.

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).

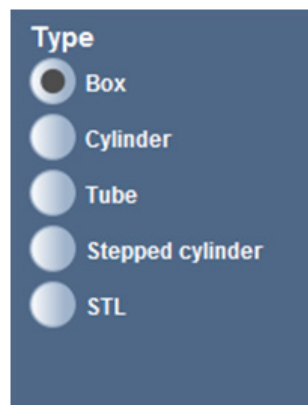
7.2.5 Position panel



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) or free form triangle mesh object (STL).

7.2.7 Geometry panel

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

Box

X mm

Y mm

Z mm

Cylinder

Length mm

Diameter mm

Tube

Length mm

Diameter mm

Diameter 1 mm

Stepped cylinder

Step count

Current step

Length mm

Diameter mm

STL

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

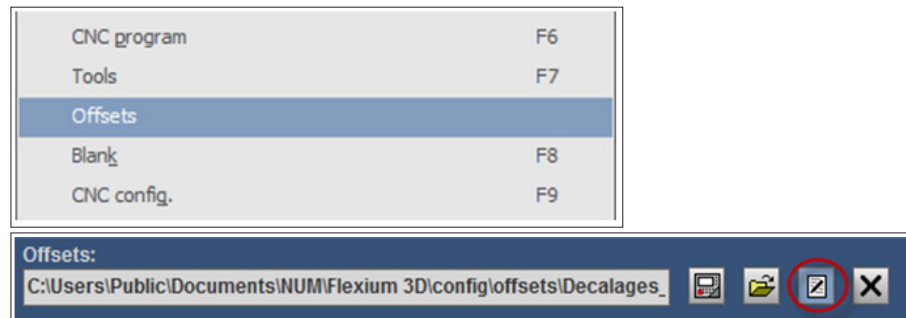
The offsets editor can be used to create or modify a special type of file that is processed during the simulation by Flexium 3D and the ISO-parser.

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.



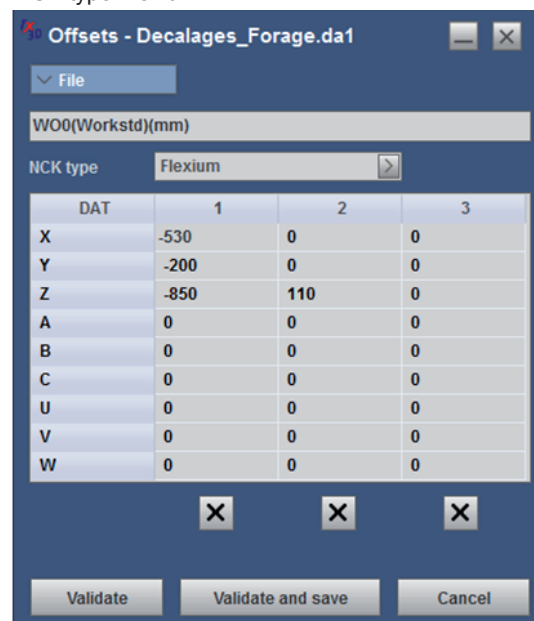
8.2 Window description

The window contains:

- 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

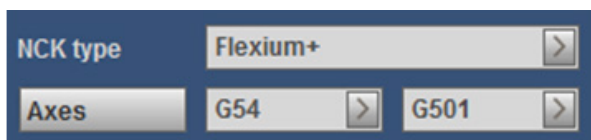


NCK type Flexium+



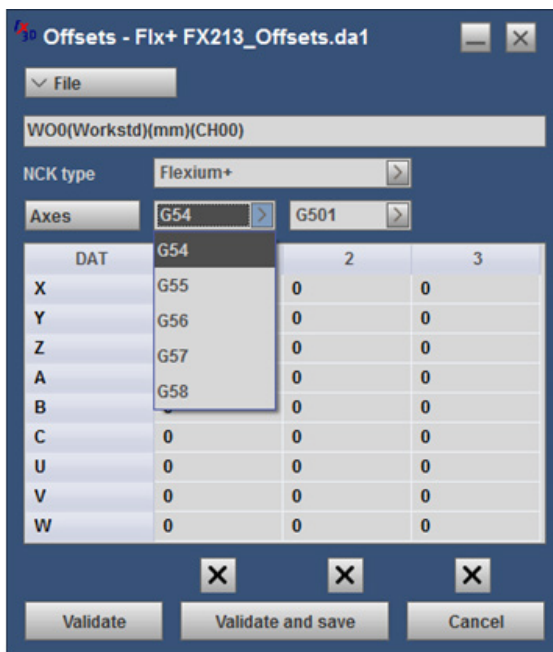
8.3 Operation

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+, 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.

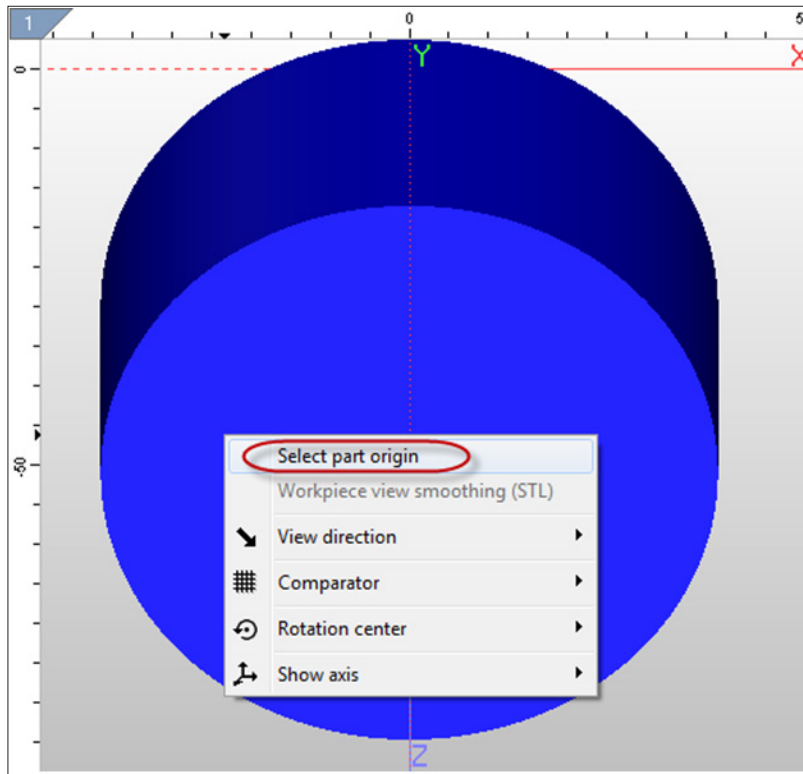


The involved axes can be selected by pressing “Axes” button that opens the following menu:

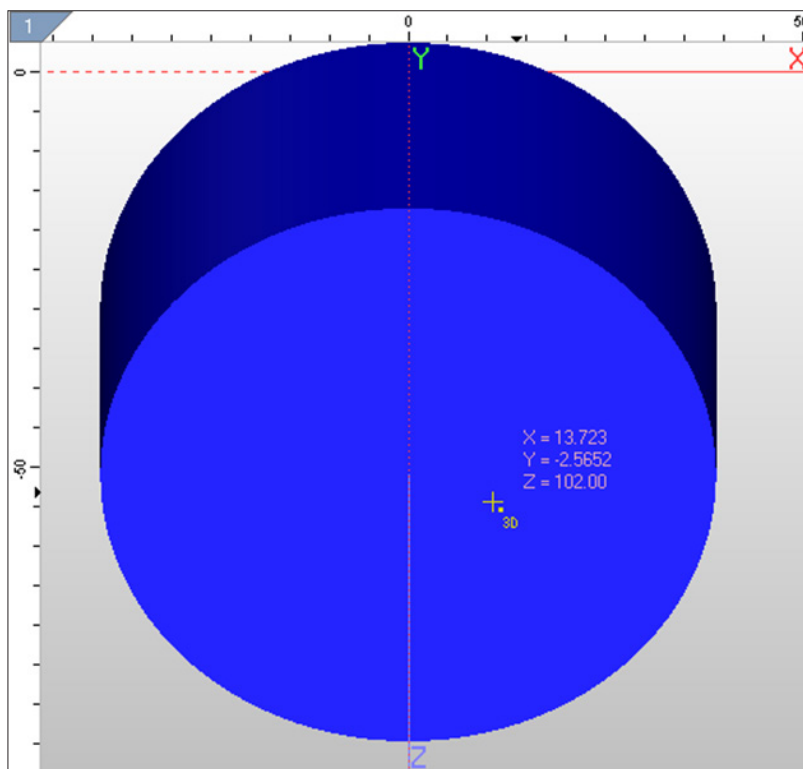


8.4 Select part origin from 3D view

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:

Offsets - Workstd_NC1.da1

File

WO0(Workstd)(mm)(CH00)

NCK type Flexium

DAT	1	2	3
X	13.723	0	0
Y	-2.5652	0	0
Z	102	0	0
A	0	0	0
B	0	0	0
C	0	0	0
U	0	0	0
V	0	0	0
W	0	0	0

Validate Validate and save Cancel

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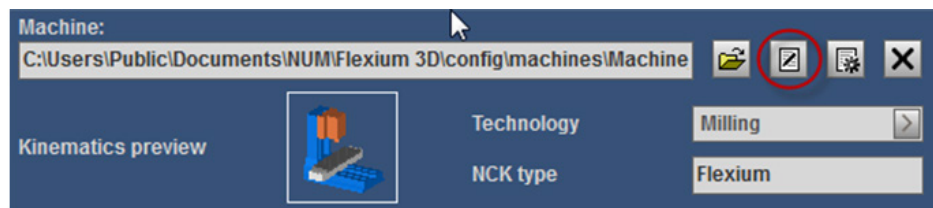
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9 Machine editor

Machine editor window 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.

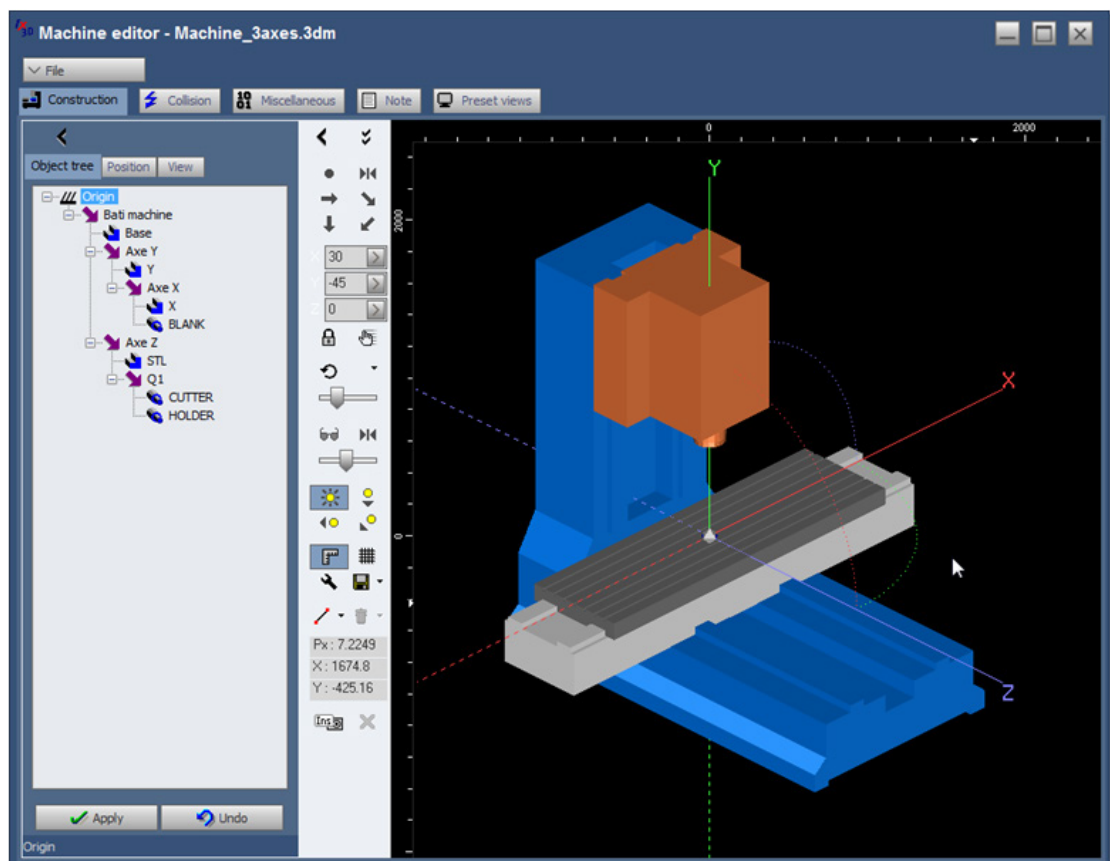
9.1 Open machine editor

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



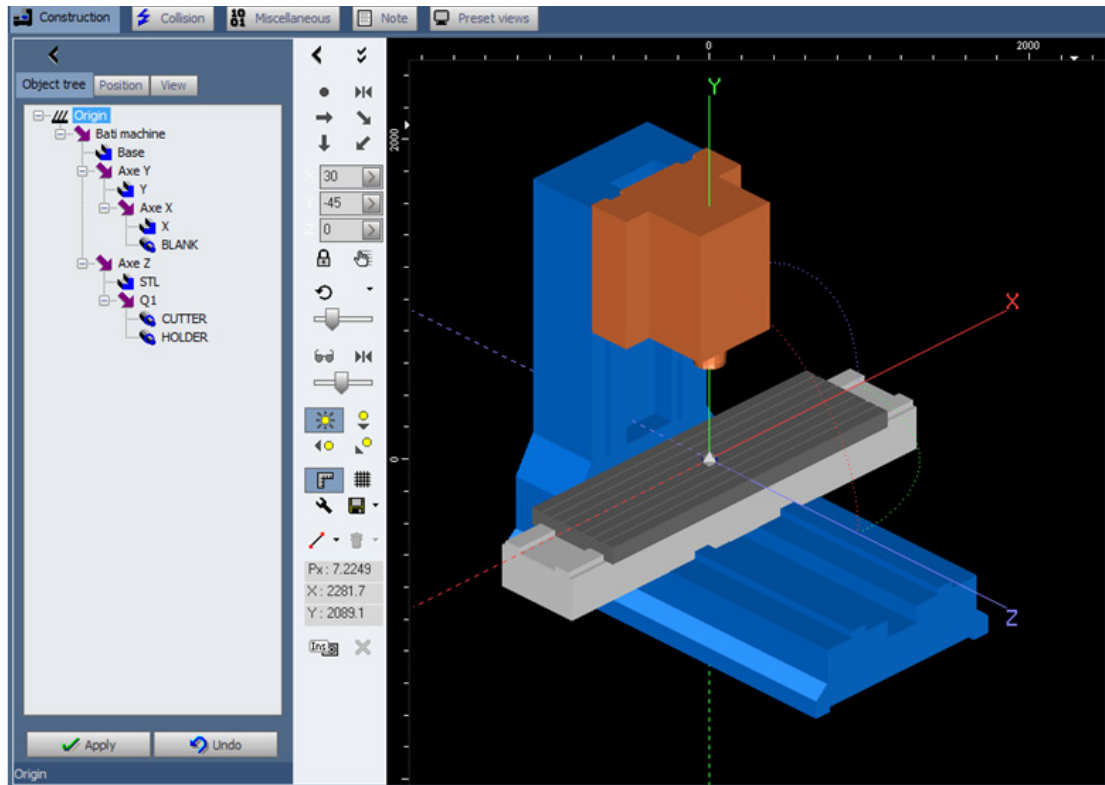
9.2 Page description

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



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.

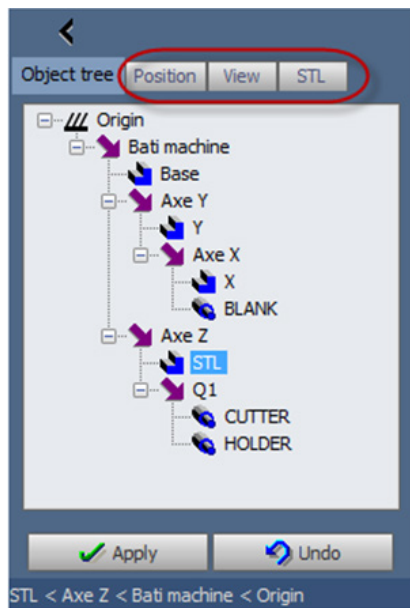
There are some special designated names for objects:

- **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).

To edit one object the object 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.

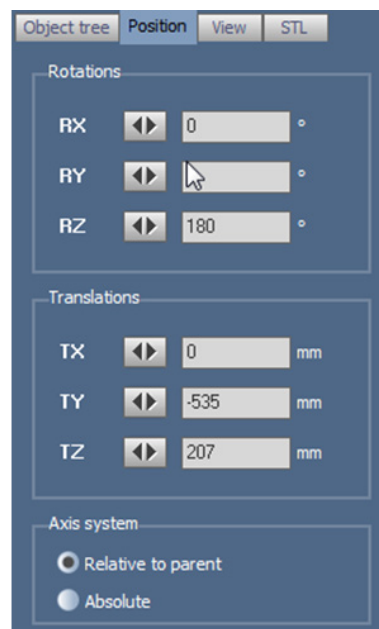
The properties of the current object can be edited on three different pages:

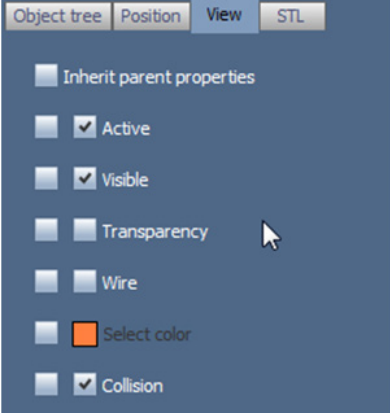
- Position, View, specific attributes.



Position

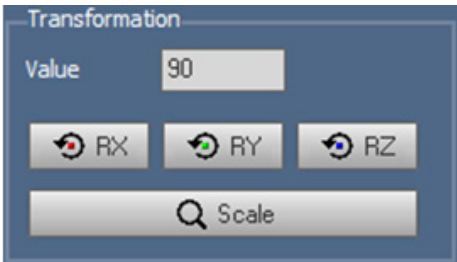
The Position page allows shifting and rotating the object. All child objects will also be moved. By default the relative position of the current object to its parent is defined. The rotations and translations are made with respect to the parent object coordinate system:

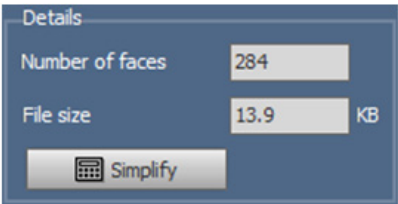
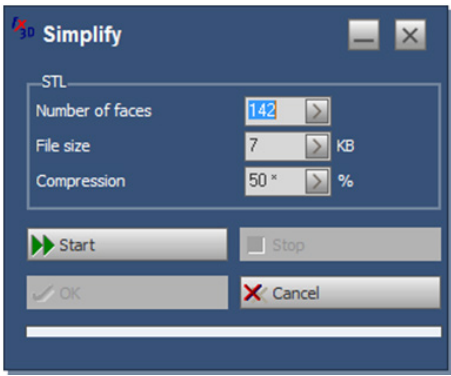


View	The View page only makes sense for physical objects since axis or group objects don't have a visual representation.  On this page some basic attributes can be set: Active: the object is visible and can generate collisions. Visible: the object is visible (an invisible object can still generate collisions). Transparent: the object is transparent. Wire: the object is displayed with wires. Select color: change the object color. Collision: the object can generate collisions.
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. Motion (only for rotation type): shortest, complete or sign direction. NC axis: the CNC axis. 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.

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

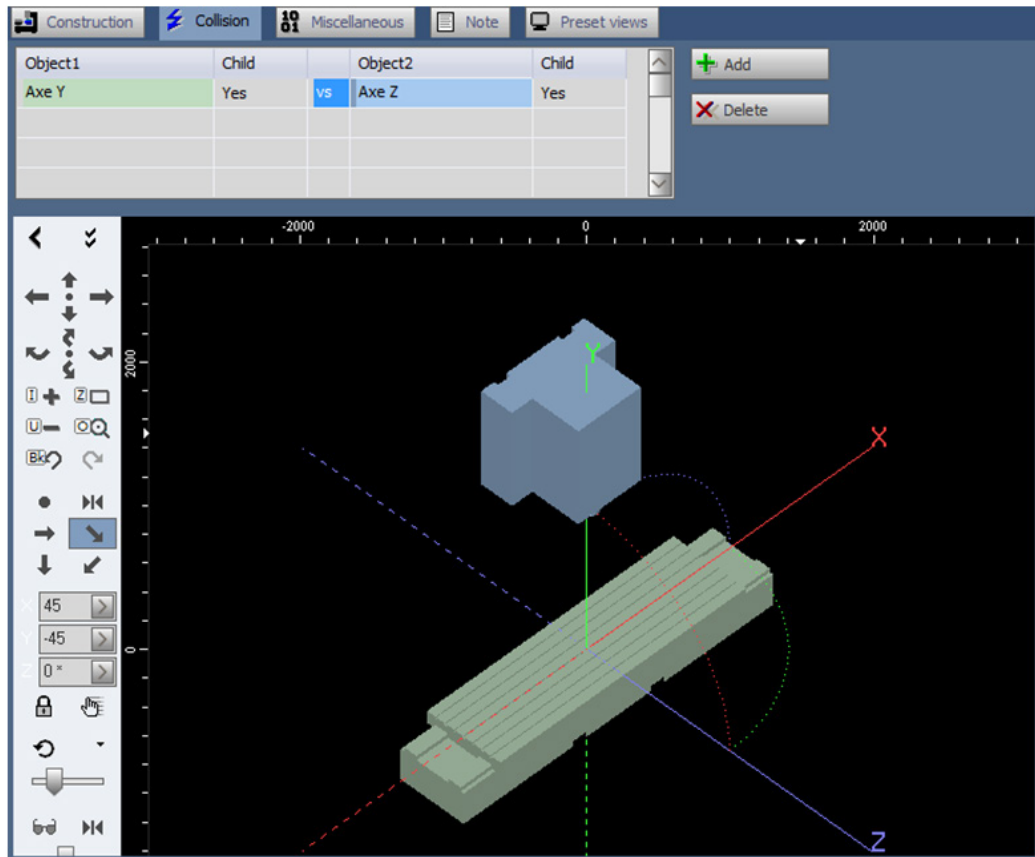


Import	 Import Import STL file.
Properties	 Visualisation coloured or not
Transformation	 Transformation settings for: • rotation around Cartesian axes • scaling factor to increase or decrease.

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 <i>Start</i> button starts the simplification process.

9.2.2 Collision

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



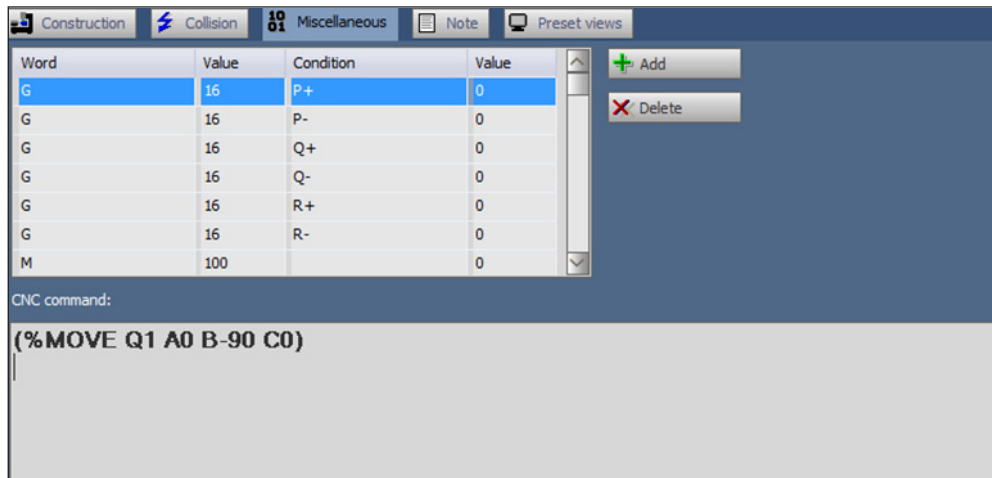
The “Child” property defines whether all children of the object should be tested for collisions.

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 “Axe Y” should be tested for collision against all child objects of “Axe Z”.

The 3D view displays the two object groups.

9.2.3 Miscellaneous

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

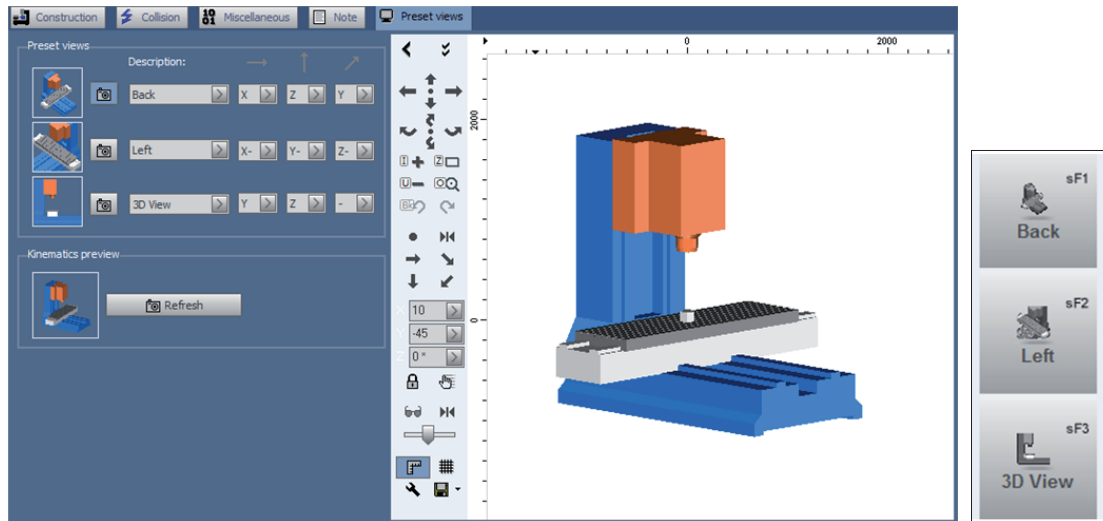


For each G or M command and an optional condition it is possible to define a small subroutine with CNC commands (only a small subset of commands is supported) or special simulation commands (as round brackets NC comment lines).

For the example pictured above, when a "G16 P+" program line is processed, the (%MOVE Q1 A0 B-90 C0) is executed, which will rotate object "Q1" around logical Y axis with minus 90 degrees.

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.

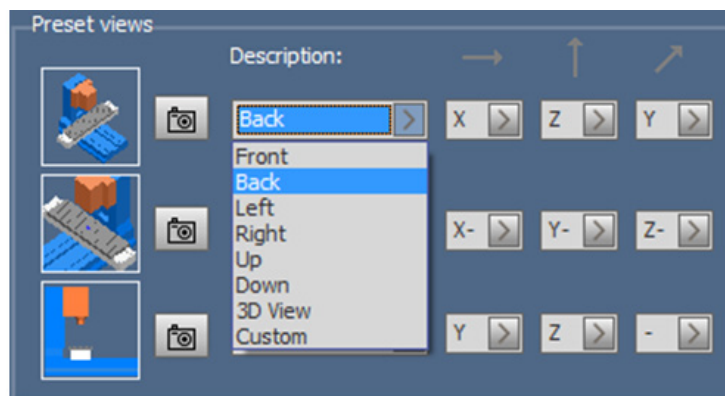


For each preset view button, you can set a 3D view of the machine as the icon and the view description as the tool tip text.

For machine view it is helpful to have predefined views to the kinematics.

Each adjustment of 3D-model 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:



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

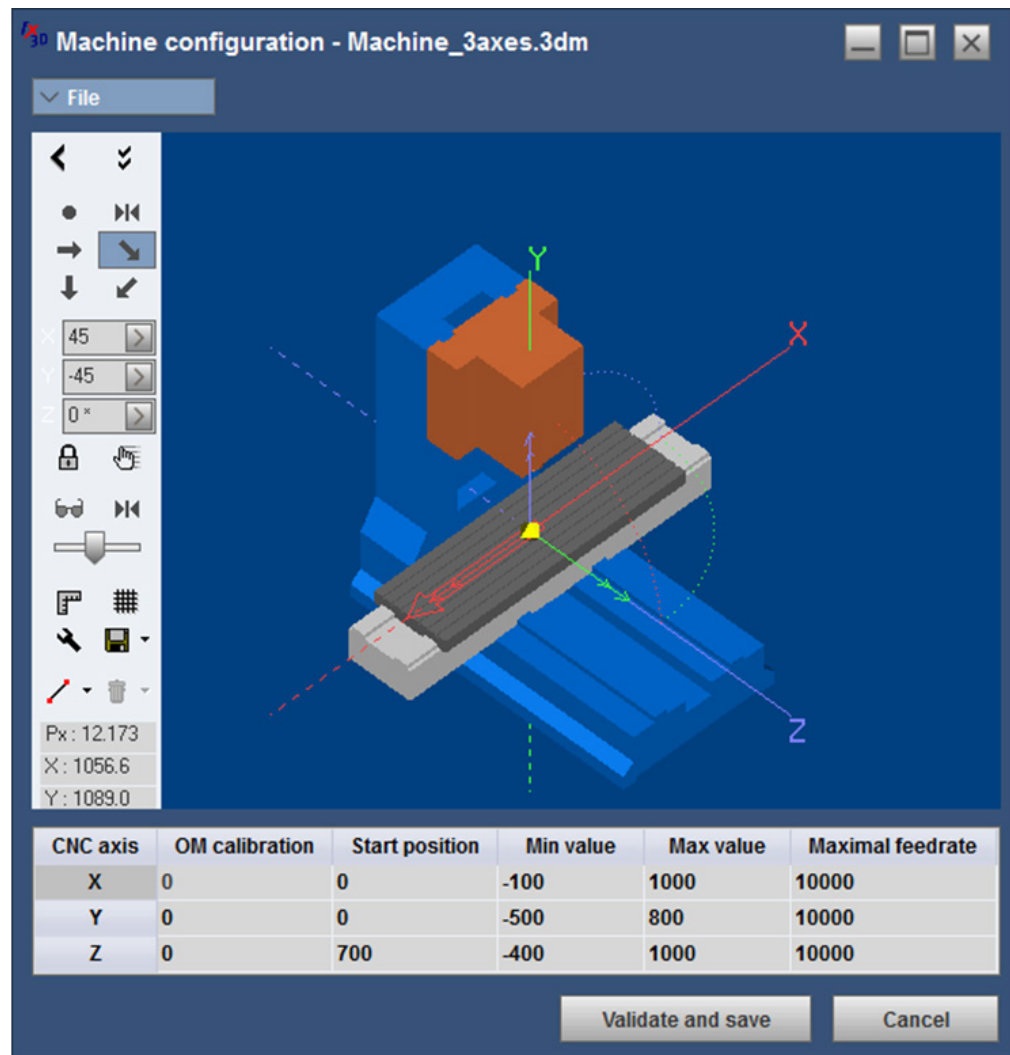
The machine axis configuration window can be opened with a designated button from the Machine section of the CNC configuration window:



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.

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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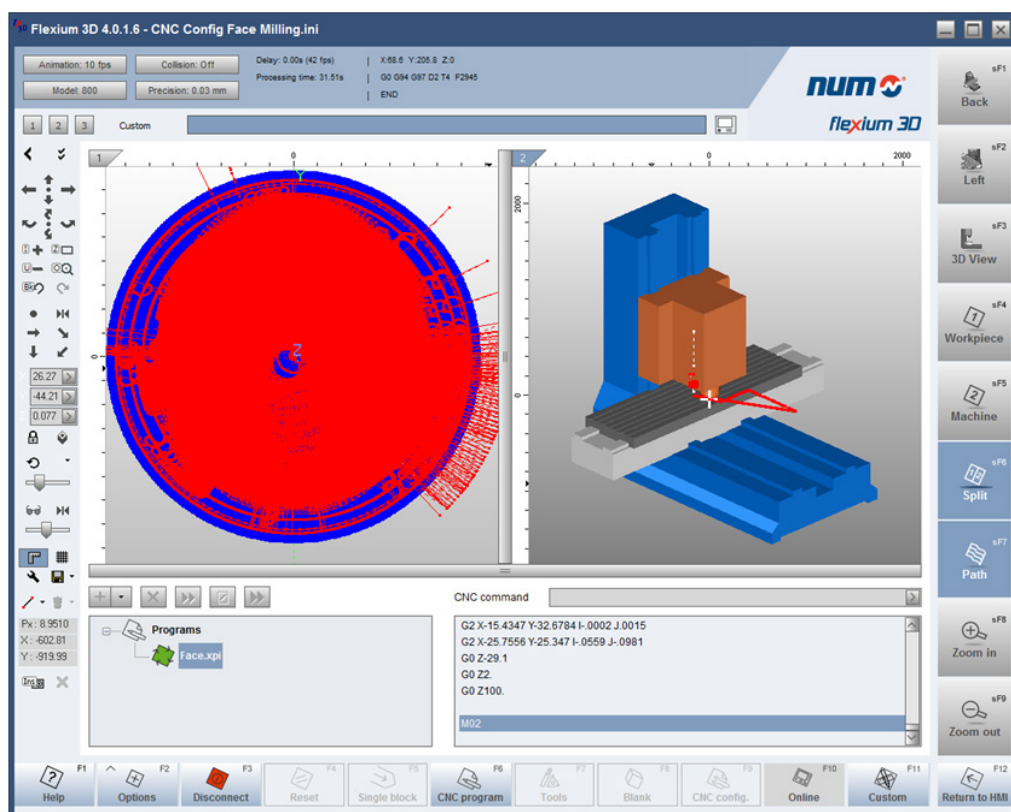
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11 Online simulation

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.



11.2 Operation

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



Online mode off

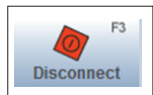


Online mode currently active

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

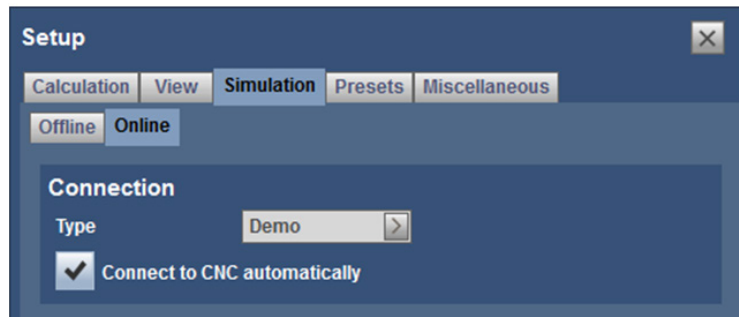


Connect to the CNC and start the online simulation



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.

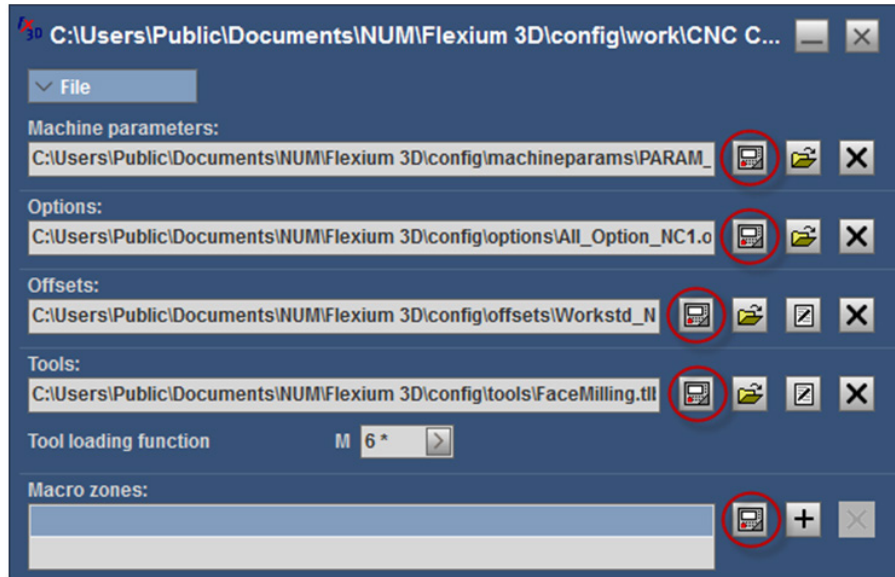
The settings for most simulation parameters are stored separately for offline and online simulation mode.

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

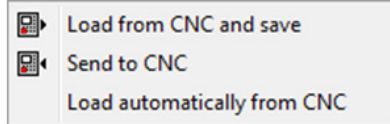
1. Activate the wire path display (**s+ [F8]**).
2. Run the simulation in offline mode (**[F3]**).
3. Open the setup dialog and disable “*Connect to the CNC automatically*” from the Simulation, Online page.
4. Activate “*Always on top*” and “*Time line*” options from the View, Path display page.
5. Switch 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.

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12 Text editor

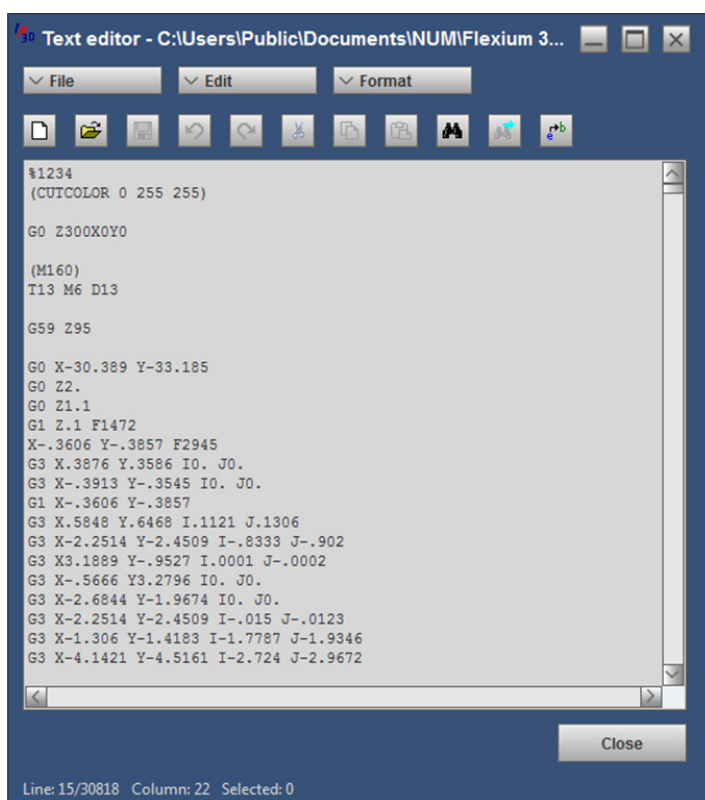
12.1 Text editor view

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:

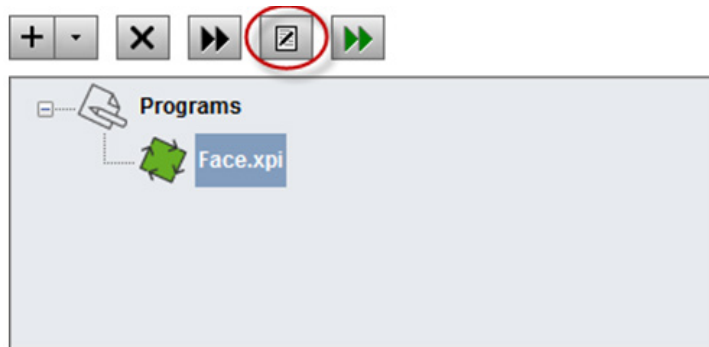
- Create new file
- Open file
- Save current file
- Undo, redo (one step only)
- Cut, copy, paste
- Find, find next
- Go to line number (available only through keyboard shortcut **Ctrl** + **G**)
- Change display font.

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.

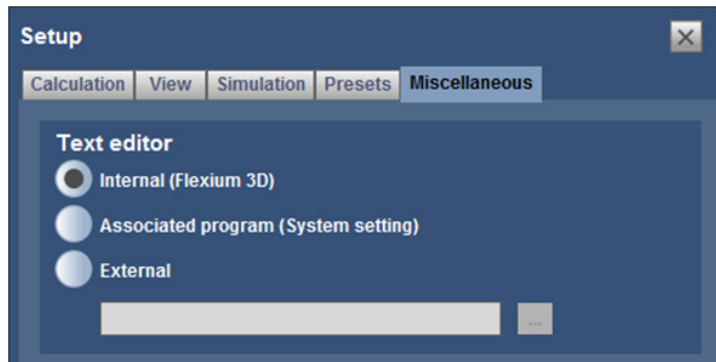


By design it has tolerable loading/saving times for files smaller than 10MB. There is a file size limit of 64KB when running in Windows 95/98/Me.

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*:



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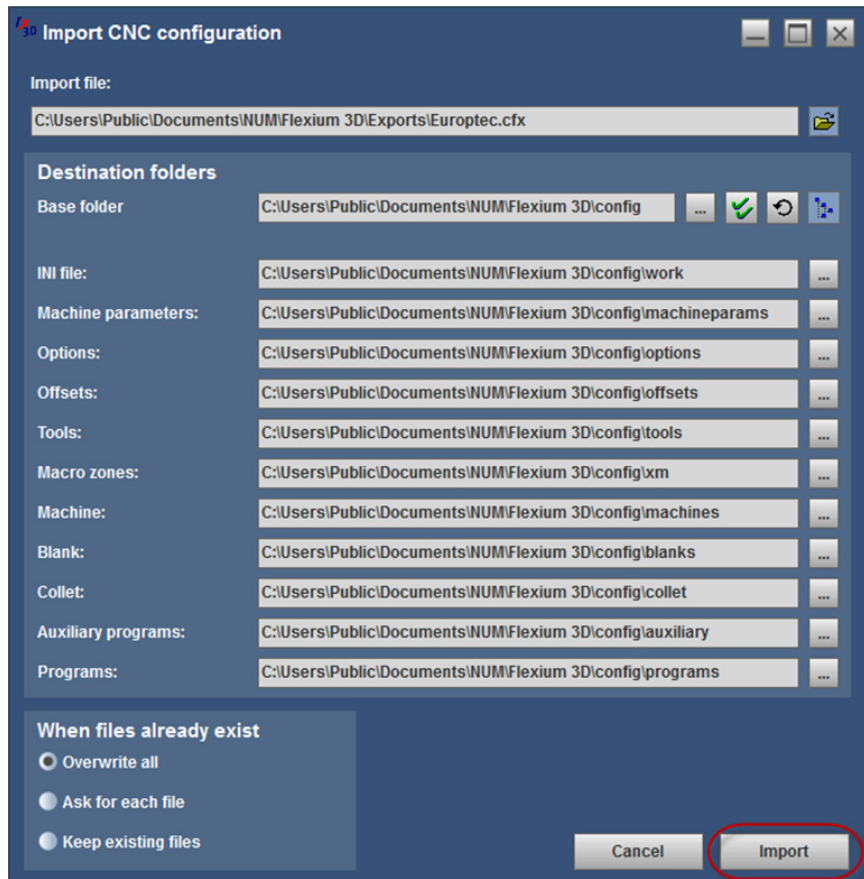
13 Import / Export CNC configuration

13.1 Import a CNC configuration

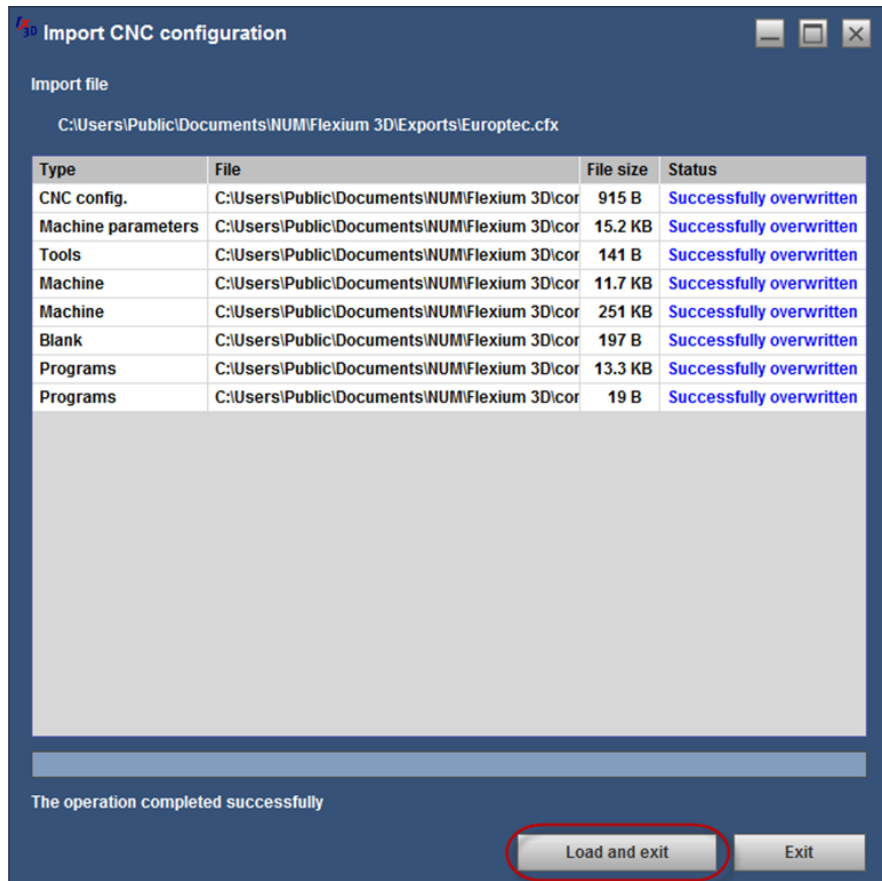
13.1.1 Command description

Unpack an exported file 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.



13.1.2 Operating steps

1	Select the archive file containing the CNC configuration to be imported (<i>Import file</i>).
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 “ <i>Status</i> ” 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 “ <i>Ask for each file</i> ” 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 “ <i>Load and exit</i> ” button*.

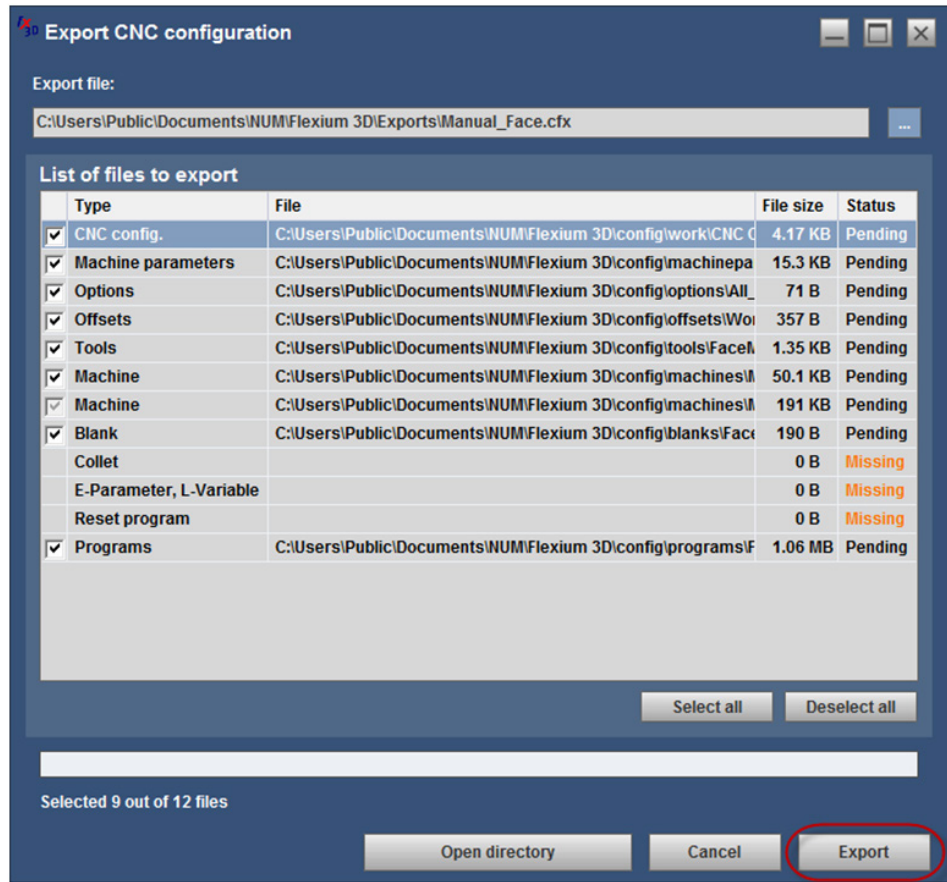
13.1.3 Important notes

Browse from base folder	When the “ <i>Browse from base folder</i> ” 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 (<i>Export file</i>).
2	Select one or multiple configuration files to be exported. Result of selection can be seen in the “ <i>Status</i> ” 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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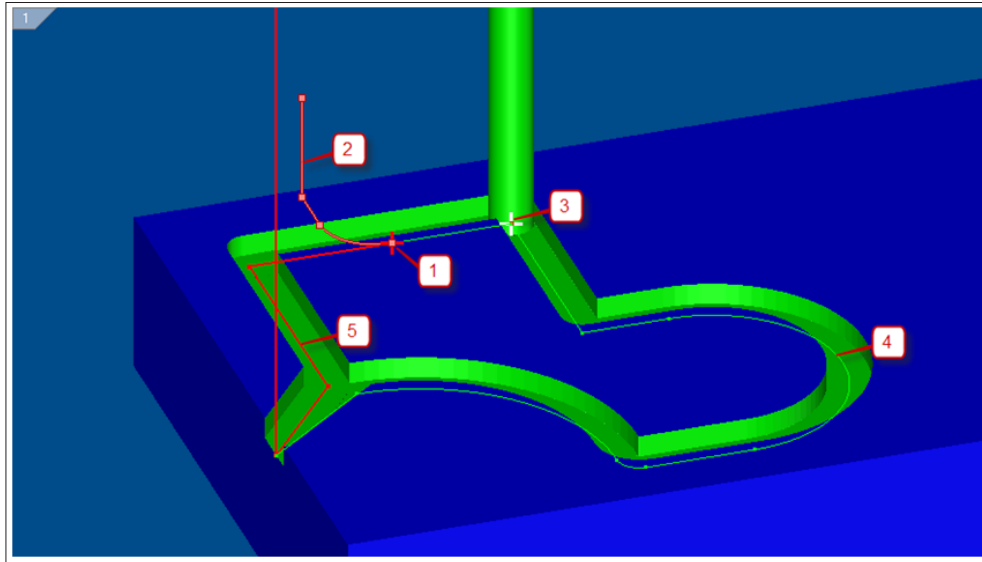
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14 Graphical supported "Search mode"

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.

Search mode

Path

Position mm

NC block

CNC program .

Line

Passes

Line progress %

NC block

Enabled channels

Approach path

Correction side (Left)

Plane

Distance mm

Radius mm

Angle °

Height mm

Feedrate mm/min

Subroutine

Code: ;10 .0 H125 ;1 EP32 G3 X-5 Y5 EX-5 EY10 I0 J5 EO100 ER10 / .2 .3 .4

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.

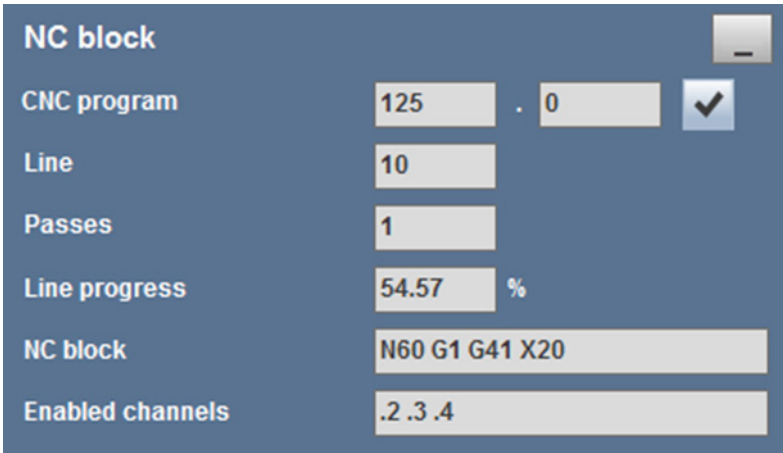
The resume point can also be defined by the path distance to the point. This can be entered as a value inside the "Position" field or it can be moved around using the track bar.



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.
	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.

Approach path

1 2 3

Correction side L R A (Left)

Plane G17 (XY)

Distance $\leftarrow \rightarrow +$ 10 * $\rightarrow$ mm

Radius $\leftarrow \rightarrow +$ 5 * $\rightarrow$ mm

Angle $\leftarrow \rightarrow +$ 90 * $\rightarrow$ °

Height $\leftarrow \rightarrow +$ 10 * $\rightarrow$ mm

Feedrate $\leftarrow \rightarrow +$ 0 * $\rightarrow$ mm/min

Subroutine 100 $\rightarrow$

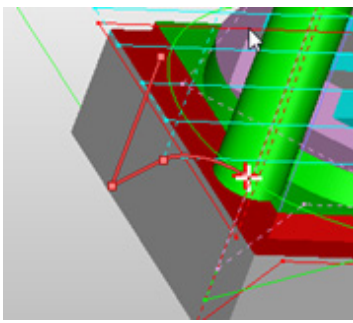
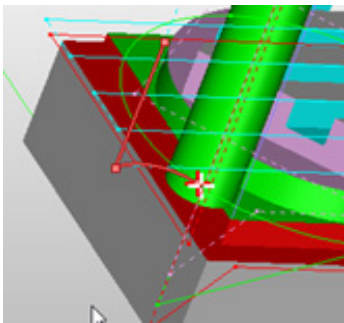
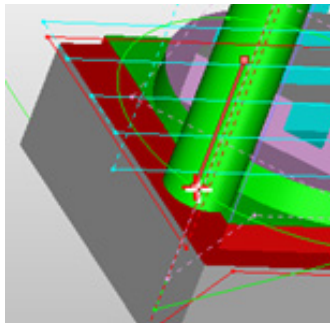
X [Red Stop] + -

User interface description:

Approach path	The user can select one of three predefined approach paths: 1 2 3 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: L R A 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 sign.
	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  button is pressed, the Online simulation is automatically started and the Search command is transferred to the NCK.

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 SINGL 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 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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