

T-FLEX[®] CAM 16

User Manual



TOP
SYSTEMS

T-FLEX CAM

User Manual

This Software and Related Documentation are proprietary to Top Systems. Any copying of this documentation, except as permitted in the applicable license agreement, is expressly prohibited.

The information contained in this document is subject to change without notice and should not be construed as a commitment by Top Systems who assume no responsibility for any errors or omissions that may appear in this documentation.

Trademarks:

Trademarks T-FLEX Parametric CAD, T-FLEX Parametric Pro, T-FLEX CAD, T-FLEX CAD 3D, T-FLEX CAM, T-FLEX NC TRACER are property of Top Systems Ltd.

Parasolid is a trademark of Siemens PLM Software.

All names of software and hardware products, company names and other trademarks or registered trademarks mentioned in this publication are property of their respective owners.

CONTENTS

Contents.....	3
About T-FLEX CAM.....	5
General Information.....	9
Standard Notations Accepted in System Description	11
Customizing CAM system.....	12
Toolpaths for 2D, 2-1/2D and 4D Machining (2D Version).....	14
Toolpaths Properties.....	14
Wire-cut Machining	15
Laser-cut Machining	23
Turning Machining	30
Drilling.....	83
Milling (2-2.5D)	87
Engraving.....	106
Punching process.....	111
Defining the Path Start Point	117
Toolpaths for 3D, 4D and 5D Machining (3D Version)	118
3D Milling	118
Bounded 3D Milling.....	126
5D Milling	154
Bounded 5D Milling.....	161
5D Drilling.....	167
Cam Milling (4D Machining).....	172
Technological Toolpaths.....	175
Drilling, Milling and Engraving	175
Turning	176
Wire Cutting and Laser Cutting	178
Punching	180
Machining Cycles	182
Machining Cycles Editor	182
Using Custom Machining Cycles	185
Predefined Machining Cycles	191

Machining Manager	227
Relative Coordinate System	237
Relative Coordinate System For 2D, 2.5D and 4D Machining.....	237
Relative Coordinate System For 3D, 4D and 5D Machining (3D Version)	238
NC program Generation	240
Machining Simulator	242
Running the Machining Simulator	242
Setting Up the Machining Simulator (2D version)	243
Setting Up the Machining Simulator (3D version)	244
Tool Editor	247
Tool Setup Chart	252
Postprocessors	254
Postprocessor Editor	254
External Postprocessors	284
Additional system functions	301
T-FLEX CAM Variables	301
T-FLEX CAM Fragments (*.grb file)	302
Data Exchange in the GTL Format.....	302
Examples of Use	304
Wire Cutting and Laser Cutting	304
Turning	323
Drilling and 2.5D Milling.....	332
Engraving.....	346
Punching	350
3D Milling	356
Bounded Machining. 3D Milling.....	359
Bounded Machining. 5D-milling	364
Position Drilling	368
Cam Milling (4D Machining).....	372

ABOUT T-FLEX CAM

For developing software for CNC centers, the company “Top Systems” offers the system T-FLEX CAM. The offered system is fully integrated with the T-FLEX CAD and differs favorably from its counterparts by the presence of through parameterization. This implies that, while modifying a part’s drawing parametrically in the system T-FLEX CAD, the developer can automatically see changes in a control sequence. Among the advantages of the T-FLEX CAM, which explain a successful development of the product on the CAD/CAM market, we can mention the modular structure of the system, a through parameterization, presence of specialized modules for working with tools and postprocessors. Five main advantages due to which the system T-FLEX CAM is an ideal solution for many manufacturers are given as follows:

- high functionality
- simple to learn
- excellent technical support (system is easily adapted to clients’ equipment, assistance for end users is provided)
- free of charge updates

The system T-FLEX CAM is supplied in two versions: T-FLEX CAM 2D and T-FLEX CAM 3D. T-FLEX CAM 2D consists of base module, module of wire-cut machining, turning machining, drilling module, module of laser machining, module of 2.5D milling, and module of punching. The system T-FLEX CAM 3D consists of base module, module of 3D and 5D milling.

The base module of the system T-FLEX CAM includes:

- mathematical core, integrated with PARASOLID;
- editor for developing tools used for machining a specific part or for creating tools database;
- module for generating postprocessors allowing the user to create his own postprocessors using tabular settings, macros, and direct programming;
- postprocessors library which contains about 400 ready-to-use postprocessors; among those are:
 - AGIE 100/200, CHARLIE, 2M43, 2M43—55 — wire-cut machining;
 - COMETA — laser machining;
 - 1A734, HLQ31, 2P22, FANUC 0/00/0M/16 — turning machining;
 - 2C150, 2C42—61(65), 65A80, BRADLEY, FANUC — drilling;
 - 2C150, 2C42—61(65), CNC 600, FIDIA-CNC20, FIDIA-CNC30, OLIVETTI, POWER AUTOMATION, SINUMERIK 820D, SINUMERIK 840D, SINUMERIK 840A, BRADLEY — milling;
 - and many other postprocessors;
- machining simulator showing the machining process from a generated control sequence without actual material removal

Wire-cut machining serves for obtaining open-ended holes as well as holes with arbitrary contour, and, also, narrow straight or curvilinear slits in blanks for manufacturing moulds, punches, profiled cutting tools, stencils and other products. For that, wire-cutting by a non-profiled wire electrode is used.

In the system T-FLEX CAD it is possible to design the following types of wire-cut machining:

- single contour cutting (2D machining);
- angular cutting (2.5D machining);
- double contour cutting (4D cutting);
- selection of material along the spiral;
- engraving a text.

For generating control sequences, the user can use postprocessors found in the postprocessors library or independently create the postprocessor, required for machining, with the use of module for generating postprocessors of the system T-FLEX CAM.

Laser machining is based on material removal by exerting on it concentrated light rays. On a machined surface, due to high temperatures, material evaporation takes place at a place of contact of light with the material. This method is used for obtaining holes of simple or profiled forms when machining diamonds, rubies, ceramics, stiff alloys and other intractable materials. Recently laser machining has been widely used for figured cutting, for example, for manufacturing parquet, decorative bars, etc.

In the system T-FLEX CAM, it is possible to design the following types of laser machining:

- single contour cutting (2D machining);
- angular cutting (2.5D machining);
- double cutting (4D machining)
- engraving a text.

For generating control sequences, the user can use postprocessors supplied with the postprocessors library, for example COMETA, TRUMPF, or independently create required for machining postprocessor by using the module for generating postprocessors of the system T-FLEX CAM.

It is important to note that by using machining types described above, it is possible to create sequences of machining by plasma and hydrojet cutting.

Turning (2D machining) — turning is machining of outer and inner, cylindrical and conic, profiled and face surfaces. Due to specific configuration of machined objects, which are the bodies of revolution, the design of machining process is reduced to a solution of problems on a plane, in the axial section.

T-FLEX CAM offers the user a set of universal cycles for machining by turning, for example, the cycles of removing allowances by cut-off cutter or pass cutter. In addition, the system supports the use of specialized cycles for turning machining station, parameters of which are set in accordance with CNC's like FANUC, SINUMERIK, and 2P22. Among them are: repeated removal of allowance, cone finish turning, axial drilling, deep axial drilling, threading, groove milling, etc.

For generating control sequences it is possible to use postprocessors supplied with the postprocessors library or independently create required for machining postprocessor by using the module of the T-FLEX CAM for generating postprocessors.

Drilling (2.5D machining) — this includes drilling, boring, countersinking and threading in holes. In case of using CNC centers, for this type of machining, marking and jigs are not used. With equipment of this type, the complex drilling-boring machining of blanks of different configurations and with various degree of accuracy becomes possible.

For drilling "slanted" channels, and also for drilling holes on different planes of the machined part, including the planes located at an angle with respect to each other, the module of 5D drilling can be used.

T-FLEX CAM offers the user a whole set of specialized machining cycles which include different types of deep drilling, drilling with traverse, repeated drilling, etc. Parameters of these cycles are set in accordance with the CNC's like: OLIVETTI, BRADLEY, POWER AUTOMATION, 2C42—61(65). However, the user can independently change these settings.

For generating control sequences it is possible to use postprocessors supplied in the postprocessors library, or independently create required for machining postprocessor by using the module of the T-FLEX CAM system for postprocessor generation.

Milling — this is the most widely used type of machining for which CNC centers are used.

The system T-FLEX CAM offers the capability to design the machining process and generate control sequences for the following types of milling:

- **2.5D milling** — is used for machining cylindrical and linear surfaces (contours) with arbitrary guides, which are either parallel to the axis of the tool or make a constant angle with the tool axis in a normal section. In addition, for this type of milling, the user can perform «pencil» milling and also carry out milling of "pockets", "manholes", "islands" and other specific technological elements. What particularly important is that the technologist-programmer is not restricted in tools selection: with the help of T-FLEX CAM it is possible to develop any desired tool which can be subsequently used when preparing a control sequence.
- **3D milling** — is suited for volumetric machining of arbitrary surfaces and solid bodies. In addition, the T-FLEX CAM system offers the capability of zone machining of solid bodies, sections, "manholes", cylindrical surfaces, capability of pencil machining and edge selection.
- **5D milling** — is suited for machining surfaces by the face or lateral side of the tool in cases when the use of standard volumetric machining is impossible or ineffective. Also, as in the case of 3D-milling, there is a possibility to use zone machining of solid bodies, sections, «manholes» and edge selection.

For generating control sequences it is possible to use postprocessors supplied with the postprocessors library or create independently a postprocessor, required for machining, by using the postprocessor generation module of the system T-FLEX CAM.

Punching – process of plastic deformation of the metal associated with changes in the form and sizes of the body. The **T-FLEX CAM** system offers capability of construction of contour punching with different types of punched holes.

To generate control sequences, it is possible to use postprocessors supplied in the library of postprocessors or independently create the postprocessor, required for machining, by using the module of generation of postprocessors of T-FLEX CAM system.

GENERAL INFORMATION

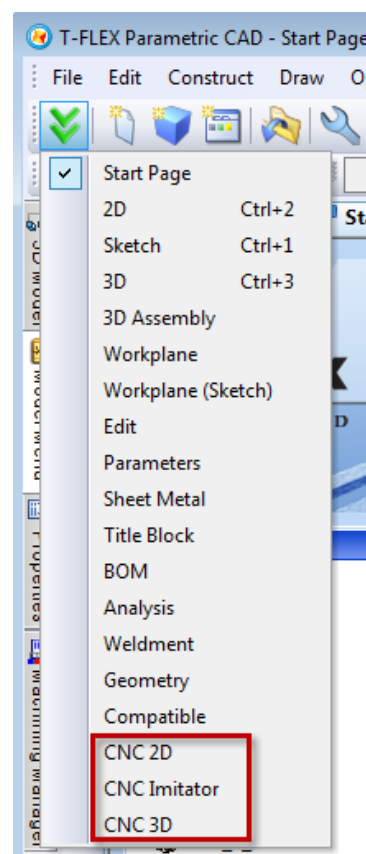
This manual is intended as a handy reference for mastering a new software product of the T-FLEX family. This suite of solutions is a specialized system for automatic generation of control sequences for various CNC centers and various types of machining. T-FLEX CAM system has modular structure, offering use of selected modules for a particular type of machining of interest to the client, plus the base module. A T-FLEX CAD installation is also required, since the two make an automated integrated CAD/CAM system. T-FLEX CAM software is available as the 2D version (for 2D, 2.5D and 4D machining), or the 3D version (normally, for 2D, 2.5D, 4D, 3D and 5D machining). The present manual introduces the T-FLEX CAM 2D and T-FLEX CAM 3D systems in what follows.

The last chapter of this manual contains description of specific examples for each type of machining provided with the system and described in the respective chapters. Note that the order of steps for creating toolpaths described in this manual is not the only one possible; therefore, you can use your own procedures for defining machining toolpaths and calculating control sequences.

After starting the T-FLEX CAM system installed on your computer, the T-FLEX CAD application window appears on the screen, providing environment for all design steps from drafting the part to automatic generation of the control sequence. The text menu of T-FLEX CAD will be expanded due to the block containing T-FLEX CAM commands – «CNC».

In addition to that, in the list of main toolbar states the new modes: «CNC 2D», «CNC Simulation» and «CNC 3D» will appear.

Each of these options will put the main toolbar to the corresponding mode with a collection of appropriate command buttons. The "CNC 2D" and "CNC 3D" modes contain the set of base commands for working with the system and, also, the set of commands for creating toolpaths of 2D and 3D machining of various types.



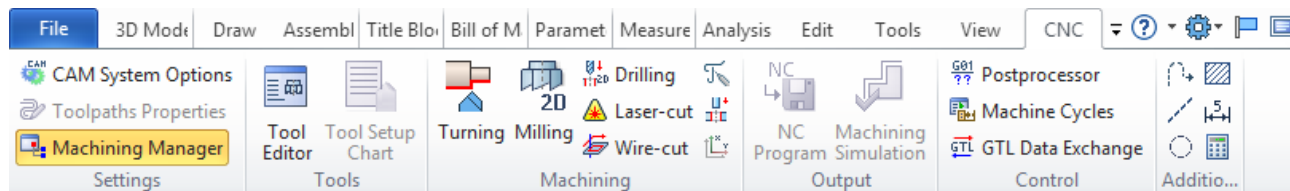
The «CNC Simulation» mode can be directly opened or closed via the list of main toolbar states, however, upon the start of the mode of control sequence simulation, the main toolbar will automatically

be switched to the «CNC Simulation» mode. In the standard mode (T-FLEX CAD) and upon creation of toolpaths, the buttons of the main toolbar in the «CNC Simulation» state are deactivated.

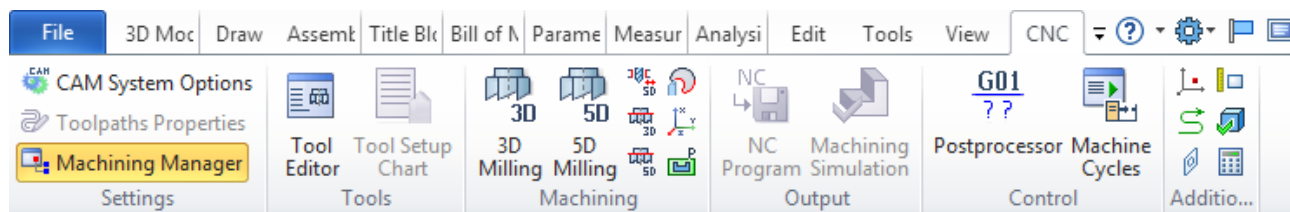
The mode of the main toolbar «CNC Simulation»



The mode of the main toolbar «CNC 2D»



The mode of the main toolbar «CNC 3D»



STANDARD NOTATIONS ACCEPTED IN SYSTEM DESCRIPTION


We use the following standard notations throughout this book:

<Enter>, <Esc>, etc. - denotes a key on the keyboard.

[OK], [Yes], etc. - denotes a graphic button in a dialog box.

 - Left mouse button.

 - Right mouse button.

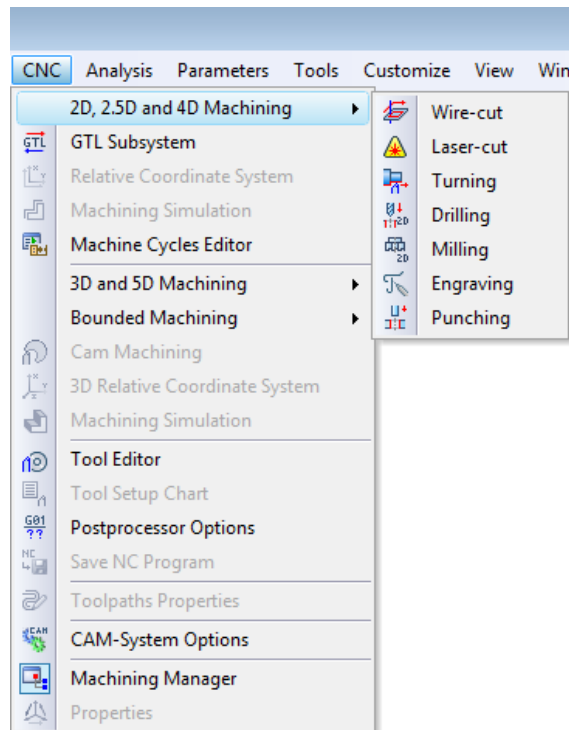
 - Double-clicking the left mouse button.

 - an icon on a toolbar.


“CNC>Machining Manager”, etc. - means selection of the entry “CNC” in the textual menu, with the subsequent selection of the pull-down menu item “Machining Manager”.

“CNC>3D and 5D machining>3D Milling” - means selection of the textual menu entry “CNC”, followed by selecting the pull-down menu item “3D and 5D machining”, and then selecting the fly-out menu item “3D Milling”.

Command calls in T-FLEX CAM are done by selections in the textual menu. In addition, the icons of the frequently used commands are placed on the toolbar. For convenience, as you move the pointer over any icon, a tooltip appears with the name of the respective command.




Some frequently used commands can also be accessed by pressing key combinations indicated in the textual menu next to the command name string. For example, <Ctrl> <O> - opens a file.

To select an icon, to press an icon, to select or focus on an input box, or to press a button means the following in this description: move the pointer over an icon, an input box, or a button and press the left mouse button .

To point at an element, to point at an icon, to point at a button means in this description to move the pointer over an element, an icon, or a button.

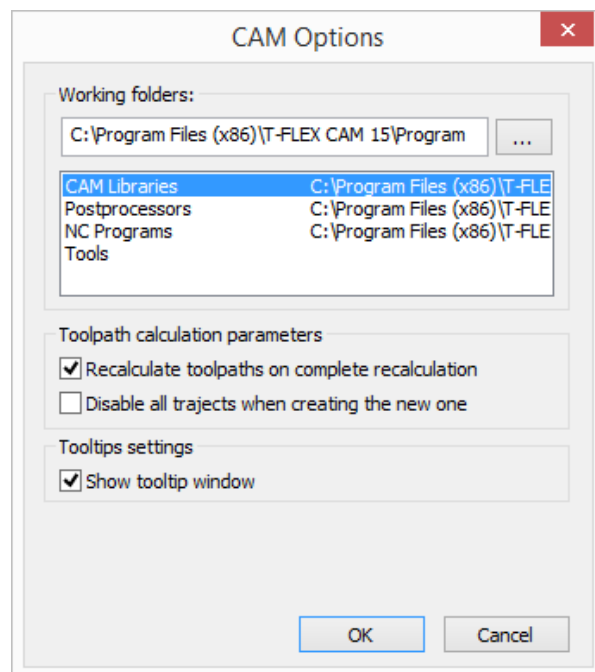
CUSTOMIZING CAM SYSTEM

The T-FLEX CAM system customization is done by the command:

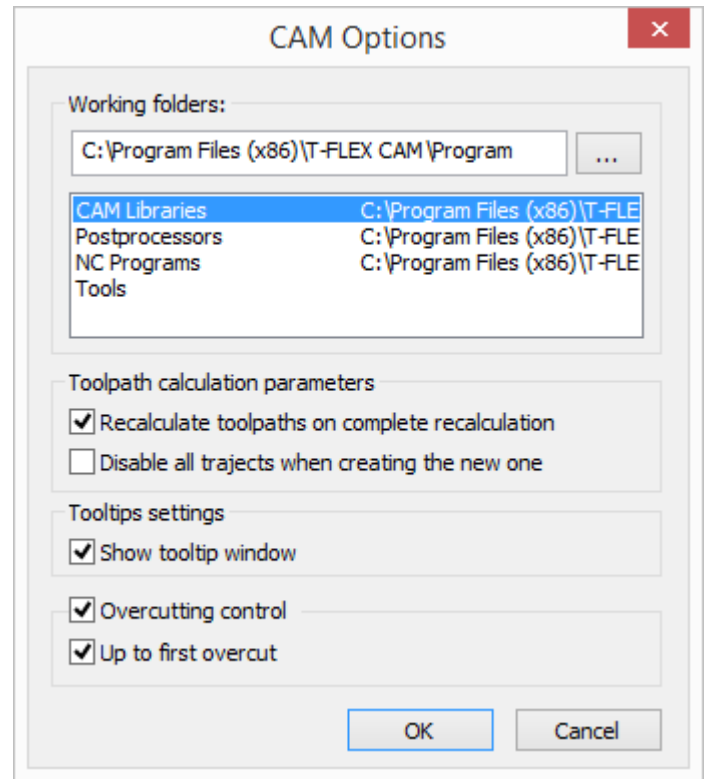
Icon	Ribbon
	CNC → Settings → CAM System Options
Keyboard	Textual Menu
	CNC>CAM System Options

The dialog box corresponding to the above command allows you to modify the default folders for searching postprocessor files (both those that are shipped with the T-FLEX CAM installation and the user-defined ones), predefined control sequences and specialized libraries of technological process. For the 2D module, define the paths to folders with 2D postprocessors and 2D NC programs. If the 3D module is installed later, then the 2D folders are automatically substituted by folders with 3D postprocessors and 3D NC programs.

For user's convenience, the settings are stored separately for 2D and 3D windows of T-FLEX.





For 3D window, the flag for control of overcutting a part with a tool and of precision of overcuts control is added to the dialog of parameters of CAM system.



Normally, all mentioned components are installed by default in the following respective folders:

- ...*T-FLEX CAM\Library of postprocessors* (predefined postprocessors);
- ...*T-FLEX CAM\Postprocessors\ Postprocessors 2D* (generic postprocessors for 2D machining);
- ...*T-FLEX CAM\Postprocessors\Postprocessors 3D* (generic postprocessors for 3D machining);
- ...*T-FLEX CAM\CNC Programs\NC Programs 2D* (control sequences for 2D machining);
- ...*T-FLEX CAM\CNC Programs\NC Programs 3D* (control sequences for 3D machining);
- ...*T-FLEX CAM\Program* (specialized libraries of technological processes).
- ...*T-FLEX CAM\Tools* (or any other folder for created tools).

To modify those directories, press the  button. In the standard tree-browsing window that appears, specify the new folder for the component being defined. To navigate through the list of components being defined, use  or keyboard keys <↑>, <↓>. As you switch the focus from one item to another, the respective path is updated in the input box in the upper portion of the dialog.

If the “Recalculate toolpaths on complete recalculation” check box is cleared, the system dynamically recalculates the machining toolpath upon any changes in the drawing geometry. Otherwise, if set, automatic toolpaths recalculation will not occur. To save the parameters modifications, press the [OK] button.


Before beginning creation of the machining toolpath and, later, a particular control sequence, we recommend setting up a postprocessor with the desired type of machining and creating the tool file (if it does not exist yet) for the machining process. First, let us review the task of designing a tool.

TOOLPATHS FOR 2D, 2-1/2D AND 4D MACHINING (2D VERSION)

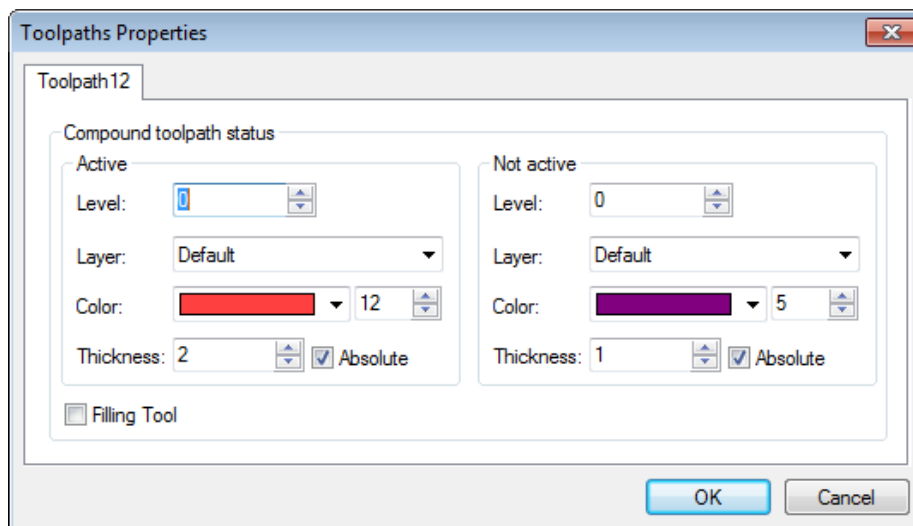
Once you have created the drawing of the part and performed the above-described settings of T-FLEX CAM system, you can proceed with generating the toolpaths and the control sequence.

TOOLPATHS PROPERTIES

Setting properties of toolpaths is done by the command:

Icon	Ribbon
	CNC → Settings → Toolpaths properties
Keyboard	Textual Menu
	CNC>Toolpaths properties


All settings are defined for the compound toolpath for various types of machining. The user can specify the properties for two groups of elements – active and nonactive toolpaths. It is possible to customize the thickness of displayed elements of toolpaths, their color, layer and the level of location of toolpaths on the drawing.












Filling Tool. This serves to fill the tool section contour with the color used for the toolpath (for punching toolpaths). Other parameters are described in the respective Help and Manual sections of T-FLEX CAD.

WIRE-CUT MACHINING

To make a toolpath and a control sequence for wire-cut machining, use the command:




Icon	Ribbon
	CNC → Machining → Wire-cut machining
Keyboard	Textual Menu
	CNC>2D, 2.5D and 4D machining>Wire-cut machining

Upon calling the command, the following options will appear in the automenu:





	<End>	Finish editing
	<T>	Technological toolpath
	<O>	Single contour cut
	<U>	Angular cut
	<D>	Double contour cut
	<R>	Select hatch for metal removal on spiral
	<A>	Engraving
	<Q>	Select point for tool moving
	<Esc>	Exit command



Toolpath Creation


Once done with setting up the default machining parameters, the user needs to select one of the three types of wire-cutting machining:

- Single contour cutting (2D machining) 
- Angle cutting (2-1/2D machining) 
- Double contour cutting (4D machining) 


Upon selecting any of the listed wire-cut machining types, the following options will appear in the automenu:

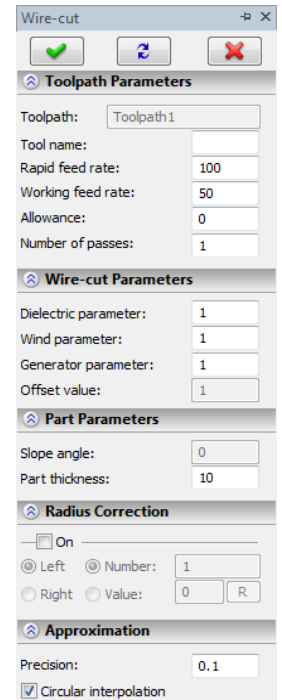
	<End>	Finish editing
	<P>	Toolpath properties
	<1>	Hatch selection filter
	<2>	Path selection filter



	<3>	Change initial point of toolpath
	<Esc>	Exit command

After the command is invoked, the "Wire-Cut" dialog window will appear on the screen. Parameters set up in this window will be enabled **by default** for all newly created wire-cutting toolpaths. When a specific toolpath is created, these parameters can be changed. The user can change them as desired and save these changes by pressing the  button.

Detailed description of all parameters is provided in the "Toolpath parameters" section below.

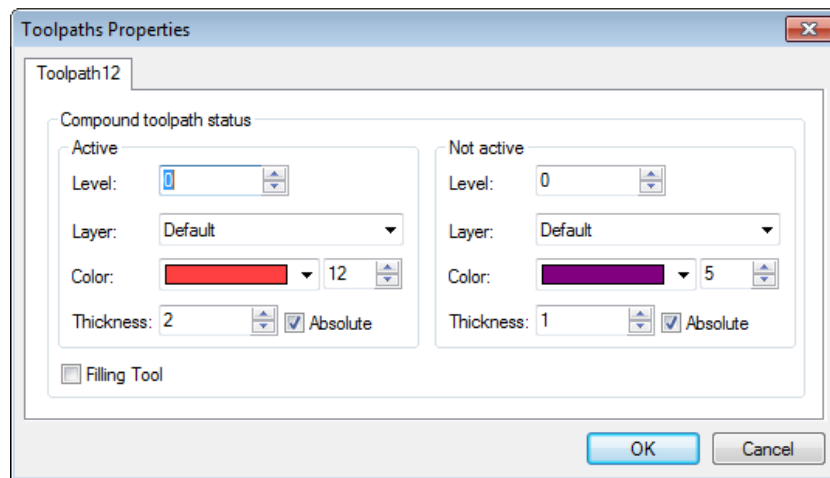
The  option is used for creating technological (service) toolpaths. Such toolpaths are necessary, for example, for defining additional commands in the control sequence for servicing technological processes. Detailed description of technological toolpath creation is provided in the "Technological toolpaths" chapter that covers all types of machining processes.



Toolpath of wire-cutting is built on the basis of standard construction elements of T-FLEX CAD "path"() or "hatch"(). The path has a direction and can be not closed. The hatch has a direction and constitutes a closed contour. Nodes of the path and the hatch after construction of a toolpath are uploaded to the control sequence in the form of support displacements of the tool relative to the part. The direction of the hatches and paths coincides with the direction of the motion of a tool relative to the part being machined.

When creating a toolpath, the "Toolpath properties" and "Change of initial point of toolpath" options will be accessible only after selection on the drawing the required construction element of T-FLEX CAD. When editing the created toolpath, these options become accessible right away.

In the **Toolpath properties** window, the user can specify the properties for two groups of elements – active and nonactive toolpaths. The user can select the thickness of displayed elements of the toolpath, their color, layer, and the level of location of the toolpath on the drawing.



Filters for selection of hatches and paths change the possibility of their selection on the drawing as a geometrical basis for construction of a toolpath. Filters can be active in pairs (in this case, both paths and hatches can be selected), and they can be active individually (only paths or only hatches). The image of the active filter is highlighted with a frame.

Depending on the construction element on the basis of which the user wants to create a toolpath, it is necessary to **select either the hatch or the path**.

Simultaneously with the automenu of wire-cut machining, the nonmodal window of toolpath parameters will appear. It is possible to indicate an element of the drawing and specify toolpath parameters in any order. That is, it is possible first to set up parameters of the toolpath, then indicate construction elements on the drawing or vice versa, first indicate construction elements, and then specify parameters. Also, it is possible to partially define parameters of the toolpath, and then indicate construction elements, and then continue specification of the toolpath parameters.

When using angle cutting, the user needs to consider the following limitation: all elements in the contour should have tangency connections in order to achieve a correct control sequence.

When selecting a hatch or path (for single-contour or angle cutting), a nonmodal dialog window of toolpath parameters will appear at the left of the drawing area (the default setting), letting the user modify those as appropriate (the detailed description of available parameters is given below).

There are a number of considerations for creating a toolpath for double-contour cutting:

- In double-contour cutting, the part is defined by two contours: the top and bottom ones. Both contours are required to be defined by the same-type elements: both "hatch"-type or both "path"-type. Combining the two types of elements is not allowed;
- The geometrical shape of the top and bottom contours ought to be different in the double-contour cutting. Otherwise, it is reduced to single-contour cutting;
- The user should first specify the bottom contour (the base of the part), and then the top contour, in order to achieve successful and correct calculation of the machining path;

Toolpath Parameters

The toolpath parameters dialog box is the same for all types of cutting. Let us describe it on the example of the single-contour cutting toolpath parameters.

Toolpath. Name of the toolpath.

Tool name. The user can specify a specific name (number) of the tool, utilized in the machining station, if such a name exists, otherwise indicate the trade-mark and thickness of the wire to be used;

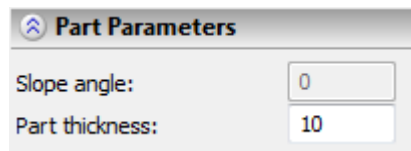
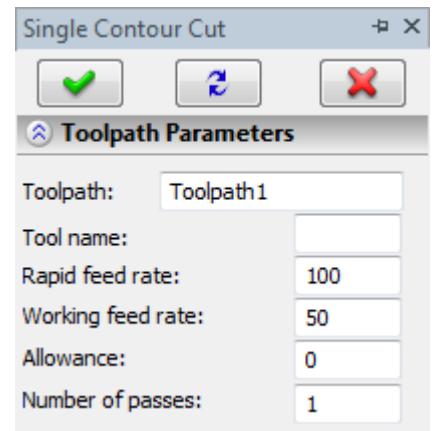
Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary displacements;

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the tool's translation when executing movements of cutting;

Allowance. This parameter defines the magnitude of the allowance – distance between original and equidistant contours;

Number of passes. The number of passes along the selected contour of the part. When using a correction by the value for the radius, the equidistant passes, in the indicated quantity, displaced with respect to each other at the value of the correction will be created.

Wire-cut parameters.



Dielectric parameter is specified as a numerical value from 0 to 9;

Wind parameter is specified as a specific numerical value;

Generator's parameter is specified as a specific numerical value;

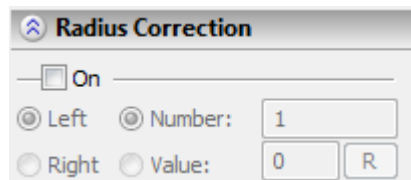
Part parameters.

Slope angle of the wire. This parameter is accessible only for **angular cutting**;

Part thickness. Uploaded to the control sequence in the form of a separate frame.

Radius correction.

Left correction. Offset from the original contour to the left side with respect to the direction of the motion;



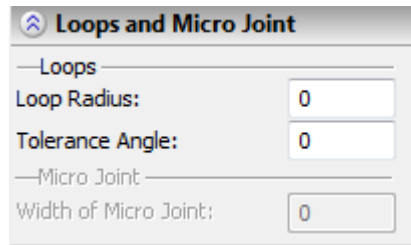
Right correction. Offset from the original contour to the right side with respect to the direction of the motion;

Number. For the selected correction, we specify the number from the table shown on the machining station;

Offset value. For the selected correction we specify the magnitude of the offset from the original contour;

Loops and Micro Joints.

Addition of circular segments of the specified **radius** to the toolpath for tangential transition from one segment to another if the **angle** between those segments exceeds the one specified among the parameters.



Loops and Micro Joint

—Loops—

Loop Radius: 0

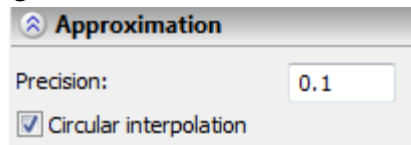
Tolerance Angle: 0

—Micro Joint—

Width of Micro Joint: 0

Approximation.

Precision of the approximation is specified as a numerical value in units supported by the CNC center and constitutes the magnitude of the caving (maximum distance from the arc of the circle to be straightened to the approximating line).



Approximation

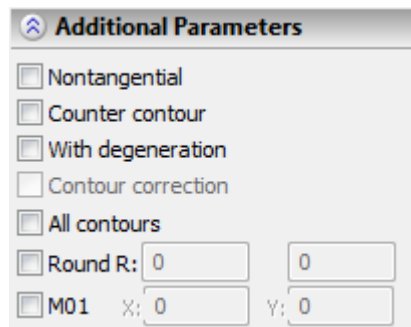
Precision: 0.1

☒ Circular interpolation

Circular interpolation. In the case when the contour to be machined has circles, arcs, splines, and other geometrical elements having similar shape, the selection of this parameter will allow us to obtain the control sequence that contains frames of the circular interpolation.

Additional parameters.

Precise tracing of angles. The set-up of this parameter specifies deletion of the tangential transitions (joining elements of the contour tangentially) in the toolpath that is created by means of an equidistant offset from the original contour (when using the allowance or correction by the value for the tool's radius).



Additional Parameters

☐ Nontangential

☐ Counter contour

☐ With degeneration

☐ Contour correction

☐ All contours

☐ Round R: 0 0

☐ M01 X: 0 Y: 0

Counter contour. Changes the direction of the tool's motion.

Without deletion of micro joints. This parameter enables restoration of the degenerated elements of the toolpath that is created by means of an equidistant offset from the original contour (when using the allowance or correction by the value for the tool's radius).

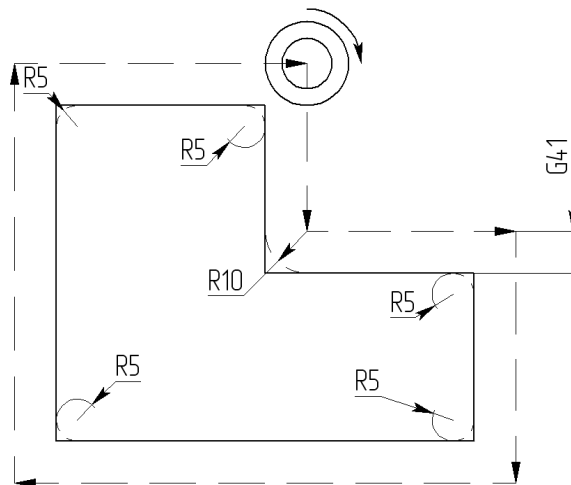
Matching contours. This parameter is accessible only for **double contour cutting** and used in case if the contours of the base and apex of the part include different number of segments.

All contours of hatching. This flag should be used when constructing a toolpath on the basis of the composite hatching that includes several contours.

R of rounding. Perform rounding of the specified radius. This parameter allows us to inscribe circles of the specified radii in the places of non-tangential transitions for internal and external angles of the contour. This parameter will be described in more detail below.


M01. Capability of the insertion, to the control sequence (at the first pass), of the frame of the conditional stopping (M01) at the point that is the nearest to the specified one (the X, Y coordinates).

Round in radius (R of rounding). In two parameter boxes the values of rounding radii of external and internal angles of a contour are set. Usually, to specify the rounding radius of external angles of contour ($>180^\circ$) the first box is used, and to specify the value of rounding radius of internal angles ($<180^\circ$) the second box is used. However, this designation of boxes can be changed depending on direction of pass along the contour and direction of correction on tool radius.





In the example shown above the radius 5 is set for rounding external angles and radius 10 for rounding internal angles.


Change of initial point of a toolpath.

Upon creation or editing of a toolpath of the contour machining based on a hatch, the user has a possibility to specify an initial point. By default, the system selects the first node of the hatch as an initial point of the toolpath. The user can refuse the given strategy of the calculation and by pressing the  button indicate another point of the contour as an initial point. If so, the machining will start and end at the indicated point.

Engraving.


The user can create the operation of the electroerosive engraving of the text by selecting the  option from the automenu of wire-cut machining. Next, it is required to indicate on the drawing the individual text or the sequence of texts, created in advance, for which the toolpath of engraving needs to be created.

The  filter turns on/off the capability of selection of the text on the drawing.

The  button in the automenu of engraving serves for cancelation of the selected text.

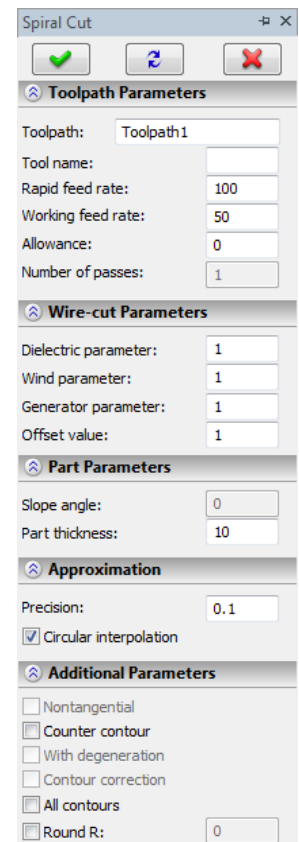
Spiral Material Removal

In the wire-cut machining there is a strategy of selection of the material along the spiral. To use this option, first it is required to construct a hatch circumscribing the contour from which the removal of the material needs to be performed.

After the preliminary constructions have been carried out, the user has to select the option of selection of the material by a spiral  from the automenu of the wire-cut machining. Next, it is required to indicate the hatch on the existing drawing and, in the window of toolpath's parameters, input the required parameters for machining. The set of toolpath's parameters here mostly coincides with that for the single contour cutting, described earlier. Below we describe only the additional parameters of selection of the material by a spiral.

Wire-cut parameters.


Offset value. This parameter specifies the distance between the passes.



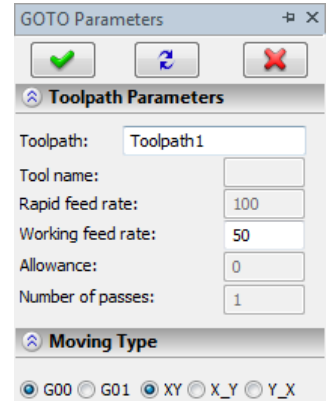
The screenshot shows the 'Spiral Cut' dialog box with the following parameters:


- Toolpath Parameters:**
 - Toolpath: Toolpath1
 - Tool name: (empty)
 - Rapid feed rate: 100
 - Working feed rate: 50
 - Allowance: 0
 - Number of passes: 1
- Wire-cut Parameters:**
 - Dielectric parameter: 1
 - Wind parameter: 1
 - Generator parameter: 1
 - Offset value: 1
- Part Parameters:**
 - Slope angle: 0
 - Part thickness: 10
- Approximation:**
 - Precision: 0.1
 - ☒ Circular interpolation
- Additional Parameters:**
 - ☐ Nontangential
 - ☐ Counter contour
 - ☐ With degeneration
 - ☐ Contour correction
 - ☒ All contours
 - ☐ Round R: 0


Going to Point with Specified Parameters

To go to a point with specified parameters in wire cutting and laser cutting, click the  icon. This activates the mode of selecting a point (node). Selecting a point brings up the command parameters dialog box.


In the dialog, you can select the motion type (fast – G00, or working feed – G01). You can also choose an axis-wise order of transition or simultaneous transition in two axes.



After all changes are introduced to the parameters of machining, it is sufficient for the user to press the  button in order to save the toolpath to the file that contains the finished drawing of the part being machined.


The calculated toolpath will be added to a special list of toolpaths of the Machining Manager. To view this list, the user has to utilize the  button. The work with this option will be described in more detail in the "Machining Manager" section.

Several important remarks concerning the system as a whole have to be made at this point:









1. In the list of compound toolpaths, only those toolpaths are indicated for which a unified control sequence will be calculated subsequently. For example, suppose that various contours are cut from one sheet of metal. To implement this, it is required to draw the contours in one file, and then for each contour generate a machining toolpath, and in addition, the types of machining can differ. That is, for one type of machining, its different subtypes can be used inside a single compound toolpath. For example, in case of wire-cut machining, one contour can be processed with a single contour cutting, and another one – with angular cutting;
2. If it is required to process the same part with two different types of machining (for example, wire-cut and laser-cut machining) then, first, the toolpath for one type of machining is saved (for example, for wire-cut machining) to the file that contains the image of the part being machined. After that, the user has to create new compound toolpath in the same file and use the other type of machining (in our case, it is laser machining);
3. If for the same part being machined it is required to create different subtypes of a single machining type (for example, single contour cutting and angular cutting), then one of the toolpaths is calculated first (for example, single contour cutting). After that the technologist-programmer must quit the automenu of the given machining (in our case, the automenu of wire-cut machining) using the  option. Then, it is necessary to open again the wire-cut machining and calculate the toolpath for the second subtype (in our example, for the angular cutting).


LASER-CUT MACHINING

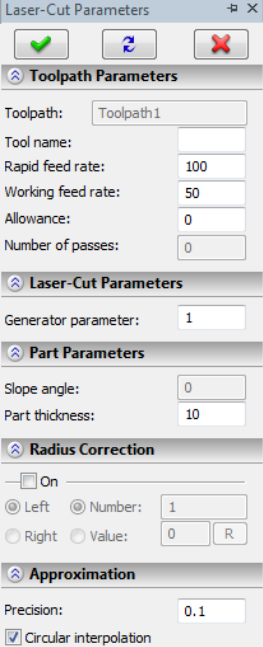
To define a toolpath and a control sequence for laser-cut machining, use the command:

Icon	Ribbon
	CNC → Machining → Laser-cut machining
Keyboard	Textual Menu
	CNC>2D, 2.5D and 4D machining>Laser-cut machining




Upon calling the command, the following options become available in the automenu:

	<End>	Finish editing
	<T>	Technological toolpath
	<O>	Single contour cut
	<U>	Angular cut
	<D>	Double contour cut
	<A>	Engraving
	<Q>	Select point for tool moving
	<Esc>	Exit command

The  option is used for creation of a technological (service) toolpath. The data of this toolpath are needed, for example, for specifying additional technological commands inside the control sequence. The process of creation of technological toolpaths for all types of machining will be described in more detail in the “Technological toolpaths” chapter.



Laser-Cut Parameters

☒ 
☐ 
☐ 

Toolpath Parameters

Toolpath:
 Tool name:
 Rapid feed rate:
 Working feed rate:
 Allowance:
 Number of passes:

Laser-Cut Parameters

Generator parameter:

Part Parameters

Slope angle:
 Part thickness:

Radius Correction




☐ On
☒ Left ☒ Number:
☐ Right ☐ Value: R

Approximation







Precision:
☒ Circular interpolation



Toolpath Creation

Once done with setting up the default machining parameters, the user needs to select one of the three types of laser-cut machining:

- Single contour cut (2D machining) 
- Angle cutting (2-1/2D machining) 
- Double contour cut (4D machining) 

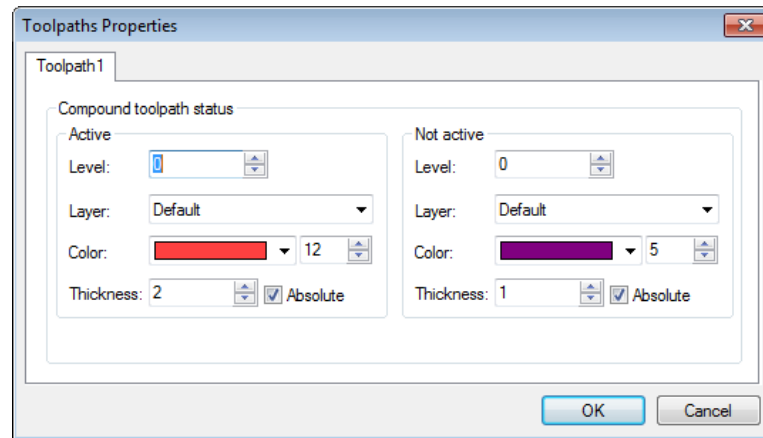
After selecting any of the listed laser-cutting types, the following options will appear in the automenu:

	<End>	Finish editing
	<P>	Toolpath properties
	<1>	Hatch selection filter
	<2>	Path selection filter
	<3>	Change of initial point of toolpath
	<Esc>	Exit command

The laser-cut toolpath is constructed on the basis of standard construction elements of T-FLEX CAD "path"() or "hatch"(). The path has a direction and can be not closed. The hatch has a direction and constitutes a closed contour. Nodes of the path and the hatch after construction of a toolpath are uploaded to the control sequence in the form of support displacements of the tool relative to the part. The direction of the hatches and paths coincides with the direction of the motion of a tool relative to the part being machined.

When creating a toolpath, the "Toolpath properties" and "Change of initial point of toolpath" options will be accessible only after selection on the drawing the required construction element of T-FLEX CAD. When editing the already created toolpath, these options become accessible right away.

In the **Toolpath properties** window, the user can specify the properties for two groups of elements – active and nonactive toolpaths. The user can select the thickness of displayed elements of the toolpath, their color, layer, and the level of location of the toolpath on the drawing.



Filters for selection of hatches and paths affect the possibility of their selection on the drawing as a geometrical basis for construction of a toolpath. Filters can be active in pairs (in this case, both paths and hatches can be selected), and individually (only paths or only hatches). The image of the active filter is highlighted with a frame.

Depending on the construction element on the basis of which the user wants to create a toolpath, it is necessary to **select either the hatch or the path**.

Simultaneously with the automenu of the laser-cut machining, the nonmodal window of toolpath parameters will appear. It is possible to indicate an element of the drawing and specify toolpath parameters in any order. That is, it is possible first to set up parameters of the toolpath, then indicate construction elements on the drawing or vice versa, first indicate construction elements, and then specify parameters. Also, it is possible to partially define parameters of the toolpath, then indicate construction elements, and then continue specification of the toolpath parameters.

There are a number of considerations for creating a toolpath for double-contour cutting:

- In double-contour cutting, the part is defined by two contours: the top and bottom ones. Both contours are required to be defined by the same-type elements: both "hatch"-type or both "path"-type. Combining the two types of elements is not allowed;
- The geometrical shape of the top and bottom contours ought to be different in the double-contour cutting. Otherwise, it will be reduced to single-contour cutting;
- The user should first specify the bottom contour (the base of the part), and then the top contour, in order to achieve successful and correct calculation of the machining path.

Toolpath Parameters

The toolpath parameters dialog box is the same for all types of cutting. Let us describe it on the example of the single-contour cutting toolpath parameters:

Toolpath. Name of the toolpath.

Tool name. The user can specify a specific name (number) of the tool, utilized in the machining station, if such a name exists, or otherwise, indicate the trade-mark and thickness of the wire being used;

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary displacements;

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the tool's translation when executing movements of cutting;

Allowance. This parameter defines the magnitude of the allowance – distance between original and equidistant contours;

Number of passes. The number of passes along a selected contour of the part. When using a correction by the value for the radius, the equidistant passes, in the specified quantity, displaced with respect to each other at the value of the correction are created.

Laser-cut machining parameters.

Generator's parameter is specified as a specific numerical value;

Part parameters.

Slope angle of wire. This parameter is accessible only for **angular cutting**;

Part thickness. Uploaded to the control sequence in the form of a separate frame.

Radius correction.

Left correction. Offset from the original contour to the left side with respect to the direction of the motion;

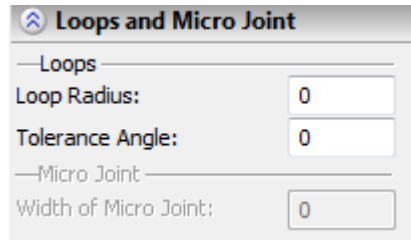
Right correction. Offset from the original contour to the right side with respect to the direction of the motion;

Number. For the selected correction, we specify the number from the table shown on the machining station;

Offset value. For the selected correction we specify the magnitude of the offset from the original contour.

Loops and Micro Joints.

Addition of circular segments of the specified **radius** to the toolpath for tangential transition between two segments that make an **angle** exceeding the one specified in the parameters.



Loops and Micro Joint

—Loops—

Loop Radius: 0

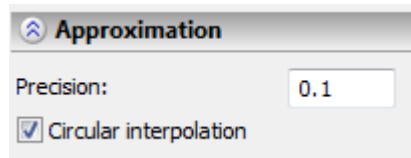
Tolerance Angle: 0

—Micro Joint—

Width of Micro Joint: 0

Approximation.

Precision of the approximation is specified as a numerical value in units supported by the CNC center and constitutes the magnitude of the caving (maximum distance from the arc of the circle to be straightened to the approximating line).



Approximation

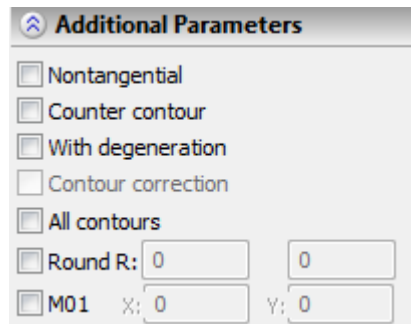
Precision: 0.1

☒ Circular interpolation

Circular interpolation. In case when the contour to be machined has circles, arcs, splines, and other geometrical elements having similar shape, the selection of this parameter will allow us to obtain the control sequence that contains frames of the circular interpolation.

Additional parameters.

Precise tracing of angles. The set-up of this parameter specifies deletion of the tangential transitions (joining elements of the contour tangentially) in the toolpath that is created by means of an equidistant offset from the original contour (when using the allowance or the correction by the value for the tool's radius).



Additional Parameters

☐ Nontangential

☐ Counter contour

☐ With degeneration

☐ Contour correction

☐ All contours

☐ Round R: 0 0

☐ M01 X: 0 Y: 0

Counter contour. Changes the direction of the tool's motion.

Without deletion of micro joints. This parameter enables restoration of the degenerated elements of the toolpath that is created by means of an equidistant offset from the original contour (when using the allowance or the correction by the value for the tool's radius).

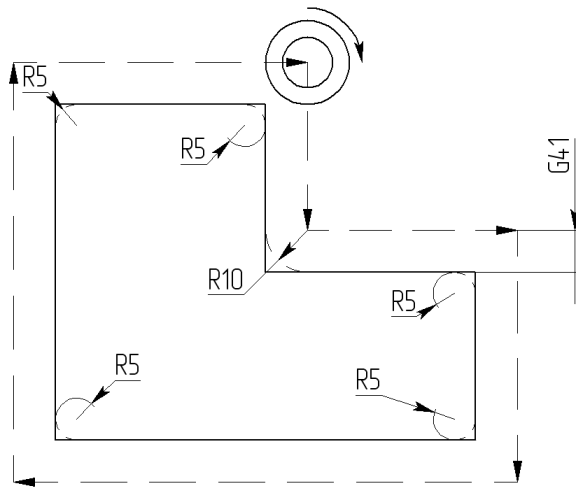
Matching contours. This parameter is accessible only for **double contour cutting** and used in case if the contours of the base and apex of the part include different number of segments.

All contours of hatching. This flag should be used when constructing a toolpath on the basis of the compound hatch that includes several contours.

R of rounding. Perform rounding with a radius. This parameter allows us to inscribe circles of the specified radii in the places of non tangential transitions for internal and external angles of the contour. This parameter will be described in more detail below.

M01. Capability of the insertion, to the control sequence (at the first pass), of the frame of conditional stopping (M01) at the point nearest to the specified one (the X, Y coordinates).


Round in radius (R of rounding). This parameter defines the rounding radius for non-tangent transitions (this command is active in all types of laser machining). In two parameters boxes the values of rounding radii of external and internal angles of a contour are set. Usually, to set the rounding radius of external angles of contour ($>180^\circ$) the first box is used, and to set the value of rounding radius of internal angles ($<180^\circ$) the second box is used. However, the designation of boxes can be changed depending on direction of pass along the contour and direction of correction on tool radius.




In the example, shown above the radius 5 is set for rounding external angles and radius 10 for rounding internal angles.


Change of initial point of a toolpath.

Upon creation or editing of a toolpath of the contour machining based on a hatch, the user has a capability to specify an initial point. By default, the system selects the first node of the hatch as an initial

point of the toolpath. The user can refuse the given strategy of the calculation and by pressing the  button indicate another point of the contour as an initial point. If so, the machining will start and end at the selected point.


Engraving.

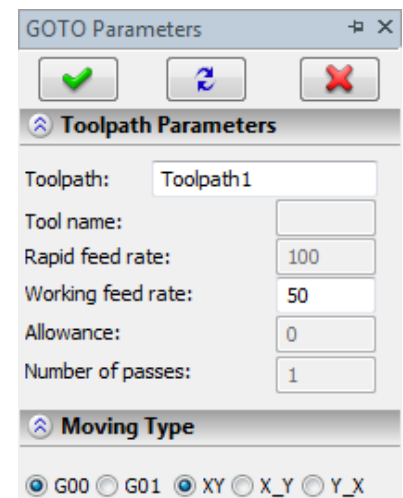
The user can create the operation of the laser-cut engraving of the text by selecting the  option from the automenu of laser-cut machining. Next, it is required to indicate on the drawing an individual text or a sequence of texts, created in advance, for which the toolpath of engraving needs to be created.


The  filter turns on/off the capability of selection of the text on the drawing.

The  button in the automenu of engraving serves for cancelation of the selected text.

Going to point with specified parameters


To go to a point with specified parameters in wire-cut and laser-cut machining, click the  icon. This activates the mode of selecting a point (node). Selecting a point brings up the command parameters dialog box. In the dialog, the user can select the type of the motion (rapid - G00 or working feed rate - G01). The user can also select the order of the displacement along the axes (simultaneous displacement along two axes or displacement along one axis by turns).



















Once all necessary changes are done to the machining parameters, the user just needs to press the [OK] button to save the toolpath in a file with the prepared drawing of the machined part. The calculated toolpath will be added to the available toolpaths list of the Machining Manager. To view this list, use the  button. Handling the list of toolpaths is described in detail in the "Machining Manager" section.


TURNING MACHINING






To make a toolpath and a control sequence for turning, use the command:


Icon	Ribbon
	CNC → Machining → Turning machining
Keyboard	Textual Menu
	CNC>2D, 2.5D and 4D machining>Turning machining

Upon calling the command, the following options appear in the automenu:

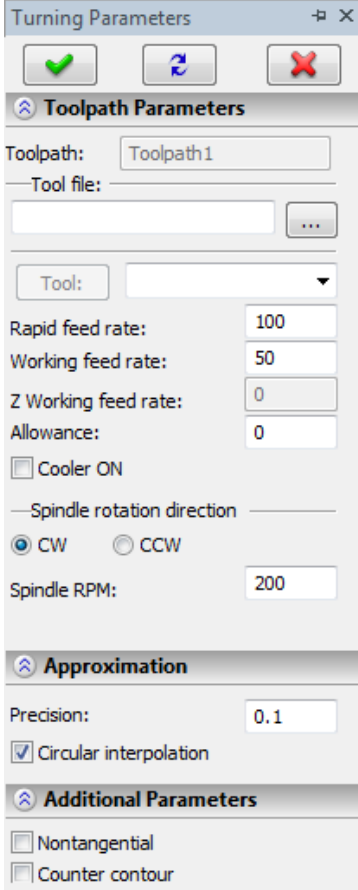
	<End>	Finish editing
	<T>	Technological toolpath
		"Finish turning" operation
	<9>	"Rough turning " group of operations
		"Pocket turning" group of operations
	<D>	"Groove turning" group of operations
	<X>	"Threading" group of operations
	<I>	"Drilling" group of operations
	<O>	"Cut off" operation
	<E>	Machining cycles EXCEL
	<W>	Machining cycles 2P22
	<N>	Machining cycles NC31
	<A>	Machining cycles NCT
	<M>	User-defined cycle of turning
	<Q>	Going to point with specified parameters
	<Esc>	Exit command

The  option is used for creating technological (service) toolpaths. Such toolpaths are necessary, for example, for defining additional commands in the control sequence for servicing technological processes. Detailed description of technological toolpath creation is provided in the "Technological toolpaths" chapter that covers all types of machining processes.

The options , , ,  and  are used for creating machining cycles. Handling these options is described in the "Machining cycles" chapter.

After the command is invoked, the "Turning machining" dialog window will appear on the screen. Parameters set up in this window will be enabled **by default** for all newly created toolpaths of machining by turning. When a specific toolpath is created, these parameters can be changed. The user can change them as desired and save these changes by pressing the .

Detailed description of all parameters is provided in the "Toolpath parameters" section below.



The "Turning Parameters" dialog window contains the following sections and controls:


- Buttons:** A green checkmark button, a refresh button, and a red X button.
- Toolpath Parameters:**
 - Toolpath:** A text field containing "Toolpath1".
 - Tool file:** A text field with a browse button (three dots).
 - Tool:** A dropdown menu.
 - Rapid feed rate:** A numeric field set to 100.
 - Working feed rate:** A numeric field set to 50.
 - Z Working feed rate:** A numeric field set to 0.
 - Allowance:** A numeric field set to 0.
 - Cooler ON:** A checkbox.
 - Spindle rotation direction:** Radio buttons for CW (selected) and CCW.
 - Spindle RPM:** A numeric field set to 200.
- Approximation:**
 - Precision:** A numeric field set to 0.1.
 - Circular interpolation:** A checked checkbox.
- Additional Parameters:**
 - Nontangential:** A checkbox.
 - Counter contour:** A checkbox.



Toolpath Creation



Note that, unlike the two previous types of machining, in most cases for turning-type machining the user can use only the graphic element of the "path" type.

Another important recommendation is positioning the part's axis horizontally for turning. In this way, the Y-coordinate will be equal 0.

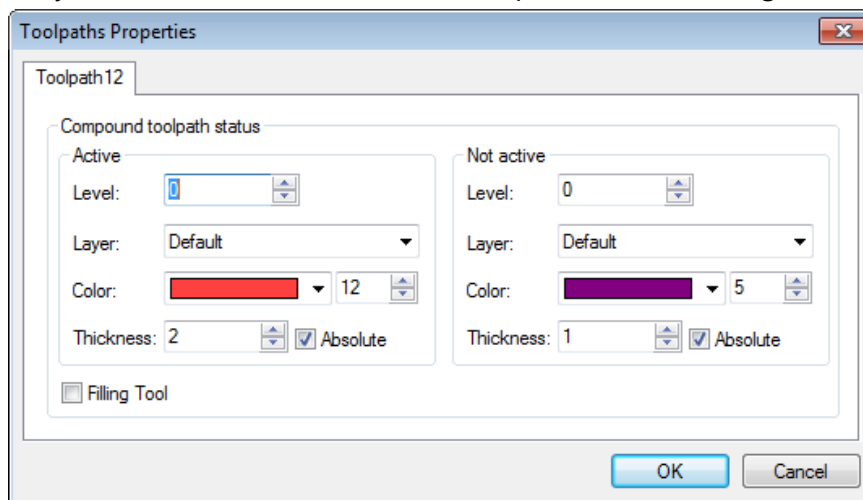
"Finish Turning" Operation

To create a toolpath of finish turning of the contour, press the  button in the automenu. On the screen the toolpath parameters dialog window will appear, which the user can change as desired, and the system will transition to the mode of contour selection.

	<End>	Finish editing
	<P>	Toolpath properties

	<2>	Path selection filter
	<Esc>	Exit command



In the **Toolpath properties** window the user can specify the properties for two groups of elements – active and nonactive toolpaths. It is possible to select the thickness of displayed elements of the toolpath, their color, layer, and level of location of the toolpath on the drawing.



Path selection **filter** changes the possibility of selection of the given element on the drawing as a geometric basis for construction of the toolpath. The image of the active filter is highlighted with a frame.

Toolpath parameters.

Toolpath. Name of the toolpath.

Tool's file. With the help of the  button, select the tool's file, and then from the drop-down list located below, - the desired tool. Start editing of the selected tool by pressing the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilized in the chosen type of machining (in this case, only turning cutters which are contained in the tool's file);

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary displacements.

Working feed rate (on X and on Z dir) is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing movements of cutting along each

of the axes separately; when executing working displacements along both axes simultaneously, the arithmetical mean value of the feed rates along each of the axis, multiplied by the coefficient, will be used (see below).

Coefficient for X/Z must be specified as a numerical value. This number will be a multiplier for the arithmetical mean value of the working feed rates when the working displacements of the machining station are carried out simultaneously. For example: if **Working feed rate on X** and **Working feed rate on Z** are equal to 30, and the **Coefficient for X/Z** is equal to 1, then the displacements of the machining station along two axes simultaneously will also be carried out with the feed rate equal to 30 ($30+30=60$; $60/2=30$ – arithmetical mean value of the feed rates; $30 \times 1=30$ – final value of the working feed rate for the displacements along two axes simultaneously).

Allowance. This parameter defines the distance between original and equidistant contours.

Cooler on. Turning on/off the supply of the cooling agent when machining the contour.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC center uses when working with this parameter.

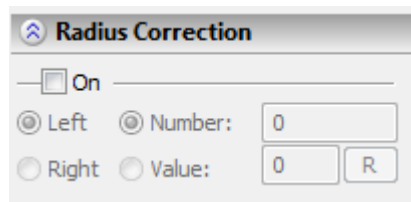
Contour's parameters.

Elementwise. Setting up this flag enables us to specify parameters of finish turning of the contour element-by-element. See below for more details.

Radius correction.

On. Setting up this flag allows us to use parameters of correction for the radius. Correction can be **left** and **right**. It is possible to specify correction **by the number** (in this case, upon machining, the value of the correction will be used, written in the table of correctors of the machining station, in the row that has a number specified by the user) and **by the value** (in this case, the user specifies the precise value of the radius of rounding of the cutting rims of the cutter).

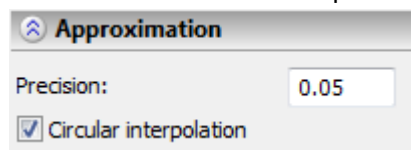
R – use the radius of the selected tool.



Approximation.

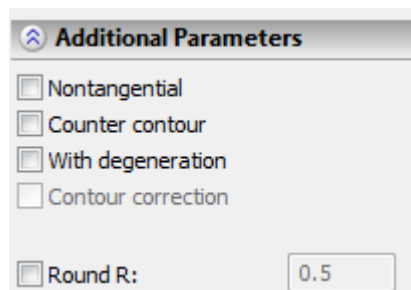
Precision of the approximation is specified as a numerical value in units supported by the CNC center and constitutes the magnitude of the caving (maximum distance from the arc of the circle to be straightened to the approximating line).

Circular interpolation. In case when the contour being machined has circles, arcs, splines, and other geometrical elements having similar shape, the selection of this parameter will allow us to obtain the control sequence that contains frames of the circular interpolation.



Additional parameters.

Precise tracing of angles. The set-up of this parameter specifies deletion of the tangential transitions (joining elements of the contour tangentially) in the toolpath that is created by means of an equidistant offset from the original contour (when using the allowance or the correction by the value for the tool's radius).



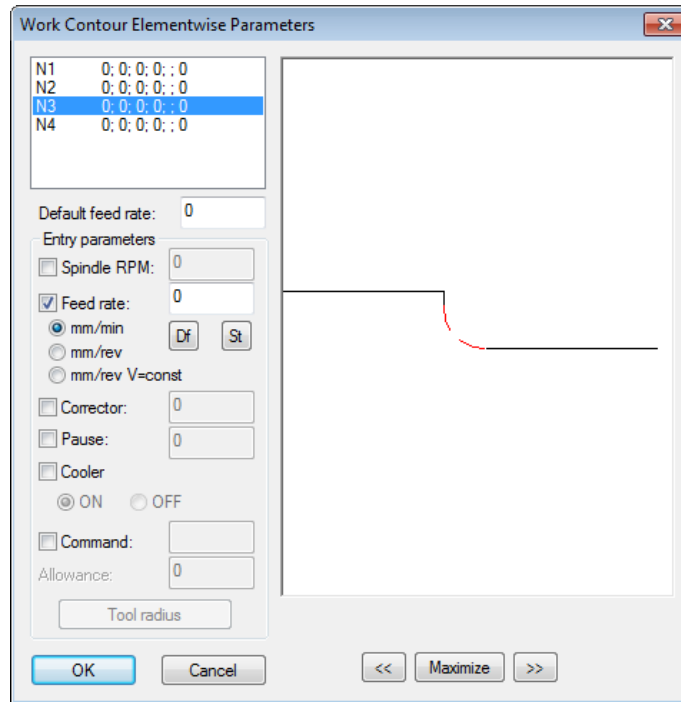
Counter contour. Changes the direction of the tool's motion.

Without deletion of micro joints. This parameter enables restoration of the degenerated elements of the toolpath that is created by means of an equidistant offset from the original contour (when using the allowance or the correction by the value for the tool's radius).

R of rounding. This parameter allows us to inscribe circles of the specified radii in the places of non tangential transition.

The main particularity of this machining is the capability of creating a toolpath for element-wise contour turning. To use it, check the "Elementwise" option. This action makes the **[Parameters]** button active. Upon clicking this button, the users gets a dialog window with element-wise contour turning parameters. If no contour was specified, then the parameters window does not appear, and the system outputs an appropriate warning.

The dialog box contains a sketch of the contour, the list of elements (contour segments) and machining parameters. The sketch of the contour is quite schematic. Each currently selected element is highlighted on the sketch. The system breaks the contour into elements (segments) on the basis of the path node positions, where the path is used for creating the machining toolpath.



The user can define the following parameters:

Default Feed Rate: the feedrate that will automatically appear in the “**Feed rate**” input box upon clicking the [Df] button; clicking the [St] button results in setting the standard feed rate to 1000 in the “**Feed rate**” input box. The feedrate units are selected below from the three options: mm/min, mm/RPM and mm/RPM at a constant cutting rate.

Spindel RPM: The spindle rotations per minute.

Corrector: sets and number of the entry in the machining station table of compensations.

Pause: there will be a frame with a user-specified dwell time after this element in the control sequence.


Cooler (On/Off): turns on/off feeding the cooling agent while machining the given element.



Command: there will be a frame with a user-specified command after this element in the control sequence.

In earlier versions of T-FLEX CAM there were two types of toolpath for finish turning of contour (“**Element-by-element finish turning**” and “**Special finish turning**”). When opening files of earlier versions the system automatically converts the old toolpaths to the toolpaths of new type displaying the warning message: “*Attention! Toolpaths shall be transformed from “Special finish turning of contour” to “Finish turning of contour”. It is possible that the user will have to manually edit first and last segments of*”


paths." After that, transformation of parameters of toolpaths of older type will be carried out: parameters "Undertravel value" and "Overtravel value" will be converted to parameters of the "Move In/Move off" option (see the "Machining Manager" section). Also, it will be required to manually change the path, on the basis of which the toolpaths was built, by removing the first and the last segments.





"Rough Turning" Group of Operations

To call the group, click the  button and hold it. After that the drop-down list of operations that can be created in the framework of this group will appear: "Allowance removal with parting (cut-off) cutter", "Allowance removal with passing cutter", "Machining corner with rounds".

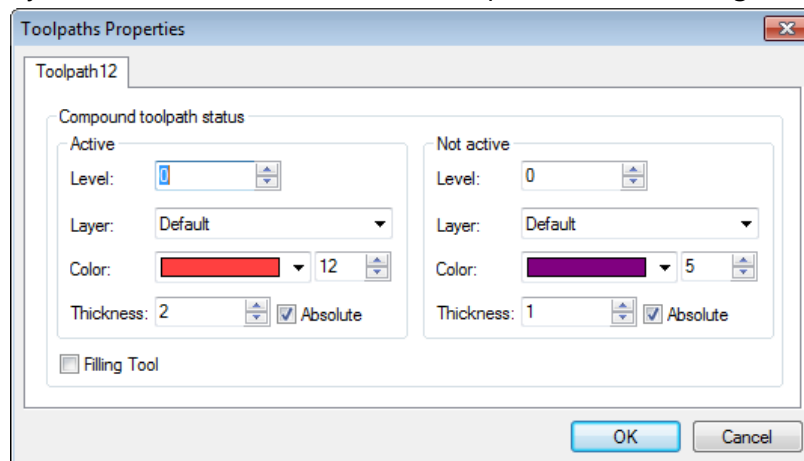
After selecting the first path (the part contour), two automenu options will become available,  and , for selecting the second path (the initial workpiece contour). At the same time, the machining parameters definition dialog will appear as well, this time much expanded as compared to the default parameters dialog.

Allowance removal with cut-off cutter.


To create a toolpath for removal of allowance with a cut-off cutter, select the  option from the drop-down list the "Allowance removal". On the screen the dialog window of the toolpath parameters will appear which the user can change as desired, and the system will transition to the mode of selection of the first path (contours of the part).

	<End>	Finish editing
	<P>	Toolpath properties
	<2>	Path selection filter
	<Esc>	Exit command

In the **Toolpath properties** window the user can specify the properties for two groups of elements – active and nonactive toolpaths. It is possible to select the thickness of displayed elements of the toolpath, their color, layer, and level of location of the toolpath on the drawing.


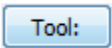


Path selection **filter** changes the possibility of selection of the given element on the drawing as a geometric basis for construction of the toolpath. The image of the active filter is highlighted with a frame.

After selection of the first path (**contour of the path**), in the automenu the  option will become accessible for specifying the second path (**contour of the workpiece**).

Toolpath parameters.

Toolpath. Name of the toolpath.

Tool's file. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilized in the chosen type of machining (in this case, only turning cutters which are contained in the tool's file);

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary displacements.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing the movements of cutting.

Allowance. This parameter defines the distance between original and equidistant contours.

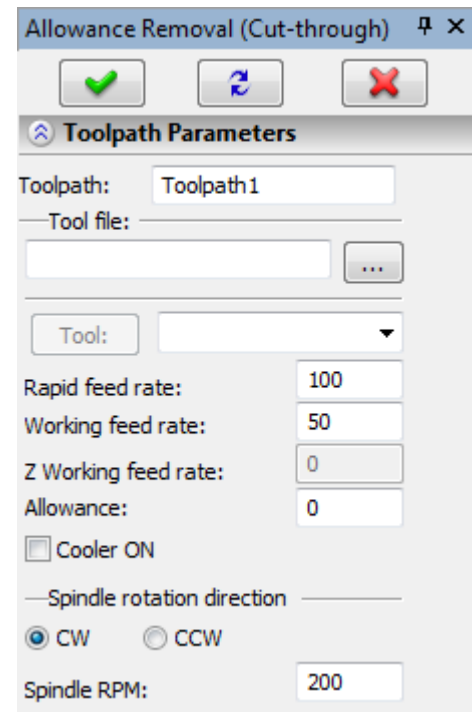
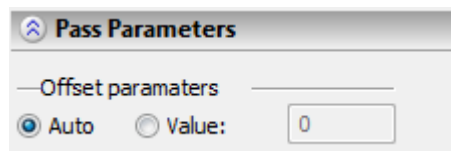
Cooler on. Turning on/off the supply of the cooling agent when machining the contour.

Spindle rotation. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC center supports when working with this parameter.

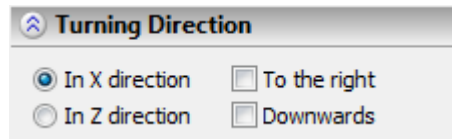
Pass parameters.

Offset parameters (Automatically/By value). This parameter controls the offset of the first and last passes from the boundary of the specified domain of machining. By default, the system calculates the offset automatically, with the offset from the part being machined equal to the rounding radius of the cutter's tip.

Direction of finish turning.

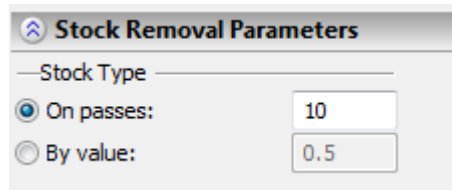
By using this option, it is possible to select the direction of passes: **along the X-axis** or **along the Z-axis**, and also it is possible to select the direction of finish turning from left to right, from right to left, downwards or upwards.



Stock removal parameters.

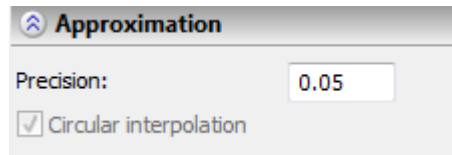
By number of passes – specifies the number of passes which are required to remove the designated stock (the magnitude of the cutoff for one pass is calculated automatically).

By value of cutoff – specifies the magnitude of the layer of the material being removed for one working pass of the tool (the required number of passes for removal of the stock is calculated automatically).








Approximation.

Precision of the approximation is specified as a numerical value in units supported by the CNC center and constitutes the magnitude of the caving (maximum distance from the arc of the circle to be straightened to the approximating line).

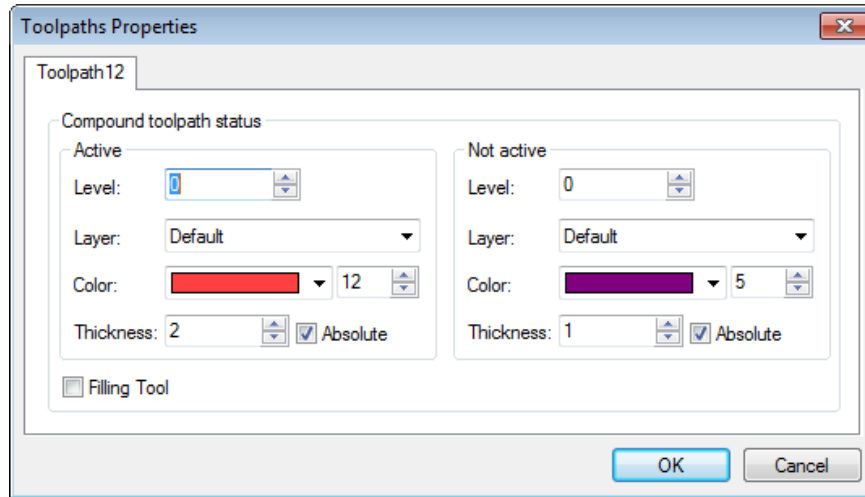


Allowance removal with passing cutter


To create a toolpath for removal of allowance with a passing cutter, select the  option from the drop-down list the "Allowance removal". On the screen the dialog window of the toolpath parameters will appear which the user can change as desired, and the system will transition to the mode of selection of the first path (contours of the part).

	<End>	Finish editing
	<P>	Toolpath properties
	<2>	Path selection filter
	<Esc>	Exit command

In the **Toolpath properties** window the user can specify the properties for two groups of elements – active and nonactive toolpaths. It is possible to select the thickness of displayed elements of the toolpath, their color, layer, and level of location of the toolpath on the drawing.





Path selection **filter** changes the possibility of selection of the given element on the drawing as a geometric basis for construction of the toolpath. The image of the active filter is highlighted with a frame.

After selection of the first path (**contour of the path**), in the automenu the  option will become accessible for specifying the second path (**contour of the workpiece**).

Toolpath parameters.

Toolpath. Name of the toolpath.

Tool's file. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilized for the chosen type of machining (in this case, only turning cutters which are contained in the tool's file);

Rapid feed rate is specified as a specific numerical value in units supported by the CNC center and the control system (the units are not specified in the parameters);

Working feed rate (on X and on Z dir) is specified as a specific numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing movements of cutting along each of the axes separately; when executing working displacements along both axes simultaneously, the arithmetical mean value of the feed rates along each of the axis multiplied by this coefficient will be used (see below).

Coefficient for X/Z must be specified as a numerical value. This number will be a multiplier for the arithmetical mean value of the working feed rates when the working displacements of the machining station are carried out simultaneously. For example: if **Working feed rate on X** and **Working feed rate on Z** are equal to 30, and the **Coefficient for X/Z** is equal to 1, then the displacements of the machining station along two axes simultaneously will also be carried out with the feed rate equal to 30 ($30+30=60$; $60/2*1=30$ – arithmetical mean value of the feed rates; $30*1=30$ – final value of the working feed rate for displacements along two axes simultaneously).

Allowance. This parameter defines the distance between original and equidistant contours.

Cooler on. Turning on/off the supply of the cooling agent when machining the contour.

Spindle rotation. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC center supports when working with this parameter.

Pass parameters.

Undertravel value. Distance from the part to the cutting rim of the tool before executing the working pass along the contour.

Overtravel value. Distance from the part to the cutting rim of the tool after executing the working pass along the contour.

Allowance Removal (Pass cutt...)

✓ ↺ ✗

Toolpath Parameters

Toolpath: Toolpath1

—Tool file: [] []

Tool: []

Rapid feed rate: 100

Working feed rate on X dir.: 30

Working feed rate on Z dir.: 30

Coefficient for X/Z: 1

Allowance: 0

☐ Cooler ON

—Spindle rotation direction []

☒ CW ☐ CCW

Spindle RPM: 200

Pass Parameters

Undertravel value: 0

Overtravel value: 0

Overtravel angle: 0

☐ Through pass 0

☐ Ignore pockets

—Offset parameters []

☒ Auto ☐ Value: 0

Overtravel angle. The value of the angle, in degrees, with which the tool digresses from the tool after execution of each working pass.

Through pass. When this option is enabled, the length of the pass of the tool is increased by a specified value.

Ignore pockets. If this option is enabled, then upon calculation of the toolpath, the system T-FLEX CAM performs analysis of the contour, specified by the user, and eliminates, from the domain being machined, the pockets with vertical walls, and also similar segments of the contour which cannot be safely machined by a specified tool.

Offset parameters (Automatically/By value). This parameter controls the offset of the first and last passes from the boundary of the specified domain of machining. By default, the system calculates the offset automatically, with an offset from the part being machined equal to the rounding radius of the cutter's tip.

Direction of finish turning.

By using this option, it is possible to select the direction of passes: **along the X-axis** or **along the Z-axis**, and also it is possible to select the direction of finish turning from left to right, from right to left, downwards or upwards.

Stock removal parameters.

By number of passes – specifies the number of passes which are required to remove the designated stock (the magnitude of the cutoff for one pass is calculated automatically).

By value of cutoff– specifies the magnitude of the layer of the material being removed for one working pass of the tool (the required number of passes for removal of the stock is calculated automatically).

Semi-finishing pass.

This pass will be carried out along the contour being machined after completion of the cycle of the stock removal. This technological operation is intended for removal of the residual roughnesses and burrs, remained after stock removal operation in order to provide the maximally smooth and uniform loading on the cutting tool while executing the finishing pass. In the parameters of the semi finishing pass it is possible to specify the **rapid** and **working feed rates**, **frequency of spindle rotation** and **the allowance**, which will be used upon execution of the semi finishing pass. Also, for the semi finishing pass the following parameters are available:

Pullout from part. Specifies the distance of the pullout of the tool from the part before execution of the semi finishing pass.

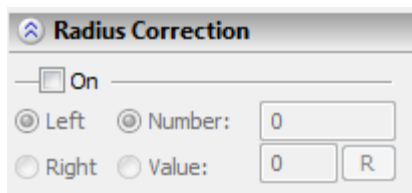
Technological stop – activation of technological stop.

Change direction. When this option is enabled, the semi finishing pass will be performed in the reverse direction (with respect to the direction of the indicated paths).

Radius correction.

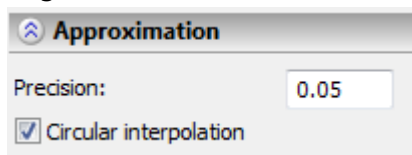
On. Setting up this flag allows us to use parameters of correction for the radius. Correction can be **left** and **right**. It is possible to specify correction **by the number** (in this case, upon machining, the value of the correction will be used, written in the table of correctors of the machining station, in the row that has a number specified by the user) and **by the value** (in this case, the user specifies the precise value of the radius of rounding of the cutting rims of the cutter).

R - use the radius of the selected tool.




Approximation.




Precision of the approximation is specified as a numerical value in units supported by the CNC center and constitutes the magnitude of the caving (maximum distance from the arc of the circle to be straightened to the approximating line).




Circular interpolation. In case when the contour being machined has circles, arcs, splines, and other geometrical elements having similar shape, the selection of this parameter will allow us to obtain the control sequence that contains frames of the circular interpolation.

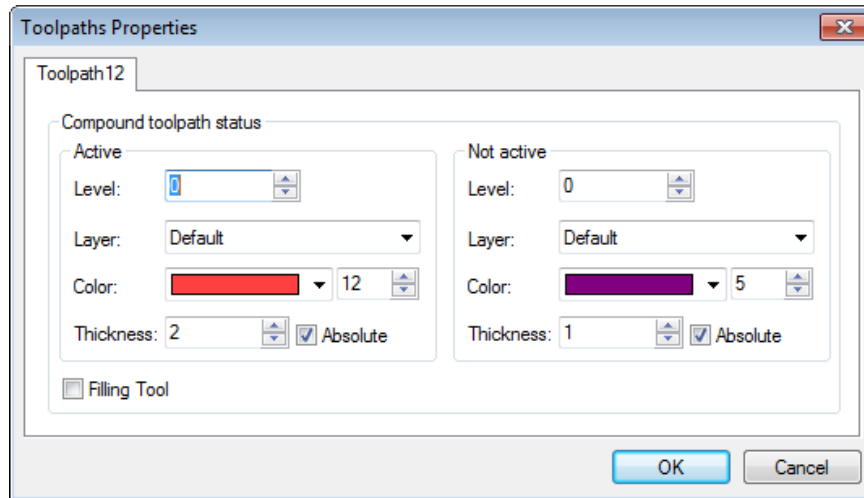
Machining a corner with a rounding

To create a toolpath for machining a corner with a rounding, select the  option from the drop-down list the "Allowance removal". On the screen the dialog window of the toolpath parameters will appear which the user can change as desired, and the system will transition to the mode of selection of the path (contour of the part).

	<End>	Finish editing
	<P>	Toolpath properties
	<2>	Path selection filter

	<Esc>	Exit the command
---	-------	------------------



In the **Toolpath properties** window the user can specify the properties for two groups of elements – active and nonactive toolpaths. It is possible to select the thickness of displayed elements of the toolpath, their color, layer, and level of location of the toolpath on the drawing.



Path selection **filter** changes the possibility of selection of the given element on the drawing as a geometric basis for construction of the toolpath. The image of the active filter is highlighted with a frame.

Toolpath parameters.

Toolpath. Name of a toolpath.

File's tool. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilized in the chosen type of machining (in this case, only turning cutters which are contained in the tool's file);

Rapid feed rate is specified as a numerical value in units supported by the CNC center and the control system (the units are not specified in the parameters);

Working feed rate (on X and on Z dir) is specified as a specific numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing movements of cutting along each of the axes separately; when executing working displacements along both axes simultaneously, the arithmetical mean value of the feed rates along each of the axes multiplied by the coefficient will be used (see below).

Coefficient for X/Z must be specified as a numerical value. This number will be a multiplier for the arithmetical mean value of the working feed rates when the working displacements of the machining station are carried out simultaneously. For example: if **Working feed rate on X** and **Working feed rate on Z** are equal to 30, and the **Coefficient for X/Z** is equal to 1, then the simultaneous displacements of the machining station along two axes will also be carried out with the feed rate equal to 30 ($30+30=60$; $60/2*1=30$ – arithmetical mean value of the feed rates; $30*1=30$ – final value of the working feed rate for displacements along two axes simultaneously).

Allowance. This parameter defines the distance between original and equidistant contours.

Cooler on. Turning on/off the supply of the cooling agent when machining the contour.

Spindle rotation. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

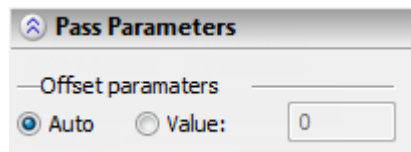
Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which are supported by the CNC center when working with this parameter.

Pass parameters.

Undertravel value. Distance from the part to the cutting rim of the tool before displacement along the given contour.

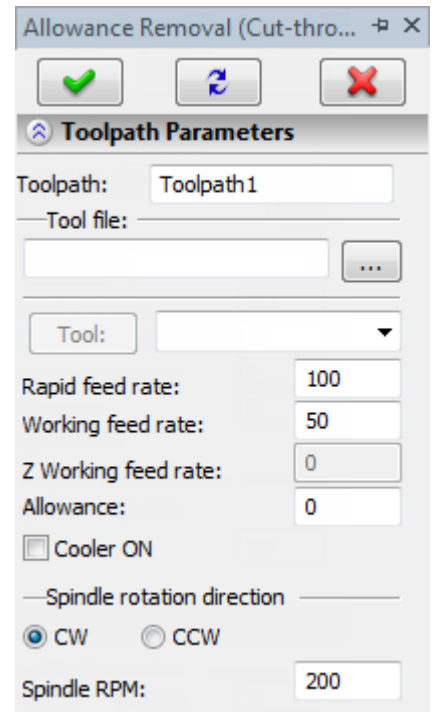
Overtravel value. Distance from the part to the cutting rim of the tool after displacement along the given contour.

Overtravel angle. The angle between the overtravel line and the last line of the contour.



Workpiece. In this field of the dialog the general allowance is indicated for machining along each of the axes (**On X** and **On Z dir**).

Start machining from (first/last) – machining starts with the first or last segment of the path.

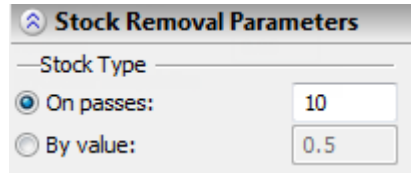


Stock removal parameters.

Cutoff type.

By number of passes – the number of passes is specified.

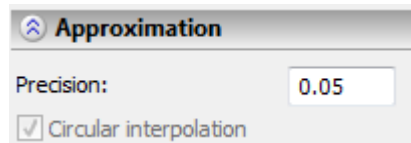
By value – specifies the depth of cutting along each of the axes; removal of all material first takes place from one side of the corner, then from the other side;



Synchronically – used when material is removed by the value. Material is first removed from one side of the corner by the specified value, then from the other side of the corner by the same value.


Approximation.

Precision of the approximation is specified as a numerical value in units supported by the CNC center and constitutes the magnitude of the caving (maximum distance from the arc of the circle to be straightened to the approximating line).




Circular interpolation. In case when the contour being machined has circles, arcs, splines, and other geometrical elements having similar shape, the selection of this parameter will allow us to obtain the control sequence that contains frames of the circular interpolation.


Once all necessary changes are done to the machining parameters, the user just needs to press the [OK] button to save the toolpath in a file with the prepared drawing of the machined part.

The calculated toolpath will be added to the available toolpaths list of Machining Manager. To view this list, use the  button.





“Pocket Turning” Group of Operations

To call the group, click and hold the  button. After that the drop-down list of operations which can be created in the framework of this group will appear: “Zigzag pocket turning”, “Spiral pocket turning”, “Special pocket turning”.

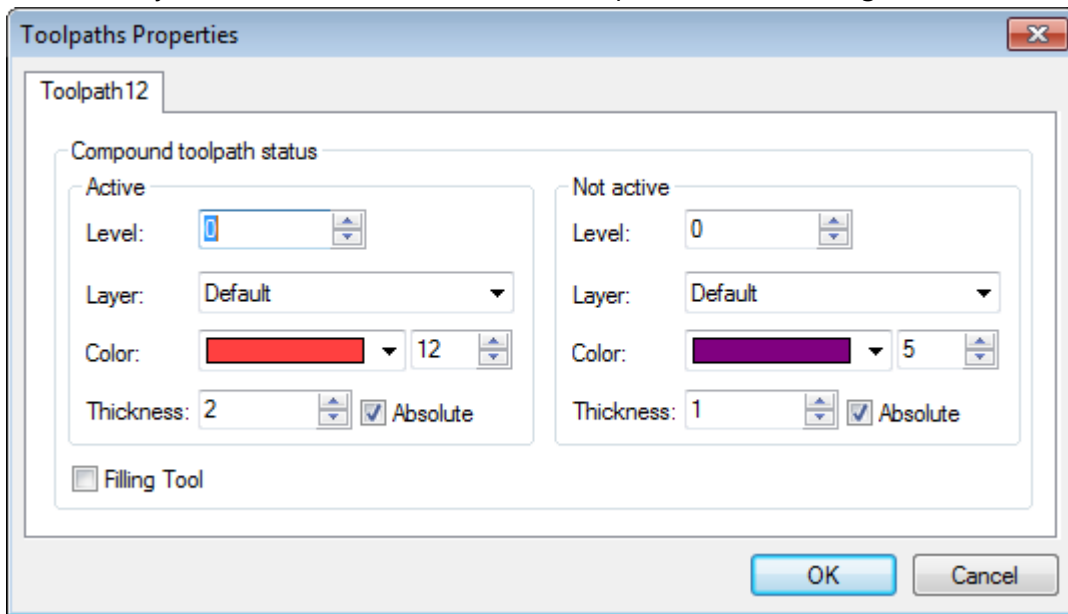
Zigzag pocket turning

To create a toolpath of zigzag pocket turning, select the  option from the drop-down list the “Pocket turning”. On the screen the dialog window of the toolpath parameters will appear

which the user can change as desired, and the system will transition to the mode of selection of the hatch.

	<End>	Finish editing
	<P>	Toolpath properties
	<1>	Hatch selection filter
	<Esc>	Exit the command

In the **Toolpath properties** window the user can specify the properties for two groups of elements – active and nonactive toolpaths. It is possible to select the thickness of displayed elements of the toolpath, their color, layer, and level of location of the toolpath on the drawing.


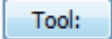


Hatch selection **filter** changes the possibility of selection of the given element on the drawing as a geometric basis for construction of the toolpath. The image of the active filter is highlighted with a frame.

For the toolpath of zigzag pocket turning, as an initial construction element, we use a **hatch** that indicates the material which must be removed from the pocket.

Toolpath parameters.

Toolpath. Name of a toolpath.

File's tool. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilizable for the chosen type of machining (in this case, only turning cutters which are contained in the tool's file);

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of translation of the tool when executing auxiliary displacements.

Working feed rate is specified as a specific numerical value in units supported by the CNC center and sets up the speed of translation of the tool when executing movements of cutting.

Allowance. This parameter defines the distance between original and equidistant contours.

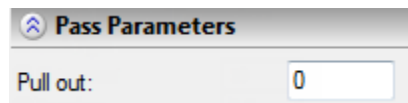
Cooler on. Turning on/off the supply of the cooling agent upon machining.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

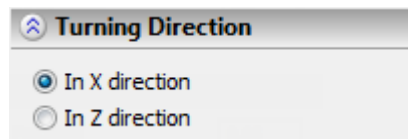
Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which are supported by the CNC center when working with this parameter.

Pass parameters.

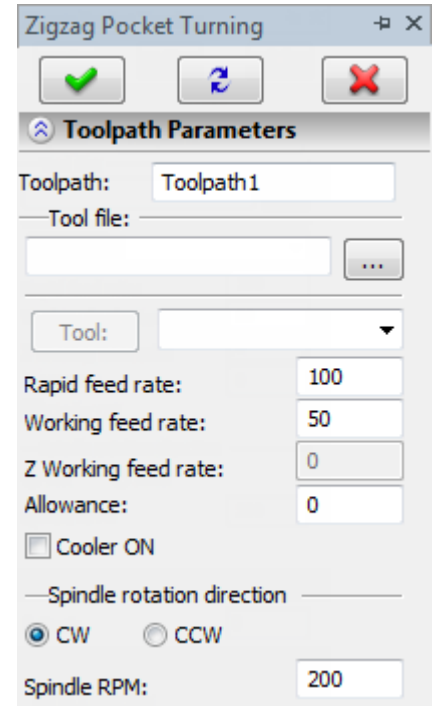
Pullout from part. Specifies the magnitude of pullout of the tool from the part after execution of the semi finishing pass.

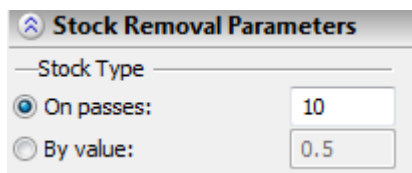
**Direction of finish turning.**

Selection of either longitudinal or transverse axis of the machining station along which the tool's passes will be directed.

**Stock removal parameters.**

By number of passes – the number of passes is specified which is required to remove the designated stock (the value of the cutoff for one pass is calculated automatically).

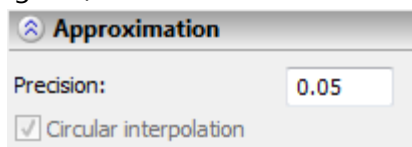





By value – the size of the layer of the material being removed for one working pass of the tool is specified (the required number of passes for removal of the stock is calculated automatically).





Approximation.

Precision of the approximation is specified as a numerical value in units supported by the CNC center and constitutes the magnitude of the caving (maximum distance from the arc of the circle to be straightened to the approximating line).

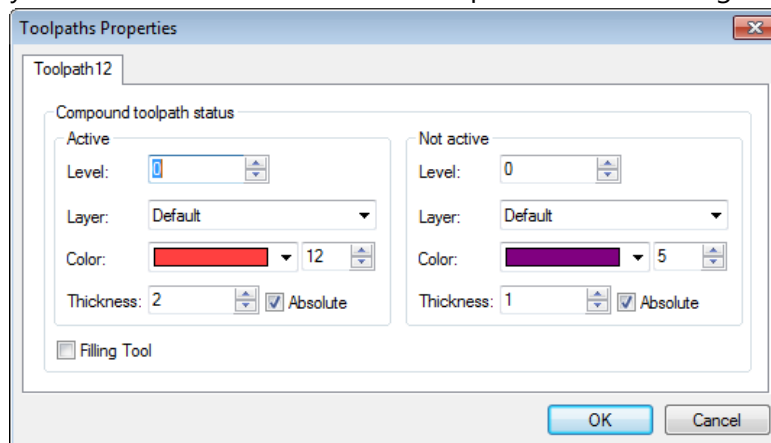


Spiral pocket turning

To create a toolpath of spiral pocket turning, select the  option from the drop-down list the "Pocket turning". On the screen the dialog window of the toolpath parameters will appear which the user can change as desired, and the system will transition to the mode of selection of the first hatch.


	<End>	Finish editing
	<P>	Toolpath properties
	<1>	Hatch selection filter
	<Esc>	Exit the command

In the **Toolpath properties** window the user can specify the properties for two groups of elements – active and nonactive toolpaths. It is possible to select the thickness of displayed elements of the toolpath, their color, layer, and level of location of the toolpath on the drawing.





Hatch selection **filter** changes the possibility of selection of the given element on the drawing as a geometric basis for construction of the toolpath. The image of the active filter is highlighted with a frame.

For the toolpath of spiral pocket turning, as an initial construction element, we use a **hatch** that indicates the material which must be removed from the pocket. In addition to this hatch, it is also required to create one more hatch above the already created one. Moreover, the second hatch must have the same number of segments (a part of the toolpath contour enclosed between two nodes) as the first hatch.

After selection of the first hatch, the  option will become available in the automenu for specifying the second hatch.

Toolpath parameters.

Toolpath. Name of a toolpath.

File's tool. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilizable for the chosen type of machining (in this case, only turning cutters which are contained in the tool's file);

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of translation of the tool when executing auxiliary displacements.

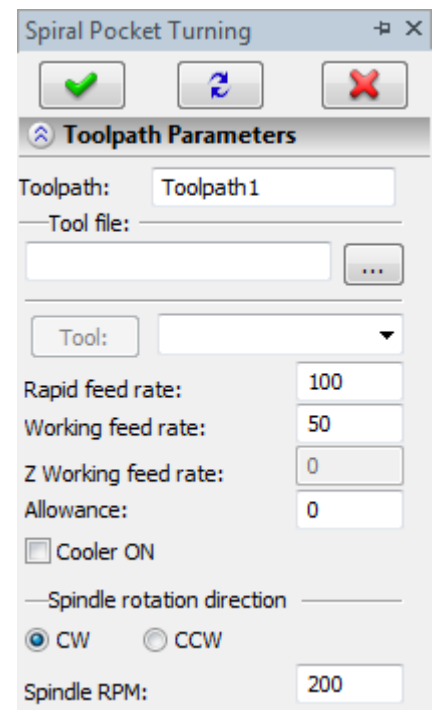
Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing movements of cutting.

Allowance. This parameter defines the distance between original and equidistant contours.

Cooler on. Turning on/off the supply of the cooling agent upon machining.

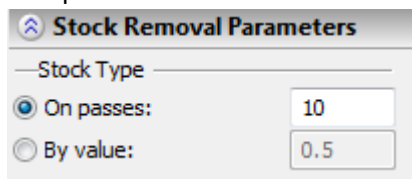
Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC center supports when working with this parameter.



Stock removal parameters.

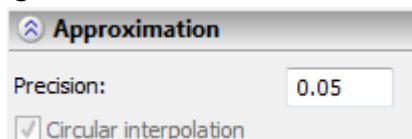
By number of passes – the number of passes is specified which is required to remove the designated stock (the value of the cutoff for one pass is calculated automatically).




By value – the size of the layer of the material being removed for one working pass of the tool is specified (the required number of passes for removal of the stock is calculated automatically).





Approximation.

Precision of the approximation is specified as a numerical value in units supported by the CNC center and constitutes the magnitude of the caving (maximum distance from the arc of the circle to be straightened to the approximating line).

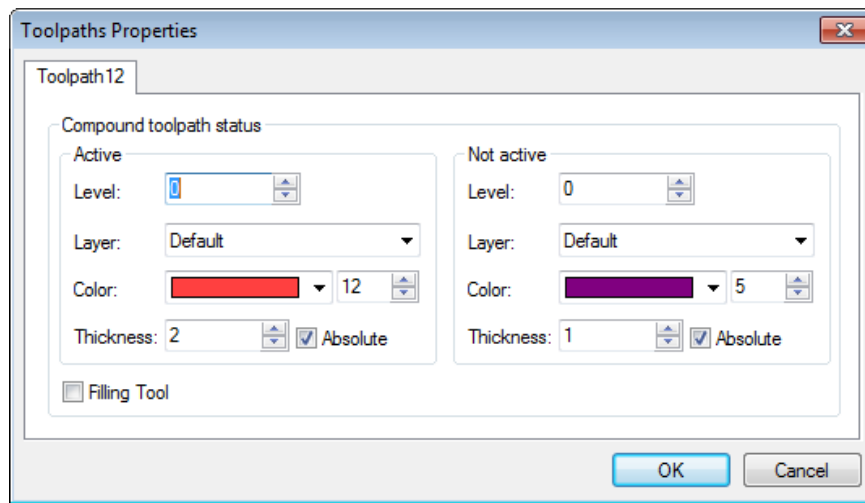


Special pocket turning

To create a toolpath of special pocket turning, select the  option from the drop-down list the "Pocket turning". On the screen the dialog window of the toolpath parameters will appear which the user can change as desired, and the system will transition to the mode of selection of the first path.

	<End>	Finish editing
	<P>	Toolpath properties
	<2>	Path selection filter
	<Esc>	Exit the command

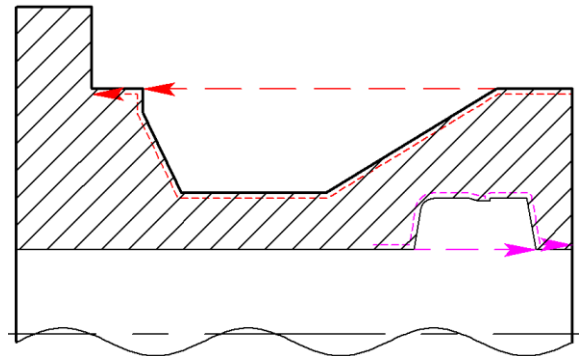
In the **Toolpath properties** window the user can specify the properties for two groups of elements – active and nonactive toolpaths. It is possible to select the thickness of displayed elements of the toolpath, their color, layer, and level of location of the toolpath on the drawing.



Path selection **filter** changes the possibility of selection of the given element on the drawing as a geometric basis for construction of the toolpath. The image of the active filter is highlighted with a frame.

To define pocket roughing, you need to create two paths. The first path follows the contour of the pocket, the second - the contour of the stock bar. The paths should satisfy the following requirements:



- external turning with the contour traversal in the counter-clockwise direction with respect to the part revolution axis;
- internal turning with the contour traversal in the clockwise direction with respect to the part revolution axis;
- the paths together make a closed contour.



The machining is defined by the upper half of the part drawing with respect to the revolution axis.

Toolpath parameters.

Toolpath. Name of a toolpath.

File's tool. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button. It is to be noted that the system will offer the user to select only that tool which can be utilized in the chosen type of machining (in the given case only turning cutters which are contained in the tool's file);

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of translation of the tool when executing auxiliary displacements.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing movements of cutting.

Working feed rate (on X and on Z dir) is specified as a specific numerical value in units supported by the CNC center and sets up the speed of the translation of the tool when executing movements of cutting along each of the axes separately; when executing working displacements along both axes simultaneously, the arithmetical mean value of the feed rates along each of the axes multiplied by the coefficient will be used (see below).

Coefficient for X/Z must be specified as a numerical value. This number will be a multiplier for the arithmetical mean value of the working feed rates when the working displacements of the machining station are carried out simultaneously. For example: if **Working feed rate on X** and **Working feed rate on Z** are equal to 30, and the **Coefficient for X/Z** is equal to 1, then the displacements of the machining station along two axes simultaneously will also be carried out with the feed rate equal to 30 ($30+30=60$; $60/2*1=30$ – arithmetical mean value of the feed rates; $30*1=30$ – final value of the working feed rate for displacements along two axes simultaneously).

Allowance. This parameter defines the distance between original and equidistant contours.

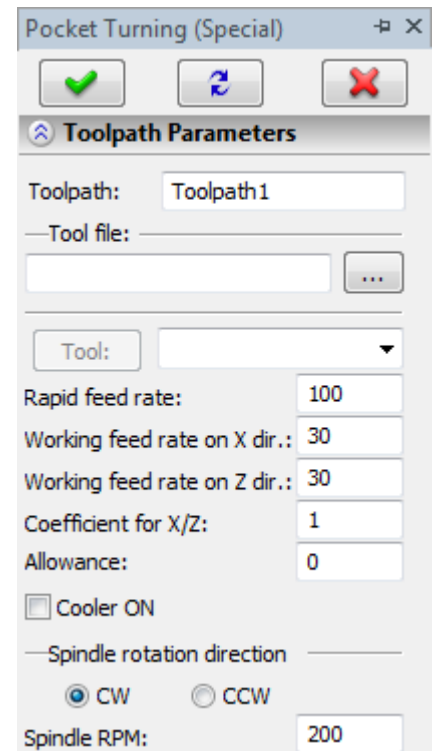
Cooler on. Turning on/off the supply of the cooling agent upon machining.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

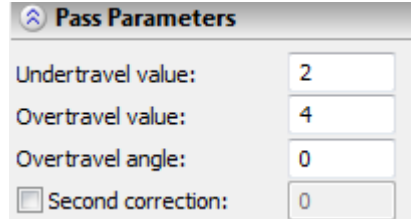
Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC center supports when working with this parameter.

Pass parameters.

Undertravel value. Distance from the part to the cutting rim of the tool before execution of the working pass along the contour.



Overtravel value. Distance from the part to the cutting rim of the tool after execution of the working pass along the contour.



The 'Pass Parameters' dialog box contains the following fields:

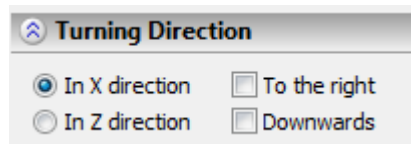
Undertravel value:	2
Overtravel value:	4
Overtravel angle:	0
<input type="checkbox"/> Second correction:	0

Overtravel angle. The angle, given in degrees, at which the tool will be pulled out after execution of each working pass.

Second correction. Correction on the second cutting rim of the tool (the number of the corrector is specified).

Direction of finish turning.

It is possible to select the direction of passes: - **along the X-axis** or **along the Z-axis**, and it is also possible to select the direction of finish turning: from left to right, from right to left, downwards, upwards.



The 'Turning Direction' dialog box contains the following options:

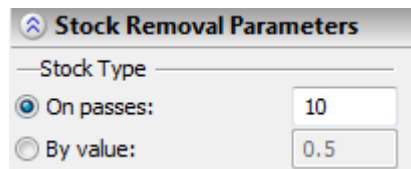
<input checked="" type="radio"/> In X direction	<input type="checkbox"/> To the right
<input type="radio"/> In Z direction	<input type="checkbox"/> Downwards

Stock removal parameters.

By number of passes – the number of passes is specified which is required to remove the designated stock (the value of the cutoff for one pass is calculated automatically).

By value – the size of the layer of the material being removed for one working pass of the tool is specified (the required number of passes for removal of the stock is calculated automatically).

Approximation.




The 'Stock Removal Parameters' dialog box contains the following options:

— Stock Type —	
<input checked="" type="radio"/> On passes:	10
<input type="radio"/> By value:	0.5


Precision of the approximation is specified as a numerical value in units supported by the CNC center and constitutes the magnitude of the caving (maximum distance from the arc of the circle to be straightened to the approximating line).





Circular interpolation. In case when the contour being machined has circles, arcs, splines, and other geometrical elements having similar shape, the selection of this parameter will allow us to obtain the control sequence that contains frames of the circular interpolation.

"Groove Turning" Group of Operations

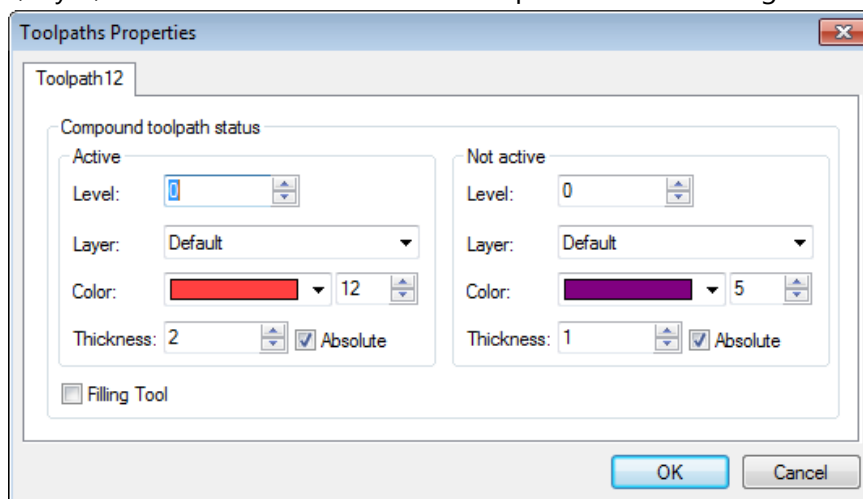
To access the group, press and hold the  button. After that the drop-down list of operations which can be created in the framework of this group will appear: "Groove turning with cut-through cutter", "Turning a groove with rounding", "Turning a groove with chamfers", "Deep groove turning", "Turning a sloping groove".

Grooving with cut-off cutter

To create a toolpath of grooving with a cut-off cutter, select the  option from the drop-down list the "Grooving". On the screen the dialog window of the toolpath parameters will appear which the user can change as desired, and the system will transition to the mode of selection of the first path.


	<End>	Finish editing
	<P>	Toolpath properties
	<2>	Path selection filter
	<Esc>	Exit the command

In the **Toolpath properties** window the user can specify the properties for two groups of elements – active and nonactive toolpaths. It is possible to select the thickness of displayed elements of the toolpath, their color, layer, and level of location of the toolpath on the drawing.





Path selection **filter** changes the possibility of selection of the given element on the drawing as a geometric basis for construction of the toolpath. The image of the active filter is highlighted with a frame.

The user needs specify two paths as the elements of a drawing. The first path describes the contour of the groove to be subject to turning, and the second one describes the material which must be removed. The paths are specified successively.

After selection of the first path, the  option for indicating the second path will become accessible in the automenu.

Toolpath parameters.

Toolpath. Name of a toolpath.

File's tool. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button. It is to be noted that the system will offer the user to select only that tool which can be utilizable in the chosen type of machining (in the given case only turning cutters which are contained in the tool's file);

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary displacements.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of translation of the tool when executing movements of cutting.

Allowance. This parameter defines the distance between original and equidistant contours.

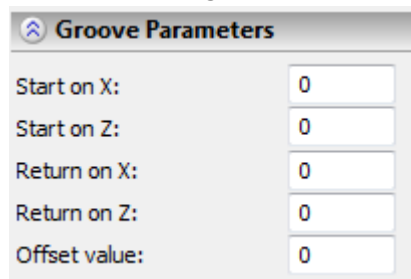
Cooler on. Turning on/off the supply of the cooling agent upon machining.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC center supports when working with this parameter.

Groove parameters.

Start on X. Specifies the location of the cutter along the X-axis before the start of machining the groove.

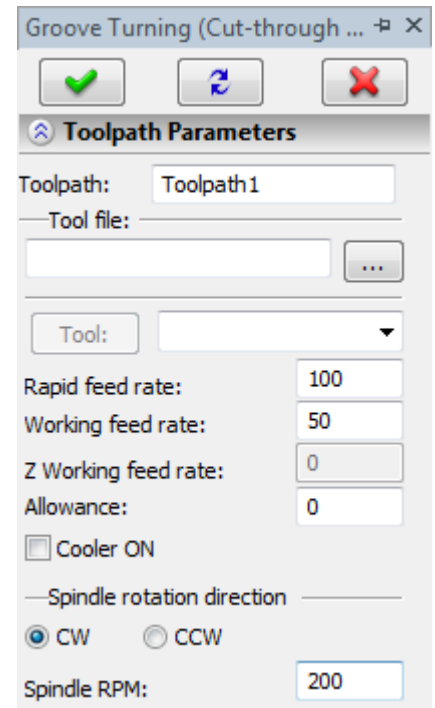


The screenshot shows the 'Groove Parameters' dialog box with the following fields and values:

Parameter	Value
Start on X:	0
Start on Z:	0
Return on X:	0
Return on Z:	0
Offset value:	0

Start on Z. Specifies the location of the cutter along the Z-axis before the start of machining the groove.

Return on X. Specifies the location of the cutter along the X-axis after machining the groove.



The screenshot shows the 'Toolpath Parameters' dialog box with the following fields and values:

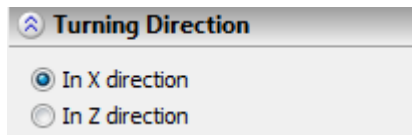
Parameter	Value
Toolpath:	Toolpath1
Tool file:	
Tool:	
Rapid feed rate:	100
Working feed rate:	50
Z Working feed rate:	0
Allowance:	0
Cooler ON	<input type="checkbox"/>
Spindle rotation direction	<input checked="" type="radio"/> CW <input type="radio"/> CCW
Spindle RPM:	200

Return on Z. Specifies the location of the cutter along the Z-axis after machining the groove.

Offset value. Specifies the magnitude of the allowance on lateral sides of the groove.

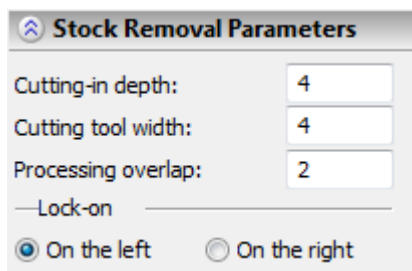
Direction of finish turning.

Selection of either longitudinal or transverse axis of the machining station along which the passes of the tool will be directed.



Stock removal parameters.

The values for **depth of cutting-in**, **width of the cutter** and **of the overlap** between neighboring passes of the tool are specified.








Lock-on left or right. Locking the cutter on the the left or right rim.

Approximation.

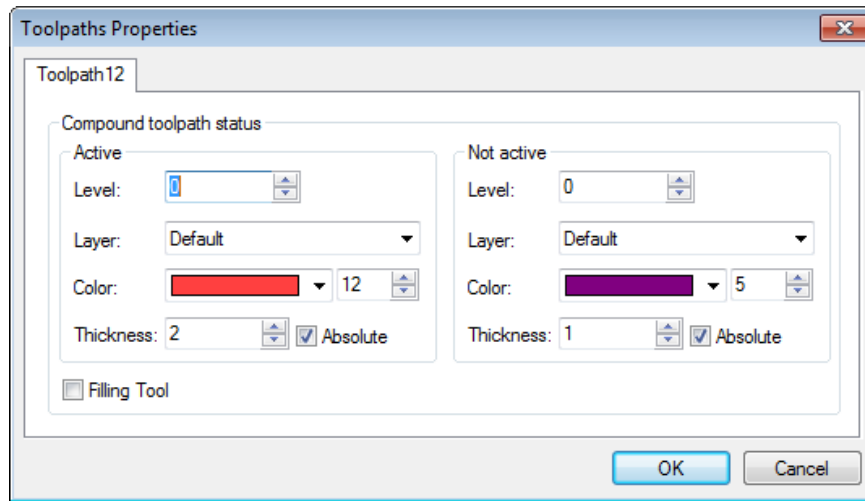
Precision of the approximation is specified as a numerical value in units supported by the CNC center and constitutes the magnitude of the caving (maximum distance from the arc of the circle to be straightened to the approximating line).

Turning a groove with rounding

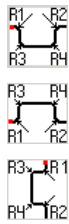
To create a toolpath of turning a groove with rounding, select the  option from the drop-down list "Grooving". On the screen the dialog window of the toolpath parameters will appear which the user can change as desired, and the system will transition to the mode of selection of the node.

	<End>	Finish editing
	<P>	Toolpath properties
	<3>	Nodes selection filter
	<Esc>	Exit the command

In the **Toolpath properties** window the user can specify the properties for two groups of elements – active and nonactive toolpaths. It is possible to select the thickness of displayed elements of the toolpath, their color, layer, and level of location of the toolpath on the drawing.



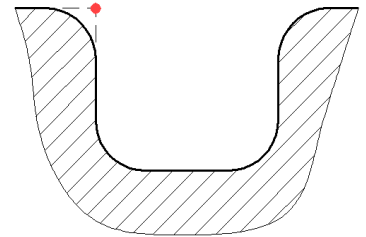
Nodes selection **filter** changes the possibility of selection of the given element on the drawing as a geometric basis for construction of the toolpath. The image of the active filter is highlighted with a frame or cutting rounded grooves, create a node as a base point, as shown on the diagram. The base point position with respect to the groove is described by the diagrams inserted in the machining dialog box:



- external grooving;


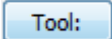
- internal grooving;

- face grooving.



Toolpath parameters

Toolpath. Name of a toolpath.

File's tool. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilizable in the chosen type of machining (in the given case only turning cutters which are contained in the tool's file);

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary displacements.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of translation of the tool when executing movements of cutting.

Working feed rate (on X and on Z dir) is specified as a specific numerical value in units supported by the CNC center and sets up the speed of the translation of the tool when executing movements of cutting along each of the axes separately; when executing working displacements along both axes simultaneously, the arithmetical mean value of the feed rates along each of the axes multiplied by the coefficient will be used (see below).

Coefficient for X/Z must be specified as a numerical value. This number will be a multiplier for the arithmetical mean value of the working feed rates when the working displacements of the machining station are carried out simultaneously. For example: if **Working feed rate on X** and **Working feed rate on Z** are equal to 30, and the **Coefficient for X/Z** is equal to 1, then the displacements of the machining station along two axes simultaneously will also be carried out with the feed rate equal to 30 ($30+30=60$; $60/2=30$ – arithmetical mean value of the feed rates; $30 \times 1=30$ – final value of the working feed rate for displacements along two axes simultaneously).

Cooler on. Turning on/off the supply of the cooling agent upon machining.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC center supports when working with this parameter.

Groove parameters.

Undertravel value. The distance from the base point to the cutting rim of the tool above the groove.

Overtravel value. The value of the pullout of the tool from the bottom surface of the groove.

Second correction. Correction on the second cutting rim of the tool.

Width. The width of the machined groove.

Depth. The depth of the machined groove.

Angle. The inclination angle of the machined groove.

Groove type. External or internal groove.

Finish turning parameters. On diameter or on face.

R1-R4 – radii of rounding of the walls of the groove according to the figure.

Groove Turning (With Roundi...)

✓ ↺ ✗

Toolpath Parameters

Toolpath: Toolpath1

Tool file: ...

Tool:

Rapid feed rate: 100

Working feed rate on X dir.: 30

Working feed rate on Z dir.: 30

Coefficient for X/Z: 1

Allowance: 0

☐ Cooler ON

Spindle rotation direction: ☒ CW ☐ CCW


Spindle RPM: 200





Approximation.

Precision of the approximation is specified as a numerical value in units supported by the CNC center and constitutes the magnitude of the caving (maximum distance from the arc of the circle to be straightened to the approximating line).

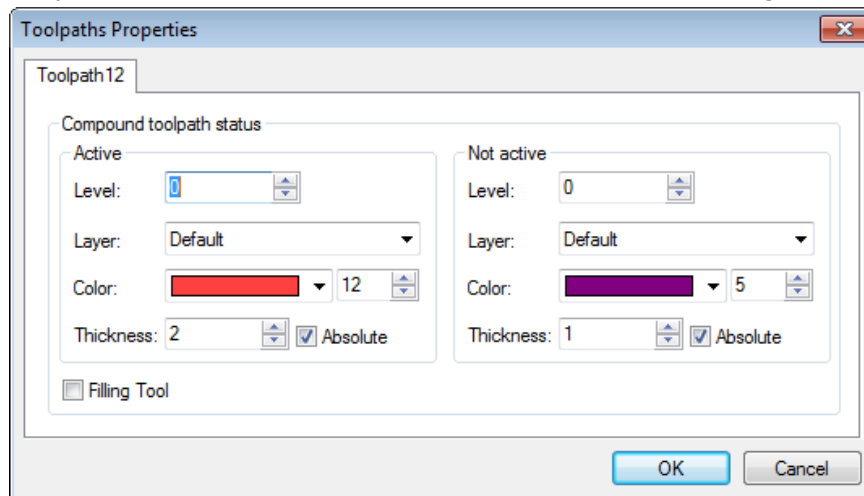
Circular interpolation. In case when the contour being machined has circles, arcs, splines, and other geometrical elements having similar shape, the selection of this parameter will allow us to obtain the control sequence that contains frames of the circular interpolation.

Cutting grooves with chamfers

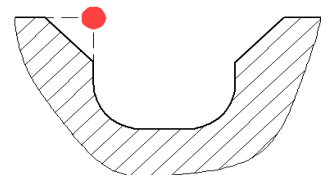
To create a toolpath of turning a groove with chamfers, select the  option from the drop-down list "Grooving". On the screen the dialog window of the toolpath parameters will appear which the user can change as desired, and the system will transition to the mode of selection of the node.

	<End>	Finish editing
	<P>	Toolpath properties
	<3>	Nodes selection filter
	<Esc>	Exit the command

In the **Toolpath properties** window the user can specify the properties for two groups of elements – active and nonactive toolpaths. It is possible to select the thickness of displayed elements of the toolpath, their color, layer, and level of location of the toolpath on the drawing.



Nodes selection **filter** changes the possibility of selection of the given element on the drawing as a geometric basis for construction of the toolpath. The image of the active filter is highlighted with a frame. For cutting grooves with chamfers, create a node as a base point, as shown on the diagram.



The base point position with respect to the groove is described by the diagrams inserted in the machining dialog box:



- external grooving;



- internal grooving;



- face grooving.

Toolpath parameters

Toolpath. Name of a toolpath.

File's tool. With the help of the button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the **Tool:** button.

It is to be noted that the system will offer the user to select only that tool which can be utilizable in the chosen type of machining (in the given case only turning cutters which are contained in the tool's file);

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary displacements.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of translation of the tool when executing movements of cutting.

Working feed rate (on X and on Z dir) is specified as a specific numerical value in units supported by the CNC center and sets up the speed of the translation of the tool when executing movements of cutting along each of the axes separately; when executing working displacements along both axes simultaneously, the arithmetical mean value of the feed rates along each of the axes multiplied by the coefficient will be used (see below).

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC center supports when working with this parameter.

Coefficient for X/Z must be specified as a numerical value. This number will be a multiplier for the arithmetical mean value of the working feed rates when the working displacements of the machining station are carried out simultaneously. For example: if **Working feed rate on X** and **Working feed rate on Z** are equal to 30, and the **Coefficient for X/Z** is equal to 1, then the displacements of the machining station along two axes simultaneously will also be carried out with the feed rate equal to 30

Grooving (With Chamfers)

✓ ↺ ✗

Toolpath Parameters

Toolpath: Toolpath1

Tool file:

Tool:

Rapid feed rate: 100

Working feed rate on X dir.: 30

Working feed rate on Z dir.: 30

Coefficient for X/Z: 1

Allowance: 0

☐ Cooler ON

Spindle rotation direction: ☒ CW ☐ CCW

Spindle RPM: 200

($30+30=60$; $60/2=30$ – arithmetical mean value of the feed rates; $30 \times 1=30$ – final value of the working feed rate for displacements along two axes simultaneously).

Cooler on. Turning on/off the supply of the cooling agent upon machining.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Groove parameters

Undertravel value. The distance from the base point to the cutting rim of the tool above the groove.

Overtravel value. The value of the pullout of the tool from the bottom surface of the groove.

Second correction. Correction on the second cutting rim of the tool.

Width. The width of the machined groove.

Depth. The depth of the machined groove.

Angle. The inclination angle of the machined groove.

Groove type. External or internal groove.

Finish turning parameters. On diameter or on face.

R1-R2 – radii of rounding according to the figure.

A1-A2 – angles of the chamfer according to the figure.


L1-L2 – length of the chamfer according to the figure.





Approximation

Precision of the approximation is specified as a numerical value in units supported by the CNC center and constitutes the magnitude of the caving (maximum distance from the arc of the circle to be straightened to the approximating line).

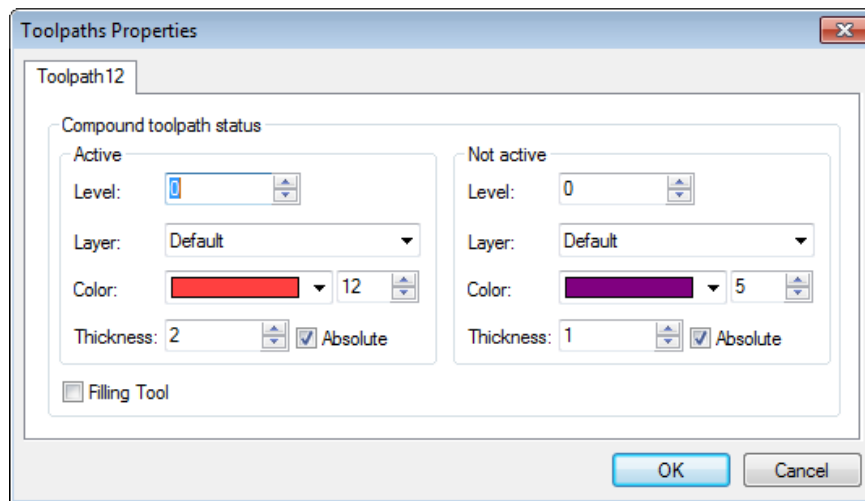
Circular interpolation. In case when the contour being machined has circles, arcs, splines, and other geometrical elements having similar shape, the selection of this parameter will allow us to obtain the control sequence that contains frames of the circular interpolation.

Deep grooving

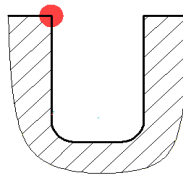
To create a toolpath for a deep grooving, select the  option from the drop-down list the "Grooving". On the screen the dialog window of the toolpath parameters will appear which the user can change as desired, and the system will transition to the mode of selection of the node.

	<End>	Finish editing
	<P>	Toolpath properties
	<3>	Nodes selection filter
	<Esc>	Exit the command

In the **Toolpath properties** window the user can specify the properties for two groups of elements – active and nonactive toolpaths. It is possible to select the thickness of displayed elements of the toolpath, their color, layer, and level of location of the toolpath on the drawing.


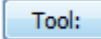


Nodes selection **filter** changes the possibility of selection of the given element on the drawing as a geometric basis for construction of the toolpath. The image of the active filter is highlighted with a frame.



Toolpath parameters.

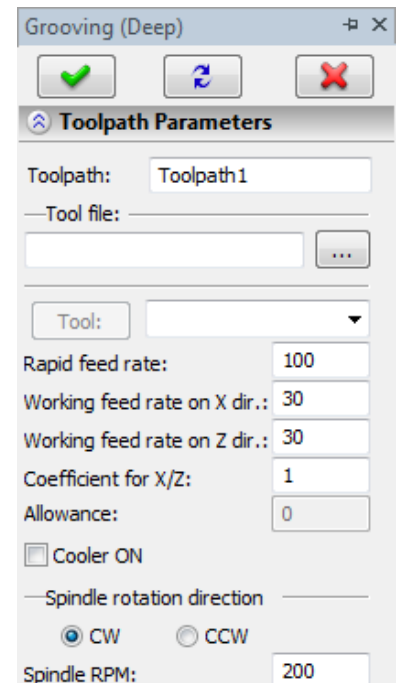
Toolpath. Name of a toolpath.

File's tool. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilizable in the chosen type of machining (in the given case only turning cutters which are contained in the tool's file);

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary displacements.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of translation of the tool when executing movements of cutting.



Working feed rate (on X and on Z dir) is specified as a specific numerical value in units supported by the CNC center and sets up the speed of the translation of the tool when executing movements of cutting along each of the axes separately; when executing working displacements along both axes simultaneously, the arithmetical mean value of the feed rates along each of the axes multiplied by the coefficient will be used (see below).

Coefficient for X/Z must be specified as a numerical value. This number will be a multiplier for the arithmetical mean value of the working feed rates when the working displacements of the machining station are carried out simultaneously. For example: if **Working feed rate on X** and **Working feed rate on Z** are equal to 30, and the **Coefficient for X/Z** is equal to 1, then the displacements of the machining station along two axes simultaneously will also be carried out with the feed rate equal to 30 ($30+30=60$; $60/2=30$ – arithmetical mean value of the feed rates; $30 \times 1=30$ – final value of the working feed rate for displacements along two axes simultaneously).

Cooler on. Turning on/off the supply of the cooling agent upon machining.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC center supports when working with this parameter.

Groove parameters.

Undertravel value. The distance from the base point to the cutting rim of the tool above the groove.

Overtravel value. The value of the pullout of the tool from the bottom surface of the groove.

Second correction. Correction on the second cutting rim of the tool.

Width. The width of the machined groove.

Depth. The depth of the machined groove.

Angle. The inclination angle of the machined groove.

Depth coefficient. The reduction coefficient for the depth of each next cutting-in.

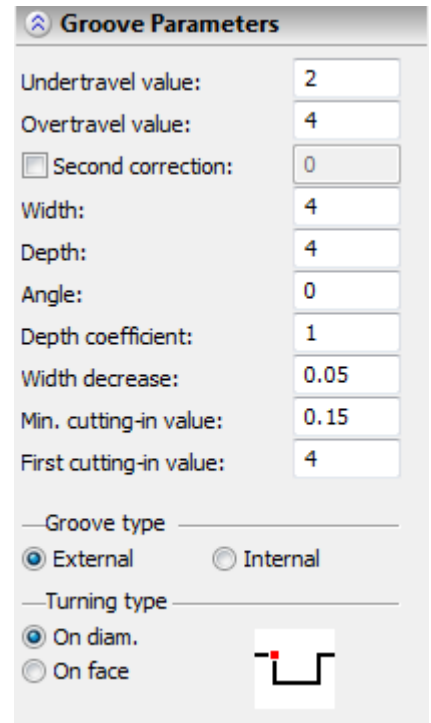
Width decrease. The magnitude of reduction of the width at each next cutting-in.

Min value of cutting-in: when this value of the step is reached, the reduction of the step via the depth coefficient will be terminated (further steps will be equal to the value of the given parameter).

Depth of cutting-in. The depth of the first cutting-in of the tool into the material.

Groove type. External or internal groove.

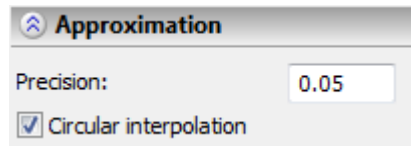
Finish turning parameters. On diameter or on face.



Groove Parameters	
Undertravel value:	2
Overtravel value:	4
<input type="checkbox"/> Second correction:	0
Width:	4
Depth:	4
Angle:	0
Depth coefficient:	1
Width decrease:	0.05
Min. cutting-in value:	0.15
First cutting-in value:	4
Groove type <input checked="" type="radio"/> External <input type="radio"/> Internal	
Turning type <input checked="" type="radio"/> On diam. <input type="radio"/> On face	


Approximation





Precision of the approximation is specified as a numerical value in units supported by the CNC center and constitutes the magnitude of the caving (maximum distance from the arc of the circle to be straightened to the approximating line).



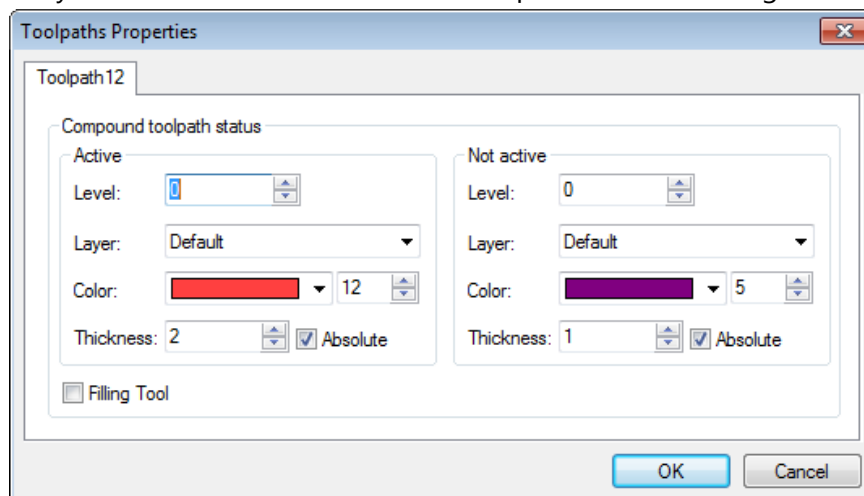
Circular interpolation. In case when the contour being machined has circles, arcs, splines, and other geometrical elements having similar shape, the selection of this parameter will allow us to obtain the control sequence that contains frames of the circular interpolation.

Cutting groove with slope

To create a toolpath of finish cutting of a groove with a slope, select the  option from the drop-down list "Grooving". On the screen the dialog window of the toolpath parameters will appear which the user can change as desired, and the system will transition to the mode of selection of the node.

	<End>	Finish editing
	<P>	Toolpath properties
	<3>	Nodes selection filter
	<Esc>	Exit the command

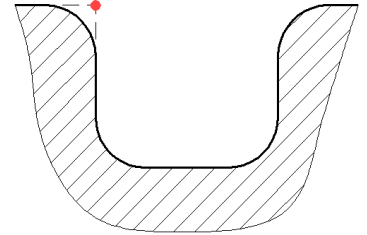
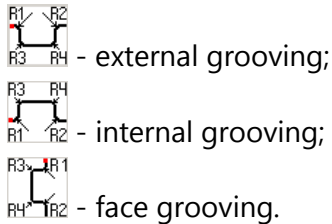
In the **Toolpath properties** window the user can specify the properties for two groups of elements – active and nonactive toolpaths. It is possible to select the thickness of displayed elements of the toolpath, their color, layer, and level of location of the toolpath on the drawing.



Nodes selection **filter** changes the possibility of selection of the given element on the drawing as a geometric basis for construction of the toolpath. The image of the active filter is highlighted with a frame.

To perform groove cutting with a slope, create a node for the base point, as shown on the diagram.

The position of the base point with respect to the groove is defined by the diagrams in the dialog box for this type of machining:



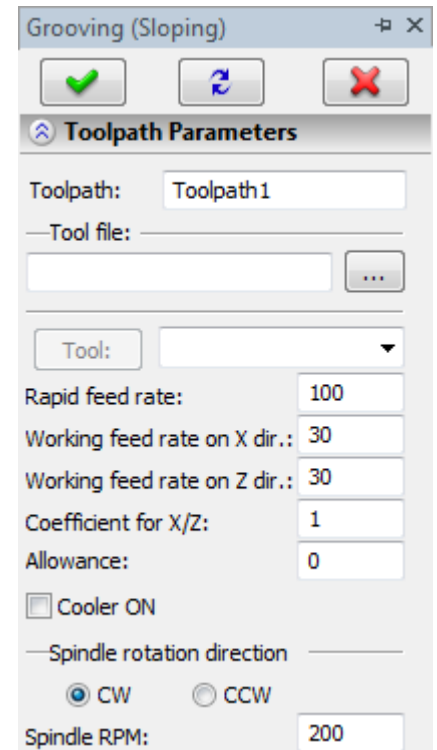
Toolpath parameters

Toolpath. Name of a toolpath.

File's tool. With the help of the button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the **Tool:** button.

It is to be noted that the system will offer the user to select only that tool which can be utilized in the chosen type of machining (in the given case only turning cutters which are contained in the tool's file);

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary displacements.



Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of translation of the tool when executing movements of cutting.

Working feed rate (on X and on Z dir) is specified as a specific numerical value in units supported by the CNC center and sets up the speed of the translation of the tool when executing movements of cutting along each of the axes separately; when executing working displacements along both axes

simultaneously, the arithmetical mean value of the feed rates along each of the axes multiplied by the coefficient will be used (see below).

Coefficient for X/Z must be specified as a numerical value. This number will be a multiplier for the arithmetical mean value of the working feed rates when the working displacements of the machining station are carried out simultaneously. For example: if **Working feed rate on X** and **Working feed rate on Z** are equal to 30, and the **Coefficient for X/Z** is equal to 1, then the displacements of the machining station along two axes simultaneously will also be carried out with the feed rate equal to 30 ($30+30=60$; $60/2=30$ – arithmetical mean value of the feed rates; $30 \times 1=30$ – final value of the working feed rate for displacements along two axes simultaneously).

Cooler on. Turning on/off the supply of the cooling agent upon machining.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC center supports when working with this parameter.

Groove parameters

Undertravel value. The distance from the base point to the cutting rim of the tool above the groove.

Overtravel value. The value of the pullout of the tool from the bottom surface of the groove.

Second correction. Correction on the second cutting rim of the tool.

Width. The width of the machined groove.

Depth. The depth of the machined groove.

Angle. The inclination angle of the machined groove.

Depth coefficient. The reduction coefficient for the depth of each next cutting-in.

Width decrease. The magnitude of the reduction of the width at each next cutting-in.

Min value of cutting-in: when this value of the step is reached, the reduction of the step via the depth coefficient is terminated (further steps will be equal to the value of the given parameter).

Depth of cutting-in. The depth of the first cutting-in of the tool into the material.

Groove type. External or internal groove.

Finish turning parameters. On diameter or on face.

Groove Parameters

Undertravel value: 2

Overtravel value: 4

☐ Second correction: 0

Width: 10

Depth: 5

Angle: 0

Undertravel to left: 0

Undertravel to right: 0

☐ Switch off finish cut

☐ Switch off rough cut

Groove type

☒ External ☐ Internal

Turning type

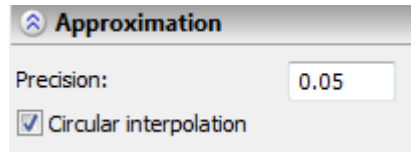
☒ On diam. ☐ On face

Diagram: A cross-section of a groove with labels R1, R2, R3, R4 for the cutting rims and A1, A2, L1, L2 for the finish turning parameters.

R1	0	A1	0
R2	0	A2	0
R3	0	L1	0
R4	0	L2	0


Approximation

Precision of the approximation is specified as a numerical value in units supported by the CNC center and constitutes the magnitude of the caving (maximum distance from the arc of the circle to be straightened to the approximating line).








Circular interpolation. In case when the contour being machined has circles, arcs, splines, and other geometrical elements having similar shape, the selection of this parameter will allow us to obtain the control sequence that contains frames of the circular interpolation.

"Threading" Group of Operations

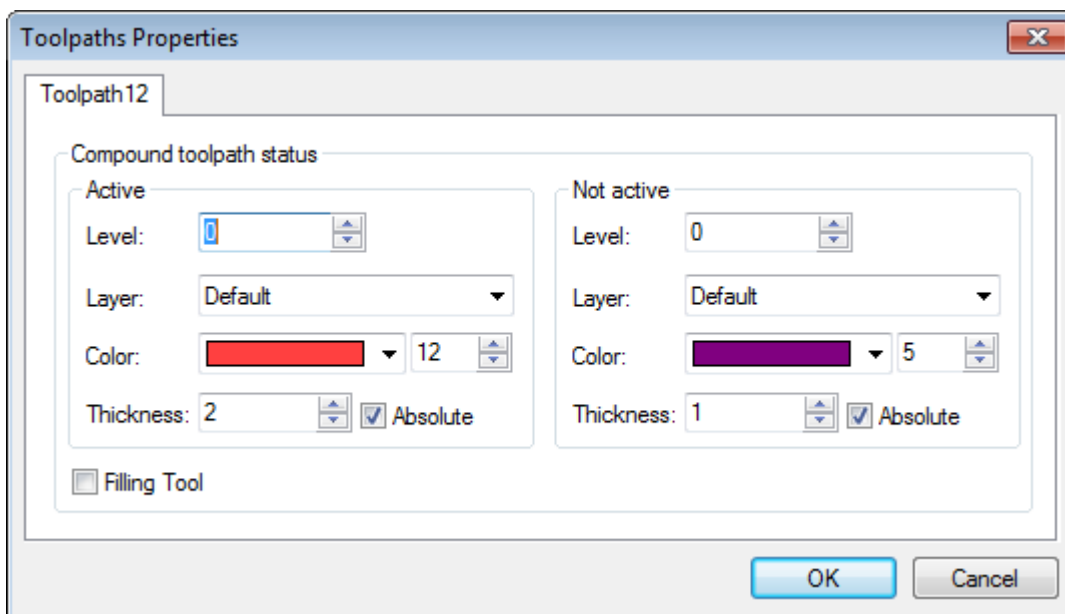
To call the group, click and hold the  button. After that, the drop-down list of operations that can be created in the framework of this group will appear: "Threading with cutter", "Threading with tap", "Cutting profile thread", "Special threading".

Threading with cutter

To create a toolpath of threading with a cutter, select the  option from the drop-down list "Threading". On the screen the dialog window of the toolpath parameters will appear which the user can change as desired, and the system will transition to the mode of selection of the path.

	<End>	Finish editing
	<P>	Toolpath properties
	<2>	Path selection filter
	<Esc>	Exit the command

In the **Toolpath properties** window the user can specify the properties for two groups of elements – active and nonactive toolpaths. It is possible to select the thickness of displayed elements of the toolpath, their color, layer, and level of location of the toolpath on the drawing.



Path selection **filter** changes the possibility of selection of the given element on the drawing as a geometric basis for construction of the toolpath. The image of the active filter is highlighted with a frame.

The path is constructed below the rotation axis of the part on the external diameter of the thread.

Toolpath parameters.

Toolpath. Name of a toolpath.

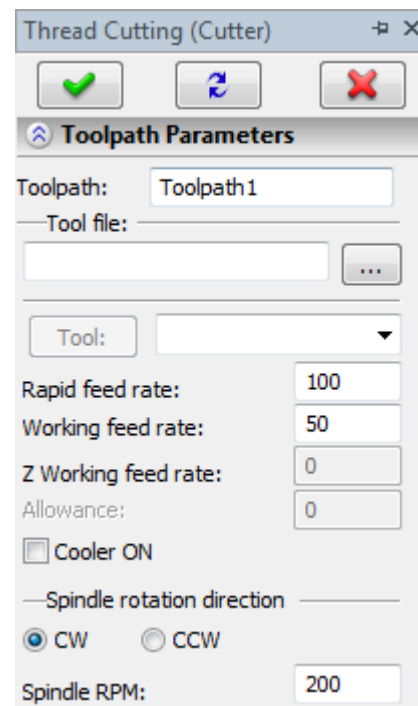
File's tool. With the help of the [...] button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the **Tool:** button.

It is to be noted that the system will offer the user to select only that tool which can be utilizable in the chosen type of machining (in the given case only turning cutters which are contained in the tool's file);

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary displacements.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of translation of the tool when executing movements of cutting.

Cooler on. Turning on/off the supply of the cooling agent upon machining.



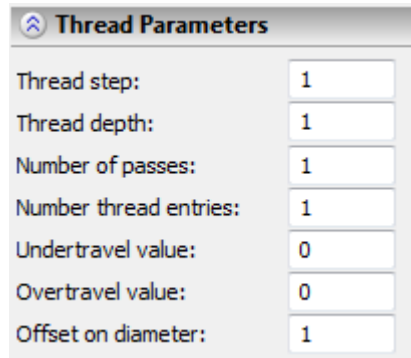
Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC center supports when working with this parameter.

Thread parameters.

Thread step. This parameters is uploaded to the control sequence.

Thread depth. Height of the thread's profile.



Thread Parameters	
Thread step:	1
Thread depth:	1
Number of passes:	1
Number thread entries:	1
Undertravel value:	0
Overtravel value:	0
Offset on diameter:	1

Number of passes. Number of passes along the height of the thread.


Number of thread entries. For multi-entry threads.





Undertravel value. The distance of the undertravel before the thread along the rotation axis.

Overtravel value. The distance of the overtravel after the thread along the rotation axis.

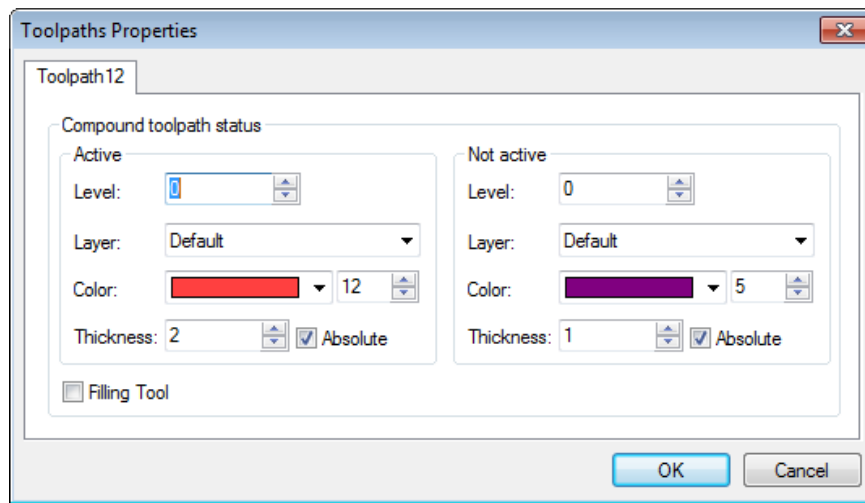
Offset on diameter. The distance from the external diameter of the thread to the initial point.

Threading with tap

To create a toolpath of threading with a tap, select the  option from the drop-down list "Threading". On the screen the dialog window of the toolpath parameters will appear which the user can change as desired, and the system will transition to the mode of selection of the path.

	<End>	Finish editing
	<P>	Toolpath properties
	<2>	Path selection filter
	<Esc>	Exit the command

In the **Toolpath properties** window the user can specify the properties for two groups of elements – active and nonactive toolpaths. It is possible to select the thickness of displayed elements of the toolpath, their color, layer, and level of location of the toolpath on the drawing.



Path selection **filter** changes the possibility of selection of the given element on the drawing as a geometric basis for construction of the toolpath. The image of the active filter is highlighted with a frame.

Toolpath parameters.

Toolpath. Name of a toolpath.

File's tool. With the help of the [...] button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the **Tool:** button.

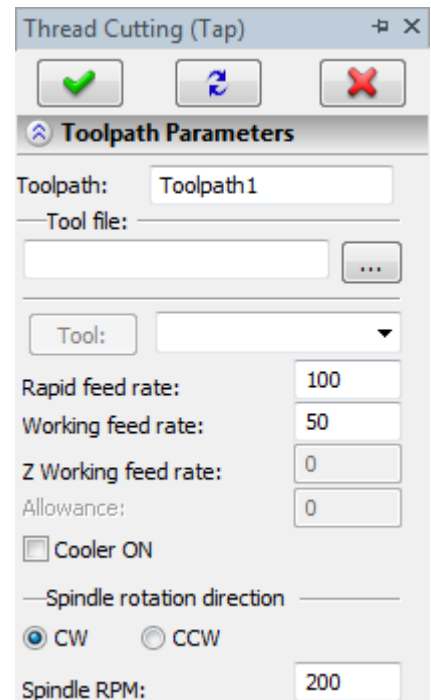
It is to be noted that the system will offer the user to select only that tool which can be utilized in the chosen type of machining (in the given case only turning cutters which are contained in the tool's file);

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary displacements.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of translation of the tool when executing movements of cutting.

Cooler on. Turning on/off the supply of the cooling agent upon machining.

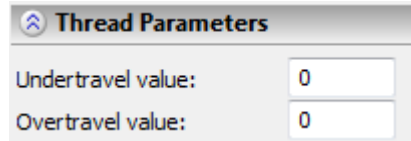
Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.



Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC center supports when working with this parameter.


Thread parameters.





Undertravel value. The distance from the start of the path to the initial point of the toolpath along the rotation axis (the safety value).



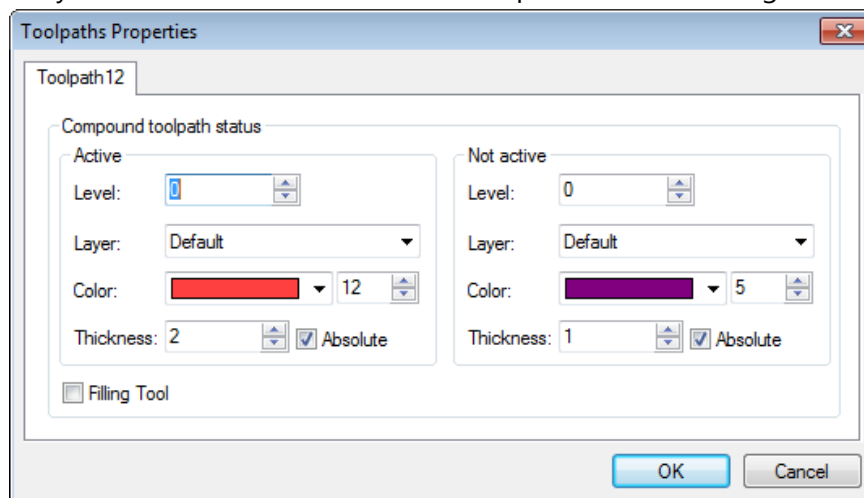
Overtravel value. The distance from the end point of the path to the end point of the toolpath along the rotation axis.

Special threading

To create a toolpath of special threading, select the  option from the drop-down list "Threading". On the screen the dialog window of the toolpath parameters will appear which the user can change as desired, and the system will transition to the mode of selection of the path.

	<End>	Finish editing
	<P>	Toolpath properties
	<2>	Path selection filter
	<Esc>	Exit the command

In the **Toolpath properties** window the user can specify the properties for two groups of elements – active and nonactive toolpaths. It is possible to select the thickness of displayed elements of the toolpath, their color, layer, and level of location of the toolpath on the drawing.





Path selection **filter** changes the possibility of selection of the given element on the drawing as a geometric basis for construction of the toolpath. The image of the active filter is highlighted with a frame.

The path is constructed at the top of the part's drawing, with respect to the rotation axis, by the nominal diameter of the thread.

Toolpath parameters

Toolpath. Name of a toolpath.

File's tool. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilizable in the chosen type of machining (in the given case only turning cutters which are contained in the tool's file);

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary displacements.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of translation of the tool when executing movements of cutting.

Allowance. This parameter defines the distance between original and equidistant contours.

Cooler on. Turning on/off the supply of the cooling agent upon machining.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.

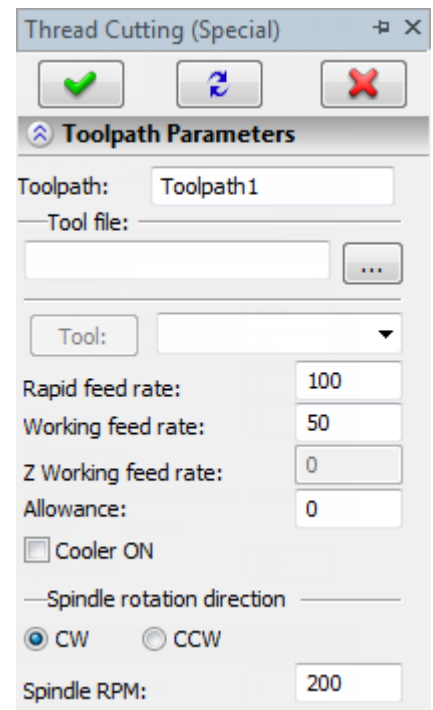
Thread parameters.

Number of threads. Additional parameter for the postprocessor.

Machining type. Additional parameter for the postprocessor.

Correctors ordering. The sequence of the correctors' numbers used in machining (specified successively by numerical values separated by commas).

Sequence number. Additional parameter for the postprocessor.



The screenshot shows a software window titled "Thread Cutting (Special)". At the top, there are three buttons: a green checkmark, a blue circular arrow, and a red X. Below this is a section titled "Toolpath Parameters". It contains the following fields and controls:

- Toolpath:** A text box containing "Toolpath1".
- Tool file:** A text box with a button containing three dots to its right.
- Tool:** A dropdown menu.
- Rapid feed rate:** A numeric input field with the value "100".
- Working feed rate:** A numeric input field with the value "50".
- Z Working feed rate:** A numeric input field with the value "0".
- Allowance:** A numeric input field with the value "0".
- Cooler ON:** A checkbox that is currently unchecked.
- Spindle rotation direction:** Two radio buttons, "CW" (which is selected) and "CCW".
- Spindle RPM:** A numeric input field with the value "200".

Number of passes. The number of passes along the height of the thread before the finishing.

Number of thread entries. For multi-entry threads.

Angle of indenting. The angle of cutting-in of the tool into the part.

Thread shift. The value that defines the angular location of the initial point of the thread.

Tool withdrawal. The pullout of the tool from the workpiece after the working pass.

Nominal diameter. The diameter of the thread (for the thread M20 – the nominal diameter is equal to 20).

Number of fair steps. The number of passes along the height of the thread during finishing.

Thread entry. The distance of undertravel before the thread along the rotation axis.

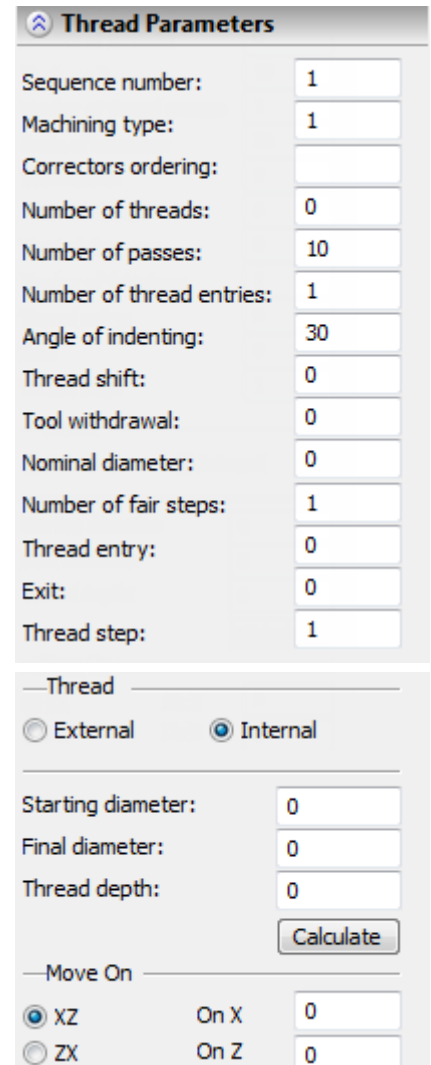
Exit. The distance of overtravel after the thread along the rotation axis.

Thread step. The magnitude of the thread step (pitch) which will be uploaded to the control sequence.


Thread external or internal.

Calculate. When this button is pressed, calculation of the **starting** and **final diameters** of the thread, and also its **depth** is performed given the nominal diameter of the thread and the step. When cutting out the conic thread, the values for the starting and final diameters of the thread are specified manually.


Approach. The value and the order in which the translations of the approach are machined.







"Axial Drilling" Group of Operations

To call the group, click and hold the  button. After that the drop-down list of operations which can be created in the framework of this group will appear: "Axial drilling", "Special axial drilling".

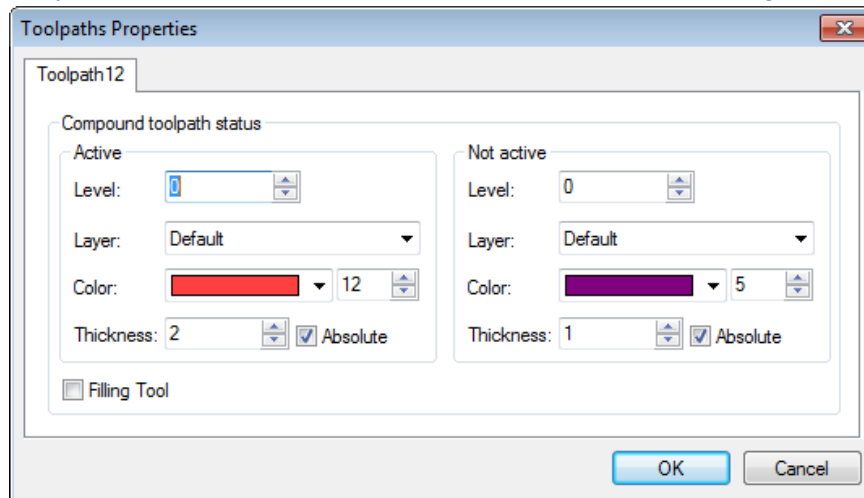
Axial drilling

To create a toolpath of axial drilling, select the  option from the drop-down list "Axial drilling". On the screen the dialog window of the toolpath parameters will appear which the user can change as desired, and the system will transition to the mode of selection of the path.

	<End>	Finish editing
	<P>	Toolpath properties

	<2>	Path selection filter
	<Esc>	Exit the command

In the **Toolpath properties** window the user can specify the properties for two groups of elements – active and nonactive toolpaths. It is possible to select the thickness of displayed elements of the toolpath, their color, layer, and level of location of the toolpath on the drawing.



Path selection **filter** changes the possibility of selection of the given element on the drawing as a geometric basis for construction of the toolpath. The image of the active filter is highlighted with a frame.

The path is constructed along the rotation axis of the part, at the depth of drilling.

Toolpath parameters.

Toolpath. Name of a toolpath.

File's tool. With the help of the [...] button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the **Tool:** button.

It is to be noted that the system will offer the user to select only that tool which can be utilizable in the chosen type of machining.

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary displacements.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of translation of the tool when executing movements of cutting.

Cooler on. Turning on/off the supply of the cooling agent upon machining.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.

Drilling parameters.

Cycle number. Selection of the drilling cycle or the cycle of deep drilling.

Safety magnitude. The height of the lift above the machined surface when making translation of the tool.

The screenshot shows the 'Axial Drilling' dialog box with the 'Toolpath Parameters' tab selected. It contains the following fields and controls:

- Toolpath:** A text field containing 'Toolpath1'.
- Tool file:** A text field with a [...] button to its right.
- Tool:** A dropdown menu.
- Rapid feed rate:** A numeric field with the value 100.
- Working feed rate:** A numeric field with the value 50.
- Z Working feed rate:** A numeric field with the value 0.
- Allowance:** A numeric field with the value 0.
- Cooler ON:** A checkbox that is currently unchecked.
- Spindle rotation direction:** Radio buttons for 'CW' (selected) and 'CCW'.
- Spindle RPM:** A numeric field with the value 200.

The screenshot shows the 'Drilling Parameters' dialog box. It contains the following fields and controls:

- Number of cycle:** A dropdown menu with 'Drill' selected.
- Safety plane height:** A numeric field with the value 5.
- Working feed rate height:** A numeric field with the value 0.
- Increment (after pull-out):** A numeric field with the value 10.
- Reduction coefficient:** A numeric field with the value 1.
- Pull out after next pass:** A numeric field with the value 100.

Z of the working feed rate. Specifies the height at which the working feed rate is turned on.

Increment (for deep drilling). The depth of the step when performing deep drilling.

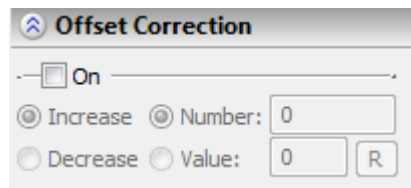
Reduction coefficient. Specifies the reduction coefficient for the depth of the step when performing deep drilling.

Pullout. The value of the pullout of the tool at each step of deep drilling. Specified as a percentage of the increment.


Offset correction.





On. When this flag is enabled, parameters of the offset correction can be used. Correction can be for the **increase** and **decrease**. It is possible to specify correction by the **number** (in this case, upon machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number indicated by the user) and by the **value** (in this case, the user specifies the precise value of the offset of the tool).

R – use the radius of the selected tool.

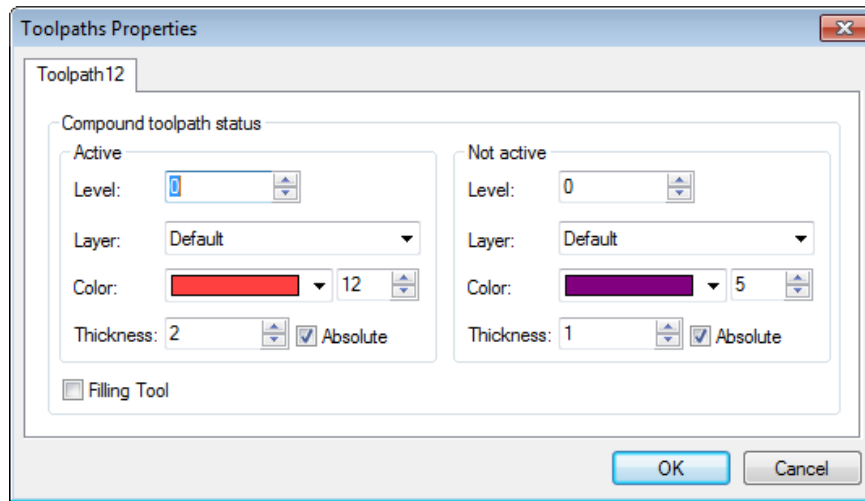


Special axial drilling

To create a toolpath of special axial drilling, select the  option from the drop-down list "Axial drilling". On the screen the dialog window of the toolpath parameters will appear which the user can change as desired, and the system will transition to the mode of selection of the path.

	<End>	Finish editing
	<P>	Toolpath properties
	<2>	Path selection filter
	<Esc>	Exit the command

In the **Toolpath properties** window the user can specify the properties for two groups of elements – active and nonactive toolpaths. It is possible to select the thickness of displayed elements of the toolpath, their color, layer, and level of location of the toolpath on the drawing.




Path selection **filter** changes the possibility of selection of the given element on the drawing as a geometric basis for construction of the toolpath. The image of the active filter is highlighted with a frame.

When creating the given toolpath, the path that goes along the axis of the hole through its entire depth is used as an element of the drawing.

Toolpath parameters.

Toolpath. Name of a toolpath.

File's tool. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the **Tool:** button.

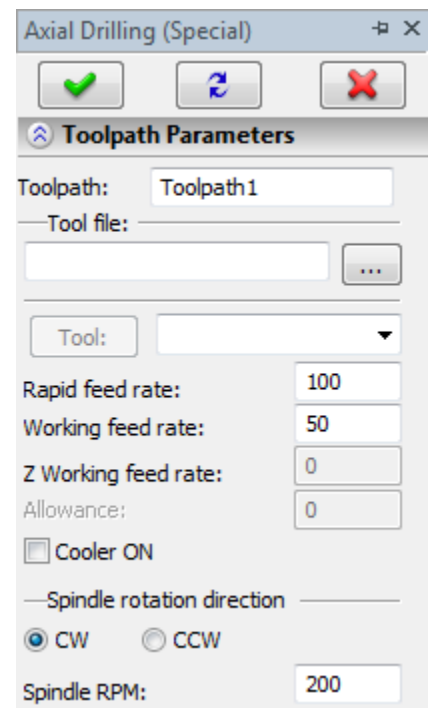
It is to be noted that the system will offer the user to select only that tool which can be utilizable for the chosen type of machining.

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary displacements.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of translation of the tool when executing movements of cutting.

Cooler on. Turning on/off the supply of the cooling agent upon machining.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.



Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.

Drilling parameters.

Undertravel value. The distance from the part to the cutting rim of the tool, before the working translation along the axis of the hole.

Drilling Parameters	
Undertravel value:	0.5
Bottom end (drilling depth):	10
Reduction coefficient:	1
Pull out after next pass:	1
<input type="checkbox"/> Retreat height:	5


Depth of cutting-in. The depth of the step for the deep drilling or the depth measured up to the end point of the hole being machined upon drilling.





Reduction coefficient. Specifies the reduction coefficient for the depth of the step when performing deep drilling.

Pullout. The value of the pullout of the tool at each step of deep drilling.

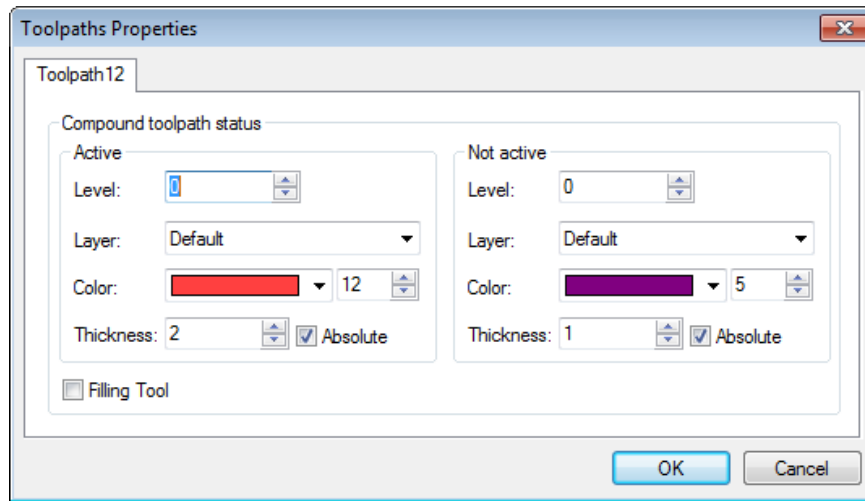
Safety value. Specifies the height of the lift above the machined surface when performing translations of the tool (optional).

"Cut off" Operation

To create a toolpath of cutoff, press the  button in the automenu. On the screen the dialog window of the toolpath parameters will appear which the user can change as desired, and the system will transition to the mode of selection of the path.

	<End>	Finish editing
	<P>	Toolpath properties
	<2>	Path selection filter
	<Esc>	Exit the command

In the **Toolpath properties** window the user can specify the properties for two groups of elements – active and nonactive toolpaths. It is possible to select the thickness of displayed elements of the toolpath, their color, layer, and level of location of the toolpath on the drawing.



Path selection **filter** changes the possibility of selection of the given element on the drawing as a geometric basis for construction of the toolpath. The image of the active filter is highlighted with a frame.

The path is constructed perpendicular to the rotation axis of the part and must be directed from the external contour towards the rotation axis of the part.

Toolpath parameters.

Toolpath. Name of a toolpath.

File's tool. With the help of the button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the **Tool:** button.

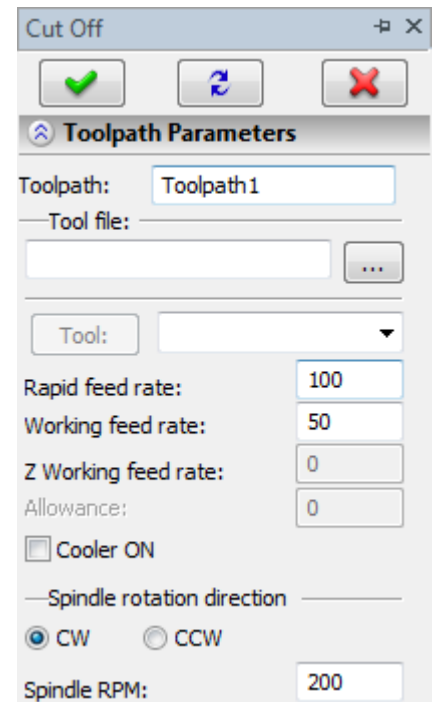
It is to be noted that the system will offer the user to select only that tool which can be utilizable for the chosen type of machining.

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary displacements.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of translation of the tool when executing movements of cutting.

Cooler on. Turning on/off the supply of the cooling agent upon machining.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

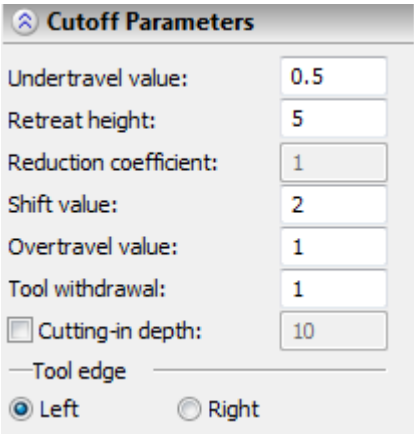


Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.

Cutoff parameters.

Undertravel value. The distance from the part to the cutting rim of the tool after which the working feed rate is turned on.

Safety height. The magnitude of the withdrawal of the tool from the part. It can be used together with the undertravel parameter.



Reduction coefficient. The reduction coefficient for the depth of the cutting-in at each pass.

Shift value. The distance between two vertical passes when machining is performed with gradual radial cutting-in.

Overtravel value. The distance from the part to the cutting rim of the tool after execution of the working pass.


Tool's pullout. The magnitude of the withdrawal of the tool.




Cutting-in depth. The magnitude of the cutting-in of the tool at the first pass (the cutoff may be completed during a single pass).


Tool edge: Left – locking the tool to the left edge.

Right – locking the tool to the right edge.

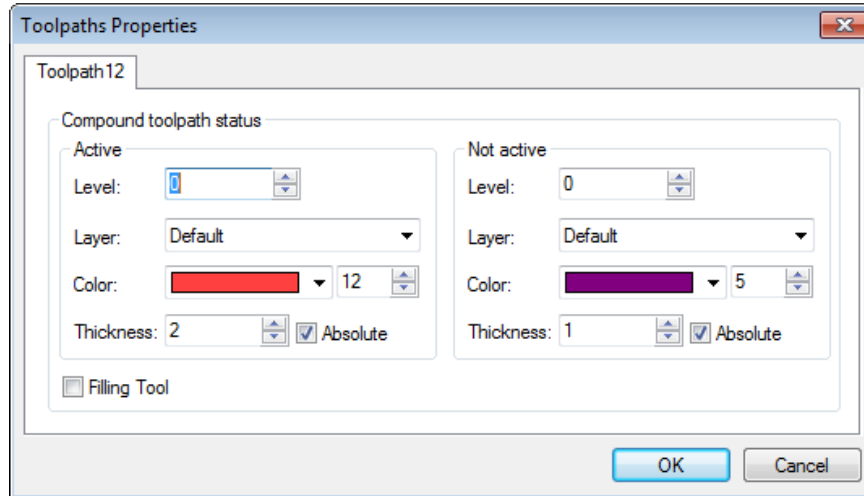
Going to a Specified Point

To create a toolpath of translation to a point with specified parameters, press the  button in the automenu. On the screen the dialog window of the toolpath parameters will appear which the user can change as desired, and the system will transition to the mode of selection of the node.

	<End>	Finish editing
	<P>	Toolpath properties
	<3>	Path selection filter

	<Esc>	Exit the command
---	-------	------------------



In the **Toolpath properties** window the user can specify the properties for two groups of elements – active and nonactive toolpaths. It is possible to select the thickness of displayed elements of the toolpath, their color, layer, and level of location of the toolpath on the drawing.



Path selection **filter** changes the possibility of selection of the given element on the drawing as a geometric basis for construction of the toolpath. The image of the active filter is highlighted with a frame.

Toolpath parameters.

Toolpath. Name of a toolpath.

File's tool. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilizable for the chosen type of machining.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of translation of the tool when executing movements of cutting. This parameter is active for the motion on working feed rate **G01**.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.

Moving type.

Selection of the method of moving (**G00** – on rapid feed rate, **G01** – on working feed rate) and the order in which the translations along the axes are performed (**XYZ** – along three axes simultaneously; **XY_Z** – first along the **X** and **Y** axes, then along the **Z**-axis; **Z_XY** – first along the **Z**-axis, then along the **X** and **Y** axes).


Offset correction

On. When this flag is enabled, parameters of the offset correction can be used. Correction can be for the **increase** and **decrease**. It is possible to specify correction by the **number** (in this case, upon machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number indicated by the user) and by the **value** (in this case, the user specifies the precise value of the offset of the tool).
















R – use the radius of the selected tool.


DRILLING


To define a toolpath and a control sequence for drilling-type machining, use the command:











Icon	Ribbon
	CNC → Machining → Drilling
Keyboard	Textual Menu
	CNC>2D, 2-1/2D and 4D machining>Drilling

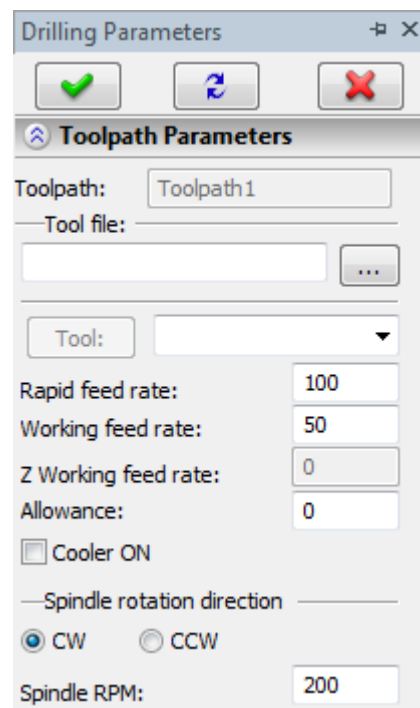
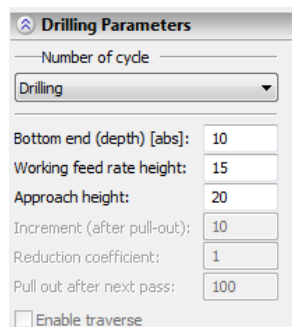
Upon calling the command, the following options become available in the automenu:

	<P>	Default toolpath parameters
	<T>	Technological toolpath
	<A>	Create drilling
	<O>	Standard OLIVETTI cycles
		Standard BREDLEY cycles
	<M>	Standard MAHO cycles
	<E>	Standard EXCEL cycles
	<C>	Select path for drilling cycle 2C42
	<R>	Select path for drilling cycle Size - 2M
	<F>	Select path for FANUC cycles
	<N>	Select path for drilling cycles E2000 CNC
	<U>	Select path for drilling cycles VECTOR 90
	<Z>	User-defined drilling cycle
	<Q>	Select point for tool moving
	<Esc>	Exit command


After the command is invoked, the dialog window "Drilling" will appear on the screen. Parameters selected in this window will be active **by default** for all newly created toolpaths of drilling. When creating a specific toolpath, however, these parameters can be modified. The user can change them as desired and save these changes by pressing the  button.





Use of the  option is described in the "Technological toolpaths" chapter.

The options , , , , , , , ,  and  are used for creating machining cycles. Working with these options is described in the "Machining cycles" chapter.

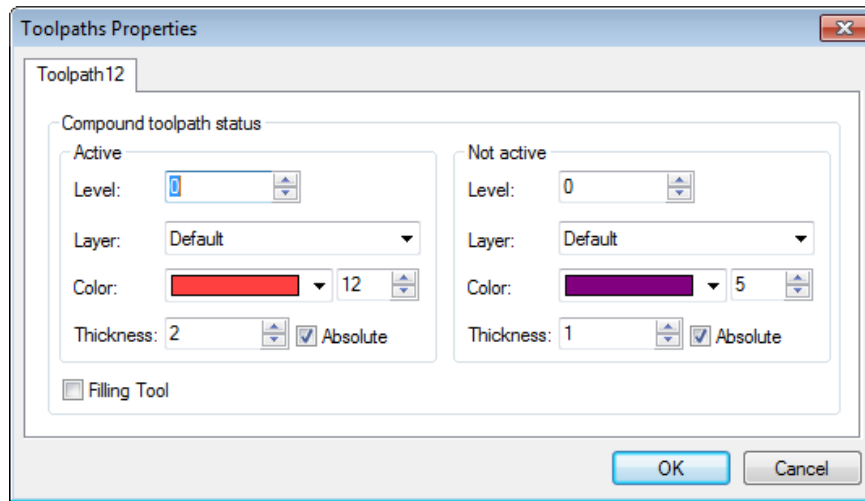


Toolpath Creation

To create a toolpath of drilling, press the  button in the automenu. On the screen the dialog window of the toolpath parameters will appear which the user can change as desired, and the system will transition to the mode of selection of the path.

	<End>	Finish editing
	<P>	Toolpath properties
	<2>	Path selection filter
	<Esc>	Exit the command

In the **Toolpath properties** window the user can specify the properties for two groups of elements – active and nonactive toolpaths. It is possible to select the thickness of displayed elements of the toolpath, their color, layer, and level of location of the toolpath on the drawing.


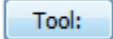


Path selection **filter** changes the possibility of selection of the given element on the drawing as a geometric basis for construction of the toolpath. The image of the active filter is highlighted with a frame.

In case of drilling, the contour being machined is merely a path of the drill's displacement (or another drilling tool). When constructing a path, it should be taken into account that at the initial node of the path the drilling will not be carried out, and the segment between the first and second nodes will be put to the control sequence as an additional translation. Thus, if the path consists of $n + 1$ nodes, the drilling will be executed n times, starting with the second node.

Toolpath parameters.

Toolpath. Name of a toolpath.

File's tool. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilizable for the chosen type of machining.

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary displacements.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of translation of the tool when executing movements of cutting.

Cooler on. Turning on/off the supply of the cooling agent upon machining.

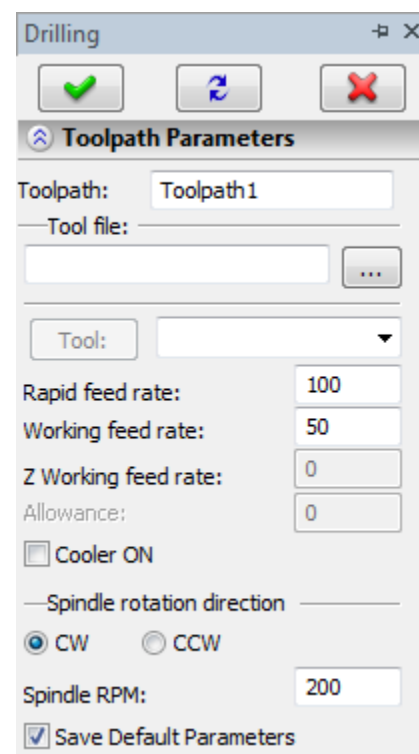
Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.

Drilling parameters.

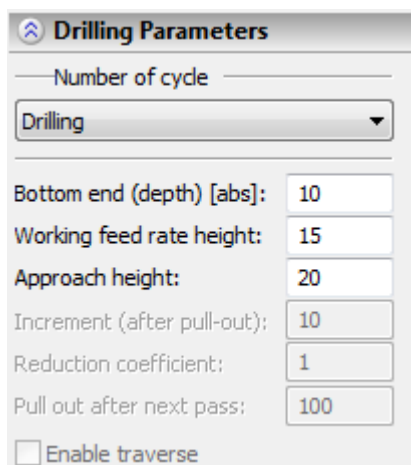
Cycle number. Selection of the drilling cycle or the cycle of deep drilling.

Depth of drilling. Specifies the height measured from the zero of the machining station up to the lowest point of the machined hole.



The screenshot shows the 'Drilling' dialog box with the 'Toolpath Parameters' tab selected. It contains the following fields and controls:

- Buttons: Green checkmark, blue refresh, and red X.
- Section: **Toolpath Parameters**
- Toolpath: Text field containing 'Toolpath1'.
- Tool file: Text field with a browse button (three dots).
- Tool: Drop-down menu.
- Rapid feed rate: Text field with value '100'.
- Working feed rate: Text field with value '50'.
- Z Working feed rate: Text field with value '0'.
- Allowance: Text field with value '0'.
- Cooler ON: Check box (unchecked).
- Spindle rotation direction: Radio buttons for 'CW' (selected) and 'CCW'.
- Spindle RPM: Text field with value '200'.
- Save Default Parameters: Check box (checked).



The screenshot shows the 'Drilling Parameters' dialog box. It contains the following fields and controls:

- Section: **Drilling Parameters**
- Number of cycle: Drop-down menu with 'Drilling' selected.
- Bottom end (depth) [abs]: Text field with value '10'.
- Working feed rate height: Text field with value '15'.
- Approach height: Text field with value '20'.
- Increment (after pull-out): Text field with value '10'.
- Reduction coefficient: Text field with value '1'.
- Pull out after next pass: Text field with value '100'.
- Enable traverse: Check box (unchecked).

Working feed rate height. Specifies the height at which the working feed rate is turned on.

Approach height. Specifies the height of the location of the plane in which the accelerated horizontal translation of the tool is carried out, i.e., the translation from one hole to the other.

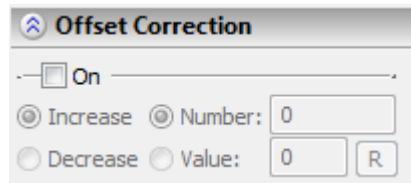
Increment. The depth of the step when performing deep drilling.

Reduction coefficient. Specifies the reduction coefficient for the depth of the step when performing deep drilling.

Pullout. The magnitude of the withdrawal of the tool at each step of the deep drilling. It is specified as a percentage of the increment.

Offset correction.


On. When this flag is enabled, parameters of the offset correction can be used. Correction can be for the **increase** and **decrease**. It is possible to specify correction by the **number** (in this case, upon machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number indicated by the user) and by the **value** (in this case, the user specifies the precise value of the offset of the tool).









R – use the radius of the selected tool.



MILLING (2-2.5D)


To define the toolpath and the control sequence for milling, use the command:

Icon	Ribbon
	CNC → Machining → Milling
Keyboard	Textual Menu
	CNC>2D, 2-1/2D and 4D machining>Milling


Upon calling the command, the following options become available in the automenu:

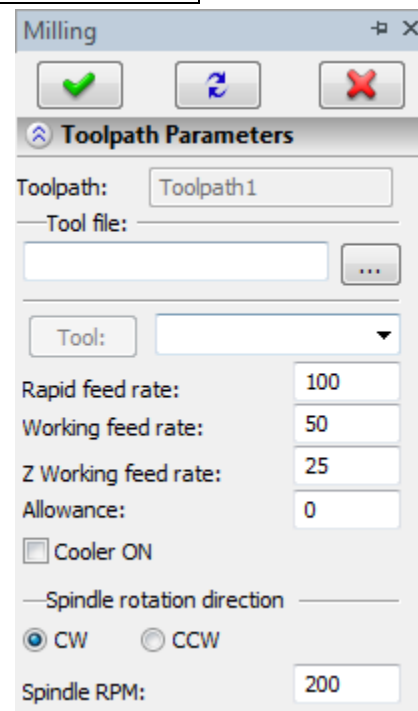
	<End>	Finish editing
	<T>	Technological toolpath
	<L>	Plane milling
	<C>	Contour milling
	<K>	Pocket milling
	<W>	Select path for machining spiral groove

	<Q>	Select point for tool moving
	<Esc>	Exit command





The  option is used for creating technological (service) toolpaths. Such toolpaths are necessary, for example, for defining additional commands in the control sequence for servicing technological processes.

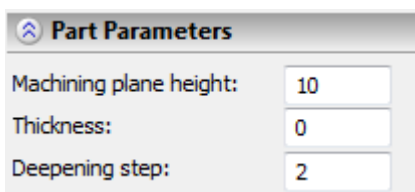
Detailed description of technological toolpath creation is provided in the "Technological toolpaths" section that covers all types of machining.

After the command is invoked, the dialog window "Milling" will appear on the screen. Parameters selected in this window will be active **by default** for all newly created toolpaths of drilling. When creating a specific toolpath, however, these parameters can be modified. The user can change them as desired and save these changes by pressing the  button.



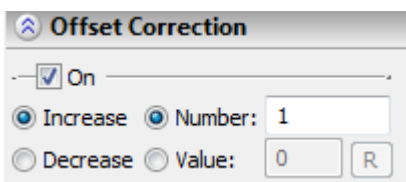
The "Milling" dialog window contains the following controls:

- Buttons:  (Save),  (Reset),  (Cancel).
- Section: **Toolpath Parameters**
- Toolpath:
- Tool file: 
- Tool:
- Rapid feed rate:
- Working feed rate:
- Z Working feed rate:
- Allowance:
- ☐ Cooler ON
- Spindle rotation direction: ☒ CW ☐ CCW
- Spindle RPM:



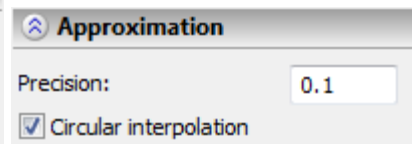
The "Part Parameters" dialog window contains the following controls:

- Machining plane height:
- Thickness:
- Deepening step:



The "Offset Correction" dialog window contains the following controls:

- ☒ On
- ☒ Increase ☒ Number:
- ☐ Decrease ☐ Value:




The "Approximation" dialog window contains the following controls:

- Precision:
- ☒ Circular interpolation


Toolpath Creation

Once the user is done with changes in Default Machining Parameters, one needs to select one of the four types of milling:


Plane milling 






Contour milling 

Pocket milling 

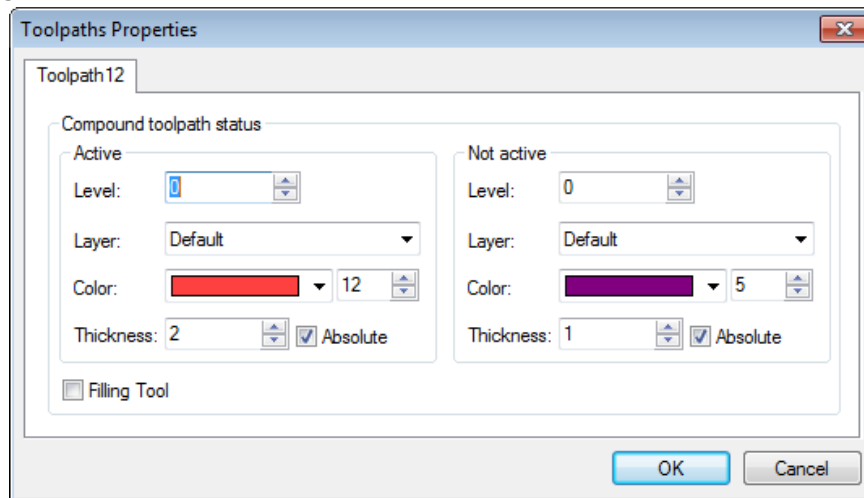
Spiral groove milling 

Plane Milling

To create a toolpath of milling a plane, press the  button in the automenu. On the screen the dialog window of the toolpath parameters will appear which the user can change as desired, and the system will transition to the mode of selection of the hatch.

	<End>	Finish editing
	<P>	Toolpath properties
	<1>	Hatch selection filter
	<3>	Change initial point of a toolpath
	<Esc>	Exit the command


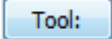
In the **Toolpath properties** window the user can specify the properties for two groups of elements – active and nonactive toolpaths. It is possible to select the thickness of displayed elements of the toolpath, their color, layer, and level of location of the toolpath on the drawing.



Hatch selection **filter** changes the possibility of selection of the given element on the drawing as a geometric basis for construction of the toolpath. The image of the active filter is highlighted with a frame.

Toolpath parameters.

Toolpath. Name of a toolpath.

File's tool. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilizable for the chosen type of machining.

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary displacements.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of translation of the tool when executing the movements of cutting.

Working feed rate (on Z dir) is specified as a specific numerical value in units supported by the CNC center and sets up the speed of translation of the tool when executing movements of cutting along the Z-axis. It is used for the external postprocessors.

Allowance. This parameter defines the distance between original and equidistant contours.

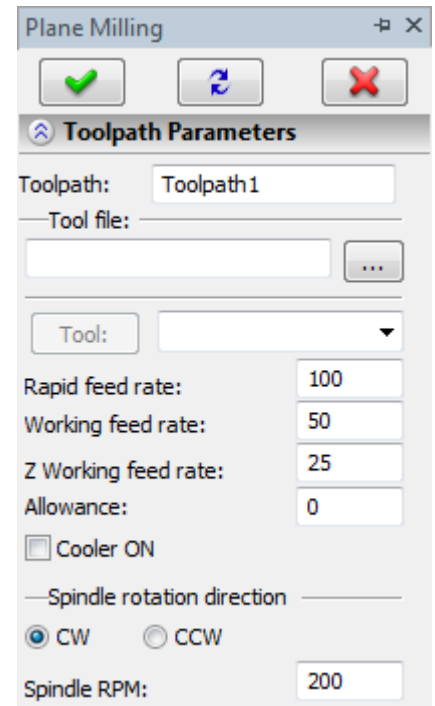
Cooler on. Turning on/off the supply of the cooling agent upon machining.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.

Pass parameters.

Pass type. Specifies one of the following types: **zigzag** (along the X-axis or along the Y-axis); **loop** (along the X-axis or along the Y-axis); **spiral** (along the contour or against the contour, and also convergence inwards or divergence outwards); **zigzag with a traversal** (along the X-axis or along the Y-axis); **double loop** (along the X-axis or along the Y-axis); **zigzag with a traversal without taking into account the contours of the cutoff** (along the X-axis or along the Y-axis).



Pass Parameters

Pass type: Zigzag in X

Tool lift: ☐ Abs. ☒ Incr. 5

☐ Step of passes: 10

☒ Num. of passes: 10

☐ Crest height: 0.5

Processing overlap: 1

Limits

Tool lift. Specifies the distance to the safety plane (in absolute coordinates or in increments).

Step of passes. Specifies the magnitude of the step between the passes.

Number of passes. Specifies the number of passes of the tool. The value of this parameter can be specified automatically based on the specified conditions: **crest height** or **processing overlap**, taking into account the tool selected by the user upon pressing the button of recalculation [N].

Limits. Described in more detail below.

Part parameters.

Here is specified the **height of machining plane** (measured from the zero plane), i.e., the height of the horizontal working translations of the mill; **Thickness** and **Deepening step** can also be specified for milling a plane.

Part Parameters

Machining plane height: 10

Thickness: 0

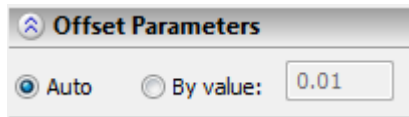
Deepening step: 2

When **Thickness** and **Deepening step** are specified, the passes of layerwise material cutoff or material selection are formed. Moreover, cutting-in between the layers of the material takes place according to the selected by the user type of the pass. The number of layers is calculated automatically, and moreover, if the thickness of the layer being removed is not a multiple of the deepening step, the system calculates the maximum possible number of layerwise passes within the framework of the selected thickness of the layer, and places the last pass along the bottom height of the material being removed.

If parameters **Thickness** and **Deepening step** are not specified, T-FLEX CAM forms an elementary toolpath of machining a plane, without the layerwise removal or selection.

Offset parameters

Automatically. The system calculates the toolpath of machining that has an offset from the boundary of the machined part equal to the radius of the mill.



By value. The specific value of the offset.

Cutting off a contour.

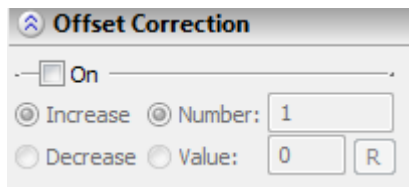
Cutoff external and internal. It is used in case of using the passes of zigzag or loop type. The external cutoff implies taking into account the external contour of the machined surface, and internal cutoff takes into account internal contours of the machined surface; the cutoff is enabled automatically when specifying parameters in the dialog window «Limits».



Offset correction

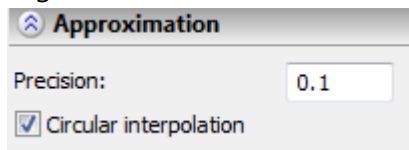
On. When this flag is enabled, parameters of the offset correction can be used. Correction can be for the **increase** and **decrease**. It is possible to specify correction by the **number** (in this case, upon machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number indicated by the user) and by the **value** (in this case, the user specifies the precise value of the offset of the tool).

R – use the radius of the selected tool.



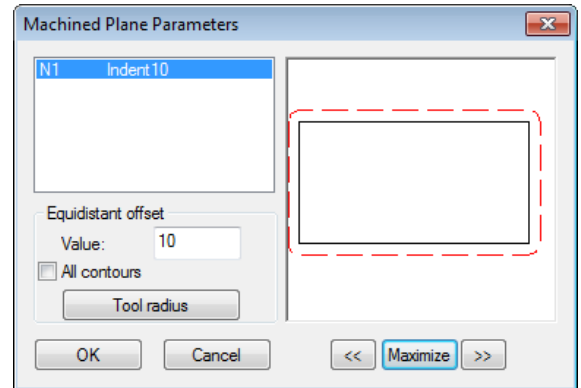
Approximation

Precision of the approximation is specified as a numerical value in units supported by the CNC center and constitutes the magnitude of the caving (maximum distance from the arc of the circle to be straightened to the approximating line).




Circular interpolation. In case when the contour being machined has circles, arcs, splines, and other geometrical elements having similar shape, the selection of this parameter will allow us to obtain the control sequence that contains frames of the circular interpolation.


In the dialog window "Machined plane parameters" (the [Limits] button), in the interactive mode, we specify the allowances for the contours of internal and external cutoffs of the surface being machined.









Change of initial point of a toolpath

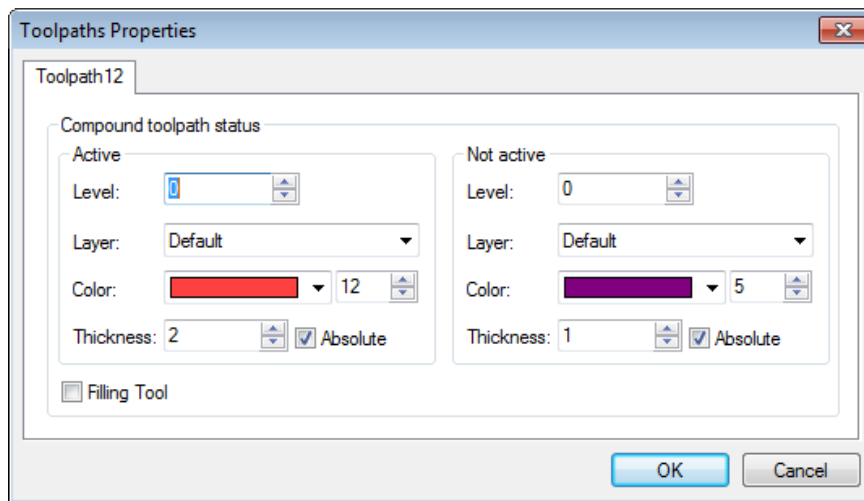
When creating or editing the toolpath of milling a plane, the user has a possibility to specify the initial point. By default, the system selects the first node of the hatch as an initial point of the toolpath. The user can refuse this strategy of calculation and, by pressing the  button, indicate another point of the contour as an initial point.

Contour Milling

To create a toolpath of milling a contour, press the  button in the automenu. On the screen the dialog window of the toolpath parameters will appear which the user can change as desired, and the system will transition to the mode of selection of the path or the hatch.

	<End>	Finish editing
	<P>	Toolpath properties
	<1>	Hatch selection filter
	<2>	Path selection filter
	<3>	Change initial point of a toolpath
	<Esc>	Exit the command


In the **Toolpath properties** window the user can specify the properties for two groups of elements – active and nonactive toolpaths. It is possible to select the thickness of displayed elements of the toolpath, their color, layer, and level of location of the toolpath on the drawing.



Filter for selection of a construction element changes the possibility of selection of the given element on the drawing as a geometric basis for construction of the toolpath. Filters can be active both in pairs (then both paths and hatches can be selected) and separately (only paths or only hatches). The image of the active filter is highlighted with a frame.

Toolpath parameters.

Toolpath. Name of a toolpath.

File's tool. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the **Tool:** button.

It is to be noted that the system will offer the user to select only that tool which can be utilizable for the chosen type of machining.

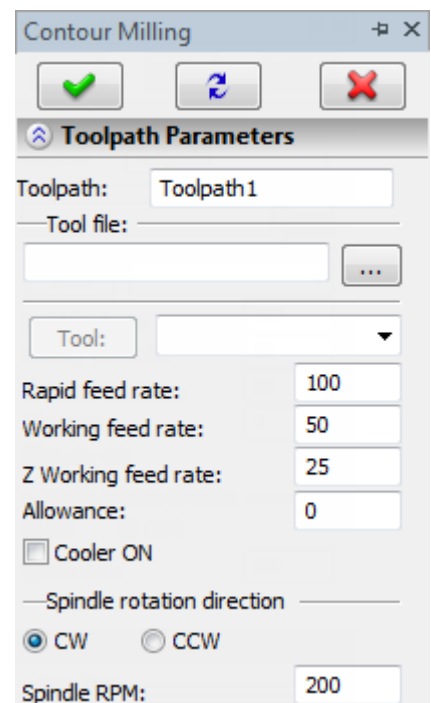
Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary displacements.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of translation of the tool when executing movements of cutting.

Working feed rate (on Z dir) is specified as a specific numerical value in units supported by the CNC center and sets up the speed of translation of the tool when executing movements of cutting along the Z-axis. It is used for external postprocessors.

Allowance. This parameter defines the distance between original and equidistant contours.

Cooler on. Turning on/off the supply of the cooling agent upon machining.



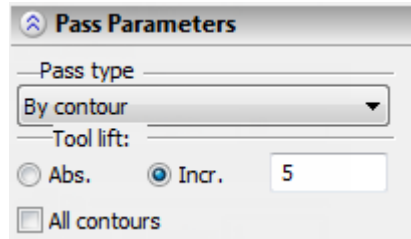
Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.

Pass parameters.

Pass type. Along the contour (coincides with the direction of the construction element) or against the contour (does not coincide).

Tool lift. Specifies the distance to the safety plane (in absolute coordinates or in increments).



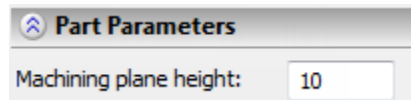
The 'Pass Parameters' dialog box contains the following controls:

- Pass type:** A dropdown menu currently set to 'By contour'.
- Tool lift:** A section with two radio buttons: 'Abs.' (unselected) and 'Incr.' (selected). To the right of the 'Incr.' button is a text input field containing the value '5'.
- All contours:** An unchecked checkbox.

All contours. It is used for milling all the contours of the compound hatches (otherwise – only external contour).

Part parameters.

Here is specified the **height of machining plane** (from the zero plane), i.e., the height of the horizontal working translations of the mill.



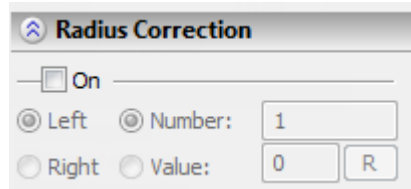
The 'Part Parameters' dialog box contains the following control:

- Machining plane height:** A text input field containing the value '10'.

Radius correction.

On. When this flag is enabled, parameters of the radius correction can be used. Correction can be **left** and **right**. It is possible to specify correction by the **number** (in this case, upon machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number indicated by the user) and by the **value** (in this case, the user specifies the precise value of the radius of rounding of the cutting edges of the mill).

R – use the radius of the selected tool.



The 'Radius Correction' dialog box contains the following controls:

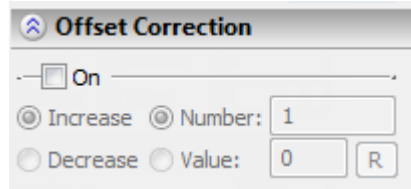
- On:** An unchecked checkbox.
- Left:** A selected radio button.
- Number:** A text input field containing the value '1'.
- Right:** An unselected radio button.
- Value:** A text input field containing the value '0'.
- R:** A button next to the 'Value' field.

Offset correction.

On. When this flag is enabled, parameters of the offset correction can be used. Correction can be for the **increase** and **decrease**. It is possible to specify correction by the **number** (in this case, upon machining, the value of the correction will be used written in the table of correctors of the

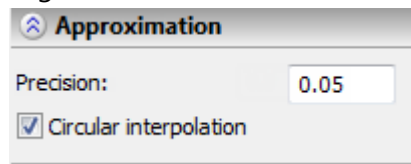
machining station in the row which has the number indicated by the user) and by the **value** (in this case, the user specifies the precise value of the offset of the tool).

R – use the radius of the selected tool.



Approximation.

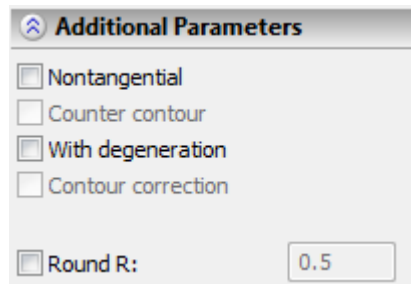
Precision of the approximation is specified as a numerical value in units supported by the CNC center and constitutes the magnitude of the caving (maximum distance from the arc of the circle to be straightened to the approximating line).



Circular interpolation. In case when the contour being machined has circles, arcs, splines, and other geometrical elements having similar shape, the selection of this parameter will allow us to obtain the control sequence that contains frames of the circular interpolation.

Additional parameters.

Precise tracing of angles. The set-up of this parameter specifies deletion of the tangential transitions (joining elements of the contour tangentially) in the toolpath that is created by means of an equidistant offset from the original contour (when using the allowance or correction by the value for the tool's radius).

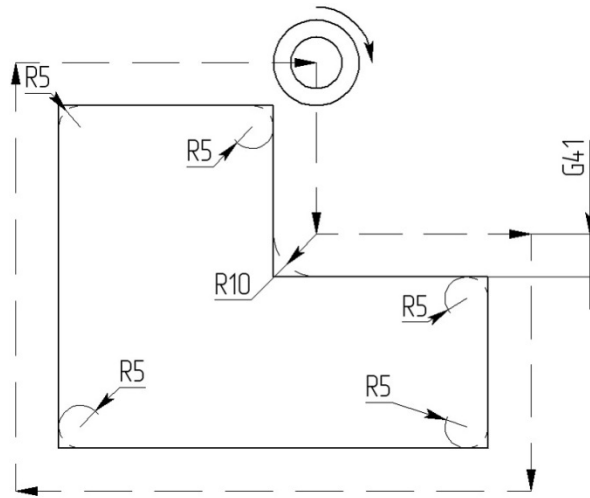


Without deletion of micro joints. This parameter enables restoration of the degenerated elements of the toolpath that is created by means of an equidistant offset from the original contour (when using the allowance or correction by the value for the tool's radius).

R of rounding. Perform rounding of the specified radius. This parameter allows us to inscribe circles of the specified radii in the places of non tangential transitions for internal and external angles of the contour. This parameter will be described in more detail below.


Round in radius (R of rounding). This parameter allows us to inscribe the circles of the specified radius in the places of non tangential transitions (this command is active in all types of wire-cut machining). In two







parameter boxes the values of rounding radii of external and internal angles of a contour are specified. Usually, the first box is used to specify the rounding radius of external angles of contour ($>180^\circ$), and the second box is used to specify the value of rounding radius of internal angles ($<180^\circ$). However, this designation of boxes can be changed depending on direction of pass along the contour and direction of correction on tool radius.



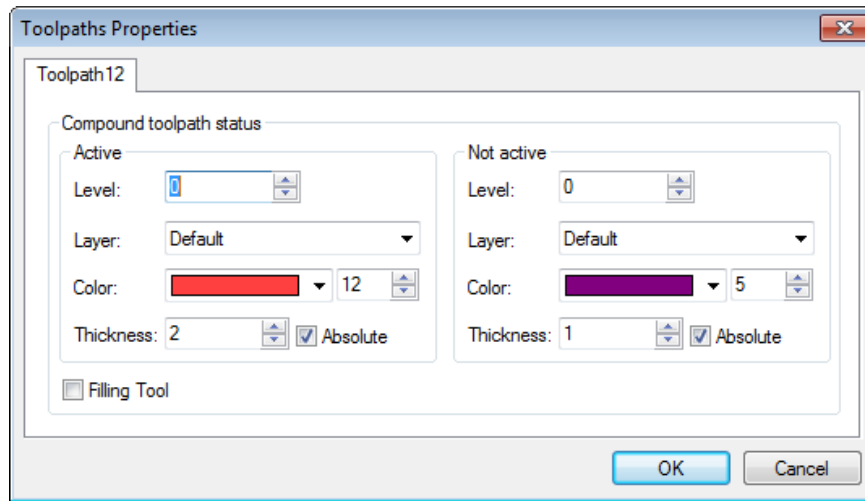
In the example shown above the radius 5 is set for rounding external angles and radius 10 for rounding internal angles.

Pocket Milling



To create a toolpath of milling a pocket, press the  button in the automenu. On the screen the dialog window of the toolpath parameters will appear which the user can change as desired, and the system will transition to the mode of selection of the path or the hatch.

	<End>	Finish editing
	<P>	Toolpath properties
	<1>	Hatch selection filter
	<2>	Path selection filter
	<3>	Change initial point of a toolpath
	<Esc>	Exit the command

In the **Toolpath properties** window the user can specify the properties for two groups of elements – active and nonactive toolpaths. It is possible to select the thickness of displayed elements of the toolpath, their color, layer, and level of location of the toolpath on the drawing.


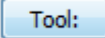


Filter for selection of a construction element changes the possibility of selection of the given element on the drawing as a geometric basis for construction of the toolpath. Filters can be active in pairs (in this case, both paths and hatches can be selected) and separately (only paths or only hatches). The image of the active filter is highlighted with a frame.

The toolpath of milling a pocket is created on the basis of either two paths or two hatches (lower contour and upper contour of the pocket). After selection of the first path, the  option for specification of the second path will become available in the automenu. Similarly, in case of hatches another option  is provided.

Toolpath parameters.

Toolpath. Name of a toolpath.

File's tool. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilizable for the chosen type of machining.

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary displacements.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of translation of the tool when executing the movement of cutting.

Working feed rate (on Z dir) is specified as a specific numerical value in units supported by the CNC center and sets up the speed of translation of the tool when executing movements of cutting along the Z-axis. It is used for external postprocessors.

Allowance. This parameter defines the distance between original and equidistant contours.

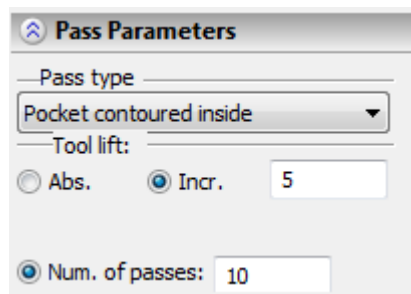
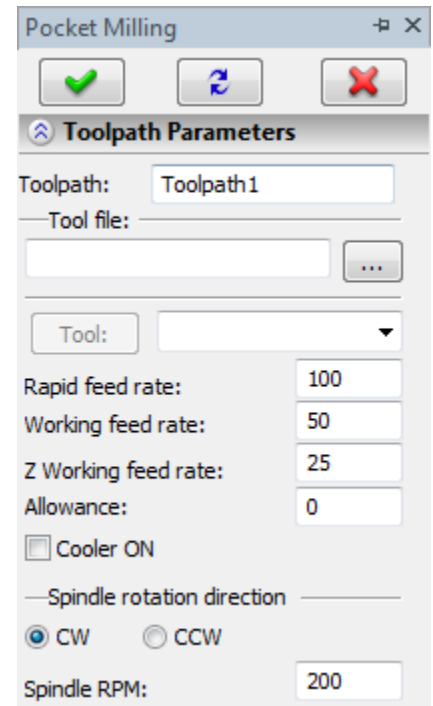
Cooler on. Turning on/off the supply of the cooling agent upon machining.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.

Pass parameters.

Pass type. Here it is possible to select how the pocket is to be milled: inwards, outwards, along the contour or against the contour in different combinations.

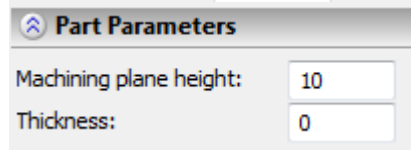


Tool lift. Specifies the distance to the safety plane (in absolute coordinates or in increments).

Number of passes. Specifies the number of passes of the tool.

Pass parameters.

Here is specified the **height of machining plane** (from the zero plane), i.e., the height of the horizontal working translations of the mill and the **thickness** of the part.

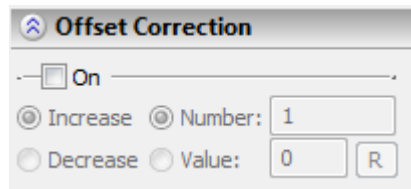


The 'Part Parameters' dialog box contains two input fields: 'Machining plane height' with a value of 10, and 'Thickness' with a value of 0.

Offset correction.

On. When this flag is enabled, parameters of the offset correction can be used. Correction can be for the **increase** and **decrease**. It is also possible to specify correction by the **number** (in this case, upon machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number indicated by the user) and by the **value** (in this case, the user specifies the precise value of the offset of the tool).

R – use the radius of the selected tool.

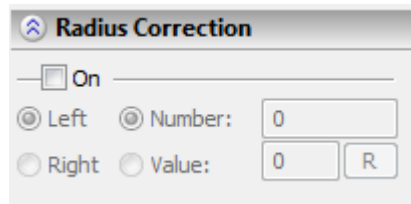


The 'Offset Correction' dialog box features an 'On' checkbox, which is checked. Below it are two radio button options: 'Increase' (selected) and 'Decrease'. To the right of 'Increase' is a 'Number' input field with the value 1. To the right of 'Decrease' is a 'Value' input field with the value 0 and an 'R' button.

Radius correction.

On. When this flag is enabled, parameters of the radius correction can be used. Correction can be **left** and **right**. It is possible to specify correction by the **number** (in this case, upon machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number indicated by the user) and by the **value** (in this case, the user specifies the precise value of the radius of rounding of the cutting edges of the mill).

R – use the radius of the selected tool.

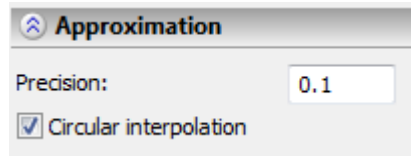


The 'Radius Correction' dialog box features an 'On' checkbox, which is checked. Below it are two radio button options: 'Left' (selected) and 'Right'. To the right of 'Left' is a 'Number' input field with the value 0. To the right of 'Right' is a 'Value' input field with the value 0 and an 'R' button.

Approximation.

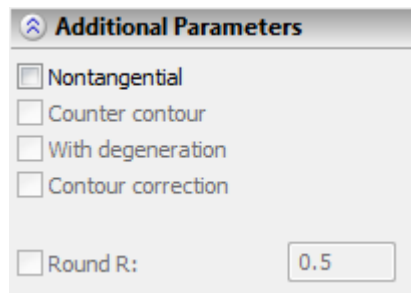
Precision of the approximation is specified as a numerical value in units supported by the CNC center and constitutes the magnitude of the caving (maximum distance from the arc of the circle to be straightened to the approximating line).

Circular interpolation. In case when the contour being machined has circles, arcs, splines, and other geometrical elements having similar shape, the selection of this parameter will allow us to obtain the control sequence that contains frames of the circular interpolation.








Additional parameters.

Precise tracing of angles. The set-up of this parameter specifies deletion of the tangential transitions (joining elements of the contour tangentially) in the toolpath that is created by means of an equidistant offset from the original contour (when using the allowance or correction by the value for the tool's radius).

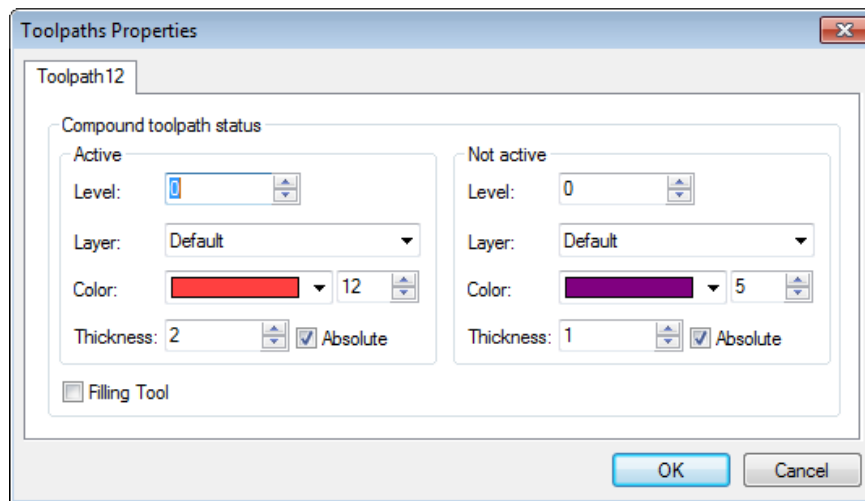


Spiral groove milling

To create a toolpath of spiral groove milling, press the  button in the automenu. On the screen the dialog window of the toolpath parameters will appear which the user can change as desired, and the system will transition to the mode of selection of the path.

	<End>	Finish editing
	<P>	Toolpath properties
	<2>	Path selection filter
	<Esc>	Exit command

In the **Toolpath properties** window the user can specify the properties for two groups of elements – active and nonactive toolpaths. It is possible to select the thickness of displayed elements of the toolpath, their color, layer, and level of location of the toolpath on the drawing.



Path selection **filter** changes the possibility of selection of the given element on the drawing as a geometric basis for construction of the toolpath. The image of the active filter is highlighted with a frame.

Toolpath parameters.

Toolpath. Name of a toolpath.

Tool's file. With the help of the button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the button.

It is to be noted that the system will offer the user to select only that tool which can be utilizable for the chosen type of machining.

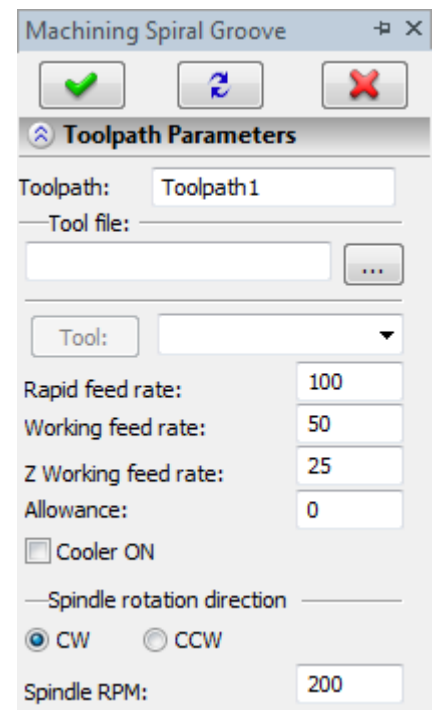
Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary displacements.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of translation of the tool when executing the movement of cutting.

Working feed rate (on Z dir) is specified as a specific numerical value in units supported by the CNC center and sets up the speed of translation of the tool when executing movements of cutting along the Z-axis. It is used for external postprocessors.

Allowance. This parameter defines the distance between original and equidistant contours.

Cooler on. Turning on/off the supply of the cooling agent upon machining.



Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.

Flute parameters.

Top point height. The Z-coordinate of the start of the spiral (it is specified with respect to the zero plane).

Flute Parameters

Top point height: 10

Bottom point height: -10

Step: 5

☒ CW ☐ CCW

Bottom point height. The Z-coordinate of the end of the spiral (it is specified with respect to the zero plane).

Step. Step of a spiral.

The direction of the coil (wind) of the spiral is specified using the **clockwise** or **counterclockwise** fields.

Offset parameters.

On. When this flag is enabled, parameters of the offset correction can be used. Correction can be for the **increase** and **decrease**. It is possible to specify correction by the **number** (in this case, upon machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number indicated by the user) and by the **value** (in this case, the user specifies the precise value of the offset of the tool).

R – use the radius of the selected tool.

Offset Correction

☒ On

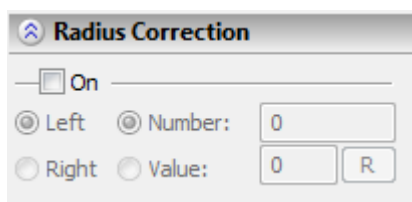
☒ Increase ☐ Number: 1

☐ Decrease ☐ Value: 0 R

Radius correction.

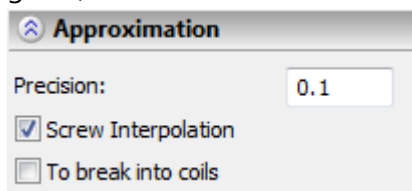
On. When this flag is enabled, parameters of the radius correction can be used. Correction can be **left** and **right**. It is possible to specify correction by the **number** (in this case, upon machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number indicated by the user) and by the **value** (in this case, the user specifies the precise value of the radius of rounding of the cutting edges of the mill).

R – use the radius of the selected tool.



Approximation.

Precision of the approximation is specified as a numerical value in units supported by the CNC center and constitutes the magnitude of the caving (maximum distance from the arc of the circle to be straightened to the approximating line).



Screw interpolation. If the CNC system of the machining station supports a screw interpolation, the use of this option will shorten the size of the text of the control sequence to more reasonable size and enhance the quality of the machining. In general case, when using the screw interpolation for machining a screw groove (flute), the frames with G22 command will be uploaded to the control sequence.

For example:

N30 G22X298.36Y44.34Z-0I112.54J0K8

Several CNC systems support the function of the screw interpolation, however, they don't support multicoil screw interpolation. In this case, it is recommended that the **Screw interpolation** option be enabled together with the **Break into coils** option. In this case, each coil of the spiral will be uploaded to the text of control sequence as a separate G22 frame.


For example:




N30 G22Z62I112.54J0


N35Z54I112.54J0

N40Z46I112.54J0

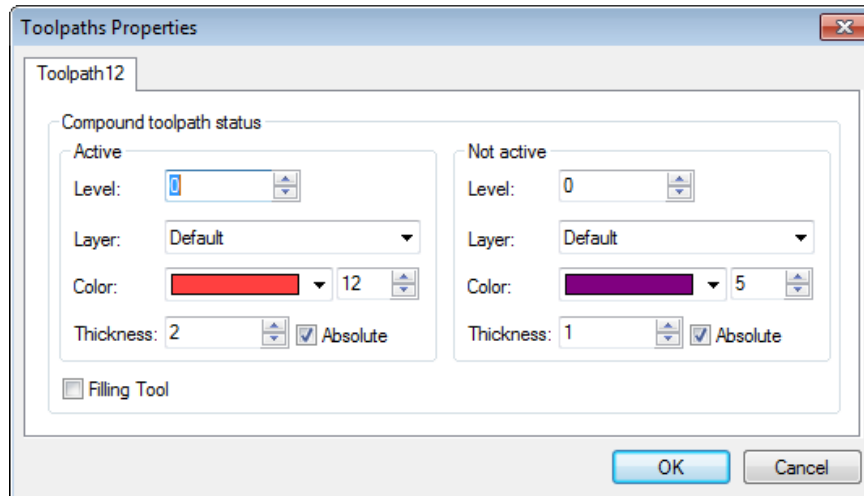
Going to a point with specified parameters

To create a toolpath of translation to a point with specified parameters, press the  button in the automenu. On the screen the dialog window of the toolpath parameters will appear which the user can change as desired, and the system will transition to the mode of selection of the node.

	<End>	Finish editing
	<P>	Toolpath properties
	<3>	Node selection filter

	<Esc>	Exit the command
---	-------	------------------


In the **Toolpath properties** window the user can specify the properties for two groups of elements – active and nonactive toolpaths. It is possible to select the thickness of displayed elements of the toolpath, their color, layer, and level of location of the toolpath on the drawing.



Node selection **filter** changes the possibility of selection of the given construction element on the drawing as a geometric basis for construction of the toolpath. The image of the active filter is highlighted with a frame.

Toolpath parameters.

Toolpath. Name of a toolpath.

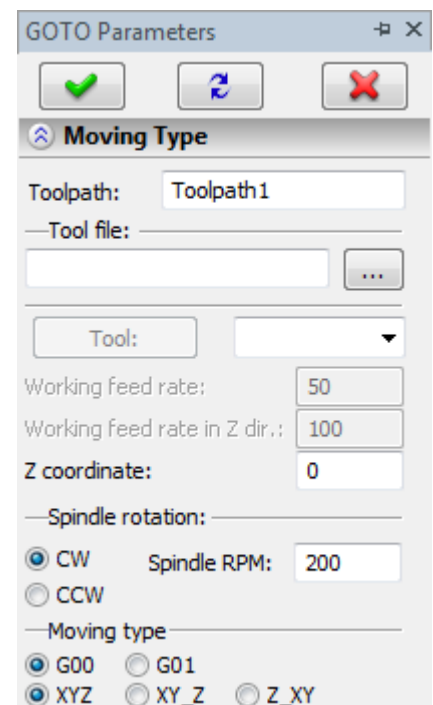
Tool's file. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the **Tool:** button.

It is to be noted that the system will offer the user to select only that tool which can be utilizable for the chosen type of machining.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of translation of the tool when executing the movement of cutting. This parameter is active for the motion on the working feed rate **G01**.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.



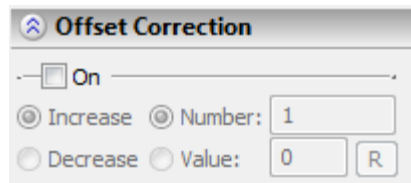
Moving type.

Selection of the method of moving (**G00** – on rapid feed rate, **G01** – on working feed rate) and the order in which translations along the axes are performed (**XYZ** – along three axes simultaneously; **XY_Z** – first along the **X** and **Y** axes, then along the **Z**-axis; **Z_XY** – first along the **Z**-axis, then along the **X** and **Y** axes).

Offset correction.

On. When this flag is enabled, parameters of the offset correction can be used. Correction can be for the **increase** and **decrease**. It is possible to specify correction by the **number** (in this case, upon machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number indicated by the user) and by the **value** (in this case, the user specifies the precise value of the offset of the tool).

R – use the radius of the selected tool.




ENGRAVING


To define a toolpath and a control sequence for engraving, use the command:

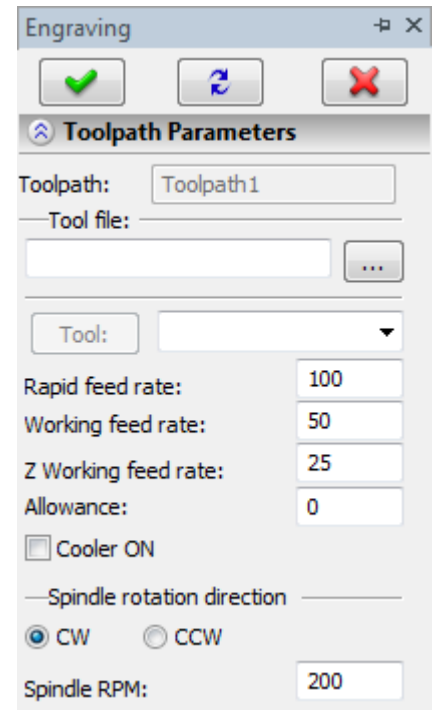
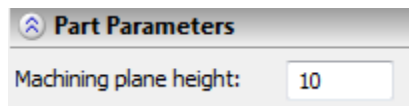
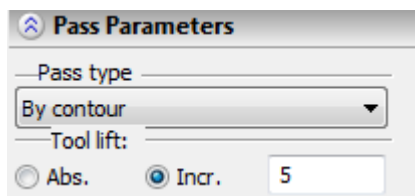
Icon	Ribbon
	CNC → Machining → Engraving
Keyboard	Textual Menu
	CNC>2D, 2-1/2D and 4D machining>Engraving


Upon calling the command, the following options become available in the automenu:






	<End>	Finish editing
	<T>	Technological toolpath
	<A>	Select text
	<Q>	Select point for tool moving
	<Esc>	Exit command

The  option is used for creating technological (service) toolpaths. Such toolpaths are necessary, for example, for defining additional commands in the control sequence for servicing technological processes. Detailed description of technological toolpath creation is provided in the "Technological toolpaths" chapter that covers all types of machining.

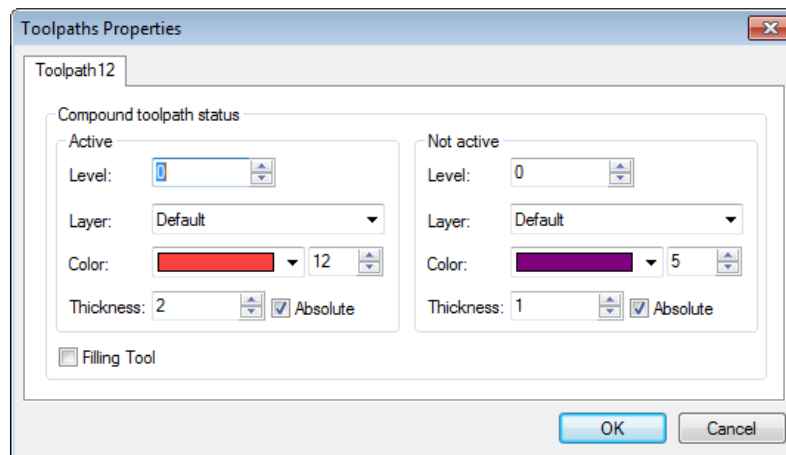
After invoking the command, the dialog window "Engraving" will appear. Parameters chosen in this window will be active **by default** for all newly created toolpaths of engraving. When creating a specific toolpath, however, these parameters can be modified. The user can change them as desired and save these changes by pressing the  button.



To create a toolpath of engraving, press the  button in the automenu. On the screen the dialog window of the toolpath parameters will appear which the user can change as desired, and the system will transition to the mode of selection of the text.


	<End>	Finish editing
	<P>	Toolpath properties
	<A>	Text selection filter
	<E>	Cancel selection of text
	<Esc>	Exit the command

In the **Toolpath properties** window the user can specify the properties for two groups of elements – active and nonactive toolpaths. It is possible to select the thickness of displayed elements of the toolpath, their color, layer, and level of location of the toolpath on the drawing.




Text selection **filter** changes the possibility of selection of the given construction element on the drawing as a geometric basis for construction of the toolpath. The image of the active filter is highlighted with a frame.

The user can indicate a single text or a sequence of texts on the drawing for which a toolpath of engraving must be created. Engraving of texts will be executed in the order they were selected.

In order to cancel selection of the text and specify another one, press  the button on the toolbar and indicate the new text for engraving.

Toolpath parameters.

Toolpath. Name of a toolpath.

File's tool. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the **Tool:** button.

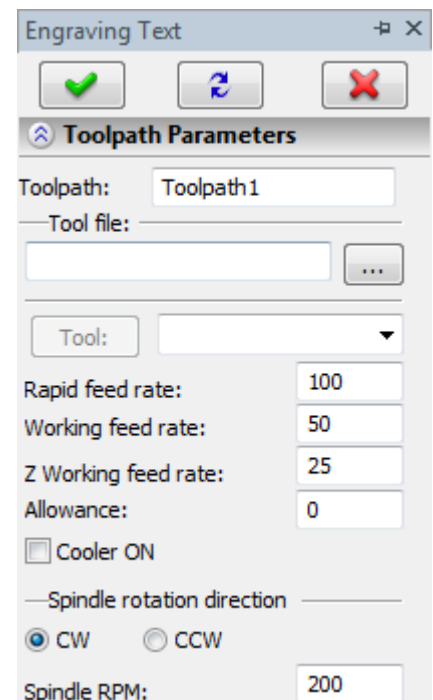
It is to be noted that the system will offer the user to select only that tool which can be utilizable for the chosen type of machining.

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary displacements.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of translation of the tool when executing the movement of cutting.

Working feed rate (on Z dir) is specified as a specific numerical value in units supported by the CNC center and sets up the speed of translation of the tool when executing movements of cutting along the Z-axis. It is used for external postprocessors.

Allowance. This parameter defines the distance between original and equidistant contours.



Cooler on. Turning on/off the supply of the cooling agent when machining the contour.

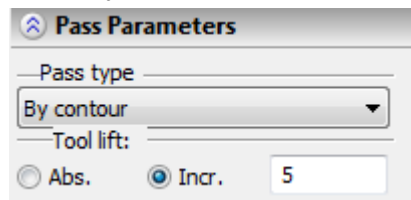
Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.

Pass parameters.

Pass type. Along the contour (coincides with the direction of the construction element) or against the contour (does not coincide).

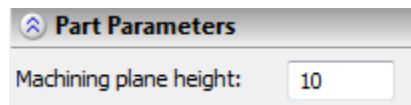
Tool lift. Specifies the distance to the safety plane (in absolute coordinates or in increments)



The 'Pass Parameters' dialog box contains the following controls: a 'Pass type' dropdown menu currently set to 'By contour'; a 'Tool lift' label; two radio buttons for 'Abs.' and 'Incr.' with 'Incr.' selected; and a text input field containing the value '5'.

Part parameters.

Here is specified the **height of machining plane** (from the zero plane), i.e., the height of the horizontal working translations of the mill.

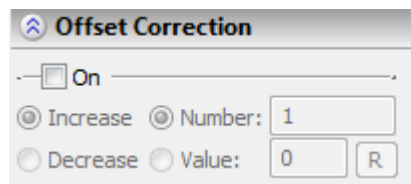


The 'Part Parameters' dialog box contains a single control: a text input field labeled 'Machining plane height:' with the value '10' entered.

Offset correction.

On. When this flag is enabled, parameters of the offset correction can be used. Correction can be for the **increase** and **decrease**. It is also possible to specify correction by the **number** (in this case, upon machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number indicated by the user) and by the **value** (in this case, the user specifies the precise value of the offset of the tool).

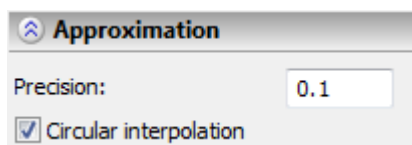
R – use the radius of the selected tool.



The 'Offset Correction' dialog box contains the following controls: a checkbox labeled 'On' which is checked; two radio buttons for 'Increase' and 'Decrease' with 'Increase' selected; a 'Number' radio button with a text input field containing '1'; a 'Value' radio button with a text input field containing '0'; and a button labeled 'R'.


Approximation.





Precision of the approximation is specified as a numerical value in units supported by the CNC center and constitutes the magnitude of the caving (maximum distance from the arc of the circle to be straightened to the approximating line).



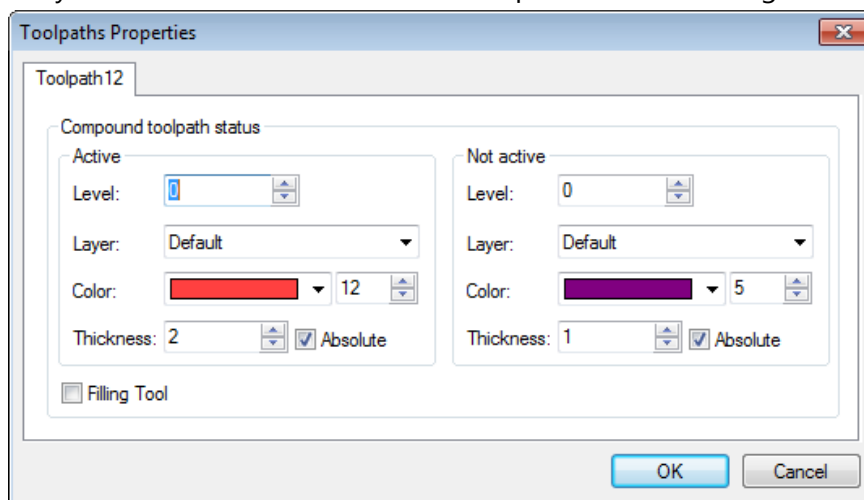
Circular interpolation. In case when the contour being machined has circles, arcs, splines, and other geometrical elements having similar shape, the selection of this parameter will allow us to obtain the control sequence that contains frames of the circular interpolation.

Going to a point with specified parameters

To create a toolpath of translation to a point with specified parameters, press the  button in the automenu. On the screen the dialog window of the toolpath parameters will appear which the user can change as desired, and the system will transition to the mode of selection of the node.

	<End>	Finish editing
	<P>	Toolpath properties
	<3>	Node selection filter
	<Esc>	Exit the command


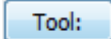
In the **Toolpath properties** window the user can specify the properties for two groups of elements – active and nonactive toolpaths. It is possible to select the thickness of displayed elements of the toolpath, their color, layer, and level of location of the toolpath on the drawing.



Node selection **filter** changes the possibility of selection of the given construction element on the drawing as a geometric basis for construction of the toolpath. The image of the active filter is highlighted with a frame.

Toolpath parameters.

Toolpath. Name of a toolpath

Tool's file. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilizable for the chosen type of machining.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of translation of the tool when executing the movement of cutting. This parameter is active for the motion on the working feed rate **G01**.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.

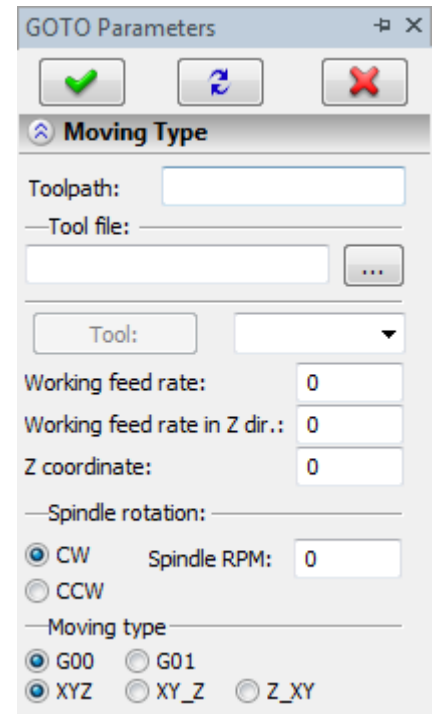
Moving type.

Selection of the method of moving (**G00** – on rapid feed rate, **G01** – on working feed rate) and the order in which translations along the axes are performed (**XYZ** – along three axes simultaneously; **XY_Z** – first along the **X** and **Y** axes, then along the **Z**-axis; **Z_XY** – first along the **Z**-axis, then along the **X** and **Y** axes).

Offset correction.

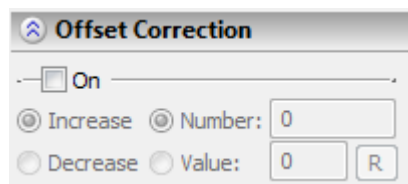
On. When this flag is enabled, parameters of the offset correction can be used. Correction can be for the **increase** and **decrease**. It is possible to specify correction by the **number** (in this case, upon machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number indicated by the user) and by the **value** (in this case, the user specifies the precise value of the offset of the tool).

R – use the radius of the selected tool.



The screenshot shows the 'GOTO Parameters' dialog box. It has a title bar with a maximize button and a close button. Below the title bar are three buttons: a green checkmark, a blue circular arrow, and a red X. The main section is titled 'Moving Type'. It contains the following fields and controls:


- Toolpath:** A text input field.
- Tool file:** A text input field with a three-dot button to its right.
- Tool:** A dropdown menu.
- Working feed rate:** A numeric input field with the value 0.
- Working feed rate in Z dir.:** A numeric input field with the value 0.
- Z coordinate:** A numeric input field with the value 0.
- Spindle rotation:** Two radio buttons, 'CW' (selected) and 'CCW'.
- Spindle RPM:** A numeric input field with the value 0.
- Moving type:** Three radio buttons: 'G00' (selected), 'G01', and 'XYZ'. Below these are three more radio buttons: 'XY_Z' and 'Z_XY'.








The screenshot shows the 'Offset Correction' dialog box. It has a title bar with a maximize button and a close button. Below the title bar is a checkbox labeled 'On'. Below the checkbox are two radio buttons: 'Increase' (selected) and 'Decrease'. To the right of the 'Increase' radio button is a numeric input field labeled 'Number' with the value 0. To the right of the 'Decrease' radio button is a numeric input field labeled 'Value' with the value 0. To the right of the 'Value' input field is a button labeled 'R'.


PUNCHING PROCESS


To define a toolpath and a control sequence for punching, use the command:

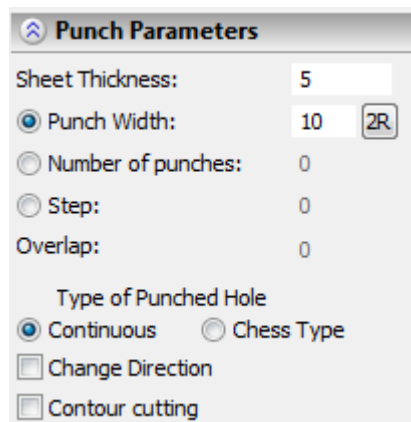
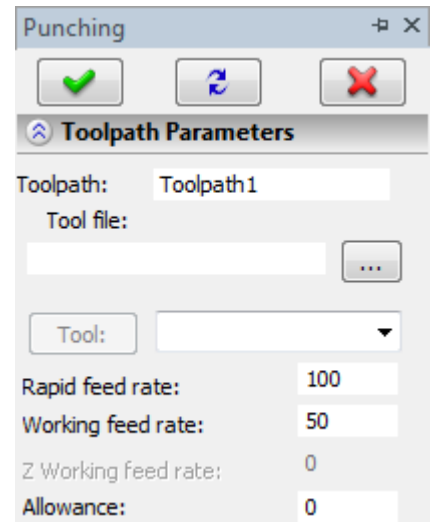
Icon	Ribbon
	CNC → Machining → Punch
Keyboard	Textual Menu
	CNC>2D, 2-1/2D and 4D machining>Punch

Upon calling the command, the following options become available in the automenu:


	<End>	Finish editing
	<T>	Technological toolpath
	<A>	Contour punch
	<Q>	Select point for tool moving
	<Esc>	Exit command








The  option is used for creating technological (service) toolpaths. Such toolpaths are necessary, for example, for defining additional commands in the control sequence for servicing technological processes. Detailed description of technological toolpath creation is provided in the "Technological toolpaths" chapter that covers all types of machining.

After invoking the command, the dialog window "Punching" will appear on the screen. Parameters chosen in this window will be active **by default** for all newly created toolpaths of punching. When creating a specific toolpath, however, these parameters can be modified. The user can change them as desired and save these changes by pressing the  button.

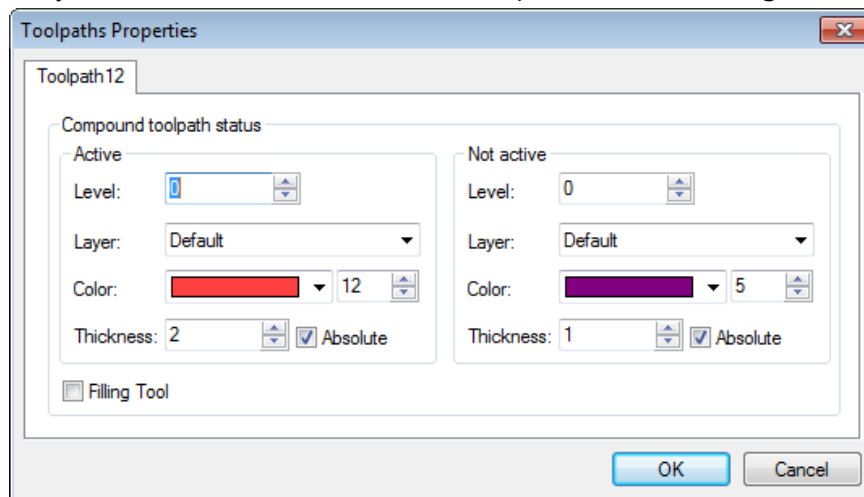


Toolpath creation

To create a toolpath of punching a contour, press the  button in the automenu. On the screen the dialog window of the toolpath parameters will appear which the user can change as desired, and the system will transition to the mode of selection of the hatches or paths.

	<End>	Finish editing
	<P>	Toolpath properties
	<1>	Hatch selection filter
	<2>	Path selection filter
	<C>	Specify cutoff points
	<X>	Remove selected cutoff points
	<Esc>	Exit the command



In the **Toolpath properties** window the user can specify the properties for two groups of elements – active and nonactive toolpaths. It is possible to select the thickness of displayed elements of the toolpath, their color, layer, and level of location of the toolpath on the drawing.




Filters for selection of hatches and paths change the possibility of their selection on the drawing as a geometric basis for construction of the toolpath. Filters can be active in pairs (in this case, it is possible to select both paths and hatches) or separately (in this case, only paths or only hatches can be selected). The image of the active filter is highlighted with a frame.

For machining a circle there are two modes of toolpath calculation:


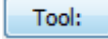
1. Selection of hatch – toolpath is calculated with a total removal of the material inside the selected contour;
2. Selection of path – toolpath is calculated with a traversal along the selected contour.

If it is required to add the cutoff point (removal of a certain strike from a series of strikes of the punch), select  and then press  in the required place of the contour.

To undo the selection of the points, press .

Toolpath parameters.

Toolpath. Name of a toolpath.

Tool's file. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilizable for the chosen type of machining.

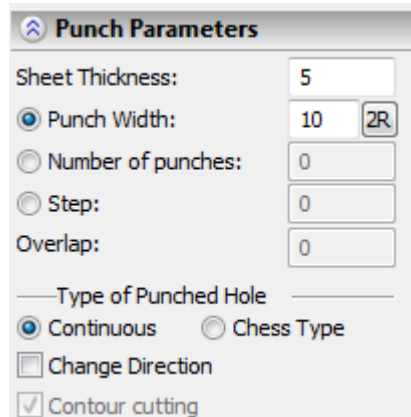
Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary displacements.




Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of translation of the tool when executing the movement of cutting.

Allowance. This parameter defines the distance between original and equidistant contours.

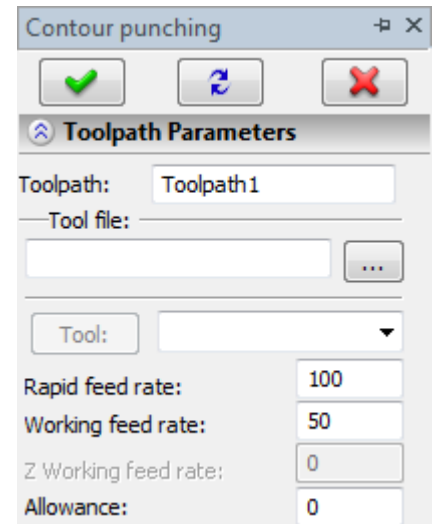
Punch parameters.

Sheet thickness. Thickness of the sheet being machined.



Punch width. Horizontal dimension of the tool's cross-section. When pressing , the horizontal dimension from the tool's file is automatically selected. If the cross-section of the tool is not circular, the  and  buttons can be used.

Number of holes. Specifies the number of strikes of the punch without taking into account the thickness of the sheet. This parameter is used for single-segment contours.



Step. Specifies the step between the strikes of the punch without taking into account the thickness of the sheet. In this mode the location of the strikes is selected symmetrically with respect to the center of the chosen contour. This parameter is used for single-segment contours.

You cannot specify the shift and bridge in this mode.

Overlap. The width of the overlap zone for two adjacent strikes of the tool (mm).

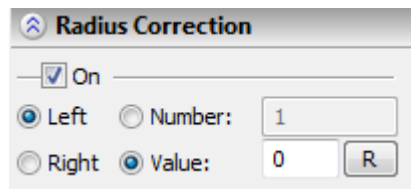
Type of punched hole. Continuous – successive strikes of the punch, chess type – alternate direct and reverse directions of punching.

Change direction. Change direction of punching.

Cutting a contour. The mode of machining a contour is turned on for a circle.

Radius correction.

On. When this flag is enabled, parameters of the radius correction can be used. Correction can be **left** and **right**. It is possible to specify correction by the **number** (in this case, upon machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number indicated by the user) and by the **value**.

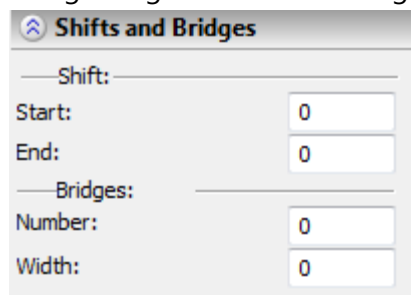


Value. For the radius correction, the value of the offset (shift) from the initial contour is specified; When pressing the **R** button, the vertical size from the tool's file is automatically selected. If the cross-section of the tool is not circular, the **A/2** and **B/2** buttons can be used.

Shifts and bridges.

Shift. Can be specified when the punch width and the number of holes parameters are used.

Start. Additional shift is set up at the beginning of the contour segment.



End. Additional shift is set up at the end of the contour segment.

Bridges. Can be specified when the punch width parameter is used.

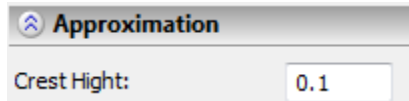
The strikes of the punch are uniformly spaced along the length of the contour segment.

Number of bridges on each segment of the contour.


Width of the bridge.








Approximation.

Crest height. When calculating the toolpaths of machining a circle or a spline, the number and location of strikes of the punch is determined from the specified height of the residual crest.

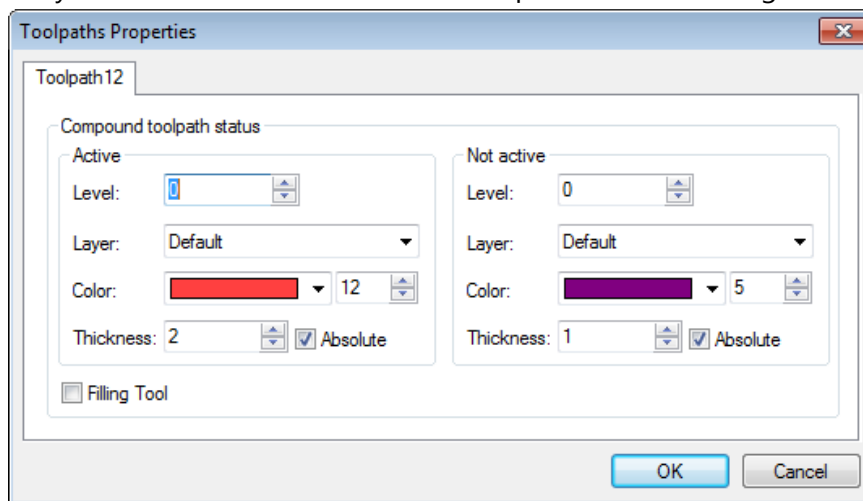


Going to a Specified Point

To go to a specified point while in punching, press the  icon. After pressing this icon, the system will transition to the mode of selection of the nodes and paths and the following options will become available in the automenu:



	<End>	Finish editing
	<P>	Toolpath properties
	<1>	Select point for tool moving
	<2>	Select path
	<Q>	Setting Snap Point of Tool
	<Z>	Delete Selected Points
	<Esc>	Exit command



In the **Toolpath properties** window the user can specify the properties for two groups of elements – active and nonactive toolpaths. It is possible to select the thickness of displayed elements of the toolpath, their color, layer, and level of location of the toolpath on the drawing.



Filters for selection of nodes and paths change the possibility of their selection on the drawing as a geometric basis for construction of the toolpath. Filters can be active in pairs (in this case, it is



possible to select both paths and hatches) or separately (in this case, only paths or only hatches can be selected). The image of the active filter is highlighted with a frame.

After specifying the tool file(s) and selecting the tool, you will see the  button to snap the vertex. By clicking the  button, you will loop through selections of subsequent section vertices and have them snap to the previously specified point.

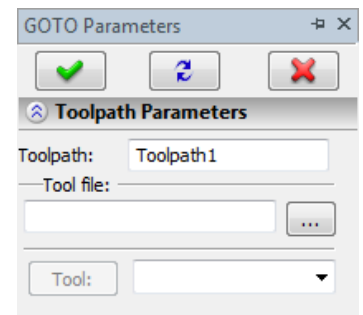
To undo the snap point selection, click  after which the tool section contour will return to the original position. To process a sequence of holes with a single tool, select the  path connecting the nodes of the punch locations.

Toolpath parameters.


Toolpath. Name of a toolpath.

Tool's file. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilizable for the chosen type of machining.



DEFINING THE PATH START POINT

There is an option to define the first point of the machining path in the cases of contour wire cutting, laser cutting and milling. By default, the system selects the start node of the geometrical path entity as the start point for the machining path (or that of the hatch, depending on what geometrical entity was selected by the user). The user can avoid such a choice by clicking the  button, and select another point on the contour to serve as the start, using the mouse pointer. The path start will automatically be reset at the specified point. In addition, if the contour was closed, then the machining will start at the specified point and finish at the same point (the contour will be fully machined). If, on the other hand, an open path was specified, then the machining will start at the user-specified point (only the contour segment from the specified point to the end point of the path will be completely machined).


In order to define the start point of the path, you do not have to construct a T-FLEX CAD node in these coordinates, even though this is recommended to more accurately define the start point.

TOOLPATHS FOR 3D, 4D AND 5D MACHINING (3D VERSION)







Before proceeding with the description of the T-FLEX CAM 3D system, note that the following items in the "CNC 3D" menu are identical to the T-FLEX CAM 2D counterparts: "CAM-system options", "Tool Editor", "Machining simulation", "Save NC program", "Postprocessor options".

3D MILLING


To define a toolpath and a control sequence for 3D milling, use the command:

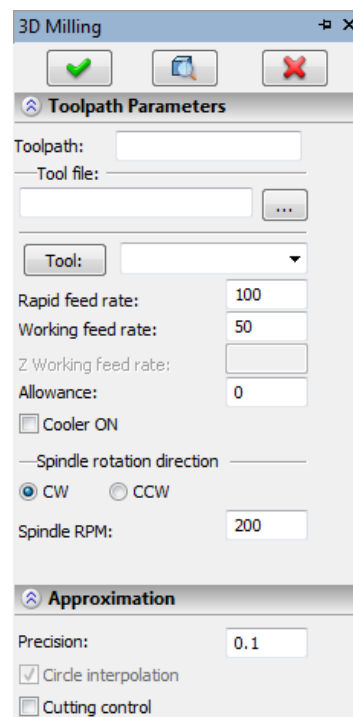
Icon	Ribbon
	CNC → Machining → 3D Milling
Keyboard	Textual Menu
	CNC>3D and 5D machining>3D Milling

Upon calling the command, the following options become available in the automenu:

	<Y>	Finish input
	<I>	Select Other Element
	<X>	Exit command
	<C>	3D contour milling
	<P>	3D surface milling
	<G>	GOTO toolpath






Toolpath creation

After the command is invoked, the "3D Milling" dialog window will appear on the screen. Parameters set up in this window will be enabled **by default** for all newly created toolpaths of 3D milling. When a specific toolpath is created, these parameters can be changed. The user can change them as desired and save these changes by pressing the  button.





Milling of 3D contour

After selection of the option of milling a 3D contour, the option of selection of the contour (3D path) being machined will become available in the automenu.

	<End>	Finish editing
	<I>	Select another closest element
	<F5>	Preview
	<E>	Selection of 3D path
	<Esc>	Exit the command

Toolpath parameters.

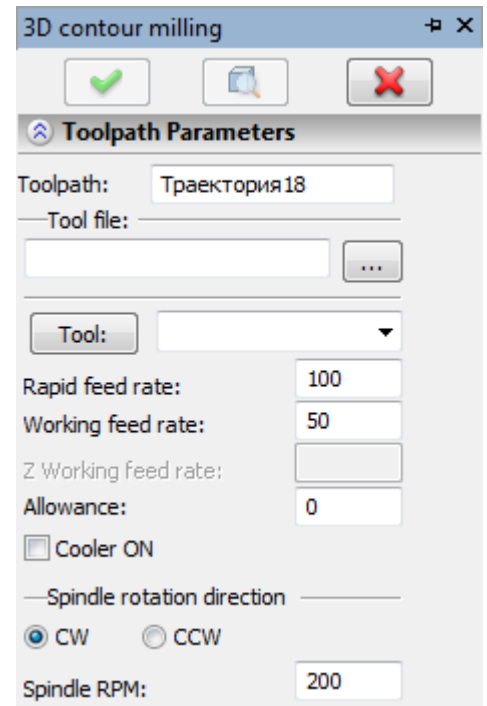
Toolpath. Name of a toolpath.

Tool's file. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilized for the chosen type of machining.

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary translations.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the tool's translation when executing the motion of cutting.



Allowance. This parameter defines the distance between the given and equidistant contours.

Cooler on. Turning on/off the supply of the cooling agent upon machining.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.

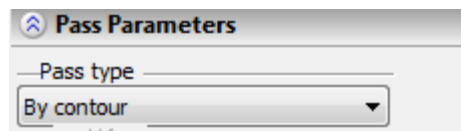
Pass parameters.

Pass type. Along the contour or against the contour.

Height of the bottom point. The Z-coordinate of the end of the spiral (specified with respect to the zero plane).

Step. Step of a spiral.

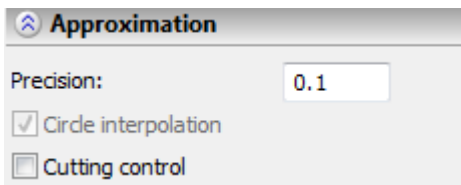
Direction of the spiral's turn is selected with the help of the **clockwise** or **counterclockwise** fields.



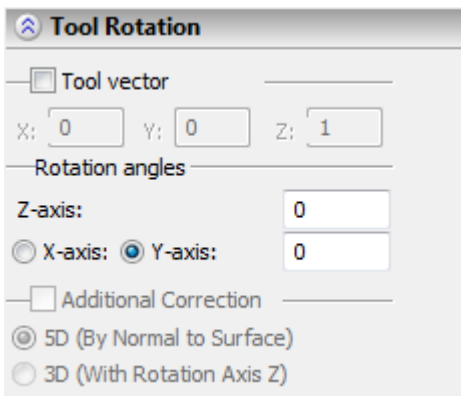
Approximation.

Precision of the approximation is specified as a numerical value in units supported by the CNC center and constitutes the magnitude of the caving (maximum distance from the arc of the circle to be straightened to the approximating line).

Overcutting control. Enabling the mechanism of control of overcuts of the part, which are produced by the tool, with the precision specified in «CAM-system Settings». The segments of the toolpath with the overcuts will be highlighted with the black color. In addition, the frames with the overcuts will be seen in «Machining Simulation».

**Tool's rotation.**

Rotation of the tool can be defined either by direct specification of the angles of rotation or by specification of the vector of the tool.



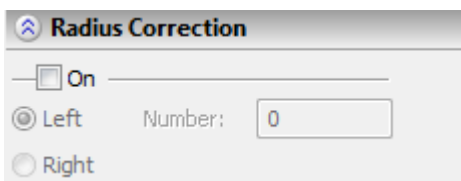
Tool's vector. Prescribe orientation of the vector that defines the location of the tool's axis upon machining. When this flag is enabled, the required angles of the tool's rotation are calculated automatically, and the windows for specification of the values of the angles will show their calculated values.

Rotation angles. Prescribe the angles of rotation around the Z-axis and then around the X or Y axis (according to the switch).

Radius correction.

On. Enabling this flag allows us to use parameters of the radius correction.

Correction can be **left** and **right**.

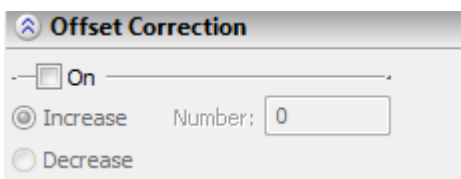


Correction is specified by the **number**. During machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number indicated by the user.

Offset correction.

On. Enabling this flag allows us to use parameters of the offset correction.






Correction can be for the **increase** and **decrease**.



Correction is specified by the **number**. During machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number indicated by the user.



Milling 3D surface

After selection of the option of milling a 3D surface, the option of selection of the face to be machined will become available in the automenu.

	<End>	Finish editing
	<I>	Select another closest element
	<F5>	Preview
	<F>	Selection of a face
	<Esc>	Exit command

Toolpath parameters.

Toolpath. Name of a toolpath.

Tool's file. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilized for the chosen type of machining.

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary translations.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the tool's translation when executing the motion of cutting.

Allowance. This parameter defines the distance between the given and equidistant contours.

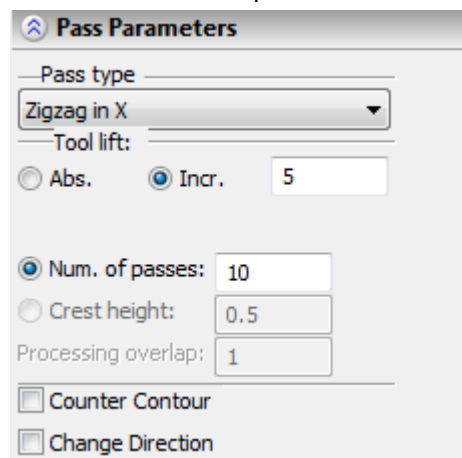
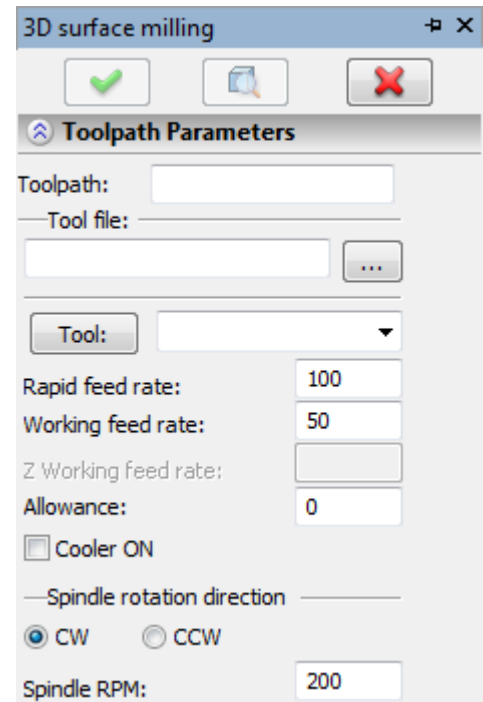
Cooler on. Turning on/off the supply of the cooling agent upon machining.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.

Pass parameters.

Pass type. Specified as one of two possible types: zigzag (along parametric U-lines or parametric V-lines) or loop (along parametric U-lines or parametric V-lines). The notation X and Y is quite arbitrary since in some cases the X-axis can correspond to U, and in other cases to V.



Tool's lift. The lift of the tool while making transitions between zones of machining. Specified by an absolute or relative value.

Number of passes. The number of tool's passes is specified.

Counter contour. Machining will be carried out in reverse order with respect to the original contour of the toolpath (along the isoparametric line U).

Change direction. Removal of the material will start in the opposite direction, with respect to the initial direction of passes, at the opposite side of the face being machined (in the direction of the isoparametric line V).

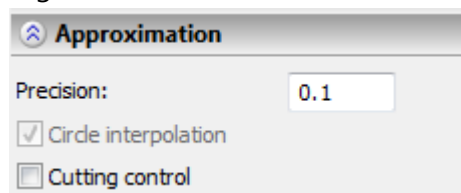
Optimized. This type of the offset provides automatic safety control of the tool's exit out of the limits of the surface's boundaries (in the direction of passes), which allows us to cancel the equidistant offset where it is not required. This type of the offset is recommended to use for milling **partially closed** along the perimeter surfaces.

Value. The value of the offset. The entry of the value of the parameter into the text field can be carried out either manually or automatically when pressing the **R** button.

In the latter case, the value of the radius of the tool being used will be inserted into the field.

Approximation.

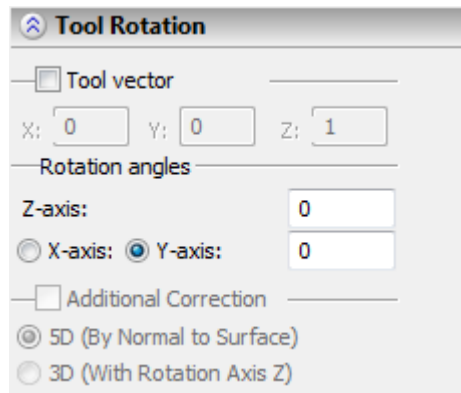
Precision of the approximation is specified as a numerical value in units supported by the CNC center and constitutes the magnitude of the caving (maximum distance from the arc of the circle to be straightened to the approximating line).



Overcutting control. Enabling the mechanism of control of overcuts of the part, which are produced by the tool, with the precision specified in «CAM-system Settings». The segments of the toolpath with the overcuts will be highlighted with black color. In addition, the frames with the overcuts will be seen in «Machining Simulation».

Tool's rotation.

To simplify execution of the operation of 3D milling on multiaxial equipment, it is possible to specify the angles of rotation of the tool (or the part, depending on the machining station kinematics) or the vector of the tool's inclination. Thus, the machining toolpath can be created on any surface of the part (top, side, end, back) regardless of the orientation of the part's model in 3D scene.



Tool's vector. Prescribe orientation of the vector that defines the location of the tool's axis upon machining. When this flag is enabled, the required angles of the tool's rotation are calculated automatically, and the windows for defining the values of the angles will show their calculated values.

Rotation angles. Prescribe the angles of rotation around the Z-axis and then around the X or Y axis (according to the switch).

Radius correction.

On. Enabling this flag allows us to use parameters of the radius correction.

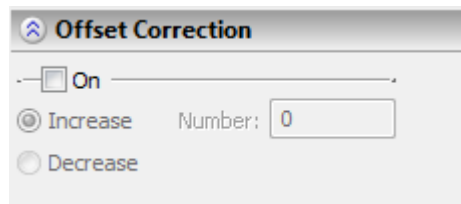
Correction can be **left** and **right**.

Correction is specified by the **number**. During machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number specified by the user.


Offset correction.


On. Enabling this flag allows us to use parameters of the offset correction.

Correction can be for the **increase** and **decrease**.




Correction is specified by the **number**. During machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number specified by the user.

To add the created toolpaths to the document containing the prepared drawing of the part to be machined, click the  icon in the automenu.











The calculated toolpath will be added to the list of available toolpaths in the Machining Manager. To view this list, click the  icon.


BOUNDED 3D MILLING

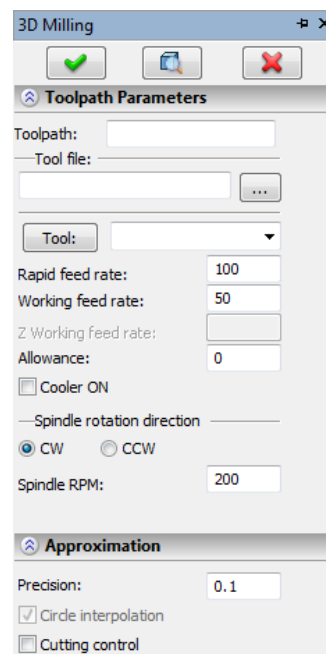
To define a toolpath and a control sequence for bounded 3D milling, use the command:

Icon	Ribbon
	CNC → Machining → 3D milling
Keyboard	Textual Menu
	CNC>Bounded machining>3D - milling


Upon calling the command, the following options become available in the automenu:








	<Y>	Finish input
	<I>	Select Other Element
	<X>	Exit command
	<R>	Rough milling
	<F>	Finish milling
	<P>	Pencil machining
	<O>	Pocket machining
	<S>	Surface of revolution
	<K>	Milling with a spiral
	<G>	GOTO toolpath



After the command is invoked, the "3D Milling" dialog window will appear on the screen. Parameters set up in this window will be enabled **by default** for all newly created toolpaths of 3D zone milling. When a specific toolpath is created, these parameters can be changed. The user can change them as desired and save these changes by pressing the  button.




Rough milling

To create a toolpath of rough milling, press the  button in the automenu of 3D zone milling. On the screen the dialog window of the toolpath parameters will appear which the user can change as desired, and the system will transition to the mode of selection of the first operand.

	<End>	Finish editing
	<I>	Choose another closest element
	<Esc>	Exit command
		Select first operand
	<S>	Select second operand
	<O>	Select orienting face
	<K>	Cancel selection


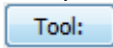
In 3D scene of T-FLEX CAD it is required to indicate the body to be machined – first operand (the  option is automatically selected). Then, if necessary, the user can indicate the second body that describes the zone of machining (the  option – second operand). If the second (bounding) body is not selected, it will automatically be created on the basis of overall parallelepiped of the workpiece. With the help of the **Offset from workpiece** parameter the user can control the magnitude of the offset from the overall parallelepiped along the X and Y axes.

Selection of the orienting face allows us to automatically specify the vector of the tool's axis. The values of the tool's vector coordinates will then be inserted into the corresponding fields of the "Tool's rotation" tab. Moreover, as an orienting face the user can select, in addition to planes, spherical, conic, cylindrical and toroidal surfaces. In this case, the axis of symmetry will be used as a tool's vector.

The  option allows us to cancel all selected elements for their subsequent repeated specification.

Toolpath parameters.

Toolpath. Name of a toolpath.

Tool's file. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilized for the chosen type of machining.

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary translations.

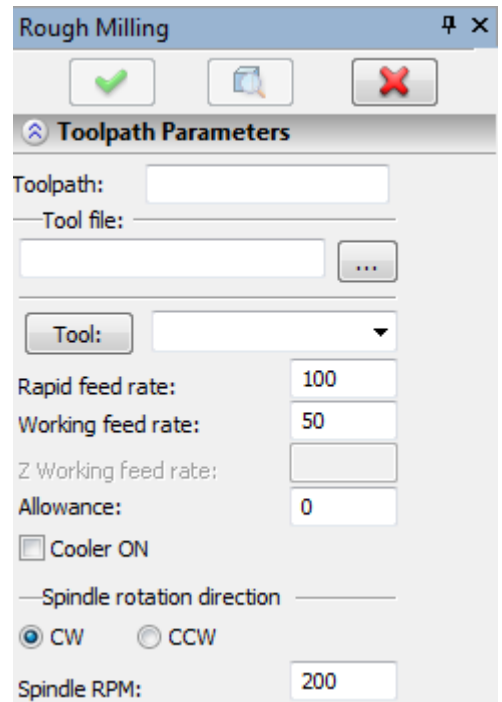
Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the tool's translation when executing the motion of cutting.

Allowance. This parameter defines the distance between the given and equidistant contours.

Cooler on. Turning on/off the supply of the cooling agent upon machining.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.



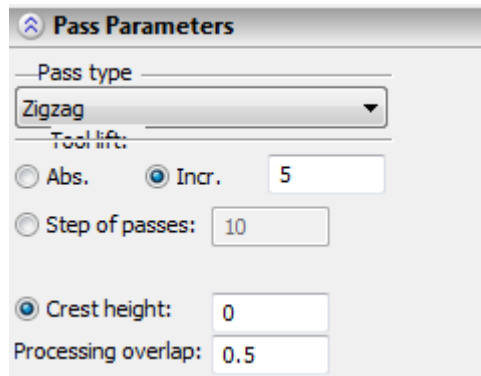
The screenshot shows the 'Rough Milling' dialog box with the 'Toolpath Parameters' tab selected. The dialog contains the following fields and controls:

- Toolpath:** A text input field.
- Tool file:** A text input field with a three-dot button to its right.
- Tool:** A dropdown menu with a 'Tool:' button to its left.
- Rapid feed rate:** A numeric input field with the value 100.
- Working feed rate:** A numeric input field with the value 50.
- Z Working feed rate:** A numeric input field.
- Allowance:** A numeric input field with the value 0.
- Cooler ON:** A checkbox.
- Spindle rotation direction:** Two radio buttons labeled 'CW' and 'CCW'.
- Spindle RPM:** A numeric input field with the value 200.

Pass parameters.

Pass type. From the list the user can select zigzag/loop/spiral or with smooth approach to the part.

Tool lift. The lift of the tool when making transitions between the zones of machining. Specified as an absolute or relative value.



Pass Parameters

Pass type: Zigzag

Tool lift:

☐ Abs. ☒ Incr. 5

☐ Step of passes: 10

☒ Crest height: 0

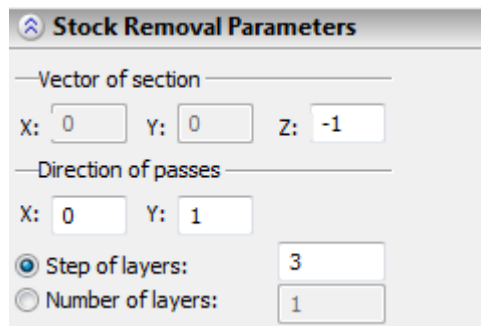
Processing overlap: 0.5

Step of passes. The value of the step between the passes of the tool is specified.

Crest height. The value of the step between the passes of the tool is calculated here based on the given crest height and the value of the **overlap** of the tool on neighbouring passes. The crest height is the height of the undercut material remaining between two neighbouring passes.

Stock removal parameters.

Vector of sections. The vector of normal to the planes in which the passes lie. It defines the direction of the layers of the material being removed and the direction of the motion of the tool from one layer to another. For example, if the vector is specified as (0,0,1) – layers of the material being removed are located parallel to the XY-plane. In the rough machining the vector is specified only along the Z-axis ($X=0$ $Y=0$, $Z \neq 0$). Therefore, positions X and Y of the **vector of sections** are not active for the rough machining, and their values are not taken into consideration upon calculation of the toolpath.



Stock Removal Parameters

Vector of section

X: 0 Y: 0 Z: -1

Direction of passes

X: 0 Y: 1

☒ Step of layers: 3

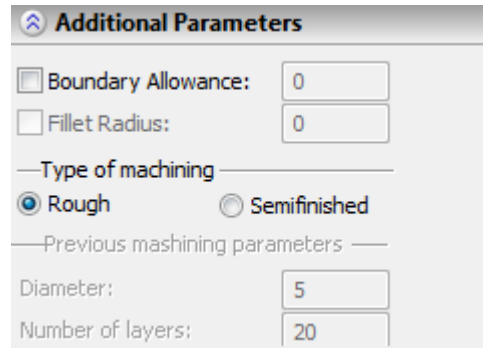
☐ Number of layers: 1

Direction of passes. Specifies the vector (X,Y) which shows the direction of the passes of the tool in one layer of the material being removed. This field is used only for rough machining for all passes except the passes of a spiral type (in this case, this element is not active). This vector must lie in the plane of the material being removed, hence, must be perpendicular to the vector of sections. Therefore, not all combinations of the components of the vectors of sections and directions of passes are admissible.

Additional parameters.

Offset from a workpiece. Here we specify the equidistant offset from the workpiece. The actual magnitude of the offset will be equal to the sum of the radius of the tool and the input value for the

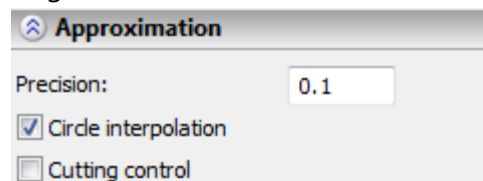
offset. It is important to remember that upon choosing the second operand (that bounds the machining zone) the positive value of the parameter will correspond to the equidistant offset from the machining zone boundaries towards the interior of the bounding body, and in the case when the second body is not selected (i.e., bounding body is created automatically on the basis of the overall parallelepiped of the machined body), correspond to the offset in the X and Y directions towards the increase of the section of the bounding body.



Type of machining. Rough or semifinished. For **rough** machining the passes of the tool are constructed layerwise on the basis of the fact that the bounding body – a model of a workpiece. Therefore, all material that is enclosed between the faces of the bounding body and the surface of the part will be removed as a result of the rough machining. For **semifinished** machining the user specifies parameters of the previous rough machining. Also, it is necessary to specify the cutting tool that is different from the one used in the previous rough machining (for example, the tool of a smaller diameter, with larger rounding radii). When calculating the toolpath, T-FLEX CAM carries out the analysis of the part being machined and searches for the segments of the part which cannot be machined with the previous tool but can be machined with a new tool. After that the calculation of the passes of the selected type takes place only in the unfinished segments of the part. The usage of the semifinished machining (or several semifinished machinings) allows us to significantly unload the tool that works in the stage of the finishing (usually – spherical mill) at a minimum consumption of the computer time.

Approximation.

Precision of the approximation is specified as a numerical value in units supported by the CNC center and constitutes the magnitude of the caving (maximum distance from the arc of the circle to be straightened to the approximating line).



Circular interpolation. In case when the contour being machined has circles, arcs, splines, and other geometrical elements having similar shape, the selection of this parameter will allow us to obtain the control sequence that contains frames of the circular interpolation.

Overcutting control. Enabling the mechanism of control of overcuts of the part, which are produced by the tool, with the precision specified in «CAM-system Settings». The segments of the toolpath with the overcuts will be highlighted with black color. In addition, the frames with the overcuts will be seen in «Machining Simulation».

Tool's rotation.

Rotation of the tool can be specified by either directly specifying the angles of rotation, or by specifying the tool's vector.

Tool's vector. Prescribe orientation of the vector that defines the location of the tool's axis upon machining. When this flag is enabled, the required angles of the tool's rotation are calculated automatically, and the windows for defining the values of the angles will show their calculated values.

Rotation angles. Prescribe the angles of rotation around the Z-axis and then around the X or Y axis (according to the switch).

Radius correction.

On. Enabling this flag allows us to use parameters of the radius correction.

Correction can be **left** and **right**.

Correction is specified by the **number**. During machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number specified by the user.

.


Offset correction.



On. Enabling this flag allows us to use parameters of the offset correction.






Correction can be for the **increase** and **decrease**.



Correction is specified by the **number**. During machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number specified by the user.

Finishing milling


To create a toolpath of the finishing milling, press the  button in the automenu of 3D zone milling. On the screen the dialog window of the toolpath parameters will appear which the user can change as desired, and the system will transition to the mode of selection of the first operand.

	<End>	Finish editing
	<I>	Select another closest element

	<Esc>	Exit command
		Select first operand
	<S>	Select second operand
	<O>	Select orienting face
	<K>	Cancel selection


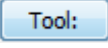
In 3D scene of T-FLEX CAD it is required to indicate the body to be machined – the first operand (the  option is automatically selected). Then, if necessary, the user can indicate the second body that describes the zone of machining (the  option – the second operand). If the second (bounding) body is not selected, it will automatically be created on the basis of overall parallelepiped of the workpiece. With the help of the **Offset from workpiece** parameter the user can control the magnitude of the offset from the overall parallelepiped along the X and Y axes.

Selection of the orienting face allows us to automatically specify the vector of the tool's axis. The values of the tool's vector coordinates will then be inserted into the corresponding fields of the "Tool's rotation" tab. Furthermore, as an orienting face the user can select, in addition to planes, spherical, conic, cylindrical and toroidal surfaces. In this case, the axis of symmetry will be used as a tool's vector.

The  option allows us to cancel all selected elements for their subsequent repeated specification.

Toolpath parameter.

Toolpath. Name of a toolpath.

Tool's file. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

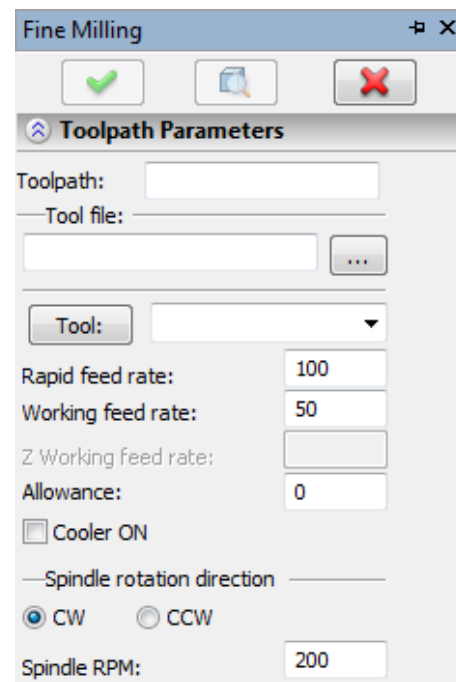
It is to be noted that the system will offer the user to select only that tool which can be utilized for the chosen type of machining.

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary translations.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the tool's translation when executing the motion of cutting.

Allowance. This parameter defines the distance between the given and equidistant contours.

Cooler on. Turning on/off the supply of the cooling agent upon machining.




Fine Milling

☐ ☐ ☐

Toolpath Parameters

Toolpath:

Tool file: 

Tool:

Rapid feed rate:

Working feed rate:

Z Working feed rate:

Allowance:

☐ Cooler ON

— Spindle rotation direction —

☒ CW ☐ CCW

Spindle RPM:

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.

Pass parameters.

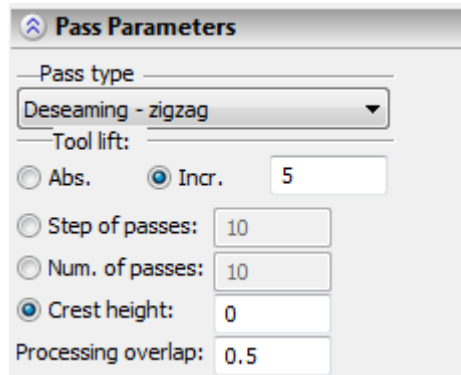
Pass type. From the list the user can select zigzag/loop/spiral or with smooth approach to the part. The crest height is the height of the undercut material remaining between two neighbouring passes.

Tool lift. The lift of the tool when making transitions between the zones of machining. Specified as an absolute or relative value.

Step of passes. Specifies the value of the step between the passes of the tool.

Number of passes. Specifies the number of the passes of the tool.

Crest height. The value of the step between the passes of the tool is calculated here based on the given crest height and the value of the **overlap** of the tool on the neighbouring passes.



Pass Parameters

Pass type: Deseaming - zigzag

Tool lift:
☐ Abs. ☒ Incr. 5

☐ Step of passes: 10

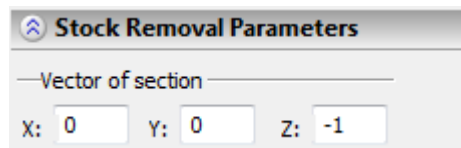
☐ Num. of passes: 10

☒ Crest height: 0

Processing overlap: 0.5

Stock removal parameters.

Vector of sections. The vector of normal to the planes in which the passes lie. It defines the direction of the layers of the material being removed and direction of the motion of the tool from one layer to another. For example, if the vector is specified as (0,0,1) – layers of the material being removed are located parallel to the XY-plane.



Stock Removal Parameters

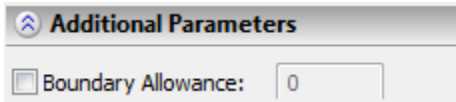
Vector of section

X: 0 Y: 0 Z: -1

Additional parameters.

Offset from a workpiece. Here we specify equidistant offset from the workpiece. The actual magnitude of the offset will be equal to the sum of the radius of the tool and the input value of the offset. It is important to remember that upon choosing the second operand (that bounds the machining zone)

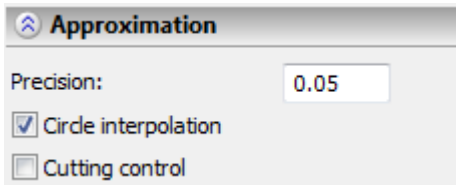
the positive value of the given parameter will correspond to the equidistant offset from the machining zone boundaries towards the interior of the bounding body, and in the case when the second body is not selected (bounding body is created automatically on the basis of the overall parallelepiped of the machined body), correspond to the offset in the X and Y directions towards the increase of the section of the bounding body.



Approximation.

Precision of the approximation is specified as a numerical value in units supported by the CNC center and constitutes the magnitude of the caving (maximum distance from the arc of the circle to be straightened to the approximating line).

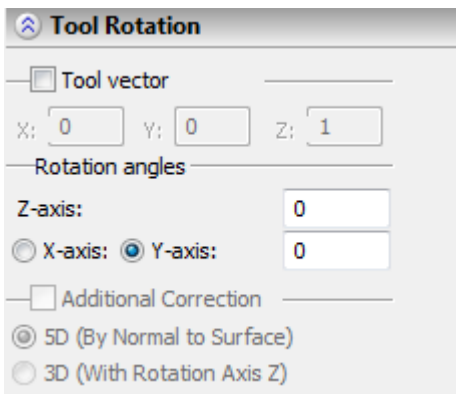
Circular interpolation. In case when the contour being machined has circles, arcs, splines, and other geometrical elements having similar shape, the selection of this parameter will allow us to obtain the control sequence that contains frames of the circular interpolation.



Overcutting control. Enabling the mechanism of control of overcuts of the part, which are produced by the tool, with the precision specified in «CAM-system Settings». The segments of the toolpath with the overcuts will be highlighted with a black color. In addition, the frames with the overcuts will be seen in «Machining Simulation».

Tool's rotation.

Rotation of the tool can be specified by either directly specifying the angles of rotation, or by specifying the tool's vector.



Tool's vector. Prescribe orientation of the vector that defines the location of the tool's axis upon machining. When this flag is enabled, the required angles of the tool's rotation are calculated automatically, and the windows for defining the values of the angles will show these calculated values.

Rotation angles. Prescribe the angles of rotation around the Z-axis and then around the X or Y axis (according to the switch).

Radius correction.

On. Enabling this flag allows us to use parameters of the radius correction.

Correction can be **left** and **right**.

Correction is specified by the **number**. During machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number specified by the user.


Offset correction.

On. Enabling this flag allows us to use parameters of the offset correction.







Correction can be for the **increase** and **decrease**.





Correction is specified by the **number**. During machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number specified by the user.


Pencil machining

To create a toolpath of pencil machining, press the  button in the automenu of 3D zone milling. On the screen the dialog window of toolpath parameters will appear which the user can change as desired, and the system will transition to the mode of selection of the body.

In pencil machining the passes of the tool are carried out as close as possible to the surface or contour of the part with the use of capabilities of 3D machining. Control of collisions of the tool with a part, in the direction of the specified tool's vector, is used in this type of machining, which eliminates the overcut of the part.

	<End>	Finish editing
	<I>	Select another closest element
	<Esc>	Exit command
		Select body
	<F>	Select face
	<E>	Select edge

	<P>	Select guiding path
	<L>	Select profile
	<O>	Select orienting face
	<K>	Cancel selection



Upon selection of 3D paths or profile as a contour of the body, first in the 3D scene of T-FLEX CAD it is required to indicate the body to be machined (the  option). When selecting edges or faces, the body being machined does not have to be selected. Furthermore, it is possible to use various combinations (for example, selection of faces and 3D-paths).

Selection of the orienting face allows us to automatically specify the vector of the tool's axis. The values of the tool's vector coordinates will then be inserted into the corresponding fields of the "Tool's rotation" tab. Furthermore, as an orienting face the user can select, in addition to planes, spherical, conic, cylindrical and toroidal surfaces. In this case, the axis of symmetry will be used as a tool's vector.

The  option allows us to cancel all selected elements for their subsequent repeated specification.

Toolpath parameters.

Toolpath. Name of a toolpath.

Tool's file. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilized for the chosen type of machining.

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary translations.

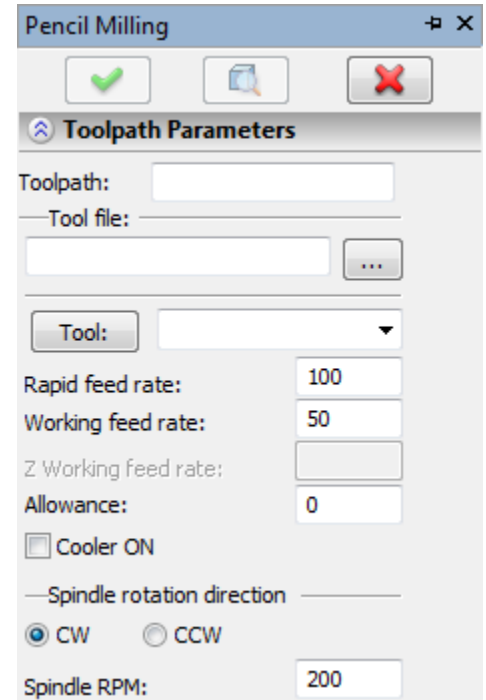
Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the tool's translation when executing the motion of cutting.

Allowance. This parameter defines the distance between the given and equidistant contours.

Cooler on. Turning on/off the supply of the cooling agent upon machining.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.



Stock removal parameters.

Length of entry. Serves to specify the segments of the tool's cutting-in before the toolpath. Additional segments are constructed along the tangent at the first point of the toolpath.

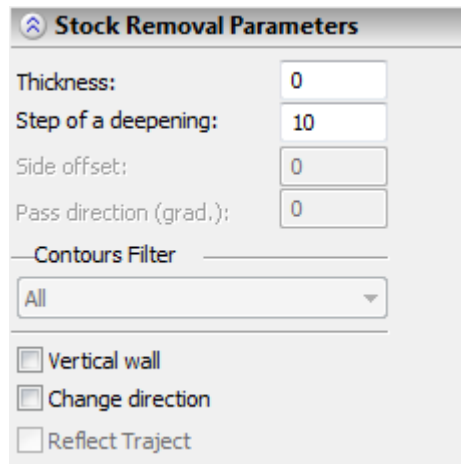
Length of exit. Serves to specify the segments of tool's exit after the toolpath. Additional segments are constructed along the tangent at the last point of the toolpath.

Side offset. Serves to specify equidistant offset. This parameter can be used for extending the boundaries of the machined surface. This parameter becomes active when the "Vertical wall" parameter is enabled.

Vertical wall. When this parameter is enabled, the equidistant offset from the given contour is created. This option speeds up the calculation of the toolpath in cases when near a 3D path there are flat walls of a 3D body (flat walls that are vertical are preferred).

Change direction. Passes of the tool will be directed in the opposite direction.

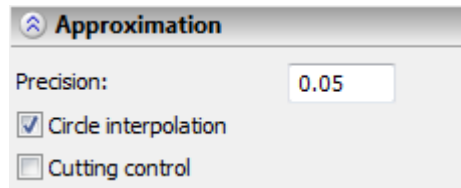
Reflect trajectory. This flag can be used when the "vertical wall" flag is enabled. The given flag allows us to change the direction of the tool's offset from the wall of the part to reverse.

**Approximation.**

Precision of the approximation is specified as a numerical value in units supported by the CNC center and constitutes the magnitude of the caving (maximum distance from the arc of the circle to be straightened to the approximating line).

Circular interpolation. In case when the contour being machined has circles, arcs, splines, and other geometrical elements having similar shape, the selection of this parameter will allow us to obtain the control sequence that contains frames of the circular interpolation.

Overcutting control. Enabling the mechanism of control of overcuts of the part, which are produced by the tool, with the precision specified in «CAM-system Settings». The segments of the toolpath with the overcuts will be highlighted with a black color. In addition, the frames with the overcuts will be seen in «Machining Simulation».

**Tool's rotation.**

Rotation of the tool can be specified by either directly specifying the angles of rotation, or by specifying the tool's vector.

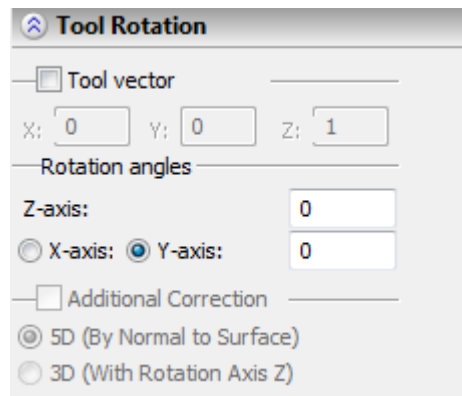
Tool's vector. Prescribe orientation of the vector that defines the location of the tool's axis upon machining. When this flag is enabled, the required angles of the tool's rotation are calculated automatically, and the windows for defining the values of the angles will show their calculated values.

Rotation angles. Prescribe the angles of rotation around the Z-axis and then around the X or Y axis (according to the the switch).

Radius correction.

On. Enabling this flag allows us to use parameters of the radius correction.

Correction can be **left** and **right**.



Correction is specified by the **number**. During machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number specified by the user.


Offset correction.

On. Enabling this flag allows us to use parameters of the offset correction.








Correction can be for the **increase** and **decrease**.



Correction is specified by the **number**. During machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number specified by the user..


Pocket machining

To create a toolpath of pocket machining, press the  button in the automenu of 3D zone milling. On the screen the dialog window of toolpath parameters will appear which the user can change as desired, and the system will transition to the mode of selection of the body.

Pocket machining serves for creation of toolpaths of selection or finishing of pockets with the use of capabilities of 3D machining. Control of collisions of the tool with a part, in the direction of the specified tool's vector, is used in this type of machining, which eliminates the overcut of the part. Control of collisions is used if the "2.5D Plane" flag is not enabled.

	<End>	Finish editing
	<I>	Select another closest element
	<Esc>	Exit command
		Select body
	<F>	Select face
	<P>	Select guiding path
	<L>	Select profile

	<O>	Select orienting face
	<K>	Cancel selection



Upon selection of 3D paths or profile as a contour of the body, first in the 3D scene of T-FLEX CAD it is required to indicate the body being machined (the  option). When selecting the faces, the body being machined does not have to be selected. Furthermore, it is possible to use various combinations (for example, selection of faces and 3D-paths).

Selection of the orienting face allows us to automatically specify the vector of the tool's axis. The values of the tool's vector coordinates will then be inserted into the corresponding fields of the "Tool's rotation" tab. Furthermore, as an orienting face the user can select, in addition to planes, spherical, conic, cylindrical and toroidal surfaces. In this case, the axis of symmetry will be used as a tool's vector.

The  option allows us to cancel all selected elements for their subsequent repeated specification.

Toolpath parameters.

Toolpath. Name of a toolpath.

Tool's file. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilized for the chosen type of machining.

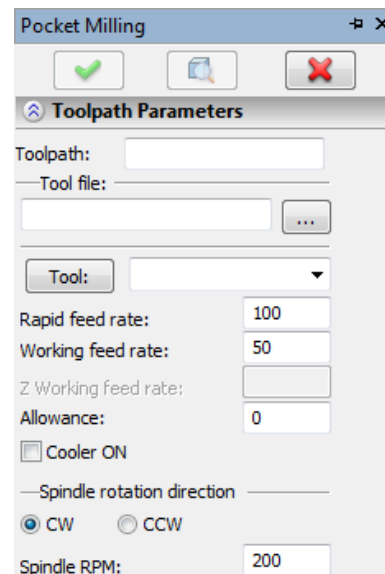
Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary translations.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the tool's translation when executing the motion of cutting.




Allowance. This parameter defines the distance between the given and equidistant contours.

Cooler on. Turning on/off the supply of the cooling agent upon machining.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.





Pocket Milling

Toolpath Parameters

Toolpath:

Tool file: 

Tool: 

Rapid feed rate:

Working feed rate:

Z Working feed rate:

Allowance:

☐ Cooler ON

— Spindle rotation direction —

☒ CW ☐ CCW

Spindle RPM:

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.

Pass parameters.

Pass type. From the list the user can select zigzag/loop/spiral.

Pass Parameters

Pass type: Outward Spiral

Tool lift:

☐ Abs. ☒ Incr. 5

☐ Step of passes: 10

☒ Crest height: 0

Processing overlap: 0.5

Tool lift. The lift of the tool when making transitions between the zones of machining. Specified as an absolute or relative value.

Step of passes. Specifies the value of the step between the passes of the tool.

Number of passes. Specifies the number of the passes of the tool.

Crest height. The value of the step between the passes of the tool is calculated here based on the given crest height and the value of the **overlap** of the tool on the neighbouring passes.

Stock removal parameters.

Thickness. The amount of the removed material in the direction of the vector of the tool. The number of the layers being removed is calculated on the basis of specified values of the "Thickness" and "Deepening step" parameters.

Stock Removal Parameters

Thickness: 0

Step of a deepening: 10

Side offset: 0

Pass direction (grad.): 0

Contours Filter: All

☐ 2.5D Plane Milling

☐ Change direction

☐ Optimised Offset

☐ Open Pocket

Deepening step. The amount of the removed material in one layer. The number of the removed layers is calculated on the basis of the values of the "Thickness" and "Deepening step" parameters.

Side offset. Serves to specify equidistant offset. It can be positive and negative.

2.5D Plane. The toolpath of the motion of the tool is constructed without taking into account collision of the tool with the part. It is used when machining the planes and through holes.

Change direction. Passes of the tool will be directed in the opposite direction.

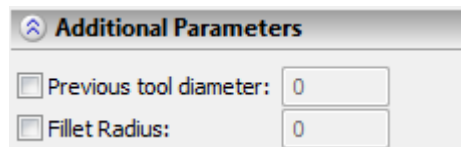
Optimized offset. Offset is carried out only for those segments of the toolpath which contain the walls (closed segments of the surface). The offset will not be carried out on the open segments of the surface.

Open pocket. This option allows us to carry out the high speed machining of the open pocket without using cutting-in. Exit/entry for each layer is carried out through the open part of the pocket. By doing this, the passes change gradually from an open contour to a closed contour that is shifted taking into account the side allowance and radius of the tool. Two methods of specifying the topology of the pocket are possible:

- 1) Two open-ended contours (open and closed) that have common start and end points. The open contour must be specified first.
- 2) Face of the pocket. For this method, selection of both contours is carried out automatically. If the face contains internal bounded contours, they will automatically be suppressed.

Additional parameters

Previous tool diameter. Diameter of the previous tool. When calculating the toolpath, T-FLEX CAM performs analysis of the part being machined and performs the search of the segments of the part which cannot be machined with the previous tool but can be machined with a new tool. After that the calculation of the passes of the selected type is carried out only in the unfinished segments of the part.



Rounding radius. When this option is enabled, the radius of curvature of the toolpath at each point will not exceed the specified value. This option is active, for example, for high speed machining.

Approximation

Precision of the approximation is specified as a numerical value in units supported by the CNC center and constitutes the magnitude of the caving (maximum distance from the arc of the circle to be straightened to the approximating line).

Circular interpolation. In case when the contour being machined has circles, arcs, splines, and other geometrical elements having similar shape, the selection of this parameter will allow us to obtain the control sequence that contains frames of the circular interpolation.

Overcutting control. Enabling the mechanism of control of the overcuts of the part, which are produced by the tool, with the precision specified in «CAM-system Settings». The segments of the

toolpath with the overcuts will be highlighted with a black color. In addition, the frames with the overcuts will be seen in «Machining Simulation».

Tool's rotation.

Rotation of the tool can be specified by either directly specifying the angles of rotation, or by specifying the tool's vector.

Tool's vector. Prescribe orientation of the vector that defines the location of the tool's axis upon machining. When this flag is enabled, the required angles of the tool's rotation are calculated automatically, and the windows for defining the values of the angles will show these calculated values.

Rotation angles. Prescribe the angles of rotation around the Z-axis and then around the X or Y axis (according to the switch).

Radius correction.

On. Enabling this flag allows us to use parameters of the radius correction.

Correction can be **left** and **right**.

Correction is specified by the **number**. During machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number specified by the user.


Offset correction.

On. Enabling this flag allows us to use parameters of the offset correction.



Correction can be for the **increase** and **decrease**.








Correction is specified by the **number**. During machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number specified by the user.


Surface of a revolution

To create a toolpath of machining of a surface of revolution, press the  button in the automenu of 3D zone milling. On the screen the dialog window of toolpath parameters will appear which the user can change as desired, and the system will transition to the mode of selection of the body.


This type of machining serves for creation of toolpaths of multiaxial, finishing and rough machining of various elements (grooves, small areas, etc.) on cylindrical surfaces and the bodies of revolution. To create such a toolpath it is desirable to place the body of revolution in the 3D scene in such a way that the axis of rotation of the body coincides with the Z-axis of the global coordinate system.

	<End>	Finish editing
	<I>	Select another closest element

	<Esc>	Exit command
		Select body
	<F>	Select face
	<P>	Select guiding path
	<L>	Select profile
	<O>	Select orienting face
	<K>	Cancel selection



Upon selection of 3D paths or a profile as a contour of the body, first in the 3D scene of T-FLEX CAD it is required to indicate the body being machined (the  option). When selecting the faces, the body being machined does not have to be selected. Furthermore, it is possible to use various combinations (for example, selection of faces and 3D-paths).

Selection of the orienting face allows us to automatically specify the vector of the tool's axis. The values for the tool's vector coordinates will then be inserted into the corresponding fields of the "Tool's rotation" tab. Furthermore, as an orienting face the user can select, in addition to planes, spherical, conic, cylindrical and toroidal surfaces. In this case, the axis of symmetry will be used as a tool's vector.

The  option allows us to cancel all selected elements for their subsequent repeated specification.

Toolpath parameters.

Toolpath. Name of a toolpath.

Tool's file. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilized for the chosen type of machining.

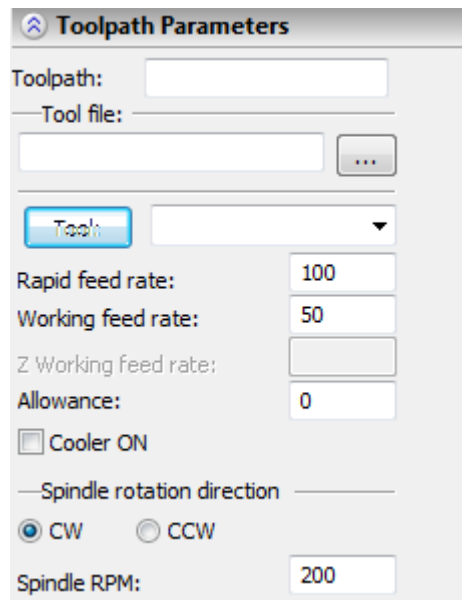
Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary translations.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the tool's translation when executing the motion of cutting.

Allowance. This parameter defines the distance between the given and equidistant contours.

Cooler on. Turning on/off the supply of the cooling agent upon machining.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.



Toolpath Parameters

Toolpath:

Tool file:

Rapid feed rate:

Working feed rate:

Z Working feed rate:

Allowance:

☐ Cooler ON

Spindle rotation direction: ☒ CW ☐ CCW

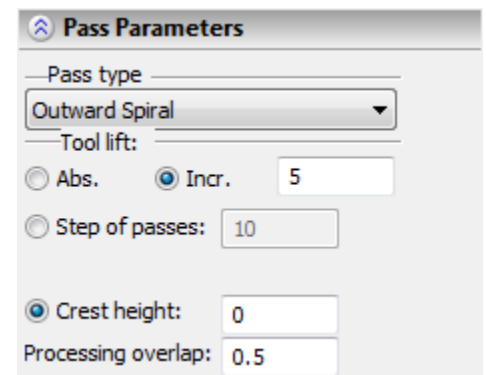
Spindle RPM:

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.

Pass parameters.

Pass type. From the list the user can select zigzag/loop/spiral.

Tool lift. The lift of the tool when making transitions between the zones of machining. Specified as an absolute or relative value.



Pass Parameters

Pass type:

Tool lift: ☐ Abs. ☒ Incr.

☐ Step of passes:

☒ Crest height:

Processing overlap:

Step of passes. Specifies the value of the step between the passes of the tool.

Number of passes. Specifies the number of the passes of the tool.

Crest height. The value of the step between the passes of the tool is calculated here based on the given crest height and the value of the **overlap** of the tool on the neighbouring passes.

Stock removal parameters.

Thickness. The amount of the removed material in the direction of the vector of the tool. The number of the layers being removed is calculated on the basis of specified values of the "Thickness" and "Deepening step" parameters.

Deepening step. The amount of the removed material for one layer. The number of the removed layers is calculated on the basis of the specified values of the "Thickness" and "Deepening step" parameters.

Side offset. Serves to specify equidistant offset. It can be positive and negative.

Cylindrical surface. This option should be enabled in cases when the machined elements of the body of revolution are located on its cylindrical portion (or in cases when the base of the machined body – is a cylinder).

Change direction. Passes of the tool will be directed in the opposite direction.

Reflect toolpath. Enabling this flag will change the calculation of the toolpath from the external surface to the internal one. The vector of the tool will then change the direction by 180 degrees.

Additional parameters.

Previous tool diameter. Diameter of the previous tool. When calculating the toolpath, T-FLEX CAM performs analysis of the part being machined and performs the search of the segments of the part which cannot be machined with the previous tool but can be machined with a new tool. After that the calculation of the passes of the selected type is carried out only in the unfinished segments of the part.

Rounding radius. When this option is enabled, the radius of curvature of the toolpath at each point will not exceed the specified value. This option is active, for example, for high speed machining.

Approximation.

Precision of the approximation is specified as a numerical value in units supported by the CNC center and constitutes the magnitude of the caving (maximum distance from the arc of the circle to be straightened to the approximating line).

Circular interpolation. In case when the contour being machined has circles, arcs, splines, and other geometrical elements having similar shape, the selection of this parameter will allow us to obtain the control sequence that contains frames of the circular interpolation.

Overcutting control. Enabling the mechanism of control of overcuts of the part, which are produced by the tool, with the precision specified in «CAM-system Settings». The segments of the toolpath with overcuts will be highlighted with a black color. In addition, the frames with the overcuts will be seen in «Machining Simulation».

Tool's rotation.

Rotation of the tool can be specified by either directly specifying the angles of rotation, or by specifying the tool's vector.

Tool's vector. Prescribe orientation of the vector that defines the location of the tool's axis upon machining. When this flag is enabled, the required angles of the tool's rotation are calculated automatically, and the windows for defining the values of the angles will show their calculated values.

Rotation angles. Prescribe the angles of rotation around the Z-axis and then around the X or Y axis (according to the switch).

Radius correction.

On. Enabling this flag allows us to use parameters of the radius correction.

Correction can be **left** and **right**.

Correction is specified by the **number**. During machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number specified by the user.


Offset correction.

On. Enabling this flag allows us to use parameters of the offset correction.





Correction can be for the **increase** and **decrease**.

Correction is specified by the **number**. During machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number specified by the user.

Spiral Milling



To create a toolpath of spiral milling, press the  button in the automenu of 3D zone milling. On the screen the dialog window of toolpath parameters will appear which the user can change as desired, and the system will transition to the mode of selection of the body.

This type of machining is mainly used for machining bodies of revolution along the concentric circles.

	<End>	Finish editing
	<I>	Choose another closest element
	<Esc>	Exit command
		Choose body

Toolpath parameters.

Toolpath. Name of a toolpath.

Tool's file. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilized for the chosen type of machining.

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary translations.

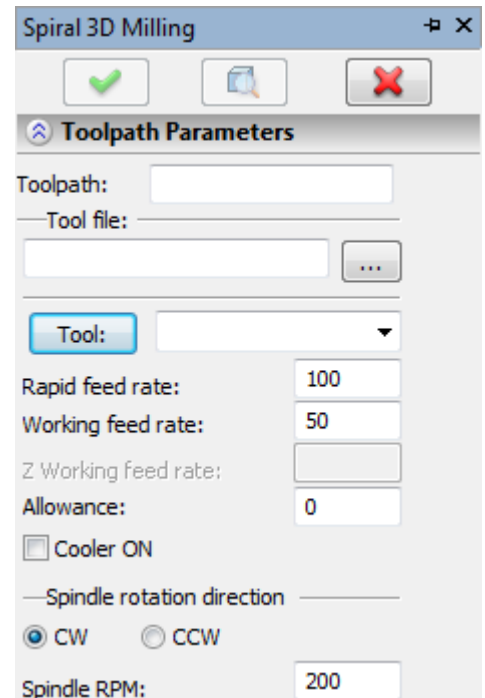
Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the tool's translation when executing the motion of cutting.

Allowance. This parameter defines the distance between the given and equidistant contours.

Cooler on. Turning on/off the supply of the cooling agent upon machining.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.

**Spiral's parameters.**

Step. Step of a spiral.

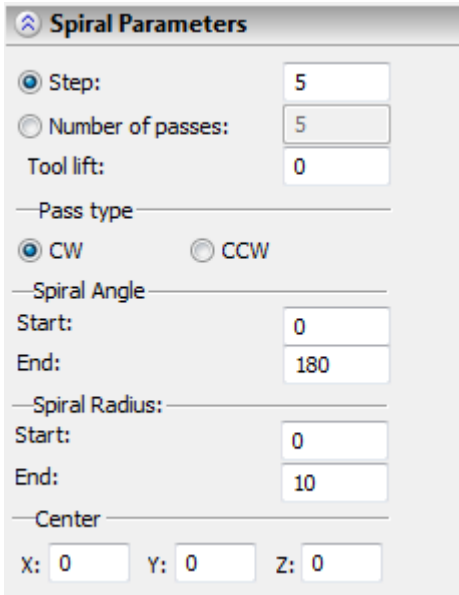
Number of passes. Specifies the number of the passes of the tool.

Tool lift. Lift of the tool when making transitions between the machining zones.

Direction of passes. Clockwise or counterclockwise.

Angle of a spiral. This group of parameters specifies the angle between the radii that cut off the sector of the body of revolution. The machining toolpath will be created for this sector. **Start** -

specifies the angle measured from the X-axis in degrees (from 0 to 360). **End** - specifies the angle measured from the Y-axis in degrees (cannot be smaller than the value of the "Start" parameter.)



Spiral Parameters

☒ Step: 5
☐ Number of passes: 5
 Tool lift: 0

— Pass type —
☒ CW ☐ CCW

— Spiral Angle —
 Start: 0
 End: 180

— Spiral Radius: —
 Start: 0
 End: 10

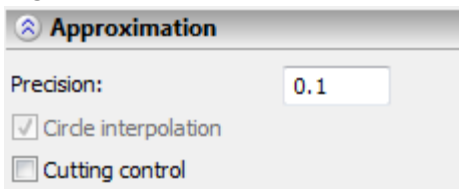
— Center —
 X: 0 Y: 0 Z: 0

Radius of a spiral. This group of parameters specifies the distances from the Z-axis of the indicated system of coordinates which bound a portion of the machined surface of the body. **Start** – specifies the distance from the Z-axis (the axis of rotation) up to the start of the machined zone. **End** – specifies distance from the Z-axis (axis of rotation) up to the last point of the machined zone (cannot be smaller than the value of the "Start" parameter).

Center of the coordinate system. Specifies absolute values of the coordinates of local CS, with respect to which the machining toolpath will be calculated. The Z-axis of the LCS is taken as the axis of rotation of the body.

Approximation.

Precision of the approximation is specified as a numerical value in units supported by the CNC center and constitutes the magnitude of the caving (maximum distance from the arc of the circle to be straightened to the approximating line).



Approximation

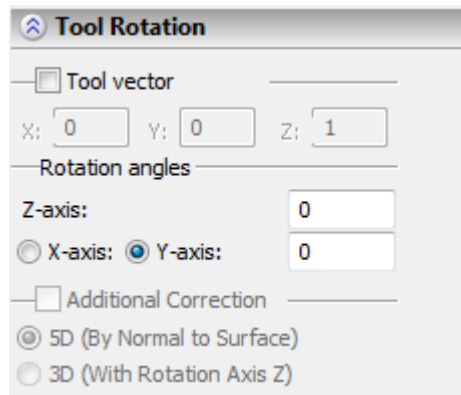
Precision: 0.1

☒ Circle interpolation
☐ Cutting control

Overcutting control. Enabling the mechanism of control of overcuts of the part, which are produced by the tool, with the precision specified in «CAM-system Settings». The segments of the toolpath with the overcuts will be highlighted with a black color. In addition, the frames with overcuts will be seen in «Machining Simulation».

Tool's rotation.

Rotation of the tool can be specified by either directly specifying the angles of rotation, or by specifying the tool's vector.



Tool's vector. Prescribe orientation of the vector that defines the location of the tool's axis upon machining. When this flag is enabled, the required angles of the tool's rotation are calculated automatically, and the windows for defining the values of the angles will show their calculated values.

Rotation angles. Prescribe the angles of rotation around the Z-axis and then around the X or Y axis (according to the switch).

Radius correction.

On. Enabling this flag allows us to use parameters of the radius correction.

Correction can be **left** and **right**.

Correction is specified by the **number**. During machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number specified by the user.

Offset correction.

On. Enabling this flag allows us to use parameters of the offset correction.

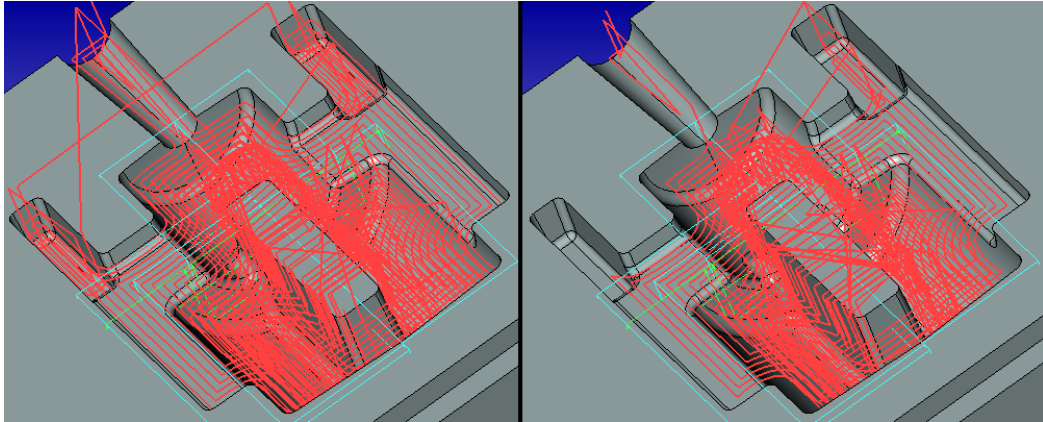
Correction can be for the **increase** and **decrease**.

Correction is specified by the **number**. During machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number specified by the user.

Examples of Using a Simplified Model for 3D Bounded Machining (Part 1)

The following examples illustrate roughing with spiral-type passes.

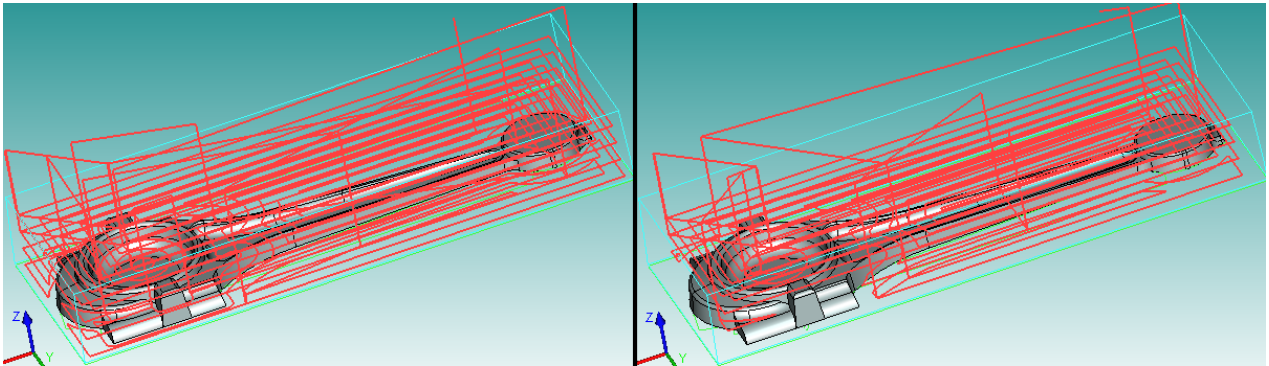
Example 1 (Right half-form)



a) mill diameter = 6.6 mm

b) mill diameter = 15 mm

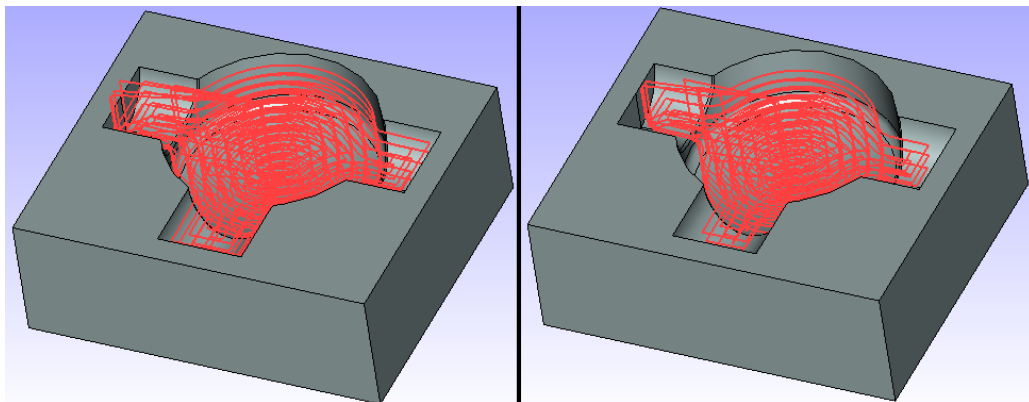
Example 2 (Connecting rod)



a) mill diameter = 8 mm

b) mill diameter = 20 mm

Example 3 (Stamped box)



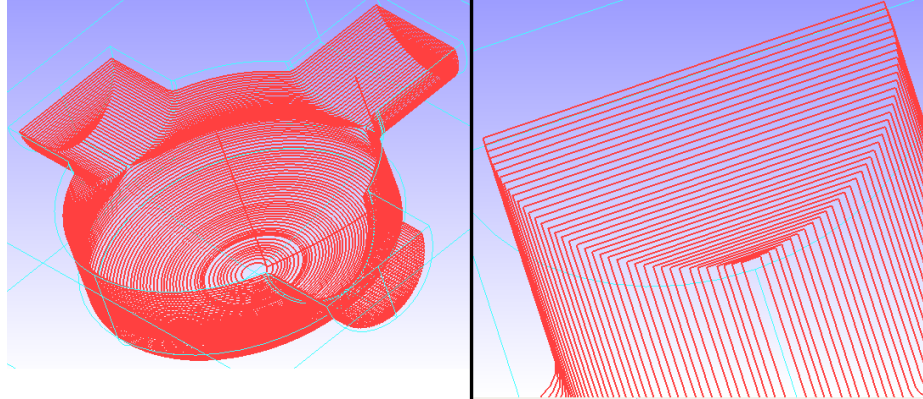
a) mill diameter = 12 mm

b) mill diameter = 30 mm

Examples of Using a Simplified Model for 3D Bounded Machining (Part 2)

The following examples illustrate using a simplified model combined with a “Deseaming – loop”-type pass.

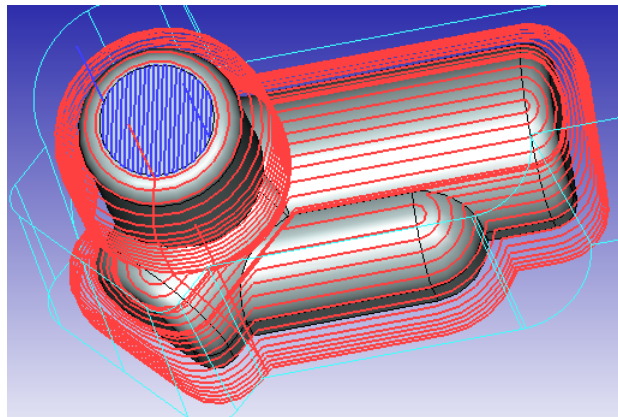
Example 1 (Stamped box)



a) Toolpath calculation result

b) The fragment illustrating a variable section step

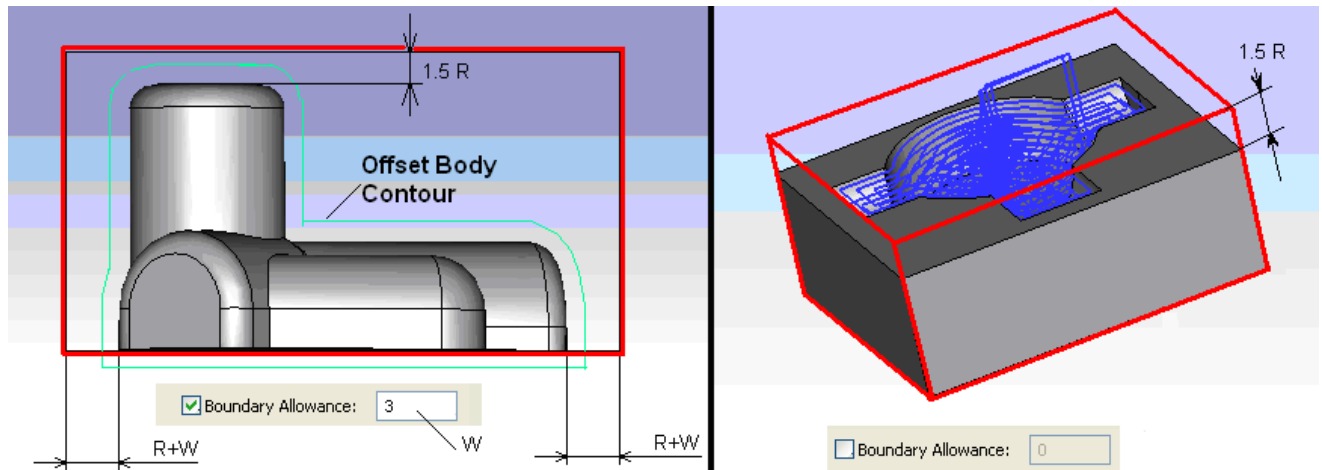
Example 2 (Cap)



Shown in Red are the toolpaths calculated using the “Deseaming – loop” + “Simplified model” machining, Blue is a flat zone machined by the 3D surface milling method.

Usage Examples (Part 3)

These examples illustrate the selection of the bounding body based on the outline box (in the case when the second body is not selected).



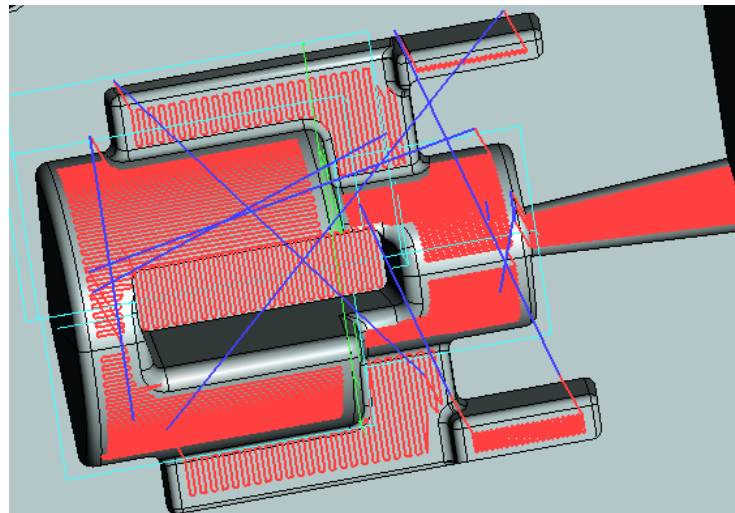
a) for a convex body we recommend an extra offset an extra offset

b) When milling cavities no extra offset is required

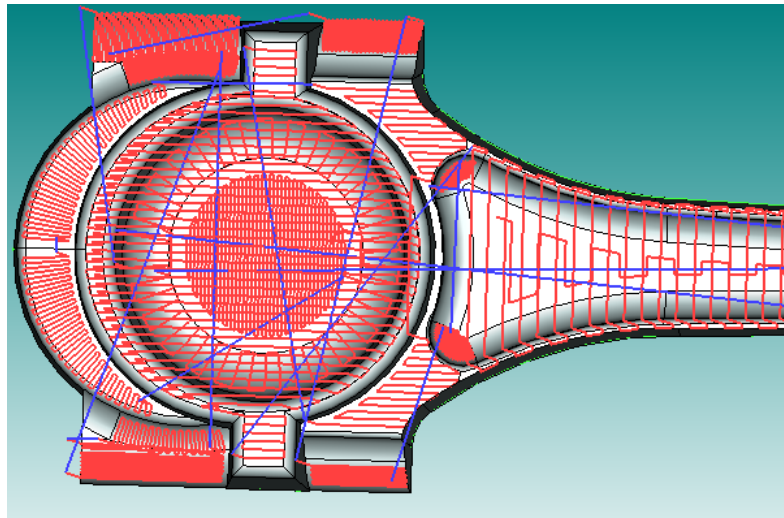
Usage Examples (Part 4)

The following examples illustrate the use of the **Surface Offset** option in 3D surface milling. **Optimized** offset. **Zigzag in X**.

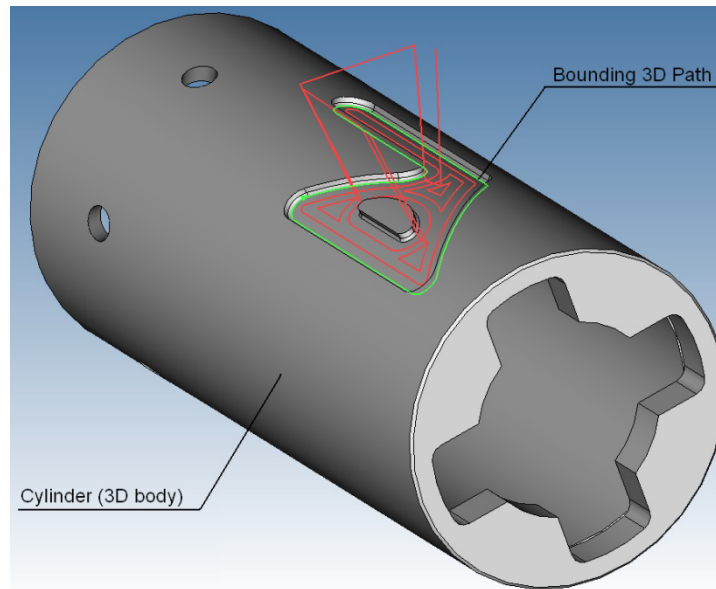
Example 1 (Right half-form)



Example 2 (Connecting rod)




Example 3 (Cylinder)










In this case machining on cylindrical surface is used.

5D MILLING


To define a toolpath and a control sequence for 5D milling, use the command:

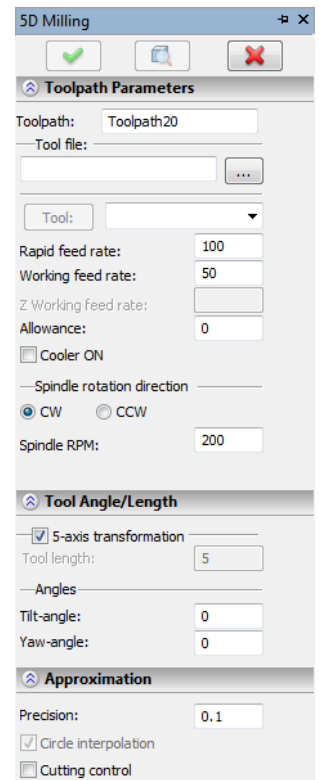
Icon	Ribbon
	CNC → Machining → 5D milling
Keyboard	Textual Menu
	CNC>3D and 5D machining>5D - milling

Upon calling the command, the following options become available in the automenu:

	<Y>	Finish input
	<P>	Set entity parameters
	<I>	Select Other Element
	<X>	Exit command
	<C>	5D contour milling
	<P>	5D surface milling
	<G>	GOTO toolpath





Creation of a toolpath

After calling the command, the dialog window "5D Milling" will appear on the screen. Parameters selected in this window will be enabled **by default** for all newly created toolpaths of 5D milling. When creating a specific toolpath, it is possible to change these parameters. The user can change them as desired and save these changes by pressing the  button.



5D contour milling



To create a toolpath of 5D contour milling, press the corresponding button in the automenu of 5D milling. On the screen the dialog window of toolpath parameters will appear which the user can change as desired, and the system will transition to the mode of selection of a 3D path.

	<Y>	Finish input
	<I>	Select another closest element
	<X>	Exit command
	<E>	Select 3D path

In the 3D scene of T-FLEX CAD it is required to specify the 3D-path that will form a contour to be machined.

Toolpath parameters.

Toolpath. Name of a toolpath.

Tool's file. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilized for the chosen type of machining.

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary translations.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the tool's translation when executing the motion of cutting.

Allowance. This parameter defines the distance between the given and equidistant contours.

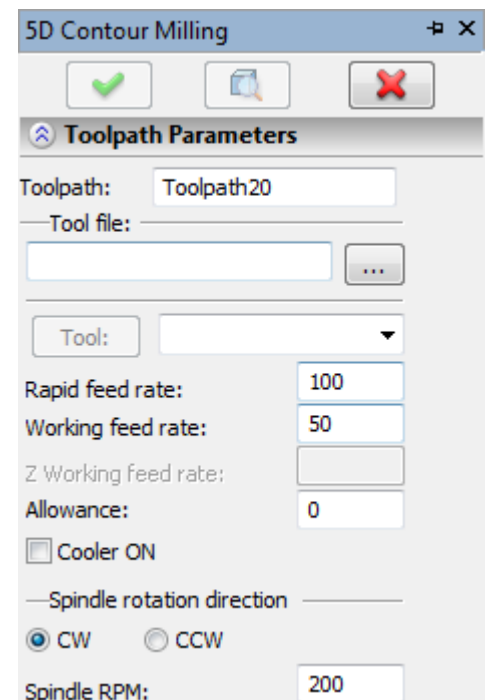
Cooler on. Turning on/off the supply of the cooling agent upon machining.

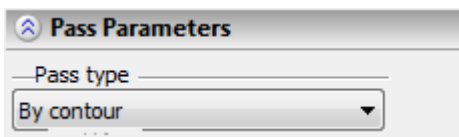
Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.

Pass parameters.

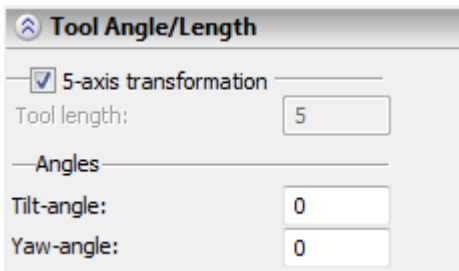
Pass type. Along the contour or against the contour.





Tool's angle of tilt/offset.

5D transformation. This parameter allows us to carry out the calculation of the toolpath in coordinates of the part or in coordinates of the machining station taking into account the tool's offset. If five-coordinate transformation is disabled, the tool's offset must be specified independently in the **adjusting parameter**.

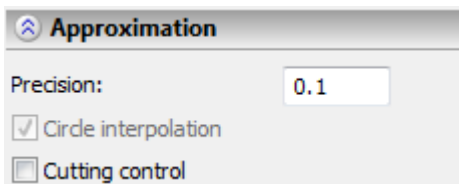


Advancing angles. They can be specified **along the direction of the motion** of the tool and **perpendicular to the direction of the motion** of the tool. By advancing angle we imply the tilt of the tool during the machining for transferring the point of cutting from the center of the tool (creation of a nonzero angular velocity).

Approximation.

Precision of the approximation is specified as a numerical value in units supported by the CNC center and constitutes the magnitude of the caving (maximum distance from the arc of the circle to be straightened to the approximating line).

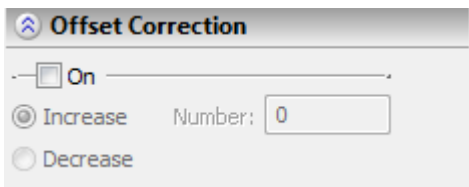
Overcutting control. Enabling the mechanism of control of overcuts of the part, which are produced by the tool, with the precision specified in «CAM-system Settings». The segments of the toolpath with the overcuts will be highlighted with a black color. In addition, the frames with the overcuts will be seen in «Machining Simulation».



Offset correction.

On. Enabling this flag allows us to use parameters of the offset correction.

Correction can be for the **increase** and **decrease**.



Correction is specified by the **number**. During machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number specified by the user.

5D surface milling

To create a toolpath of 5D surface milling, press the corresponding button in the automenu of 5D milling. On the screen the dialog window of the toolpath parameters will appear, which the user can change as desired, and the system will transition to the mode of selection of the face.

	<Y>	Finish input
	<I>	Select another closest element
	<X>	Exit the command
	<F>	Select a face

In the 3D scene of T-FLEX CAD it is required to indicate the face to be machined.

Toolpath parameters.

Toolpath. Name of a toolpath.

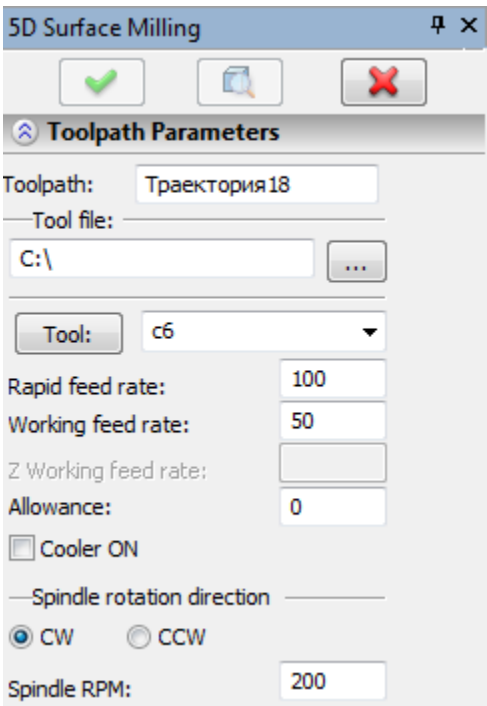
Tool's file. With the help of the button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the button.

It is to be noted that the system will offer the user to select only that tool which can be utilized for the chosen type of machining.

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary translations.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the tool's translation when executing the motion of cutting.

Allowance. This parameter defines the distance between the given and equidistant contours.



Cooler on. Turning on/off the supply of the cooling agent upon machining.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.

Pass parameters.

Pass type. Zigzag or loop.

The screenshot shows a 'Pass Parameters' dialog box with the following settings:

- Pass type:** Zigzag in X (selected from a dropdown menu)
- Tool lift:**
 - ☐ Abs.
 - ☒ Incr. 5
- Num. of passes:** 10 (selected with a radio button)
- Crest height:** 0.5 (radio button is unselected)
- Processing overlap:** 1
- Counter Contour:** ☐ (unselected)
- Change Direction:** ☐ (unselected)

Tool lift. The lift of the tool when making transitions between the zones of machining. Specified as an absolute or relative value.

Number of passes. Specifies the number of passes of the tool.

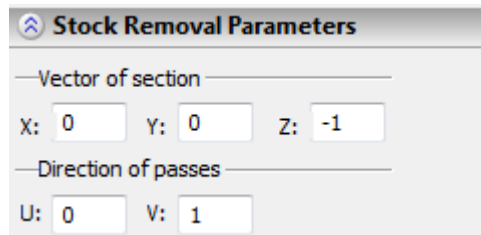
Crest height. The value of the step between the passes of the tool is calculated here based on the given crest height and the value of the **overlap** of the tool on the neighbouring passes.

Counter contour. Machining will be carried out in reverse order with respect to the initial contour of the toolpath (along the isoparametric line U).

Change direction. The removal of the material will start in the opposite direction, with respect to the initial direction of passes, at the opposite side of the face being machined (in the direction of the isoparametric line V).

Stock removal parameters.

Vector of sections. The vector of normal to the planes in which the passes lie. It defines the direction of the layers of the material being removed and the direction of the motion of the tool from one layer to another. For example, if the vector is specified as (0,0,1) – the layers of the material being removed are located parallel to the XY-plane. In the rough machining the vector is specified only along the Z-axis ($X=0$ $Y=0$, $Z \neq 0$). Therefore, the positions X and Y of the **vector of sections** are not active for the rough machining, and their values are not taken into consideration upon calculation of the toolpath.



Stock Removal Parameters

Vector of section

X: 0 Y: 0 Z: -1

Direction of passes

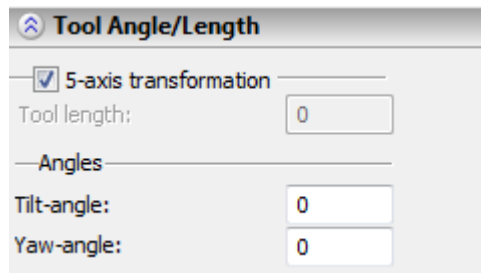
U: 0 V: 1

Direction of passes. Specifies the vector (X,Y) which shows the direction of the passes of the tool in one layer of the material that is being removed. This field is used only for rough machining for all passes except for passes of a spiral type (in this case, this element is not active). This vector must lie in the plane of the layer being removed, hence, must be perpendicular to the vector of sections. Therefore, not all combinations of the components of the vectors of sections and directions of passes are admissible.

Tool's angle of tilt/offset.

5D transformation. This parameter allows us to carry out the calculation of the toolpath in coordinates of the part or in coordinates of the machining station taking into account the tool's offset. If five-coordinate transformation is disabled, the tool's offset must be specified independently in the **adjusting parameter**.

Advancing angles. They can be specified **along the direction of the motion** of the tool and **perpendicular to the direction of the motion** of the tool. By advancing angle we imply the tilt of the tool during the machining for transferring the point of cutting from the center of the tool (creation of a nonzero angular velocity).



Tool Angle/Length

☒ 5-axis transformation

Tool length: 0

Angles

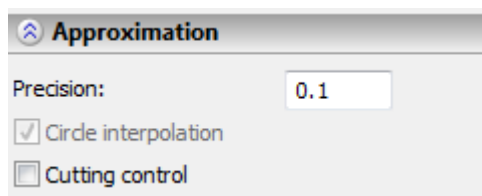
Tilt-angle: 0

Yaw-angle: 0

Approximation.

Precision of the approximation is specified as a numerical value in units supported by the CNC center and constitutes the magnitude of the caving (maximum distance from the arc of the circle to be straightened to the approximating line).

Overcutting control. Enabling the mechanism of control of overcuts of the part, which are produced by the tool, with the precision specified in «CAM-system Settings». The segments of the toolpath with the overcuts will be highlighted with a black color. In addition, the frames with the overcuts will be seen in «Machining Simulation».

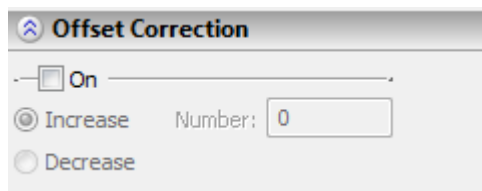




Offset correction.


On. Enabling this flag allows us to use parameters of the offset correction.

Correction can be for the **increase** and **decrease**.

Correction is specified by the **number**. During machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number specified by the user.




After that, the candidate toolpath will be calculated and displayed on the screen. To add the created toolpaths to the document, click the  icon in the automenu. If the wrong type of machining was selected from the beginning, you can cancel all changes and exit by pressing .


Once all necessary changes are done to the machining parameters, the user just needs to click the **[OK]** button to save the toolpath in a file with the prepared drawing of the part being machined. The calculated toolpath will also be added to the list of available toolpaths in the Machining Manager. To view this list, click the  icon.

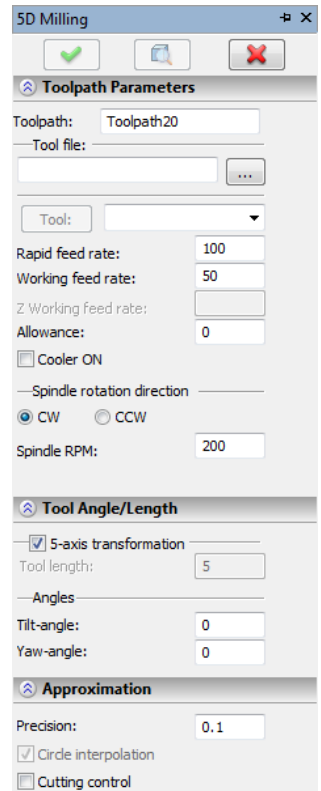
BOUNDED 5D MILLING

To define a toolpath and a control sequence for bounded 5D milling, use the command:

Icon	Ribbon
	CNC → Machining → 5D milling
Keyboard	Textual Menu
	CNC>Bounded machining>5D milling











Creation of toolpath



After calling the command, the dialog window "5D Milling" will appear on the screen. Parameters selected in this window will be enabled **by default** for all newly created toolpaths of 5D milling. When creating a specific toolpath, these parameters can be changed. The user can change them as desired and save these changes by pressing the  button.








Domain milling

When selecting the option of milling a domain, the following options will appear in the automenu:

	<Y>	Finish input
	<I>	Select another closest element
	<X>	Exit command
		Select a body
	<F>	Select a face
	<P>	Select a guiding path
	<F>	First bounding path
	<S>	Second bounding path
	<T>	Third bounding path
	<N>	Orienting path

After invoking an option, it is required to indicate the body being machined () or the face being machined (.



When the body to be machined is selected, the user can specify additional guiding and bounding paths.

- guiding path . When only the guiding path is indicated, machining by this path in the direction of the normal to the surface will be created.
- first bounding path . When indicating the guiding path and the first path - the finishing with passes between these paths will be calculated.
- second bounding path . In addition to the previous case, this path defines orientation of the tool.
- third bounding path . When specifying four paths, the rough machining is generated. This type of machining is mainly used for machining of blades or hubs.
- orienting path . For rough machining, when four paths are specified, the fifth path will be considered as orienting path for the tool.

After selecting one or another element to be machined, the user must specify certain parameters of machining in the «5D Domain Milling» window. It is recommended first to specify the required geometric elements; depending on their selection, the interface of dialogs of parameters of the toolpath will change.

Toolpath parameters.

Toolpath. Name of a toolpath.

Tool's file. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilized for the chosen type of machining.

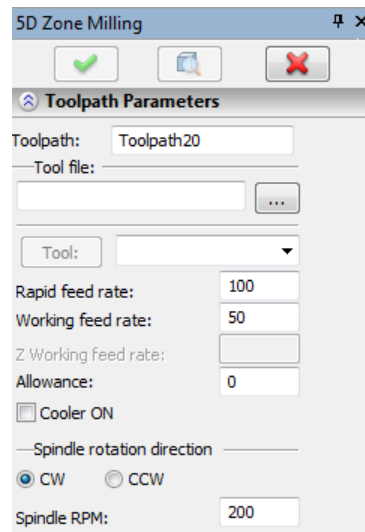
Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary translations.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the tool's translation when executing the motion of cutting.

Allowance. This parameter defines the distance between the given and equidistant contours.

Cooler on. Turning on/off the supply of the cooling agent upon machining.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.



Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.

Machining parameters.

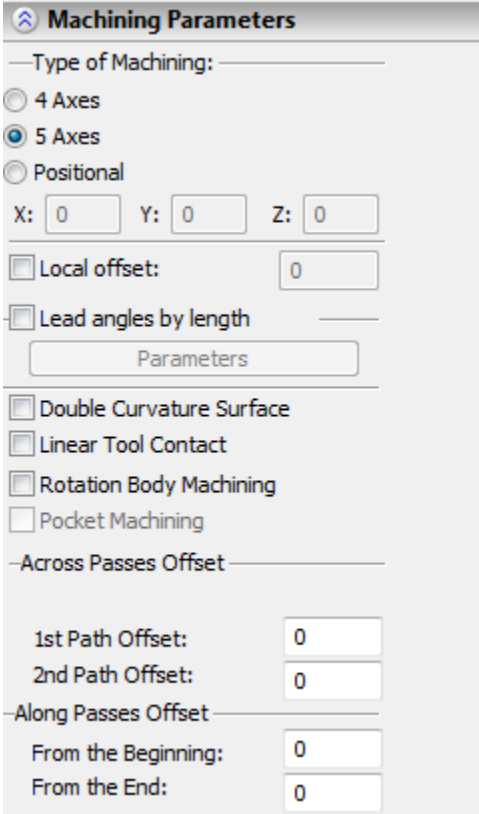
Type of machining. Selection of the type of machining is directly related to the "Double curvature surface" flag. When this flag is enabled, the toolpath is constructed along the normal to the surface, and then lateral surfaces are not taken into account. Furthermore, in **5-coordinate machining** – all five coordinates are uploaded to the control sequence; in **4-coordinate machining** – the tool is always located in the plane perpendicular to the Z-axis, and 4 coordinates are uploaded to the control sequence; **positional machining** – allows us to arbitrarily orient the vector of the tool. If the "Double curvature surface" flag is disabled, the tilt angle of the tool is initially oriented by the first two paths.

Direction. The flag is active only in the mode of "Positional" milling. In combination with the **Along the path length** option, it allows us to change, according to the specified law, the tilt angle of the tool from one pass to another, and not from a point to a point, which can significantly (up to 5-10 times) reduce the machining time. If the **Along the path length** option is not active (i.e., advancing angles are active), then the "Along the path length" angle will define the maximum deviation towards the middle pass (this allows us to avoid overcuts on the boundaries of the blade and walls), and the "Perpendicular to the motion" angle (B) – deviation from the base vector of the tool towards the current normal (i.e., when $B=0$ the tool is located strictly by the specified vector).

Local allowance. This parameter allows us to shift the points of the toolpath along the axis of the tool which simplifies construction of semifinished (with allowance) toolpaths without overcuts, on the basis of finishing toolpaths.

Parameters along the path length. For multiaxial machining of the most complicated parts (for example, spinner, turbine wheels etc.), it is possible to apply the capability of modifying the tilt angle of the tool during the course of machining. For further detail see here.

Linear contact. This option is intended for multiaxial machining of parts by a biconic mill. Enabling this option provides for the maximum area of contact of the lower conic surface of the mill with the surface being machined.



Machining Parameters

Type of Machining:

☐ 4 Axes

☒ 5 Axes

☐ Positional

X: 0 Y: 0 Z: 0

☐ Local offset: 0

☐ Lead angles by length

Parameters

☐ Double Curvature Surface

☐ Linear Tool Contact

☐ Rotation Body Machining

☐ Pocket Machining

Across Passes Offset

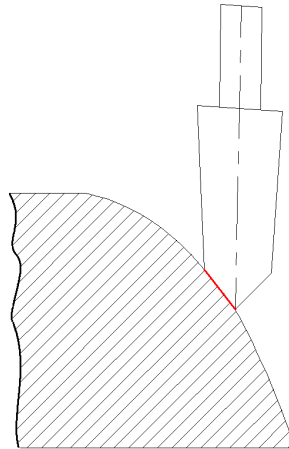
1st Path Offset: 0

2nd Path Offset: 0

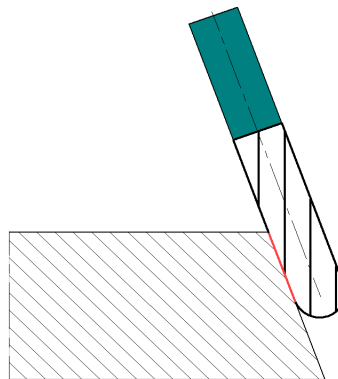
Along Passes Offset

From the Beginning: 0

From the End: 0



Machining body of revolution. Enabling this option allows us to calculate the toolpath of machining of a body of revolution with the maximum fit between the lateral surface of the cutting tool and the surface being machined.



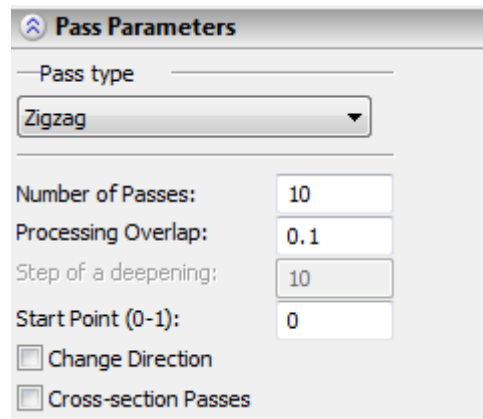
Pocket machining. This option allows us to form the passes with control of collisions with the part and with enabling all 5D options (offsets, transverse passes, control of the tilt angle of the tool, etc.). The specified 3D paths can then be arbitrary (unlike the standard 5D machining) and do not have to belong to the body of the part. The passes are formed by means of the projection of a contour (with control of collisions with the part). The contour is specified by two basis 3D paths and the current number of the pass along the tool's vector, which is defined by the initial vector (X,Y,Z) and by the "Along the path length" parameters (for Positional machining). For Positional machining the projection vector can change from one pass to another (if the "Direction" and "Along the pass length" flags are enabled). For 5D machining (or Positional machining with disabled flags "Direction" and/or "Along the path length") the vector of projection is constant but the tool's vector itself can change while being formed, as in standard 5D machining, taking into account all remaining options. In this case, however, there is no total guarantee that the overcuts are absent and the angle of projection differs from the current tilt angle of the tool.

Transverse offset. Capability to define the offset along and across the passes. This parameter is specified by the magnitude or by the number of passes. It allows us to:

- a) easily divide one machining into two (for example, for optimal selection of the tool's vector on each side) which often makes the smooth change of the angle from one lateral wall to another unnecessary;
- b) carry out machining with a tool of larger diameter without constructing roundings with subsequent machining of zones adjacent to the corners with a tool of a smaller diameter.

Pass parameters.

Pass type. **Zigzag** – layerwise or surface, equidistant zigzaglike passes; **Loop** – layerwise or surface, equidistant threadlike passes with traverse of the tool in the air for providing single-direction cutting; **Spiral** – layerwise or surface, spiral-like passes.



Sort out passes. Toolpaths are calculated by parametric lines of the surface, to which the face belongs (similar to 5D milling of a face), but with possibility to use most of the capabilities of 5D zone milling (4D, 5D or positional milling, parameters along the length path, local allowance, offsets, cutting-in, points of retreat/approach in traverses, loop with return, etc.). The "sort out passes" option allows us to obtain the optimum order of connection between individual passes with minimum total length of traverses.

Number of passes. Specifies the number of passes of the tool.

Crest height. The value of the step between the passes of the tool is calculated here based on the given crest height and the value of the **overlap** of the tool on the neighbouring passes.

Counter contour. Machining will be carried out in reverse order with respect to the initial contour of the toolpath (along the isoparametric line U).

Change direction. The removal of the material will start in the opposite direction, with respect to the initial direction of passes, at the opposite side of the face being machined (in the direction of the isoparametric line V).

Tool's angle of tilt/offset.

5D transformation. This parameter allows us to carry out the calculation of the toolpath in coordinates of the part or in coordinates of the machining station taking into account the tool's

offset. If the five-coordinate transformation is disabled, the tool's offset must be specified independently in the **adjusting parameter**.

Advancing angles. They can be specified **along the direction of the motion** of the tool and **perpendicular to the direction of the motion** of the tool. By advancing angle we imply the tilt of the tool during the machining for transferring the point of cutting from the center of the tool (creation of a nonzero angular velocity).

Point of approach/retreat in traverses.

Tool lift. Specifies the distance up to the safety plane upon each traverse.

Point of approach in traverse – point to which the tool will be withdrawn before each subsequent pass.

Point of retreat in traverse – point to which the tool will be withdrawn after completion of each pass.

Approximation.

Precision of the approximation is specified as a numerical value in units supported by the CNC center and constitutes the magnitude of the caving (maximum distance from the arc of the circle to be straightened to the approximating line).

Overcutting control. Enabling the mechanism of control of overcuts of the part, which are produced by the tool, with the precision specified in «CAM-system Settings». The segments of the toolpath with the overcuts will be highlighted with a black color. In addition, the frames with the overcuts will be seen in «Machining Simulation».

Offset correction.


On. Enabling this flag allows us to use parameters of the offset correction.

Correction can be for the **increase** and **decrease**.








Correction is specified by the **number**. During machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number specified by the user.

5D DRILLING










To define a toolpath and a control sequence for 5D drilling, use the command:




Icon	Ribbon
	CNC → Machining → 5D drilling
Keyboard	Textual Menu
	CNC > 3D and 5D machining > 5D - drilling

Upon calling the command, the following options become available in the automenu:

	<Y>	Finish input
	<P>	Set entity parameters
	<I>	Select Other Element
	<X>	Exit command
	<K>	5D Drilling
	<C>	5D Drilling Cycles
	<G>	GOTO toolpath

After selection of the required item of the automenu (drilling or machining cycles of drilling), the options will become available for selection of geometric elements of 3D-scene construction which are required for creation of a toolpath:

	<Y>	Finish editing
	<P>	Specify parameters of the element
	<I>	Choose another closest element
	<X>	Exit the command
	<F>	Select a face
	<E>	Select an edge
	<L>	Select relative coordinate system
	<O>	Select orienting face
	<K>	Cancel selection

5D drilling toolpaths can be created on the basis of relative coordinate system  (it is used when there are holes with different orientation), on the basis of edges  (in the 3D-scene we select directly the holes, without auxiliary constructions of the coordinate system) or, alternatively, it is possible to select a face  that contains several holes, and they will be selected automatically.

The depth of holes is determined automatically for each hole individually or it is taken fixed (depending on the state of the "Depth" flag). For each toolpath, orientation of holes is assumed to be the same (if the relative coordinate systems are not used) and it is determined by the tool's vector (to specify the tool's vector use the corresponding dialog). The tool's vector can be set up manually or by means of selection of orienting face, and upon selection of a face the tool's vector is set up automatically along the averaged normal to the first selected face.


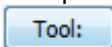
It is important to note that upon selection of skew holes, the orientation of the face being selected does not coincide with the orientation of the holes. Therefore, orientation of the holes must be inputted manually or with the help of orienting face. If the cylindrical face of the hole is selected as an orienting face, then the axis of the cylinder is taken as the tool's vector. Otherwise, the averaged normal to the surface of the face is used.

Selection of faces can be used not only for adding but also for elimination of individual holes. Elimination of the hole occurs in the case if it was selected 2 or more times: for example, the first time with the help of the face and the second - with the help of the edge or twice with the help of the edge (for example, by means of selection of two circles of the hole's chamfer or upper and lower circle of the cylindrical portion of the hole). Selection of one circle twice is impossible, which eliminates accidental repeated selection of the same circle.

Optimization of traversal of holes is carried out automatically with the help of the special algorithm.

Toolpath parameters.

Toolpath. Name of a toolpath.

Tool's file. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilized for the chosen type of machining.

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary translations.

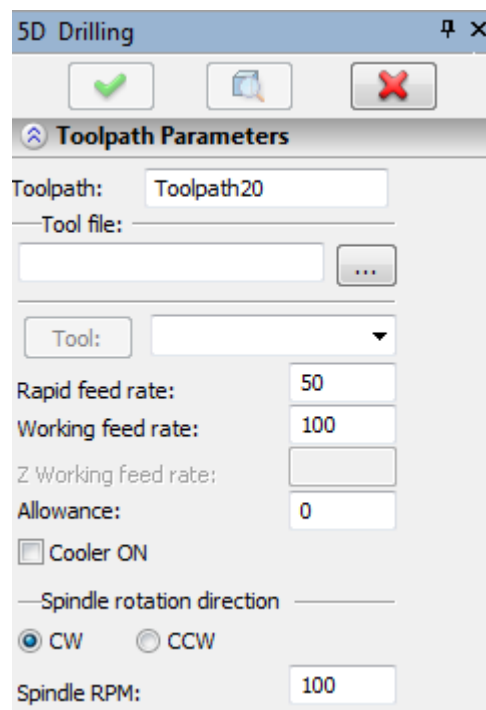
Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the tool's translation when executing the motion of cutting.

Allowance. This parameter defines the distance between the given and equidistant contours.

Cooler on. Turning on/off the supply of the cooling agent upon machining.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.



5D drilling parameter

Cycle number. Selection of drilling cycle (Preliminary facing - facing an area for centering a hole.

Drilling - centering and drilling a hole. **Boring** – boring (drilling out) a hole. **Final facing** - facing the bottom of the hole. **Cutting internal thread with a mill** - cutting internal thread with the help of the threaded mill. Cutting is carried out with the help of frames of spiral interpolation with subdivision into turns).

Cycle parameters given below can be partially enabled or disabled depending on the selected cycle.

Vertical displacement. Distance along the Z-axis from the axis of rotation of the part to the base plane.

Thread diameter. Diameter of the hole by which cutting of a thread will be carried out.

Thread depth. Depth of cutting of a thread.

Deep channel depth. Depth of a hole, obtained by drilling, specified from the center of the relative coordinate system.

Facing diameter. Diameter of the area being faced;

Thread pitch. The height of the cutting portion of the threaded mill.

Depth of facing/Depth of preliminary facing. Distance from the center of the relative coordinate system to the area being faced;

Depth of drilled-out hole. The depth of the drilled-out (widened) hole specified from the center of the relative coordinate system;

Allowance. The value of the allowance upon cutting-in (facing with cutting-in).

Length of cutting-in. Length of a segment on which the cutting-in of the tool is carried out (facing with cutting-in).

Angle. Value of the angle at which cutting-in is carried out (facing with cutting-in).

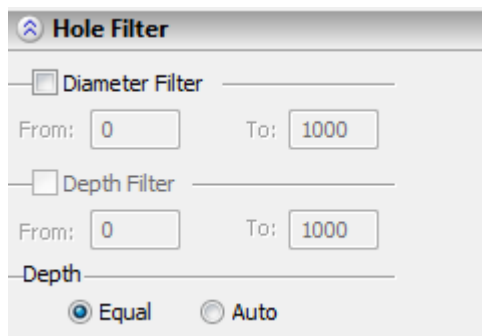
Z – rotation. The Z-coordinate to which the tool is withdrawn for rotation of the part and subsequent drilling (facing);

Z – reference plane. Distance from the center of the relative system of coordinates to the safety plane.

5D Drilling Parameters	
Number of cycle: Preliminary facing	
Vertical displacement:	100
Deep channel diameter:	5
Deep channel depth:	30
Facing diameter:	20
Facing depth:	25
Drilling-out hole diameter:	10
Drilling-out hole depth:	10
Preliminary facing depth:	20
Allowance:	0
Plunge cutting length:	10
Cutting-in angle:	0
Z- Tool change:	450
Z- Rotation:	350
Z - Reference plane:	7
<input type="radio"/> CW <input checked="" type="radio"/> CCW	
<input type="checkbox"/> Tool radius correction:	0

Hole's filter.

Diameters filter. Specify initial and final value of diameters of holes in the range of which filtration of holes will be carried out.



Depth filter. Specify initial and final value of the depth of the channel in the range of which filtration of holes will be carried out.

Depth. Fixed – this value is taken from the parameter of depth of (facing/channel/hole/thread). **Auto** – calculated automatically for "deaf" holes.

Tool's rotation.

Rotation of the tool can be specified by either directly specifying the angles of rotation, or by specifying the tool's vector.

Tool's vector. Prescribe orientation of the vector that defines the location of the tool's axis upon machining. When this flag is enabled, the required angles of the tool's rotation are calculated automatically, and the windows for defining the values of the angles will show their calculated values.

Rotation angles. Prescribe the angles of rotation around the Z-axis and then around the X or Y axis (according to the switch).

Offset correction.

On. Enabling this flag allows us to use parameters of the offset correction.

Correction can be for the **increase** and **decrease**.

Correction is specified by the **number**. During machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number specified by the user.


Parameters of 5D drilling cycles

For 5D drilling the following cycles are available: Drilling, quick exit (**G81**), Drilling with pause (**G82**), Deep drilling (**G83**), Cutting a thread by a tap (**G84**), Boring (drilling out) (**G85**).








Parameters of these cycles coincide with parameters of drilling.


CAM MILLING (4D MACHINING)

To define a toolpath and a control sequence for cam milling, use the command:



Icon	Ribbon
	CNC → Machining → Cam milling
Keyboard	Textual Menu
	CNC> Cam milling

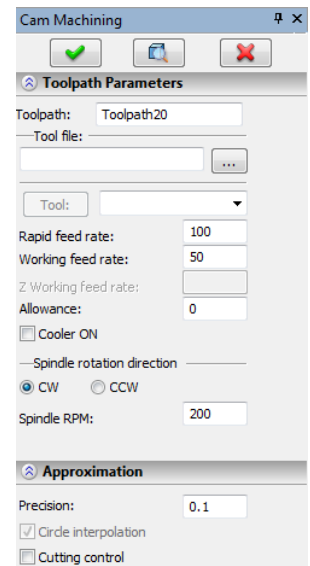
Upon calling the command, the following options become available in the automenu:

	<Y>	Finish input
	<P>	Set entity parameters
	<I>	Select Other Element
	<X>	Exit command
	<I>	Cam profile import
	<K>	Cam machining
	<G>	GOTO toolpath


By clicking the  icon, you get the dialog box with default milling parameters.

As usual, the user can modify those as desired and save the changes by pressing the **[OK]** button. You can safely leave the parameters unchanged at this time, as you can do this later in the workflow.

Once the default parameter modifications are done, select one of the two options:  or .



Cam Profile Import

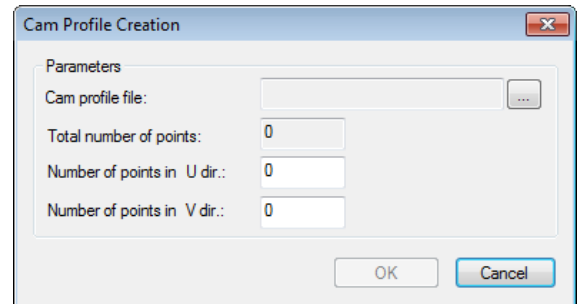
Selecting  brings up the dialog box:

Cam profile file is a text file containing the characteristic points of the surface;


Total number of points parameter is filled automatically upon selecting the cam profile file;

Number of points in U dir. is the number of points along the U-parameter direction of the cam;

Number of points in V dir. is the number of points along the V-parameter direction of the cam.






Note that the [OK] button becomes accessible only when the product $U \times V$ is equal to the total number of points.

Upon closing the dialog box, confirm your input by using the <Y> key or the  button.

After that, the user needs to complete creation of the solid model of the cam based on the generated work surface.

Cam Machining

Press the  button, and then select a solid body. The option for this purpose, , is turned on by default. Upon selecting a solid body, confirm your input by using the <Y> key or the  button. The dialog box will then appear on the screen.

Tool name. The user can specify a particular name of the tool used in the given machining station, from the list provided in the tool file;

Part/Preform (Stock bar) Allowance. Defines the allowance amount – the distance between the original and the offset bodies (or between the original and offsets surfaces);

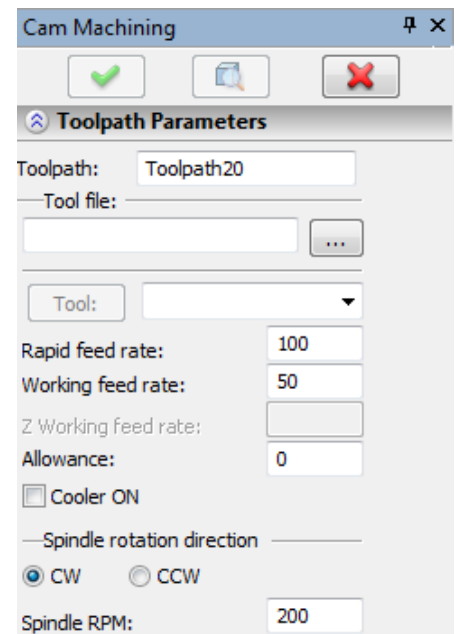
Number of passes Defines the number of passes by the tool;

Tool lift. Defines the distance to the safety plane;

Pass type. Available is “loop-removal” pass type.

Stock removal parameters. The user defines a **vector** that defines the start angle of the toolpath.

Pullout from the end face - defines the distance from the borders of the part to nearest passes;



5-axis transformation allows calculating the toolpath in the part coordinates or in the machining station coordinates with compensation for the tool length. If the 5-axis transformation is turned off, you will have to define the tool length manually.

Tool file. The user selects the necessary file with tool data. The file must be prepared in advance, using "Tool editor";

Rapid feed rate is defined by a specific numerical value in the units supported by a particular CNC center (units are not specified);

Working feed rate is defined by a specific numerical value in the units supported by a particular CNC center (units are not specified);

Spindle RPM – number of spindle rotation per unit of time;



Spindle rotation direction - clockwise or counterclockwise;

Allowance for machining. Specify the allowance amount to add to the part allowance and the stock bar allowance;


Outside surfaces. This implies accounting for the external contour of the surface being machined;

Cooler On Inserts commands in the control sequence for turning On/Off the cooling system of the machining station;


Approximation precision is defined by a specific numerical value in the units supported by a particular CNC center (units are not specified);

After that, the candidate toolpath will be calculated and displayed on the screen. To add the created toolpaths to the document, click the  icon in the automenu. If the wrong type of machining was selected from the beginning, you can cancel all changes and exit by clicking the  icon.

Once all necessary changes are done to the machining parameters, the user just needs to click the **[OK]** button to save the toolpath in a file with the prepared drawing of the part being machined.

The calculated toolpath will be added to the list of available toolpaths in the Machining Manager (the  button).

TECHNOLOGICAL TOOLPATHS

The system now allows creating technological toolpaths for flat types of machining. For this purpose, the  option is provided in the automenu of wire-cutting, laser-cutting, turning, drilling and 2.5D milling machining types. Such toolpaths are necessary, for example, for specifying additional commands in the control sequence for servicing technological processes or for other purposes.

DRILLING, MILLING AND ENGRAVING

The following parameters can be defined for technological toolpaths of drilling and 2.5D milling:

Toolpath parameters.

Toolpath. Name of a toolpath.

Working plane. Enable uploading, to the control sequence, of the working plane with the selection of a specific one (XY, XZ, YZ).

Working system of coordinates. Enable uploading, to the control sequence, of the working system of coordinates with the selection of a specific one (G54-G58).

Coordinate system. **Absolute** or **Relative**.

New command. The user can define any command or macro which are uploaded to the control sequence.

Stop. Selection of the technological stop.

Feed rate. Possibility of specifying the feed rate with a numerical value.

Pause. Possibility of specifying the pause with a numerical value.

Counter contour. Change the direction of the motion to the opposite one.

Technological parameters.

Work parameters of a **spindle**:

On/off/off with fixation of the spindle.

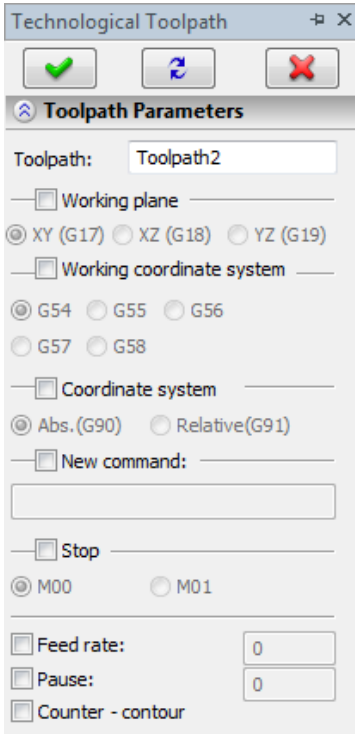
Fixation number. The number of the position of the spindle in which the stop of the spindle is required.

Spindle RPM.

Direction of spindle rotation: clockwise or counterclockwise.

Cooling. Optional activation or deactivation of the cooling.

Safety plane. It is used for external postprocessors.



Technological Parameters

☐ Spindle

☒ On ☐ Off ☐ Off with fix.

Number of fixation:

Spindle RPM:

☐ Rotation direction

☒ CW ☐ CCW

☐ Cooler

☒ On ☐ Off

☐ Safety plain:

☐ Disable

☐ Movement

☒ G00 ☐ G01 Feed rate:

☐ Height

☒ Before ☐ After Value:

Movement. The user indicates the **path**, defines **command type** and **feed rate**, based on which the movement takes place. In addition, it is required to indicate at what **height** the movement takes place (option «**Before**») or first make a movement, and then indicate at what **height** it is required to descend (option «**After**»);

TURNING

Toolpath parameters.

Trajectory. Name of a toolpath.

Working plane. Enable uploading, to the control sequence, of the working plane with the selection of a specific one (XY, XZ or YZ).

Working system of coordinates. Enable uploading, to the control sequence, of the working system of coordinates with the selection of a specific one (G54-G58).

Coordinate system. **Absolute** or **Relative**.

New command. The user can define any command or macro which are uploaded to the control sequence.

Stop. Selection of the technological stop.

Feed rate. Possibility of specifying the feed rate with a numerical value.

Pause. Possibility of specifying the pause as a numerical value.

Counter contour. Change the direction of the motion to the opposite one.

Technological parameters.

Work parameters of a **spindle**:

On/off/off with fixation of the spindle.

Fixation number. The number of the position of the spindle in which the stop of the spindle is required.

Technological Toolpath

☒ ☐ ☐

Toolpath Parameters

Toolpath:

☐ Working plane

☒ XY (G17) ☐ XZ (G18) ☐ YZ (G19)

☐ Working coordinate system

☒ G54 ☐ G55 ☐ G56

☐ G57 ☐ G58

☐ Coordinate system

☒ Abs.(G90) ☐ Relative(G91)

☐ New command:

☐ Stop

☒ M00 ☐ M01

☐ Feed rate:

☐ Pause:

☐ Counter - contour

Technological Parameters

☐ Spindle

☒ On ☐ Off ☐ Off with fix.

Number of fixation:

Spindle RPM:

☐ Rotation direction

☒ CW ☐ CCW

☐ Cooler

☒ On ☐ Off

☐ Safety plain:

☐ Disable

☐ Movement

☒ G00 ☐ G01 Feed rate:

☐ Height

☒ Before ☐ After Value:

Spindle RPM.

Direction of spindle rotation: clockwise or counterclockwise.

Cooling. Optional activation or deactivation of the cooling.

Safety plane. It is used for external postprocessors.

Movement. The user specifies the **path**, defines **command type** and **feed rate**, based on which the movement will take place. In addition, it is required to indicate at what **height** the movement will take place (option «**Before**») or first make a movement, and then indicate at what **height** it is required to descend (option «**After**»);

Selecting quadrant.

Enable in postprocessor. It is used for external postprocessors.

Cutter Position

☐ Enable in postprocessor

☐ Cutter position

5 6 7

4 9 8

3 2 1

☐ Sweep in X dir.:

☐ Sweep in Z dir.:

Cutter's position. The user specifies the location of the cutter upon machining. This parameter is used for compatibility with previous versions of T-FLEX CAM; starting with T-FLEX CAM 10 this parameter is implemented inside the tool's editor.

X-offset (sweep) – the magnitude of the tool's offset along the X-axis.

Z-offset (sweep) – the magnitude of the tool's offset along the Z-axis.

WIRE CUTTING AND LASER CUTTING

Wire cutting and laser cutting require a different set of parameters than mechanical machining. Some of the parameters are the same as those in turning, drilling and 2.5D milling. A number of parameters are specific to wire cutting and laser cutting.

Wire Cutting

The user needs to specify the following parameters for wire cutting:

Toolpath parameters.

Toolpath. Name of a toolpath.

Working plane. Enable uploading, to the control sequence, of the working plane with the selection of a specific one (XY, XZ or YZ).

Working system of coordinates. Enable uploading, to the control sequence, of the working system of coordinates with the selection of a specific one (G54-G58).

Coordinate system. **Absolute** or **Relative**.

New command. The user can define any command or macro which are uploaded to the control sequence.

Stop. Selection of the technological stop.

Feed rate. Possibility of specifying the feed rate as a numerical value.

Pause. Possibility of specifying the pause as a numerical value.

Counter contour. Change direction of the motion to the opposite one.

Wire-cut parameters.

Dielectric system. Turning on/off the system and the system's parameter.

Generator. Turning on/off the generator and the generator's parameter.

Winding of wire. Turning on/off the winding of the wire and the wire's parameter.

Compound shaping. Turning on/off the mode of complex shape forming.

Technological Toolpath

✓ ↺ ✗

Toolpath Parameters

Toolpath: Toolpath1

☐ Working plane

☒ XY (G17) ☐ XZ (G18) ☐ YZ (G19)

☐ Working coordinate system

☒ G54 ☐ G55 ☐ G56

☐ G57 ☐ G58

☐ Coordinate system

☒ Abs.(G90) ☐ Relative(G91)

☐ New command:

☐ Stop

☒ M00 ☐ M01

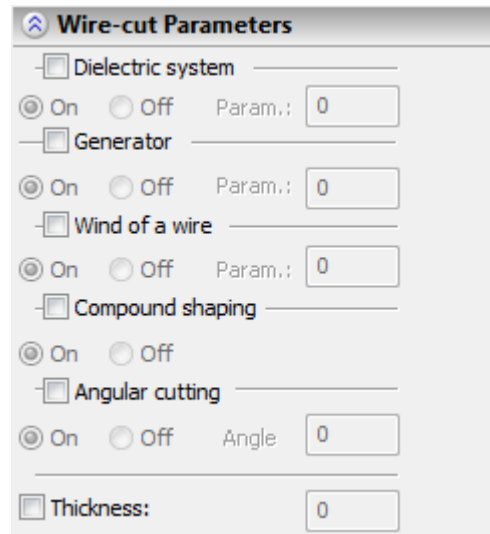
☐ Feed rate: 0

☐ Pause: 0

☐ Counter - contour

Angular cutting. Turning on/off the mode of angular cutting with the possibility of specification of the angle of inclination of the wire.

Part's thickness. Possibility of specifying the thickness of the part with the uploading of the corresponding command to the control sequence.



Laser Cutting

In the case of laser cutting, the user specifies the following set of parameters:

Toolpath parameters.

Toolpath. Name of a toolpath.

Working plane. Enable uploading, to the control sequence, of the working plane with the selection of a specific one (XY, XZ or YZ).

Working system of coordinates. Enable uploading, to the control sequence, of the working system of coordinates with the selection of a specific one (G54-G58).

Coordinate system. **Absolute** or **Relative**.

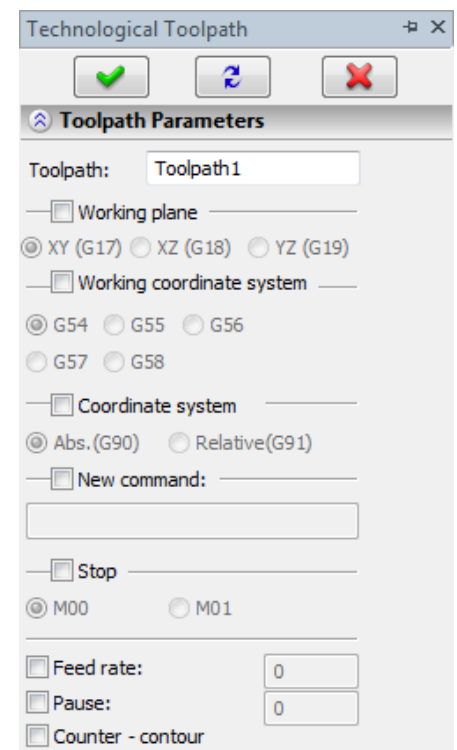
New command. The user can define any command or macro which are uploaded to the control sequence.

Stop. Selection of the technological stop.

Feed rate. Possibility of specifying the feed rate as a numerical value.

Pause. Possibility of specifying the pause as a numerical value.

Counter contour. Change direction of the motion to the opposite one.



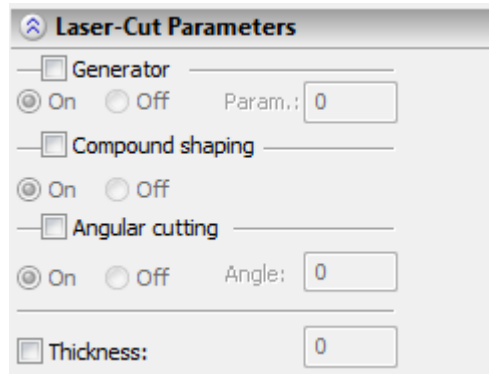
Laser-cut parameters.

Generator. Turning on/off the generator and the generator's parameter.

Compound shaping. Turning on/off the mode of complex shape forming.

Angular cutting. Turning on/off the mode of angular cutting with the possibility of specification of the angle of inclination of the wire.

Part's thickness. Possibility of specifying the thickness of the part with the uploading of the corresponding command to the control sequence.



PUNCHING

For punching we specify the following parameters of technological toolpath:

Toolpath parameters.

Toolpath. Name of a toolpath.

Working plane. Enable uploading, to the control sequence, of the working plane with the selection of a specific one (XY, XZ or YZ).

Working system of coordinates. Enable uploading, to the control sequence, of the working system of coordinates with the selection of a specific one (G54-G58).

Coordinate system. **Absolute** or **Relative**.

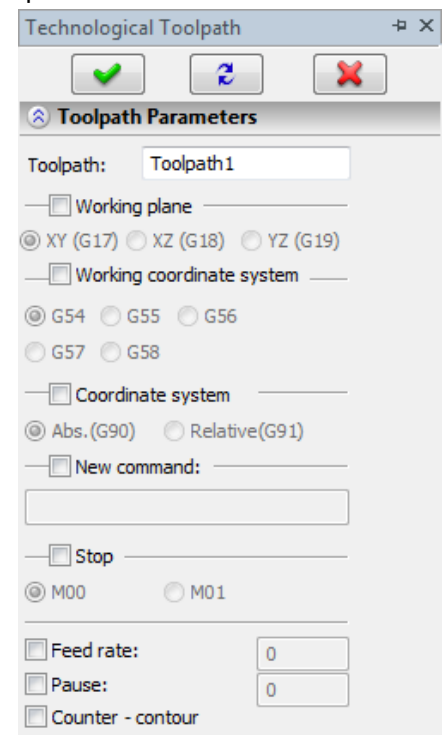
New command. The user can define any command or macro which are uploaded to the control sequence.

Stop. Selection of the technological stop.

Feed rate. Possibility of specifying the feed rate as a numerical value.

Pause. Possibility of specifying the pause as a numerical value.

Counter contour. Change direction of the motion to the opposite one.



Technological parameters.

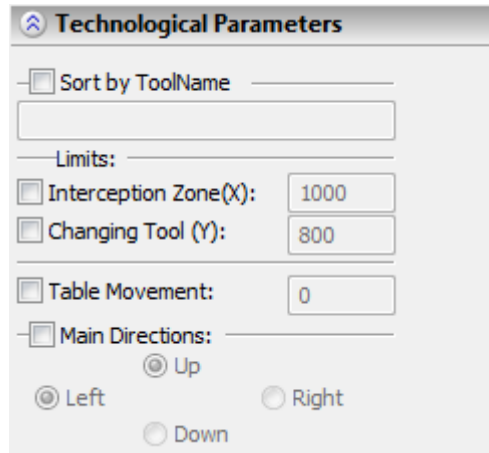
Tool's priority. Enter the sequence of the numbers of the tools according to which the toolpaths will be sorted during the postprocessing.

Interception zone (X). Horizontal dimension that limits the working zone of punching.

Change of tool (Y). Vertical dimension that limits the working zone of punching.

Table's shift. Adding, to the end of the control sequence, the shift (move) of the table by a prescribed value.

Machining direction. Selection of the main directions of machining of the toolpaths that determine the sequence of strikes of the punch.



The image shows a software dialog box titled "Technological Parameters". It contains several settings for toolpath processing:


- A checkbox labeled "Sort by ToolName" is unchecked, followed by an empty text input field.
- A section header "Limits:" is followed by two checked checkboxes with associated input fields:
 - "Interception Zone(X):" with a value of "1000".
 - "Changing Tool (Y):" with a value of "800".
- A checked checkbox "Table Movement:" is followed by an input field with the value "0".
- A checked checkbox "Main Directions:" is followed by four radio button options: "Up", "Left", "Right", and "Down". The "Left" option is currently selected.

MACHINING CYCLES

To maximize the built-in capabilities of a CNC center, the user is provided a number of standard machining cycles supported by most of the modern CNC centers. The base installation of T-FLEX CAM includes the cycles tuned up for certain brands of CNC centers. The machining cycles can also be customized by us to the customer choice of equipment.

MACHINING CYCLES EDITOR

In order for the user to create the machining cycles that are not implemented in T-FLEX CAM system, the machining cycle editor can be used. With the help of this editor, it is possible to create the machining cycles of various complexities including multi-entry cycles for any CNC center.

Icon	Ribbon
	CNC → Control → Machine Cycles Editor
Keyboard	Textual Menu
	CNC>Machine Cycles Editor

After calling the command, the working window of the machining cycles editor will appear. The editor allows us to create the user's machining cycles, save them into a file, open from a file and edit them.

The dialog window of the machining cycle editor consists of three tabs – “Primary properties”, “Comments” and “Additional properties”.

On the “Main properties” tab the user can specify the name of the cycle being created or edited, its type and parameters. Parameters of the cycle can be divided into three groups: announcement, definition and termination.

A machining cycle can consist only of definition, i.e., specification of the announcement and termination is optional. If the announcement (or termination) is not required, the input field for the G-command should be left empty.

In all three groups, the user can specify the name of the G-command, the notation for the X and Y coordinates and indicate the letter symbols for the parameters of the G-command. The numerical values for these parameters can be specified later, during creation of the toolpath on the basis of the user's machining cycles.

In the "Definition" group it is possible to enable the "Repeated cycles" option and specify the letter symbols for the first and the last frames of the repeated (multi-entry) machining cycle. The values themselves will automatically be calculated in the control sequence.

To add a cycle to the list of cycles, press the **[Add to list]** button.

When editing a cycle from the list, the **[Add to list]** button will be changed to **[Apply]**. This button should be pressed to save modified data of the cycle.

In the lower part of the tab, the list of created user's cycles with indication of their names and types of machining will be shown.

Parameters of G-commands must be separated with a "space", for example "U W F S T".

To move along the list of the user's cycles, use  or the keys *Up* and *Down*.

To change parameters of the cycle, it is required to select it in the list and click   or **[Enter]**.

By pressing the **[Open...]** button the user can open the file with the saved user's cycles for their subsequent editing. When pressing this button, the «Guide» window of Windows will appear in which it is required to indicate the file to be opened.

With the help of the **[Save]** or **[Save as...]** buttons the user can save the created list of cycles into a file.

It is possible to create the copy of already existing cycle by selecting it in the list and pressing the **[Copy]** button.

Removal of the user's cycle is carried out by selecting it in the list and pressing the **[Delete]** button.

To clear the entire list of the user's machining cycles, press the **[Delete all]** button.

With the help of the **[Save]** or **[Save as...]** buttons, the user can save the created list of cycles into a file. The file will be saved in a binary format with an extension ".cyl". By pressing the **[Save...]** button, the user can open the file with the saved user's cycles for their subsequent editing.

By pressing the **[Add...]** button, the user can add the cycles, saved in another file, to the list of cycles.

Machine cycles editor

Primary properties | **Comments** | Additional properties

Announcement

Par.	Comment
U1	Cut Depth
R1	Distance

Definition

Par.	Comment
U	Limit on X
W	Limit on Y3
F	Feedrate
S	Spindel RPM
T	Tool

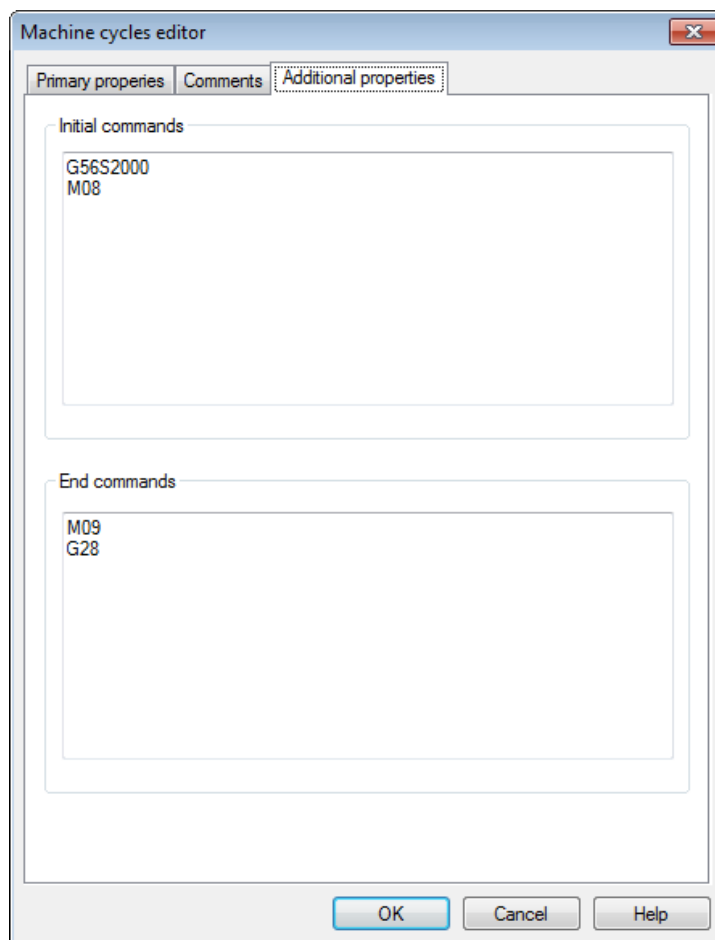
End

Par.	Comment

OK Cancel Help

On the "Comments" tab the user can specify comments for parameters of G-commands if it is necessary. These comments will appear on the screen next to the corresponding parameters when creation of the toolpath is carried out with the user's machining cycle.


To create comments, it is required to click on the "Comments" field opposite the name of the required parameter and enter its specification.

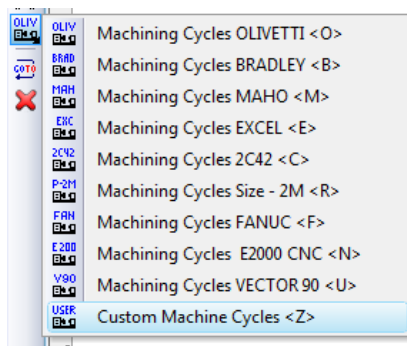


On the "Additional properties" tab it is possible to specify initial and terminal G-commands. The initial G-commands will be uploaded before the cycle announcement, the terminal ones – after the end of a cycle. The initial and terminal commands may consist of several lines.

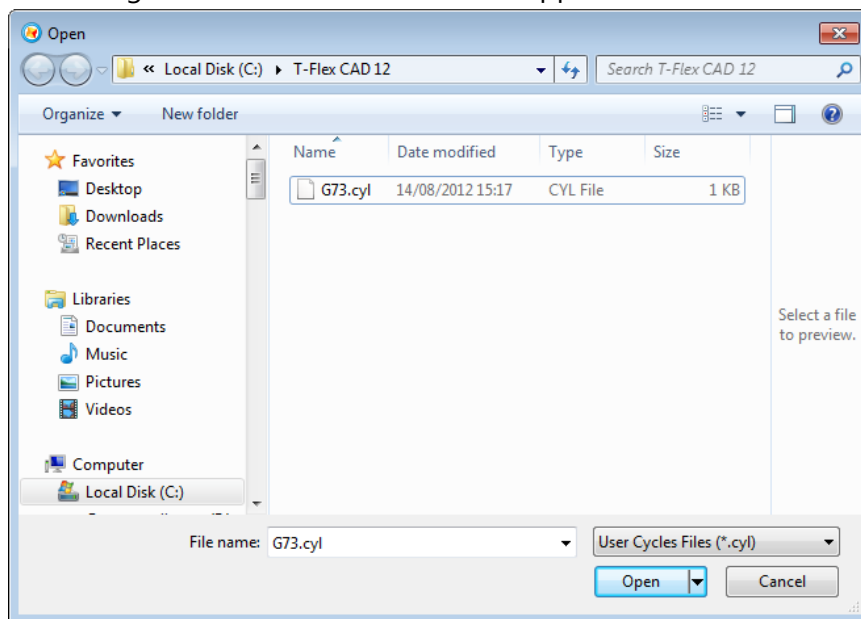
By using the [**C**lose] button the window of the user's machining cycles editor will be closed.

USING CUSTOM MACHINING CYCLES

To use the machining cycles that are created with the help of the editor, select the  option in the automenu of the corresponding type of machining (turning, drilling, milling)..










After that the standard dialog window for file selection will appear:



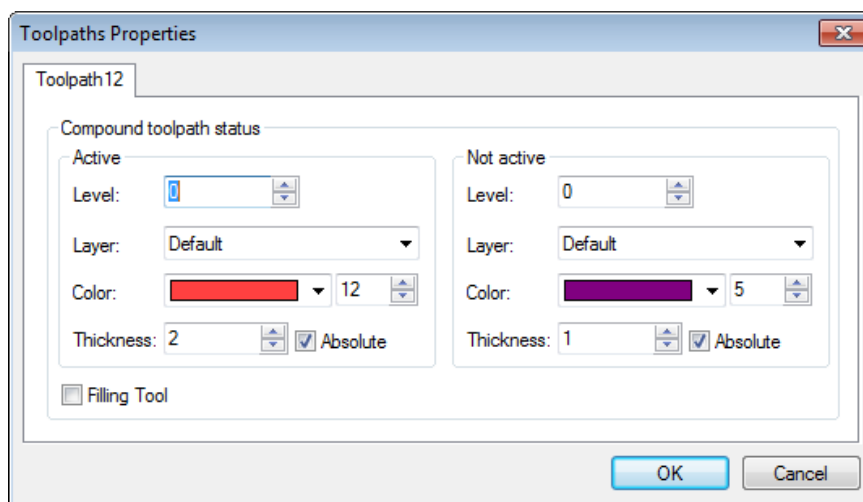
In the window that appears it is necessary to select the file with the saved user's cycles. After that, on the screen the dialog window of the toolpath parameters will appear, which the user can change as desired, and the system will transition to the contour specification mode.

At the same time the following options will become accessible in the automenu:

	<End>	Finish editing
	<P>	Toolpath properties
	<2>	Path selection filter
	</>	Select a point for definition of the cycle
	<[>	Select a point for announcement of the cycle
	<]>	Select a point for termination of the cycle

	<Esc>	Exit the command
---	-------	------------------

In the **Toolpath properties** window, the user can specify the properties for two groups of elements – active and nonactive toolpaths. Activity of toolpaths is described in more detail in the Machining manager section. It is possible to customize the thickness of displayed elements of the toolpath, their color, layer and level of location of the toolpath on the drawing.




A toolpath with a custom cycle can be created in several ways:

1. **Machining cycle without coordinates output.** In this case, the user initializes the parameters that were defined at the time of creating the cycle in the machining cycles editor, but doesn't select any construction in the drawing. The NC output format is as follows:


<G-command><Parameters>

Please note that a G-command will be written in this format, if at the time of its creation no definition was given to the symbolic IDs for the coordinates.

2. **Machining cycle with node coordinates output.** To create a toolpath with this kind of cycle, the user shall activate the  option (the cycle definition point) and select a node in the drawing. The NC output format is as follows:

<G-command><X><Y><Parameters>

When creating a cycle in the machining cycles editor, you shall define the symbolic IDs for the coordinates of the cycle definition.

3. **Machining cycle with toolpath node coordinates output.** To create a toolpath with this kind of cycle, the user shall activate the  option and select a path in the drawing.



The NC output format is as follows:

<G-command><X1><Y1><Parameters>

...

<G-command><Xn><Yn>

When creating a cycle in the machining cycles editor, you shall define the symbolic IDs for the coordinates of the cycle definition.

4. **Machining cycle with node coordinates output and linear moves between the path nodes.** To create a toolpath with this kind of cycle, the user shall activate the  option (the cycle definition point) and select a node in the drawing. This node's coordinates will be written to the NC with the appropriate parameters of the cycle definition. Next, you need to activate the  option and select a path in the drawing.

The NC output format is as follows:

<G-command><X><Y><Parameters>


<G01><X1><Y1>

...

<G01><Xn><Yn>

Creation of this kind of cycle is possible for turning and milling. For the drilling, instead of the G01 command, the output will be the G-command defined in the cycle definition.

When creating a cycle in the machining cycles editor, you shall define the symbolic IDs for the coordinates of the cycle definition.

5. **Repetitive machining cycle.** In this case you need to activate the  option and select a path in the drawing.

The NC output format is as follows:

<G-command><Pi><Qj><Parameters>



<Ni> <G01><X1><Y1>

...

< Nj> <G01><Xn><Yn>

Creation of this kind of cycle is possible for turning and milling. For the drilling, instead of the G01 command, the output will be the G-command defined in the cycle definition.

When creating a cycle in the machining cycles editor, you shall define the symbolic IDs for the first and last frames of the NC. The numerical values of those parameters will be calculated automatically based on the path geometry.

6. **Repetitive machining cycle with an approach node.** In this case, the user needs to activate the  option, select a path in the drawing, and then, using the  option, select a node in the drawing, whose coordinates will be outputted into the NC as the approach point.

The NC output format is as follows:

<G-command><Pi><Qj><Parameters>



```

< Ni> <G00><X1><Y1>
< N(i+1)> <G01><X2><Y2>
...
< Nj> <G01><Xn><Yn>


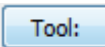
```

Creation of this kind of cycle is possible for turning and milling. For the drilling, instead of the G01 command, the output will be the G-command defined in the cycle definition.

When creating a cycle in the machining cycles editor, you shall define the symbolic IDs for the first and last frames of the NC. The numerical values of those parameters will be calculated automatically based on the path geometry.

For any of the cycles mentioned above, except for the cycle without coordinates output, one can additionally specify the nodes whose coordinates will be written into the NC in the announcement and in the end of cycle. To achieve this, you need to activate the respective option,  or . When creating a cycle in the machining cycles editor, the symbolic coordinate IDs must be defined for both the announcement and the end of cycle.

Toolpath. Name of a toolpath.

Tool's file. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilized for the chosen type of machining.

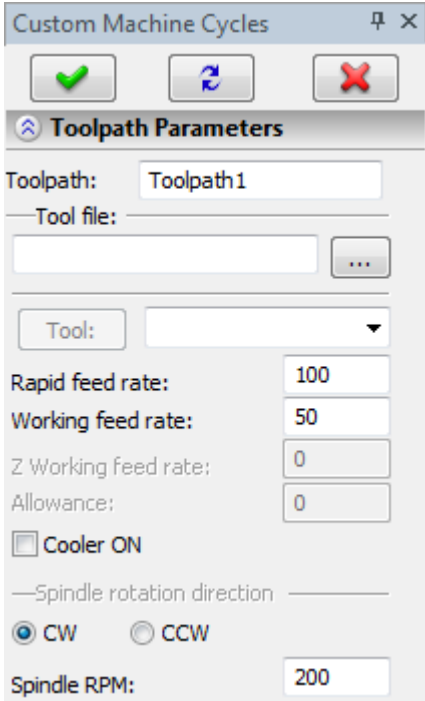
Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary translations.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the tool's translation when executing the motion of drilling.

Cooler on. Turning on/off the supply of the cooling agent upon machining.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.



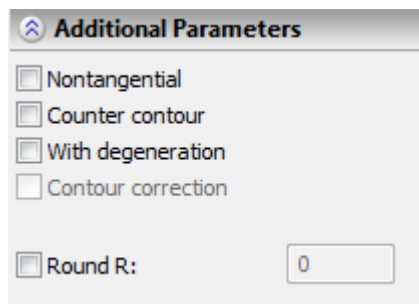
Radius correction.

On. When this flag is enabled, parameters of the radius correction can be used. Correction can be **left** and **right**. It is possible to specify correction by the **number** (in this case, upon machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number indicated by the user) and by the **value** (in this case, the user specifies the precise value of the radius of curvature of the cutting edges of the mill).

R – use the radius of the selected tool.

Additional parameters.

Precise tracing of angles. The set-up of this parameter specifies deletion of the tangential transitions (joining elements of the contour tangentially) in the toolpath that is created by means of an equidistant offset from the original contour (when using the allowance or correction on the value of the tool's radius).



Counter contour. Changes the direction of the tool's motion.

Without deletion of micro joints. This parameter enables restoration of the degenerated elements of the toolpath that is created by means of an equidistant offset from the original contour (when using the allowance or correction on the value of the tool's radius).

R of rounding. Execute rounding. This parameter allows us to inscribe circles of the specified radii in the places of non-tangential transitions for internal and external angles of the contour. This parameter will be described in more detail below.

Cycle parameters.

Type. Type of a cycle (turning/drilling/milling).

Name/number. Name of a cycle.

Dynamically created windows for inputting the values of parameters of the cycle with their description are located next (if they were specified upon creation of the cycle).

Cycle Parameters

Type - Turning

Name/Number of cycle

Rough machining cycle G73

Announcement

U1(Cut Depth) 0

R1(Distance) 0

Definition

U(Limit on X) 0

W(Limit on Y3) 0

F(Feedrate) 0

S(Spindel RPM) 0

T(Tool) 0


Approximation.

Precision of the approximation is specified as a numerical value in units supported by the CNC center and constitutes the magnitude of the caving (maximum distance from the arc of the circle to be straightened to the approximating line).

Circular interpolation. In case when the contour being machined has circles, arcs, splines, and other geometrical elements having similar shape, the selection of this parameter will allow us to obtain the control sequence that contains frames of the circular interpolation.

PREDEFINED MACHINING CYCLES


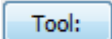
“OLIVETTI” Machining Station with CNC Center “OLIVETTI 8600”

This type of equipment is supplied with drilling and deep drilling cycles. To define parameters of these cycles, press the  icon in drilling automenu. The automenu of machining cycles OLIVETTI will appear.

To create a toolpath, it is required to indicate the path, constructed in advance, at the nodes of which the drilling operation will be carried out. When constructing a path it should be taken into account that at the start node the drilling will not be executed, and the segment of the path between the first and the second nodes will be uploaded to the control sequence as an additional translation on the rapid feed rate. Therefore, if the path consists of $n+1$ nodes, the drilling will be executed n times, starting with the second node.

Toolpath parameters.

Toolpath. Name of a toolpath.

Tool's file. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilized for the chosen type of machining (in this case, only the drills that are contained in the tool's file).

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary translations.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the tool's translation when executing the motion of drilling.

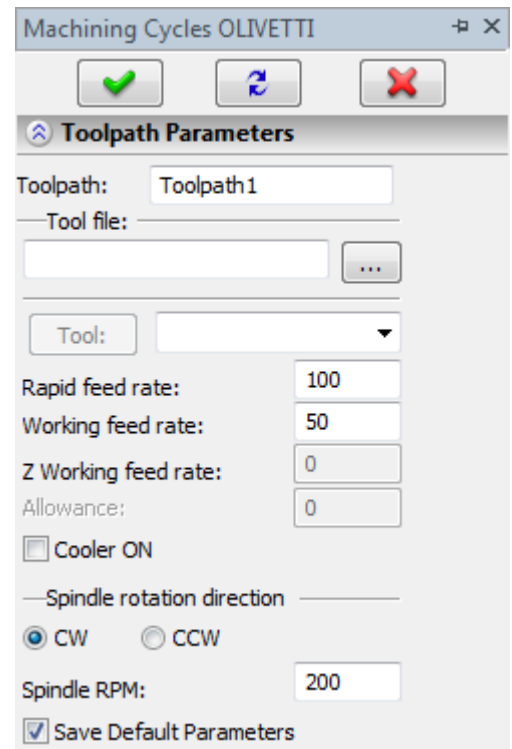
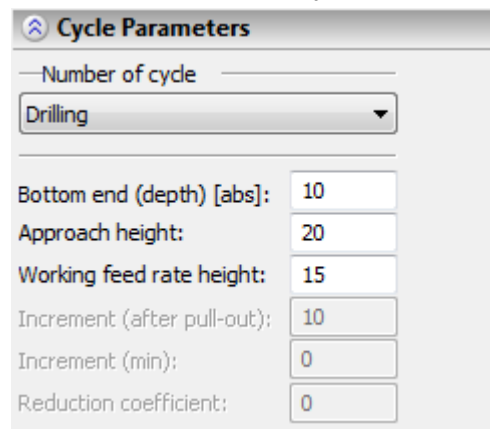
Cooler on. Turning on/off the supply of the cooling agent upon machining.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.

Cycle parameters.

Cycle number. Selection of the specific number of the cycle.

Depth of drilling. The maximum height at which the drill descends upon drilling.

Approach height. On this height the tool approaches the site of drilling.

Pullout height. On this height the tool is withdrawn from the site of drilling. If the pullout height is not specified, it is set equal to the approach height.

Increment. The value of the drilling step.

Increment (min). The magnitude of the minimum step of drilling (does not have to be specified).

Reduction coefficient. Numerical factor used for calculation of the magnitude of the subsequent steps of drilling as compared to the previous ones.


To get the length of the next drilling pass, multiply the current pass length by the reduction coefficient. Upon reaching the minimum drilling pass length, all further passes have this fixed length. This entry can be skipped.

When using the described machining cycles, the following special commands will be inserted in the control sequence:

- Drilling cycle G81 R[approach height] R[retreat height] Z[Final hole depth];
- Deep drilling cycle G83 R[approach height] R[retreat height] Z[Final hole depth] I[increment] J[increment (min)] K[reduction].

To specify the position of the hole, a line with the hole center coordinates is written in the control sequence after the row that defines the cycle. (Only the coordinates are written in this line, no command name). In this case, replace the command for drilling cycles G79 by a blank in the postprocessor. The drilling cycles cancellation command is G80.


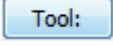
Milling Machining Station “ALLEN BRADLEY” with CNC Center “ALLEN BRADLEY 9 SERIES”

This type of equipment is supplied with drilling, deep drilling and reaming cycles. To define parameters of these cycles, press the  icon in the drilling automenu. The automenu of machining cycles BRADLEY will appear.

To create a toolpath, it is required to indicate the path, constructed in advance, at the nodes of which the drilling operation will be carried out. When constructing a path it should be taken into account that at the start node the drilling will not be executed, and the segment of the path between the first and the second nodes will be uploaded to the control sequence as an additional translation on the rapid feed rate. Therefore, if the path consists of $n+1$ nodes, the drilling will be executed n times, starting with the second node.

Toolpath parameters.

Toolpath. Name of a toolpath.

Tool's file. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilized for the chosen type of machining (in this case, only the drills that are contained in the tool's file).

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary translations.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the tool's translation when executing the motion of drilling.

Cooler on. Turning on/off the supply of the cooling agent upon machining.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.

Cycle parameters.

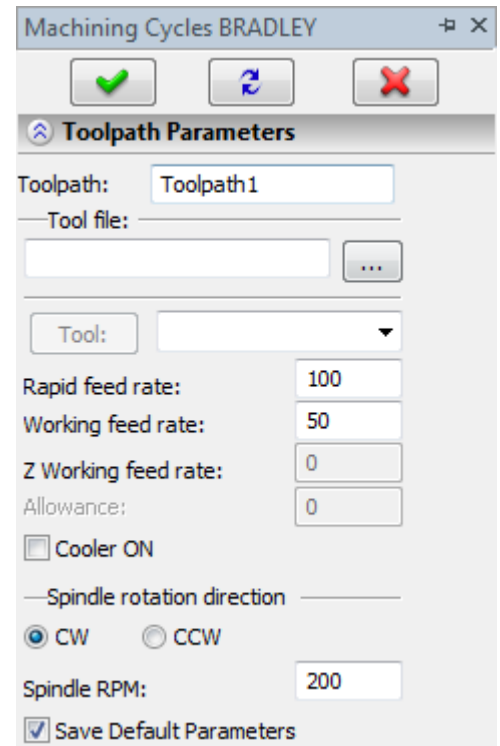
Cycle number. Selection of the specific number of the cycle.

Depth of drilling. The maximum height at which the drill descends upon drilling.

Height. On this height the tool approaches the site of drilling.


Dwell time. The time of the pause between the drillings in units supported by the CNC center.


Increment. The magnitude of the drilling step.



Machining Cycles BRADLEY

Toolpath: Toolpath1

Tool file: 

Tool: 

Rapid feed rate: 100

Working feed rate: 50

Z Working feed rate: 0

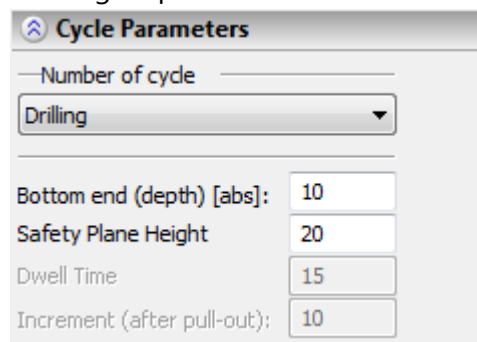
Allowance: 0

☐ Cooler ON

Spindle rotation direction: ☒ CW ☐ CCW

Spindle RPM: 200

☒ Save Default Parameters



Cycle Parameters

Number of cycle: Drilling

Bottom end (depth) [abs]: 10

Safety Plane Height: 20

Dwell Time: 15


Increment (after pull-out): 10

The following special commands will be inserted in the control sequence when using the above described machining cycles:

- Drilling cycle G81 X Y[hole position] Z[Final hole depth] R[Safety clearance height];
- Deep drilling cycle G83 X Y[hole position] Z[Final hole depth] R[Safety clearance height] Q[increment];
- Reaming cycle G86 X Y[hole position] Z[Final hole depth] R[Safety clearance height] P[dwell time].

Drilling and reaming cycles are canceled by the command G80.


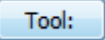
Machining Station “MAHO” with CNC Center “MAHO CNC 432”

This type of equipment is supplied with drilling, deep drilling and reaming cycles. To define parameters of these cycles, press the  icon in the drilling automenu. The automenu of machining cycles MAHO will appear.

To create a toolpath, it is required to indicate the path, constructed in advance, at the nodes of which the drilling operation will be carried out. When constructing a path it should be taken into account that at the start node the drilling will not be executed, and the segment of the path between the first and the second nodes will be uploaded to the control sequence as an additional translation on the rapid feed rate. Therefore, if the path consists of $n+1$ nodes, the drilling will be executed n times, starting with the second node.

Toolpath parameters.

Toolpath. Name of a toolpath.

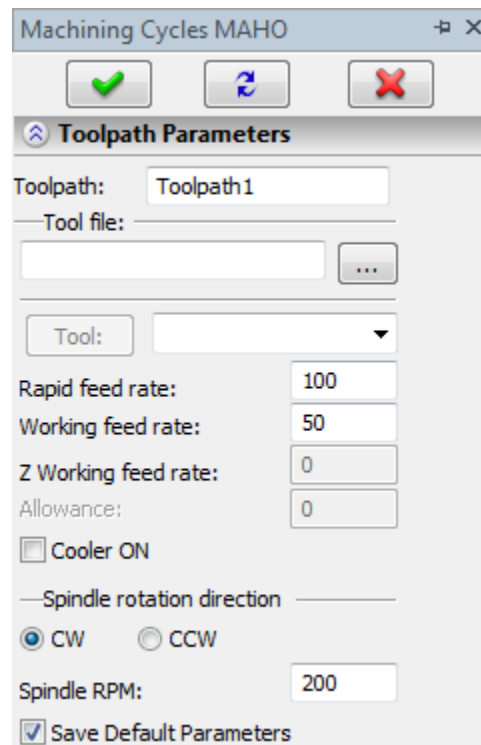
Tool's file. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilized for the chosen type of machining (in this case, only the drills that are contained in the tool's file).

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary translations.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the tool's translation when executing the motion of drilling.

Cooler on. Turning on/off the supply of the cooling agent upon machining.

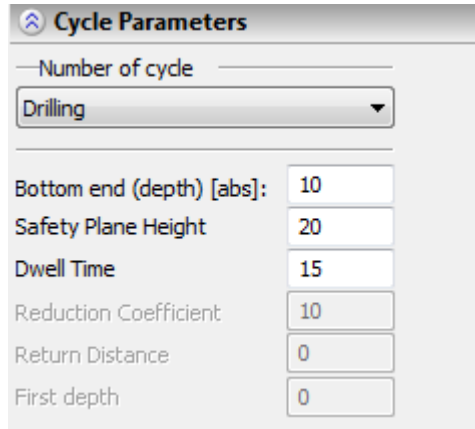


Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.

Cycle Parameters.

Cycle number. Selection of the specific number of the cycle.



Cycle Parameters	
Number of cycle	
Drilling	
Bottom end (depth) [abs]:	10
Safety Plane Height	20
Dwell Time	15
Reduction Coefficient	10
Return Distance	0
First depth	0

Depth of drilling. The maximum height at which the drill descends upon drilling.

Height. On this height the tool approaches the site of drilling.

Dwell time. The time of the pause between the drillings in units supported by the CNC center.

Reduction coefficient. Numerical factor used for calculation of the magnitude of the subsequent steps of drilling as compared to the previous ones.

Return (retreat) distance. The lift of the tool after each step of drilling (cycle of deep drilling).

First depth. The depth of the first drilling step (cycle of deep drilling).

When using the described machining cycles, the following special commands will be inserted in the control sequence:


- Drilling cycle G81 X [dwell time] Y[Safety clearance height] Z[Final hole depth];
- Deep drilling cycle G83 X [dwell time] Y[Safety clearance height] Z[Final hole depth] J[return distance] K[first depth];
- Reaming cycle G86 X [dwell time] Y[Safety clearance height] Z[reaming depth].

To specify the position of the hole, a line with the hole center coordinates is written in the control sequence after the line that defines the cycle (G79 X... Y...).

Turning Machining Station "EXCEL SL" with CNC Center "FANUC 0/00/0M/6/6M/16"

This type of equipment is supplied with drilling and turning cycles.

Drilling cycles

Drilling cycles include: endface drilling, diameter drilling, front drilling, front threading, front boring, endface drilling, endface threading, endface boring. To define parameters for these cycles, press the  icon in the drilling automenu. The automenu of machining cycles EXCEL will appear.

To create the toolpath, it is required to specify the path constructed in advance. For endface operations of drilling, boring and threading the path must be constructed parallel to the Z-axis of the machining station. For front face operations, the path must be constructed parallel to the X-axis of the machining station.


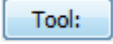
G74	Drilling cycle
G75	Drilling cycle
G83	Drilling cycle
G84	Front threading cycle
G85	Front boring cycle
G87	Endface drilling cycle
G88	Endface threading cycle
G89	Endface boring cycle

Endface drilling cycle

For endface drilling, specify a path parallel to the Z-axis of the machine, with the length corresponding to the Final hole depth. In case of using the cycle of drilling a diameter it is required to indicate the path (the path must be located parallel to the X-axis of the machining station), whose length must correspond to the drilling depth.

Toolpath parameters.

Toolpath. Name of a toolpath.

Tool's file. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilized for the chosen type of machining (in this case, only those drills that are contained in the tool's file).

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary translations.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the tool's translation when executing the motion of drilling.

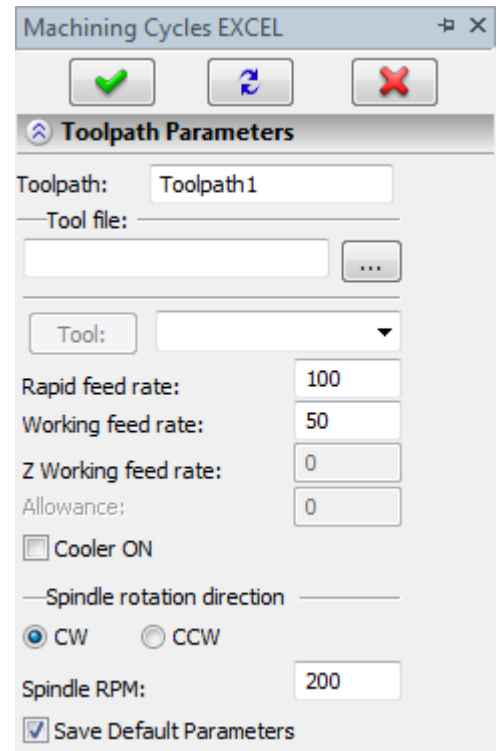
Cooler on. Turning on/off the supply of the cooling agent upon machining.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.

Cycle parameters.

Cycle number. Selection of the specific number of the cycle.



The screenshot shows the 'Machining Cycles EXCEL' dialog box with the 'Toolpath Parameters' tab selected. The dialog has a title bar with a maximize button and a close button. Below the title bar are three buttons: a green checkmark, a blue refresh icon, and a red X. The 'Toolpath Parameters' section includes the following fields and controls:

- Toolpath:** A text field containing 'Toolpath1'.
- Tool file:** A text field with a button containing three dots to the right.
- Tool:** A dropdown menu.
- Rapid feed rate:** A numeric field with the value '100'.
- Working feed rate:** A numeric field with the value '50'.
- Z Working feed rate:** A numeric field with the value '0'.
- Allowance:** A numeric field with the value '0'.
- Cooler ON:** A checkbox that is currently unchecked.
- Spindle rotation direction:** A section with two radio buttons: 'CW' (selected) and 'CCW'.
- Spindle RPM:** A numeric field with the value '200'.
- Save Default Parameters:** A checkbox that is checked.

The screenshot shows a 'Cycle Parameters' dialog box. At the top, there is a dropdown menu labeled 'Number of cycle' with 'G74 Endface drilling' selected. Below this, there are several input fields:

- Return Value: 0
- X Dir Incr. Amount: 0
- Z Dir Incr. Amount from: (empty)
- Delta (i): 0
- Delta(k): 0
- Delta (d): 0

Return distance. The magnitude of the return distance (retreat) is uploaded to the control sequence. **along X from A to B.** The distance from the first point of drilling to the last one along the X-axis is specified;

along Z from A to B. The distance from the first point of drilling to the last one along the Z-axis is specified;

Delta (i) the magnitude of translation in the direction of the X (Z) - axis.

Delta (k) the depth of cutting in the direction of the Z (X) -axis (without sign).

Delta (d) the magnitude of the tool's pullout at the end of cutting.

The format of the record in a control sequence is as follows:

G74 R[return amount]

G74 X[in X direction from A to B] Z[Z-coordinate of the hole center] P[delta (i)] Q[delta (k)] R[delta (d)].

Diameter drilling cycle

For diameter drilling cycle, specify a path parallel to the X-axis of the machine, with the length corresponding to the Final hole depth. After selecting the path, select the option G75, and then specify the machining parameters in the coming up dialog box of the cycle parameters.

Take a special note on correct filling-in of the entries in the diameter drilling cycle:

- **Return value** is a typical value that depends on the next entry and should not be defined before the latter;
- In Z direction from A to B is the distance from the first point of drilling to the last point;
- **Delta (i)** is the motion in the Z direction;
- **Delta (k)** is the cutting depth in the X direction (unsigned);
- **Delta (d)** is the amount of the tool pullout in the end of a cutting pass.

The format of the record in a control sequence is as follows:

G75 R[return amount]

G75 X[X-coordinate of the hole center] Z[in Z direction from A to B] P[delta (i)] Q[delta (k)] R[delta (d)].

Cycles of front/end-face drilling, threading and boring



In case of using the cycle of front drilling it is necessary to specify the path whose length must correspond to the drilling depth (the path must be located parallel to the X-axis of the machining station).

In case of using the cycle of front boring it is required to specify the path whose length must correspond to the depth of boring (the path must be located parallel to the X-axis of the machining station).

In case of using the cycle of end-face threading it is required to specify the path whose length must correspond to the depth of threading (the path must be located parallel to the Z-axis of the machining station).

Toolpath parameters.

Toolpath. Name of a toolpath.

Tool's file. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilized for the chosen type of machining (in this case, only those drills that are contained in the tool's file).

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary translations.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the tool's translation when executing the motion of cutting.

Cooler on. Turning on/off the supply of the cooling agent upon machining.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.

Cycle Parameters.

Initial angle. The magnitude of the turn angle of the spindle for drilling the first hole.

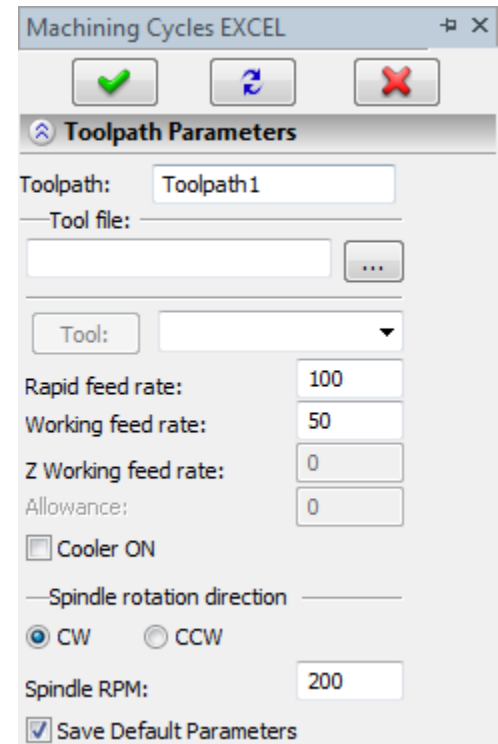
Safety plane height. The distance from the initial level to the level of cutting-in point.

Depth of cutting-in. Step of drilling.

Dwell time. Time of idling at the base of the hole;

Code for C-axis clamp. Specialized parameter of the machining station.

Rotation angle. Angle of the turn between the previous hole and the next one.



Cycle Parameters

Number of cycle

G83 Front drilling

Initial Angle: 0

Safety Plane Height: 0

Depth of Cut:

Dwell Time: 0

Code for C-axis: 0

Rotation Angle: 0

Number of repetitions: 0

Number of repetitions. The number of holes.

The format of the record of front drilling cycle in a control sequence:

G83 X[X-coordinate of hole's center] C[initial angle] Z[Z-coordinate of hole's center] R[safety plane height] Q[depth of cutting-in] P[dwell time] M[code for C-axis clamp].

The format of the record of front threading cycle in a control sequence:

G84 X[X-coordinate of hole's center] C[initial angle] Z[Z-coordinate of hole's center] R[safety plane height] P[dwell time] M[code for C-axis clamp].

The format of the record of front boring cycle in a control sequence:

G85 X[X-coordinate of hole's center] C[initial angle] Z[Z-coordinate of hole's center] R[safety plane height] P[dwell time] M[code for C-axis clamp].

The format of the record of endface drilling cycle in a control sequence:

G87 Z[Z-coordinate of hole's center] C[initial angle] X[X-coordinate of hole's center] R[safety plane height] Q[depth of cutting-in] P[dwell time] M[code for C-axis clamp].

The format of the record of endface threading cycle in a control sequence:

G88 Z[Z-coordinate of hole's center] C[initial angle] X[X-coordinate of hole's center] R[safety plane height] P[dwell time] M[code for C-axis clamp].


The format of the record of endface boring cycle in a control sequence:

G89 Z[Z-coordinate of hole's center] C[initial angle] X[X-coordinate of hole's center] R[safety plane height] P[dwell time] M[code for C-axis clamp].

Before enabling the machining cycles from G83 to G89, use the command M51 to enable the mode of marking the axis C. The command G80 cancels the machining cycles from G83 to G89. Afterwards, use the command M50 to disable the mode of marking the axis C.

Turning Cycles

The cycles of machining by turning include: the cycle of linear (straight) cutting (G90), the cycle of tapered (conic) cutting (G90'), the cycle of threading (G92), the cycle of conic threading (G92'), the cycle of endface turning (G94), the cycle of endface turning of a cone (G94'), the cycle of allowance removal (G71), the cycle of allowance removal of the endface (G72), the cycle of repetitive commands (G73). To


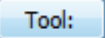
specify parameters of these cycles, it is required to press the  button in the menu of turning machining. Then, it is necessary to indicate the path, constructed in advance, that describes the contour.

G90	Straight (linear) cutting cycle
G90'	Tapered (cone) cutting cycle
G92	Threading cycle
G92'	Conic threading cycle
G94	Endface turning cycle
G94'	Cone endface turning cycle
G71	Allowance removal cycle
G72	Endface allowance removal cycle
G73	Repetitive commands cycle

Cycles of allowance removal and repetitive commands

Toolpath parameters.

Toolpath. Name of a toolpath.

Tool's file. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilized for the chosen type of machining (in this case, only the cutters that are contained in the tool's file).

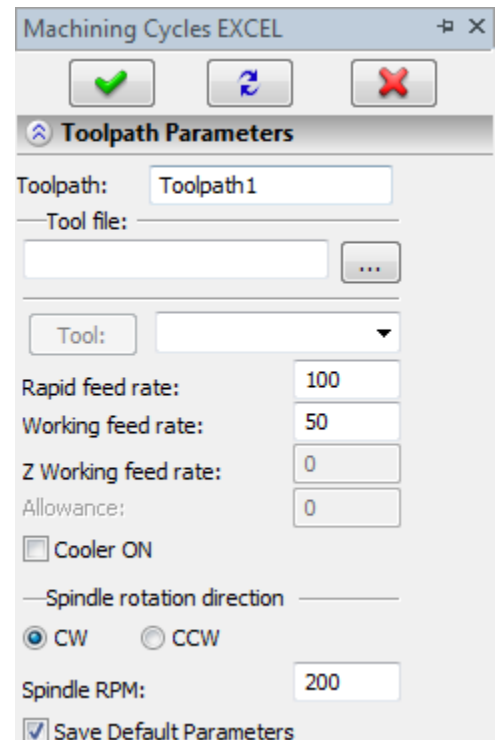
Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary translations.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the tool's translation when executing the motion of cutting.

Cooler on. Turning on/off the supply of the cooling agent upon machining.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.





Machining Cycles EXCEL

✓ ↺ ✗

Toolpath Parameters

Toolpath: Toolpath1

Tool file: 

Tool: 

Rapid feed rate: 100

Working feed rate: 50

Z Working feed rate: 0

Allowance: 0

☐ Cooler ON

— Spindle rotation direction —

☒ CW ☐ CCW

Spindle RPM: 200

☒ Save Default Parameters

Cycle parameters.

Cycle number. Selection of the specific cycle number.

Cutting depth. The magnitude of the material's removal.

Overtravel in X direction. The distance and direction of the relief in the direction of the X-axis.

Overtravel in Z direction. The distance and direction of the relief in the direction of the Z-axis.

Number of divisions. The number of iterations inside the cycle of the repetitive commands.

Allowance in X direction. The distance and direction of allowance on the finishing in the direction of the X-axis.

Allowance in Z direction. The distance and direction of allowance on the finishing in the direction of the Z-axis.

Chips volume. This parameter is uploaded to the control sequence inside the announcement of the cycle.

Cycle of finishing G70. Possibility of adding, to the selected cycle, the additional cycle of finishing G70 with the specified **spindle RPM**, **working feed rate** and **allowance**.

The format of the record of allowance removal cycle in a control sequence:

G71 U[cutting depth] R[chips volume]

G71 P[ns] Q[nf] U[allowance in X] W[allowance in Z], where

ns – the sequence number of the first block for the program of the last machining of the profile;

nf - the sequence number of the last block for the program of the last machining of the profile.

The format of the record of endface allowance removal cycle in a control sequence:

G72 U[cutting depth] R[chips volume]

G72 P[ns] Q[nf] U[allowance in X] W[allowance in Z], where

ns – the sequence number of the first block for the program of the last machining of the profile;

nf - the sequence number of the last block for the program of the last machining of the profile.

The format of the record of cycle of repetitive commands in a control sequence:

G73 U[relief in X] **W**[relief in Z] **R**[number of divisions]

G73 P[ns] **Q**[nf] **U**[allowance in X] **W**[allowance in Z], where

ns – the sequence number of the first block for the program of the last machining of the profile;

nf - the sequence number of the last block for the program of the last machining of the profile.

If the flag of the cycle of finishing **G70** is enabled in the dialog, the rows of the finishing of the given contour will be added to the control sequence.


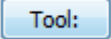
The format of the record of cycle of finishing **G70** in a control sequence:

G70 P[ns] **Q**[nf] **U**[allowance].

Cycles of cutting, finish turning and threading

Toolpath parameters.

Toolpath. Name of a toolpath.

Tool's file. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilized for the chosen type of machining (in this case, only the cutters that are contained in the tool's file).

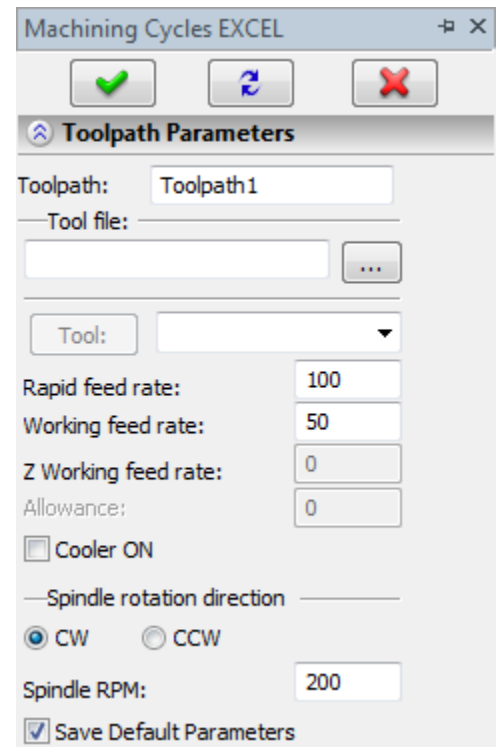
Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary translations.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the tool's translation when executing the motion of cutting.

Cooler on. Turning on/off the supply of the cooling agent upon machining.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.



Cycle parameters.

Conicity. The difference in radii of the endfaces of a truncated cone.

The screenshot shows a software interface titled 'Cycle Parameters'. At the top, there is a dropdown menu set to 'G90' Cone cutting'. Below this, several parameters are listed with corresponding input fields:

- Conic:** 1
- Relief in X direction:** 5
- Relief in Z direction:** 5
- Number of deviation:** 2
- Allowance in X direction:** 0.1
- Allowance in Y direction:** 0.1
- Chips volume:** 1
- ☐ **Finishing cycle G70**
- Spindle RPM:** 50
- Working feed rate:** 50
- Tolerance:** 0.1

The format of the record of linear cutting cycle in a control sequence:

G 90 X[X-coordinate of the end point of the specified path] Z[Z-coordinate of the end point of the specified path].

The format of the record of tapered cutting cycle in a control sequence:

G 90 X[X-coordinate of the end point of the specified path] Z[Z-coordinate of the end point of the specified path] R[conicity].

The format of the record of threading cycle in a control sequence:

G 92 X[X-coordinate of the end point of the specified path] Z[Z-coordinate of the end point of the specified path].

The format of the record of conic threading cycle in a control sequence:

G 92 X[X-coordinate of the end point of the specified path] Z[Z-coordinate of the end point of the specified path] R[conicity].

The format of the record of endface turning cycle in a control sequence:

G 94 X[X-coordinate of the end point of the specified path] Z[Z-coordinate of the end point of the specified path].


The format of the record of cycle of endface turning of a cone in a control sequence:

G 94 X[X-coordinate of the end point of the specified path] Z[Z-coordinate of the end point of the specified path] R[conicity].

Turning Cycles for CNC Centers NCT, 2R22 and NC-31

The system supports machining cycles built in the CNC center 2R22, NCT and NC-31. To use those cycles, the user needs to select the respective option in the turning automenu, then specify the path and define the set of parameters.

CNC Center 2R22



To specify parameters for the cycles that are realized for CNC center 2R22, it is required to press the  button in the menu of turning and indicate the path, constructed in advance, on the drawing.

Command	Cycle
L01	Cycle of cylindrical and conic threading with automatic partitioning into passes.
L02	Cycle of groove turning with automatic partitioning into passes
L03	Cycle of external machining with automatic pullout and return to the initial point at a safe run.
L04	Cycle of internal machining along the Z-coordinate with automatic pullout and return to the initial point at a safe run.
L05	Cycle of endface machining with automatic pullout and return to the initial point at a safe run.
L06	Cycle of deep drilling with automatic partitioning into passes.
L07	Cycle of threading with a tap or a die.
L08	Cycle of multi-pass machining from a cylindrical workpiece with automatic partitioning into passes.
L09	Cycle of multi-pass machining of forged pieces with automatic partition into passes.

Threading cycle

Toolpath parameters.

Toolpath. Name of a toolpath.

Tool's file. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilized for the chosen type of machining (in this case, only the cutters that are contained in the tool's file).

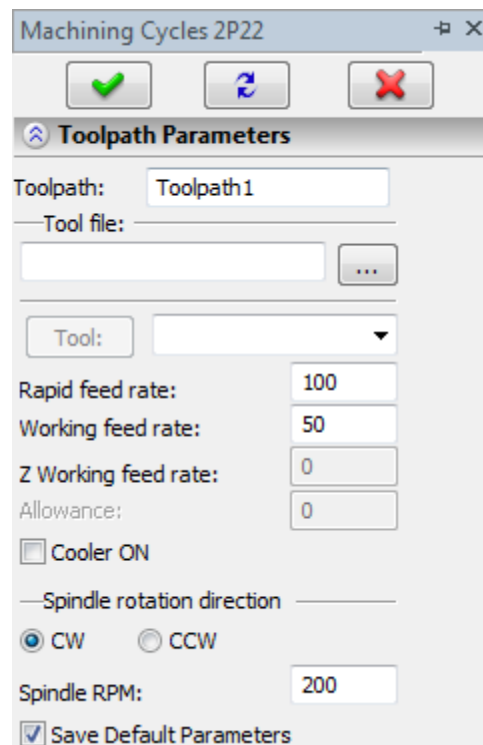
Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary translations.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the tool's translation when executing the motion of cutting.

Cooler on. Turning on/off the supply of the cooling agent upon machining.

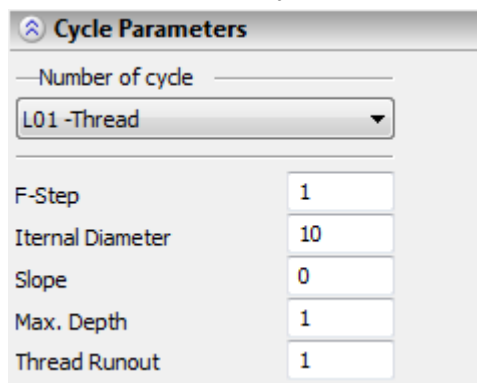
Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.



Cycle parameters.

Cycle number. Selection of a specific number of the cycle.



F-step. Step of a thread (pitch).

Internal diameter. Internal diameter of the thread.

Slope. The magnitude of the cone's angle for conic thread.

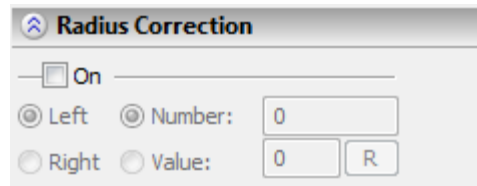
Max. depth. Maximum cutting depth.

Thread runout. Distance set for exit from the thread.

Radius correction.

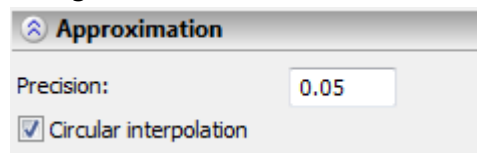
On. When this flag is enabled, parameters of the radius correction can be used. Correction can be **left** and **right**. It is possible to specify correction by the **number** (in this case, upon machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number indicated by the user) and by the **value** (in this case, the user specifies the precise value of the radius of rounding of the cutting edges of the mill).

R – use the radius of the selected tool.



Approximation.

Precision of the approximation is specified as a numerical value in units supported by the CNC center and constitutes the magnitude of the caving (maximum distance from the arc of the circle to be straightened to the approximating line).





Circular interpolation. In case when the contour being machined has circles, arcs, splines, and other geometrical elements having similar shape, the selection of this parameter will allow us to obtain the control sequence that contains frames of the circular interpolation.

Cycle of groove turning

Toolpath parameters.

Toolpath. Name of a toolpath.

Tool's file. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilized for the chosen type of machining (in this case, only the cutters that are contained in the tool's file).

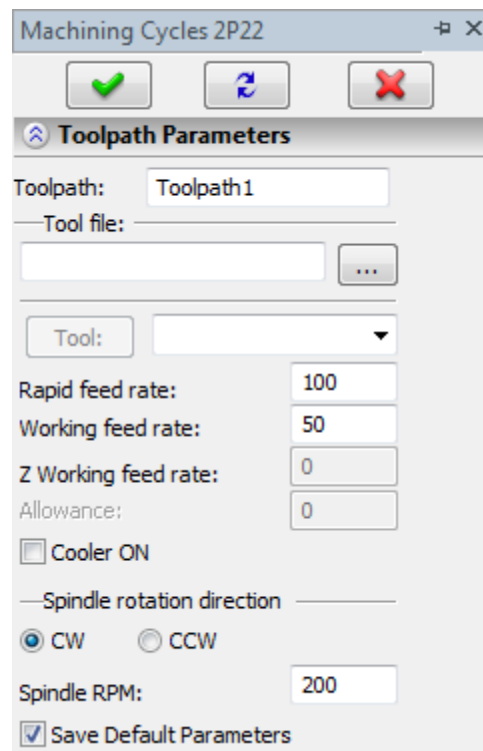
Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary translations.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the tool's translation when executing the motion of cutting.

Cooler on. Turning on/off the supply of the cooling agent upon machining.

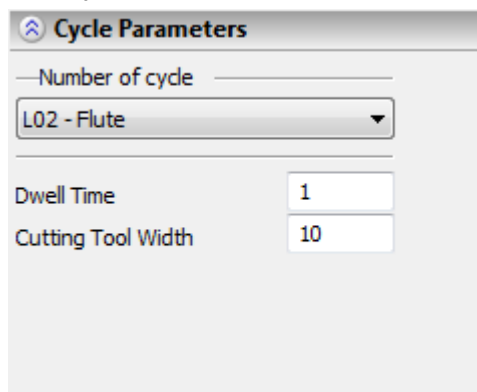
Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.



Cycle parameters.

Cycle number. Selection of a specific cycle number.



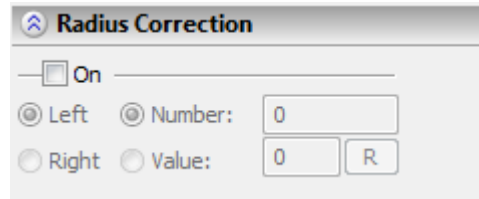
Dwell time. Pause time, in sec.

Cutting tool width. Width of the cutter used for finish turning.

Radius correction.

On. When this flag is enabled, parameters of the radius correction can be used. Correction can be **left** and **right**. It is possible to specify correction by the **number** (in this case, upon machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number indicated by the user) and by the **value** (in this case, the user specifies the precise value of the radius of rounding of the cutting edges of the mill).

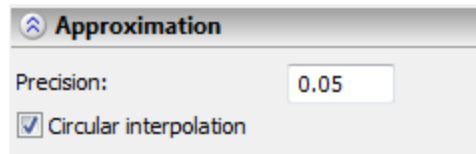
R – use the radius of the selected tool.



Approximation.

Precision of the approximation is specified as a numerical value in units supported by the CNC center and constitutes the magnitude of the caving (maximum distance from the arc of the circle to be straightened to the approximating line).



Circular interpolation. In case when the contour being machined has circles, arcs, splines, and other geometrical elements having similar shape, the selection of this parameter will allow us to obtain the control sequence that contains frames of the circular interpolation.



Cycle of external, internal and endface machining

Toolpath parameters.

Toolpath. Name of a toolpath.

Tool's file. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilized for the chosen type of machining (in this case, only the cutters that are contained in the tool's file).

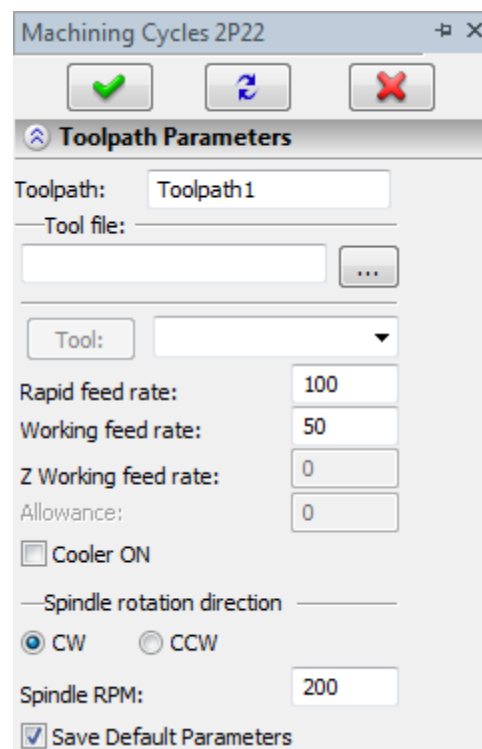
Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary translations.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the tool's translation when executing the motion of cutting.

Cooler on. Turning on/off the supply of the cooling agent upon machining.

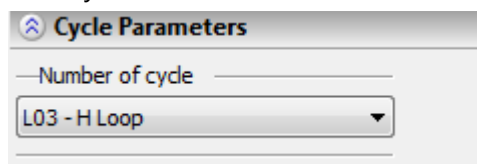
Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.



Cycle parameters.

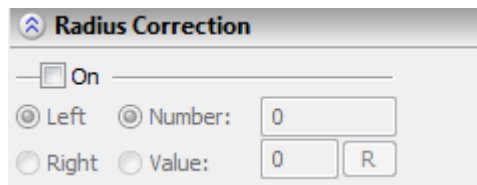
Cycle number. Selection of a specific cycle number.



Radius correction.

On. When this flag is enabled, parameters of the radius correction can be used. Correction can be **left** and **right**. It is possible to specify correction by the **number** (in this case, upon machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number indicated by the user) and by the **value** (in this case, the user specifies the precise value of the radius of curvature of the cutting edges of the mill).

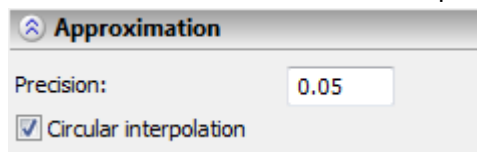
R – use the radius of the selected tool.



Approximation.

Precision of the approximation is specified as a numerical value in units supported by the CNC center and constitutes the magnitude of the caving (maximum distance from the arc of the circle to be straightened to the approximating line).

Circular interpolation. In case when the contour being machined has circles, arcs, splines, and other geometrical elements having similar shape, the selection of this parameter will allow us to obtain the control sequence that contains frames of the circular interpolation.





Cycle of deep drilling

Cycle of threading with a tap

Toolpath parameters.

Toolpath. Name of a toolpath.

Tool's file. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

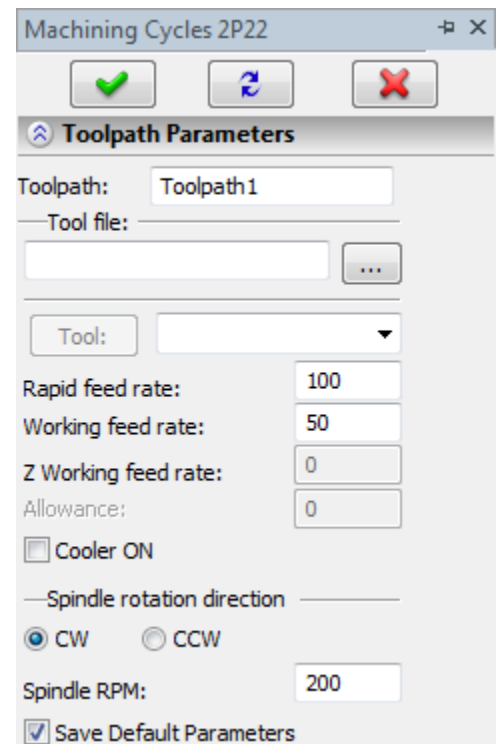
It is to be noted that the system will offer the user to select only that tool which can be utilized for the chosen type of machining (in this case, only the cutters that are contained in the tool's file).

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary translations.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the tool's translation when executing the motion of cutting.

Cooler on. Turning on/off the supply of the cooling agent upon machining.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

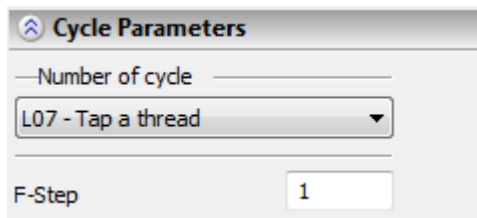


Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.

Cycle parameters.

Cycle number. Selection of a certain cycle number.

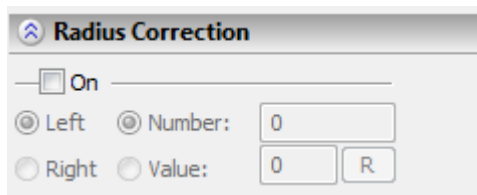
F-step. Step of a thread (pitch).



Radius correction.

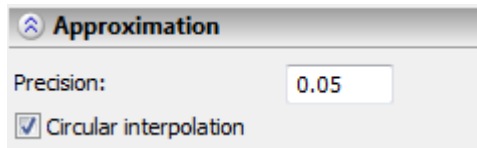
On. When this flag is enabled, parameters of the radius correction can be used. Correction can be **left** and **right**. It is possible to specify correction by the **number** (in this case, upon machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number indicated by the user) and by the **value** (in this case, the user specifies the precise value of the radius of curvature of the cutting edges of the mill).

R – use the radius of the selected tool.



Approximation.

Precision of the approximation is specified as a numerical value in units supported by the CNC center and constitutes the magnitude of the caving (maximum distance from the arc of the circle to be straightened to the approximating line).





Circular interpolation. In case when the contour being machined has circles, arcs, splines, and other geometrical elements having similar shape, the selection of this parameter will allow us to obtain the control sequence that contains frames of the circular interpolation.

Cycle of allowance removal by roughing

Toolpath parameters.

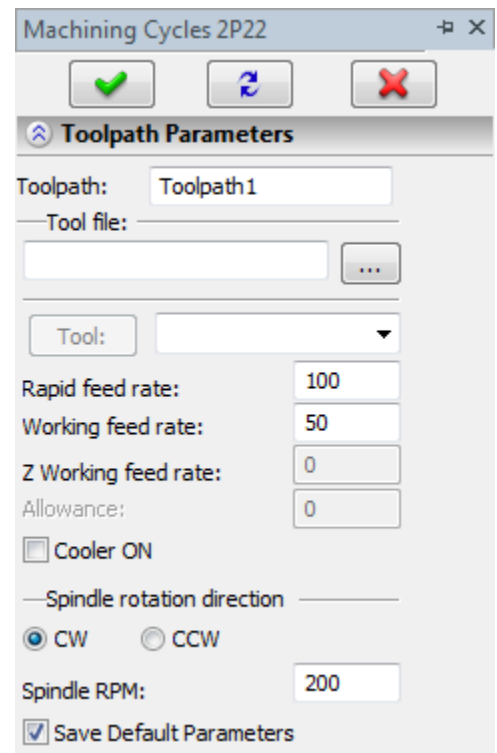
Toolpath. Name of a toolpath.

Tool's file. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilized for the chosen type of machining (in this case, only the cutters that are contained in the tool's file).

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary translations.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the tool's translation when executing the motion of cutting.



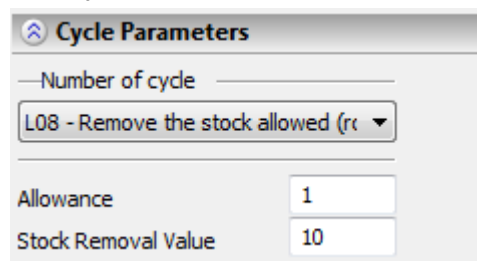
Cooler on. Turning on/off the supply of the cooling agent upon machining.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.

Cycle parameters.

Cycle number. Selection of a specific cycle number.



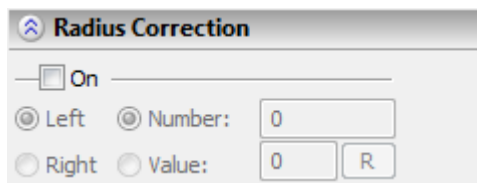
Allowance. The magnitude of the allowance. This parameter is uploaded to the control sequence.

Stock removal. This parameter is uploaded to the control sequence.

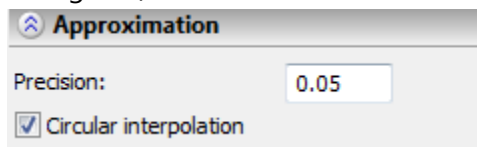
Radius correction.

On. When this flag is enabled, parameters of the radius correction can be used. Correction can be **left** and **right**. It is possible to specify correction by the **number** (in this case, upon machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number indicated by the user) and by the **value** (in this case, the user specifies the precise value of the radius of curvature of the cutting edges of the mill).

R – use the radius of the selected tool.

**Approximation.**

Precision of the approximation is specified as a numerical value in units supported by the CNC center and constitutes the magnitude of the caving (maximum distance from the arc of the circle to be straightened to the approximating line).





Circular interpolation. In case when the contour being machined has circles, arcs, splines, and other geometrical elements having similar shape, the selection of this parameter will allow us to obtain the control sequence that contains frames of the circular interpolation.

Cycle of allowance removal by finishing

Toolpath parameters.

Toolpath. Name of a toolpath.

Tool's file. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilized for the chosen type of machining (in this case, only the cutters that are contained in the tool's file).

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary translations.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the tool's translation when executing the motion of cutting.

Cooler on. Turning on/off the supply of the cooling agent upon machining.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

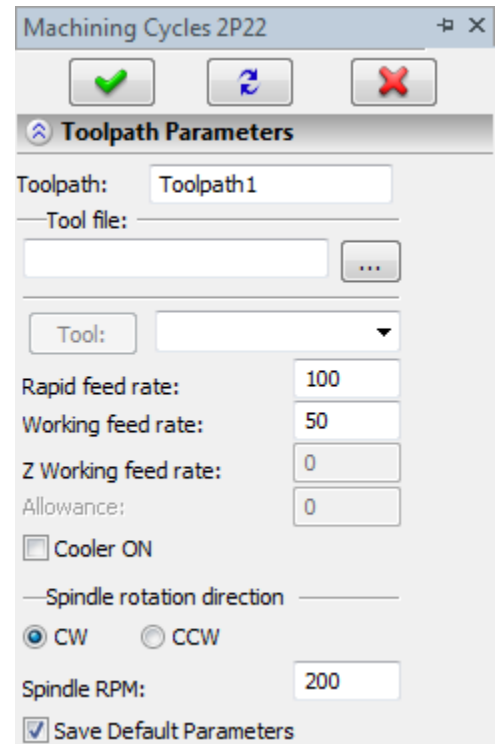
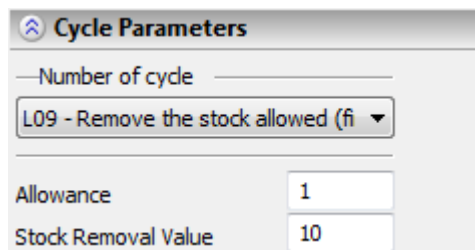
Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.

Cycle parameters.

Cycle number. Selection of a specific cycle number.

Allowance. Magnitude of allowance. This parameter is uploaded to the control sequence.

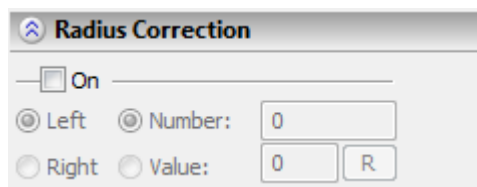
Stock removal value. This parameter is uploaded to the control sequence.

Radius correction.

On. When this flag is enabled, parameters of the radius correction can be used. Correction can be **left** and **right**. It is possible to specify correction by the **number** (in this case, upon machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number indicated by the user) and by the **value** (in this case, the user specifies the precise value of the radius of curvature of the cutting edges of the mill).

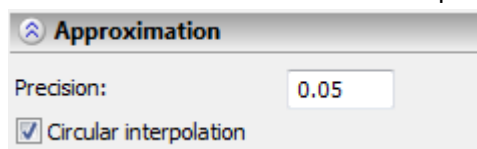
R – use the radius of the selected tool.




Approximation.

Precision of the approximation is specified as a numerical value in units supported by the CNC center and constitutes the magnitude of the caving (maximum distance from the arc of the circle to be straightened to the approximating line).

Circular interpolation. In case when the contour being machined has circles, arcs, splines, and other geometrical elements having similar shape, the selection of this parameter will allow us to obtain the control sequence that contains frames of the circular interpolation.



CNC Center NC-31


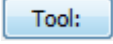
The following cycles are supported for NC-31 center: turning cycle; turning cycle with angled grooves; crosscut (grooving) cycle; crosscut (grooving) cycle with angled grooves. To define parameters of those cycles, press the  button in the turning automenu.

G77	Turning cycle
G77'	Turning cycle with angled grooves
G78	Crosscut (grooving) cycle
G78'	Crosscut (grooving) cycle with angled grooves

If you need to use one of those cycles, select the respective cycle option, and then specify the machining toolpath. Also, you should define machining parameters.

Toolpath parameters.

Toolpath. Name of a toolpath.

Tool's file. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilized for the chosen type of machining (in this case, only the cutters that are contained in the tool's file).

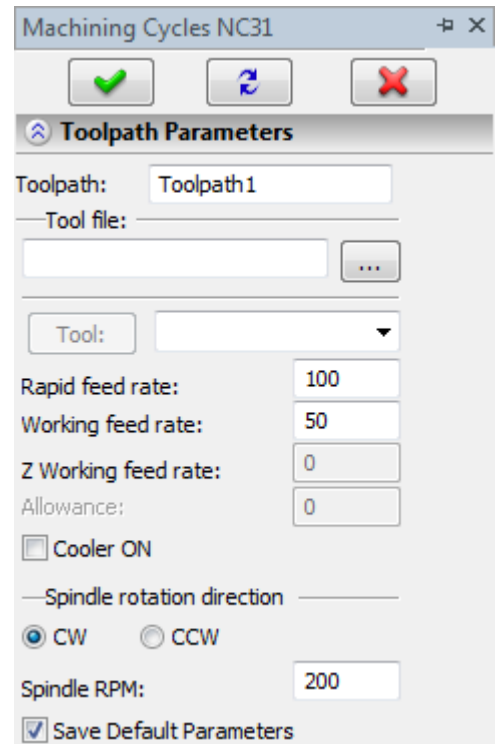
Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary translations.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the tool's translation when executing the motion of cutting.

Cooler on. Turning on/off the supply of the cooling agent upon machining.

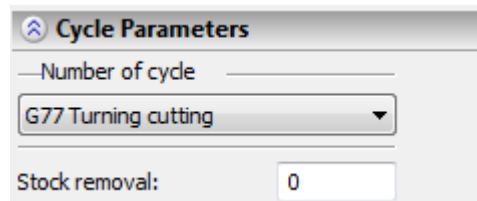
Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.


Cycle parameters.

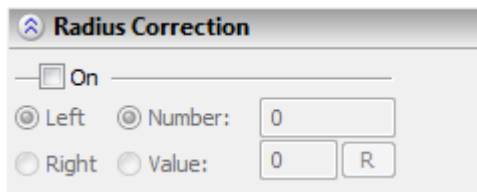
Cycle number. Selection of a specific cycle number.

Stock removal value. Maximum depth of cutting.


Radius correction.

On. When this flag is enabled, parameters of the radius correction can be used. Correction can be **left** and **right**. It is possible to specify correction by the **number** (in this case, upon machining, the value of the correction will be used written in the table of correctors of the machining station in the row which has the number indicated by the user) and by the **value** (in this case, the user specifies the precise value of the radius of curvature of the cutting edges of the mill).

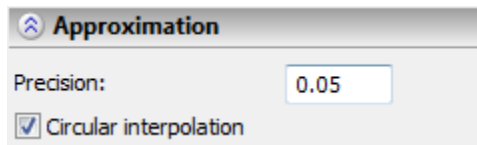
R – use the radius of the selected tool.




Approximation.

Precision of the approximation is specified as a numerical value in units supported by the CNC center and constitutes the magnitude of the caving (maximum distance from the arc of the circle to be straightened to the approximating line).

Circular interpolation. In case when the contour being machined has circles, arcs, splines, and other geometrical elements having similar shape, the selection of this parameter will allow us to obtain the control sequence that contains frames of the circular interpolation.



CNC Center NCT



The following cycles are supported for the NCT center: allowance removal cycle; endface allowance removal cycle; repetitive commands cycle. To define parameters of those cycles, press the  button in the turning automenu. The following options will become available in the automenu for selecting a path required in the cycles mentioned above:

G71	Allowance removal cycle
G72	Endface allowance removal cycle
G73	Repetitive commands

If you need to use one of those cycles, specify the machining path, and then select the appropriate cycle option. After that, a dialog box appears where you need to define machining parameters:

Toolpath parameters.

Toolpath. Name of a toolpath.

Tool's file. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilized for the chosen type of machining (in this case, only the cutters that are contained in the tool's file).

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary translations.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the tool's translation when executing the motion of cutting.

Cooler on. Turning on/off the supply of the cooling agent upon machining.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.

Cycle parameters.

Cycle number. Selection of a specific cycle number.

Cutting depth. Thickness of the material layer removed at each iteration of the cycle of allowance removal.

Overtravel in X. Pullout of the tool from the workpiece after the working pass along the X-axis.

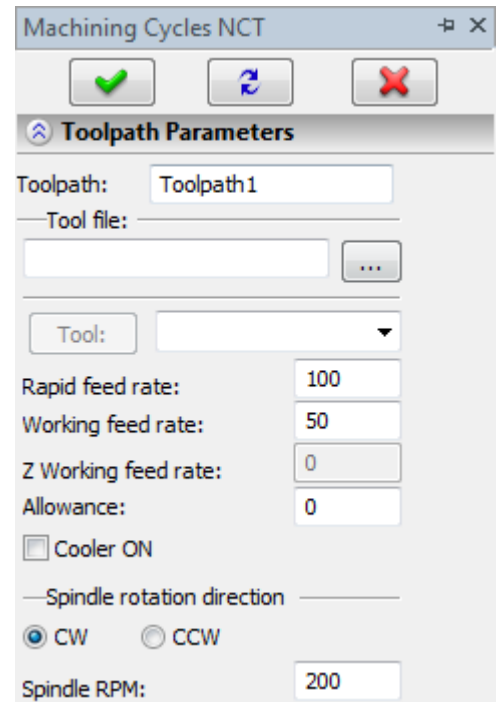
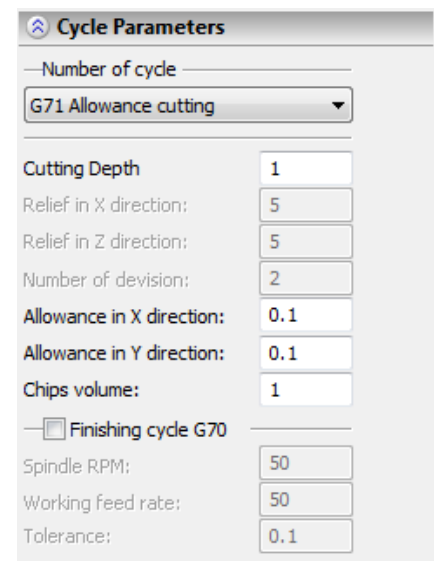
Overtravel in Z. Pullout of the tool from the workpiece after the working pass along the Z-axis.

Number of divisions. Number of passes of the tool.

Allowance in X direction. The distance and direction of the allowance for the finishing in X-direction.

Allowance in Z-direction. The distance and direction of the allowance for the finishing in Z-direction.

Chips volume. Parameter uploaded to the control sequence.

Cycle of finishing G70. Possibility of adding, to the selected cycle, an additional cycle of finishing G70 with the specified **RPM**, **working feed rate** and **allowance**.



Drilling Cycles for CNC Centers 2C42, VECTOR 90, P-2M, FANUC, E 2000 CNC

Drilling is supported in a number of standard machining cycles tuned to the following brands of CNC centers: 2C42, R-2M and Fanuc, Vector 90, E 2000 CNC used by the user for calculating machining toolpaths.

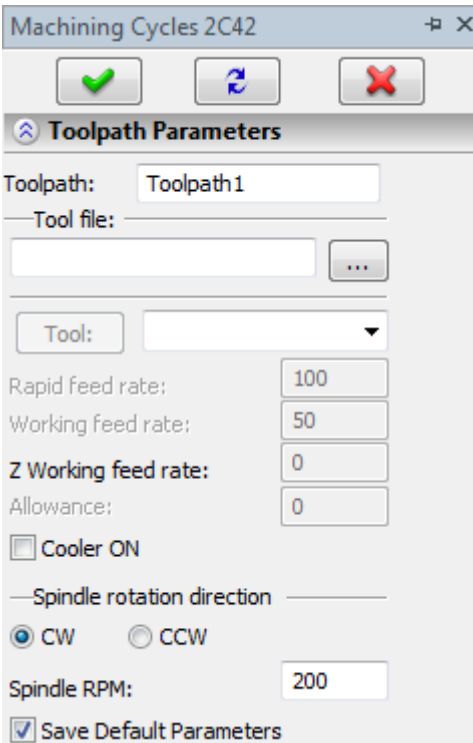
Machining Cycles for CNC Center 2C42

To use machining cycles for the 2C42 CNC center, select the desired option in the drilling automenu. To create a toolpath it is necessary to indicate the path, constructed in advance, at the nodes of which the drilling operation will be performed. When constructing a path, it should be taken into account that at the initial node of the path the drilling will not be carried out. Thus, if the path consists of $n + 1$ nodes, the drilling will be executed n times, starting with the second node.

Toolpath. Name of a toolpath.

Tool's file. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilized for the chosen type of machining (in this case, only the drills that are contained in the tool's file).



The screenshot shows a software window titled "Machining Cycles 2C42". It has three buttons at the top: a green checkmark, a blue refresh icon, and a red X. Below is a section titled "Toolpath Parameters". It includes a "Toolpath:" field with the value "Toolpath1", a "Tool file:" field with an empty text box and a three-dot button, and a "Tool:" dropdown menu. Below these are input fields for "Rapid feed rate:" (100), "Working feed rate:" (50), "Z Working feed rate:" (0), and "Allowance:" (0). There is a checkbox for "Cooler ON". A "Spindle rotation direction" section has two radio buttons, "CW" (selected) and "CCW". The "Spindle RPM:" field is set to 200. At the bottom is a checked checkbox for "Save Default Parameters".

Cooler on. Turning on/off the supply of the cooling agent upon machining.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.

Cycle parameters.

Cycle number. Selection of a specific cycle number.

R0 Working feed rate in Z. The value of the working feed rate for translations along the Z-axis.

R1 Z – approach. The value of the coordinate of the safety plane along the Z-axis (height at which all auxiliary translations are taking place) in case of approach of the tool.

R2 Depth of drilling. Numerical value of the depth of drilling.


R3 Z – retreat. The value of the coordinate of the safety plane along the Z-axis (height at which all auxiliary translations are taking place) in case of retreat of the tool.

R4 Dwell time. Numerical value of the pause in units supported by the CNC center.

R5 Feed rate of a tap. Numerical value of the feed rate corresponding to the working run of the tap.



R6 Unit depth. Numerical value of the depth of the unit cutting-in.

Machining Cycles for CNC Center R-2M

To specify parameters of the machining cycles for the CNC center R-2M, it is required to press the  icon in the automenu of drilling; after that the automenu of machining cycles R 2-M will appear.

To create a toolpath it is necessary to indicate the path, constructed in advance, at the nodes of which the drilling operation will be performed. When constructing a path, it should be taken into account that at the initial node of the path the drilling will not be carried out. Thus, if the path consists of $n + 1$ nodes, the drilling will be executed n times, starting with the second node.

Toolpath. Name of a toolpath.

Tool's file. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilized for the chosen type of machining.

Cooler on. Turning on/off the supply of the cooling agent upon machining.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.

Cycle parameters.

Cycle number. Selection of a specific cycle number.

Z-depth of drilling. This numerical value is specified from the surface in which the drilling is taking place.


R - approach on fast run. The magnitude of the rapid feed rate with which the approach will take place.

F- working feed rate. The magnitude of the working feed rate.

K – number of passes. Number of iterations in the cycle.


The image shows a software interface titled "Cycle Parameters". It contains a dropdown menu labeled "Number of cycle" with the selected option "G84 Thread cutting with tap". Below this are four input fields, each with a label and a value of "0":
 Z - Final hole depth: 0
 R - Traverse: 0
 F - Working feed rate: 0
 K - Number of passes: 0

Machining cycles for CNC center Fanuc

To specify parameters for machining cycles of the CNC center Fanuc, it is required to press the  icon in the automenu of drilling; the automenu of machining cycles Fanuc will appear.


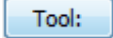
To create a toolpath it is necessary to indicate the path, constructed in advance, at the nodes of which the drilling operation will be performed. When constructing a path, it should be taken into account that at the initial node of the path the drilling will not be carried out. Thus, if the path consists of $n + 1$ nodes, the drilling will be executed n times, starting with the second node.

Machining Cycles for E 2000 CNC Center

The following cycles are available for this type of equipment: "Drilling, fast retreat", "Drilling with dwelling, fast retreat", "Deep drilling", "Threading with tap", "Boring". To define parameters of those cycles, select the  option in the drilling menu. The automenu of E 2000 CNC machining cycles will appear.

To create a toolpath, it is necessary to indicate the path, constructed in advance, at the nodes of which the drilling operation will be performed. When constructing a path, it should be taken into account that at the initial node of the path the drilling will not be carried out. Thus, if the path consists of $n + 1$ nodes, the drilling will be executed n times, starting with the second node.

Toolpath. Name of a toolpath.

Tool's file. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

It is to be noted that the system will offer the user to select only that tool which can be utilized for the chosen type of machining.

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary translations.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the tool's translation when executing the motion of drilling.

Cooler on. Turning on/off the supply of the cooling agent upon machining.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.

Cycle parameters.

Cycle number. Selection of a specific cycle number.

Safety height. Height of the lift of the tool for horizontal translations.

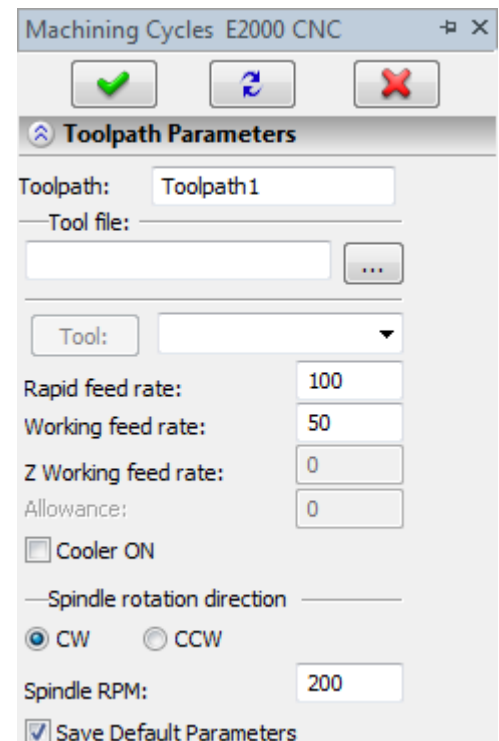
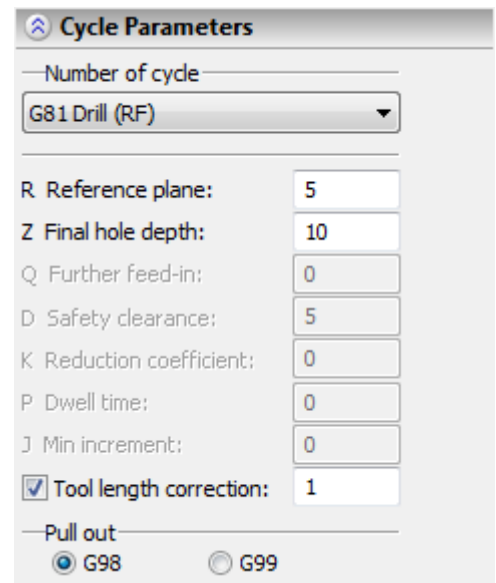
Depth of drilling. The maximum height at which the drill descends upon drilling.

Depth of iteration. The magnitude of the drilling step.

Undertravel value. Height at which the working feed rate will be turned on.

Reduction coefficient. Numerical factor used for calculation of the magnitude of subsequent drilling steps as compared to the previous steps.

Dwell time. The time of a pause between the drillings in units supported by the CNC center.


Minimum increment. The magnitude of the minimum drilling step (does not have to be specified).

Offset correction. Optional specification of the tool's offset correction with the indication of the corrector's number.

Pull out. G98 or G99.



To get the value of the next drilling step, multiply the value of the previous drilling step by the specified reduction amount. Upon reaching the value of the minimal drilling step, all further drilling steps are done with this value.

Machining Cycles for Vector 90 CNC Center

To specify parameters for machining cycles of the CNC center Vector 90, it is required to press the  icon in the automenu of the drilling; the automenu of machining cycles Vector 90 will appear.

To create a toolpath it is necessary to indicate the path, constructed in advance, at the nodes of which the drilling operation will be performed. When constructing a path, it should be taken into account that at the initial node of the path the drilling will not be carried out. Thus, if the path consists of $n + 1$ nodes, the drilling will be executed n times, starting with the second node.

Toolpath. Name of a toolpath.

Tool's file. With the help of the  button, select the required tool's file, and then from the drop-down list located below, - the desired tool. To start editing of the selected tool, press the  button.

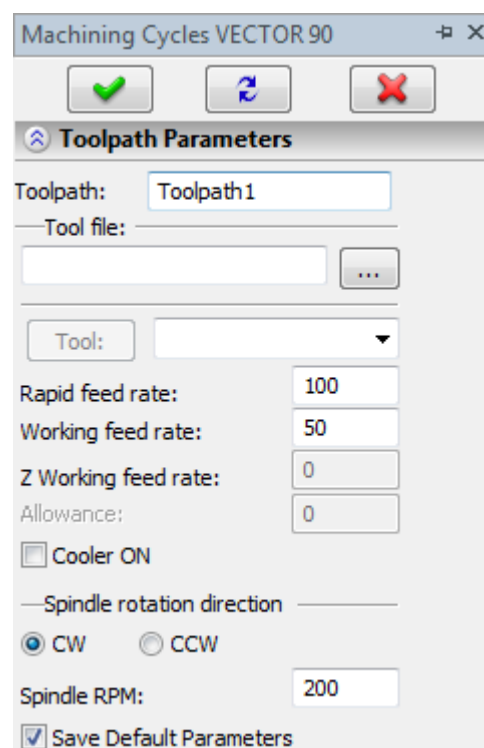
It is to be noted that the system will offer the user to select only that tool which can be utilized for the chosen type of machining.

Rapid feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the motion of the tool when executing auxiliary translations.

Working feed rate is specified as a numerical value in units supported by the CNC center and sets up the speed of the tool's translation when executing the motion of drilling.

Cooler on. Turning on/off the supply of the cooling agent upon machining.

Spindle rotation direction. Setting up the direction of the spindle rotation – either clockwise or counterclockwise.

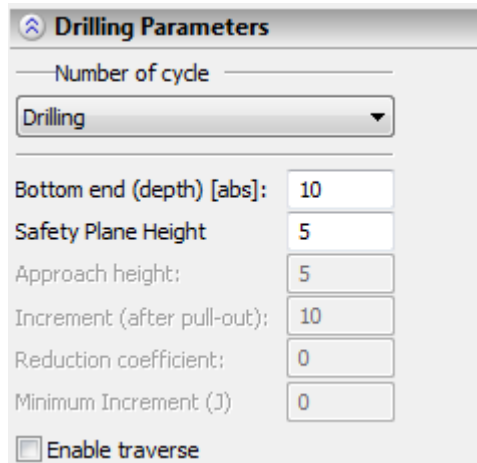


Spindle RPM. The value of the speed of the spindle rotation must be specified in those units which the CNC system supports when working with this parameter.

Cycle parameters.

Cycle number. Selection of a specific cycle number.

Depth of drilling (Z). The magnitude of this parameter is specified from the surface in which the drilling takes place.



The screenshot shows a dialog box titled "Drilling Parameters". It contains a "Number of cycle" section with a dropdown menu set to "Drilling". Below this are several input fields: "Bottom end (depth) [abs]:" with a value of 10, "Safety Plane Height" with a value of 5, "Approach height:" with a value of 5, "Increment (after pull-out):" with a value of 10, "Reduction coefficient:" with a value of 0, and "Minimum Increment (J)" with a value of 0. At the bottom, there is a checkbox labeled "Enable traverse" which is currently unchecked.

Safety height (R). The height of the lift of the tool for horizontal translations.

Traverse height (R). When the "Enable traverse" flag is enabled, the transition at the given height is added to the control sequence.


Increment (I). The depth of the step for deep drilling.


Reduction coefficient (K). The coefficient of reduction for the depth of the step upon deep drilling.


Minimum increment. Minimum drilling step. Does not have to be specified.

Enable traverse. Addition of the extra transition to the control sequence.


MACHINING MANAGER


Created toolpaths are added to the machining manager that constitutes a tree-like list of toolpaths and different types of machining. To view this list, the user needs to press the  icon or use the command:

Icon	Ribbon
	CNC → Settings → Machining Manager
Keyboard	Textual Menu
	CNC>Machining Manager

A dialog box will appear, displaying all existing machining toolpaths in the given file. To navigate through the list of toolpaths, use  or keyboard keys <↑>, <↓>.

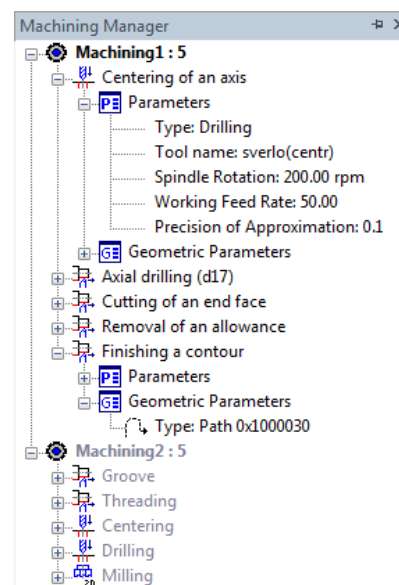
The structure of the constructed toolpaths is displayed in the window as a tree. According to the conventions of T-FLEX CAM/CAM 3D, several *machining* processes can be packed in one file, which consists of different types of *toolpaths*. In this way, wire cutting, laser cutting and mechanical machining can be combined in one machining. Machining is an arbitrary combination of the following processes: 2.5D Milling, Drilling, Turning-type machining and engraving (fig. 1).

Each element in the Manager is assigned a unique name, which by default consists of its type and number, for example, "Machining_1"/"Toolpath_3". The "plus" mark may precede an element name. This means that the manager tree branch is not expanded and the element contains enclosed elements. To expand the Manager tree, point and click  on the "plus" mark. The mark will then change to the "minus", revealing an element or a list of elements standing down the hierarchy.


Accordingly, to close a branch, point and click  on the "minus" mark. If there is no mark at the left of an element name, that means that the element is the last on this Manager branch.

The following types of elements are contained in the Machining Manager tree:

1. **Machining** is a compound toolpath, consisting of wire cutting, laser cutting or mechanical machining.
2. **Toolpath** is a calculated toolpath of the machining tool movement, arrival to a specified point (GOTO), coordinate system, machining commands/toolpath, auxiliary toolpaths (arrays of copies).
3. **Machining parameters** are the principal parameters defining the toolpath.

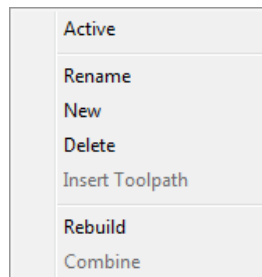


4. **Geometric parameters** are the geometrical elements used for calculating toolpaths.

By expanding branches of the Manager tree, one can access any element in the structure. The tree helps conveniently select objects. You can call the context menu by pointing at a tree element and right-clicking . The context menu composition is different for machining and toolpath elements.

The context menu of machinings and toolpaths

Active – activation/deactivation of machining. At any moment in time only one type of machining can be active in one document. All toolpaths created by default are added to the active type of machining. When selecting the Active option in the context menu of machining, the machining will be activated. If several types of machining are created for the document, when activating one type of machining, another type of machining will automatically be deactivated (font and icon change their appearance in the tree of machinings). When activating a machining using the context menu, the pointer that signifies that the selected type of machining is active at the given moment will appear next to the Active option.



Rename – edit the name of machining.

Create - creation of a new type of machining.

Delete – delete the selected type of machining and all toolpaths that are included into the structure of the machining.

Insert toolpath – add the toolpath, copied onto the exchange buffer, to the given type of machining.

Rebuild – recalculate compound toolpaths of the machining.

Combine. This command of the context menu is available when the <Ctrl> button is pressed and several types of machinings are selected. After combining the corresponding types of machining, one single type of machining that includes all toolpaths of the selected machinings will be created.

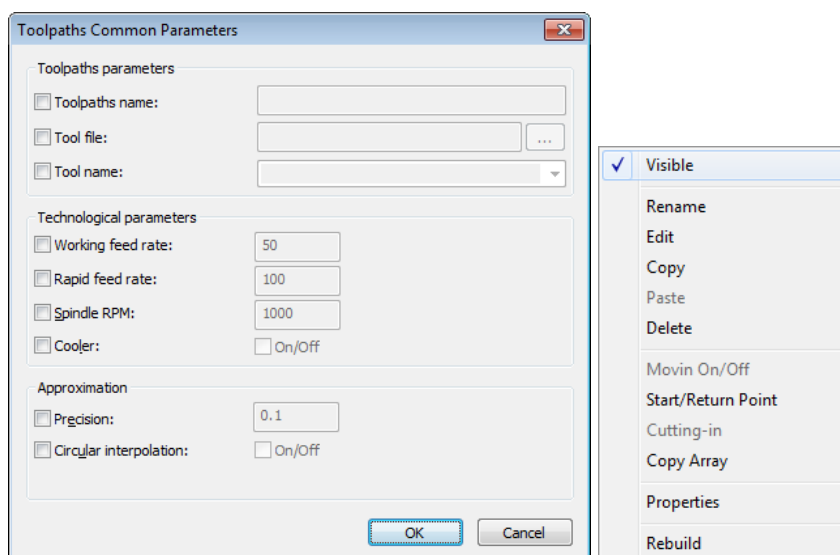
Context menu of the toolpath.

Visible – determines the visibility of the toolpath and inclusion of the toolpath into the control sequence. By default, the created toolpath always remains visible. The user can make the toolpath invisible on the part's drawing and eliminate the frames, which describe it, from the structure of the control sequence. To do so, it is necessary to select the Visible option in the context menu of the toolpath. Toolpath will become invisible on the drawing, its icon and font will change

appearance in the tree of machinings. The next time this option is selected, the toolpath will become visible.

Rename – edit the name of the toolpath.

Edit – invoke the mode of editing the toolpath. In this mode the user can respecify geometric elements, which form the basis of the toolpath (the ones that are automatically highlighted in the 3D scene or on 2D drawing, depending on the type of machining), and also change the technological parameters of the toolpath. If a group of toolpaths is selected, the Edit command opens the dialog window in which the general technological parameters of the selected toolpaths are available for modifications (for example, feed rate, tool's file, the tool's name, the toolpath's name, etc.).



Copy – copy selected toolpath to the exchange buffer.

Insert – insert a toolpath, which was copied to the buffer, into the tree of toolpaths, after the selected toolpath.

Delete – remove the selected toolpath.

Exit/entry - adding exit and/or the entry.

Start/Return point - adding the start and/or return point of the tool.

Cutting-in – adding the movements of cutting-in of the tool.

Array – adding the array of copies of the toolpath.

Properties – invoke the toolpath properties dialog.

Rebuild– recalculate the toolpath.

ARRAYS OF TOOLPATHS' COPIES.

Array constitutes the multiple copying of the toolpath or a group of toolpaths. This command opens nonmodal dialog window with parameters of multiple copying. The user can specify the center,

with respect to which the element being copied will be constructed, using two different methods. The first one – specify a node on the drawing. In this case, the node's coordinates will automatically be inserted into the corresponding fields of the dialog window of parameters. The second method – enter the coordinates in the dialog window of parameters manually which will lead to automatic construction of the node – the center of copying defined in these coordinates.

Parameters of 2D, 2.5D array.

Toolpath. Name of array's toolpath.

Linear array / circular array / completion of array. Selection of the corresponding type of an array.

Linear array.

With the help of a linear array, it is possible to define a certain number of copies of the initial machining toolpath. Copies of the machining toolpaths are located in this case along the guiding vector with a defined shift. The direction **vector** is specified as a numerical value with a sign in the following fields: **Vx, Vy, Vz**.

Setting type. Selection of the method of defining an array by the set of parameters.

Number of copies. Specifies the quantity of copies (elements) in the linear array.

Total shift. In the field of this parameter, enter the numerical value of the length of the segment on which the elements of the linear array (copies) are intended to be located.

Step. Specifies the value of the step with which the array elements (copies) will be created.

Limits. When specifying this limitation along the Z-axis of the toolpath, the toolpaths will be copied with the specified step and creation of the last copy will be carried out at the given height of the limitation.

Vector. Vector of direction of generation of array's elements.

Circular array.

Circular array allows us to create copies of the initial machining toolpath centered with respect to the specified point around which the copies at a certain angle will be placed. The center of the array is specified as numerical values in the fields: **X** and **Y** (these coordinates are specified with respect to the global coordinate system of the drawing).

Setting type. Selection of the method of defining an array by the set of parameters.

Number of copies. Specifies the quantity of copies (elements) in the circular array.

Total angle. In the field of this parameter you can specify the value of the angle within the limits of which the elements of the circular array (copies) are intended to be placed. Depending on the selected method of specification, this angle can automatically be calculated by the system.

Step. Specifies the angle between the elements of the circular array. Depending on the selected method of specification, the step can automatically be calculated by the system.

Center. Coordinates of the center of the circular array.

End of array creation.

When enabling this flag opposite the **End** field, the toolpath that defines the completion of the array is created. This means that upon creation of an array, the «End» toolpath is automatically generated by the system after the machining toolpaths of the array.

Parameters of 3D array.


Toolpath. Name of array's toolpath.


Type of array. Linear/Circular/Symmetry.

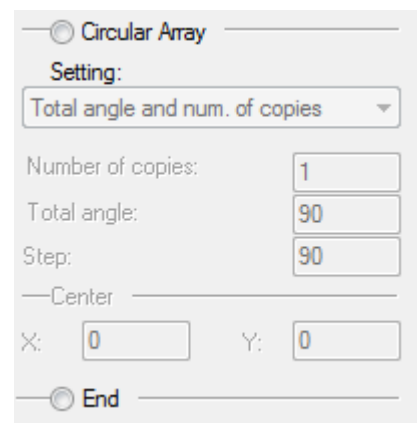
Step. Specifies the angle or distance between the elements of the array. It can be negative: in this case the direction of copying will be changed.

Number of copies. The quantity of toolpaths (array elements) that must be copied.

Total shift. In the field of this parameter enter the numerical value of the length of the segment on which the elements of the array (copies) are intended to be placed.

Start point (the  icon in the automenu). It is specified in terms of coordinates or as a 3D node. The point where the copying starts.

End point (the  icon in the automenu). It is specified in terms of coordinates or as a 3D node. The point where the copying ends.



Circular Array

Setting: Total angle and num. of copies

Number of copies: 1

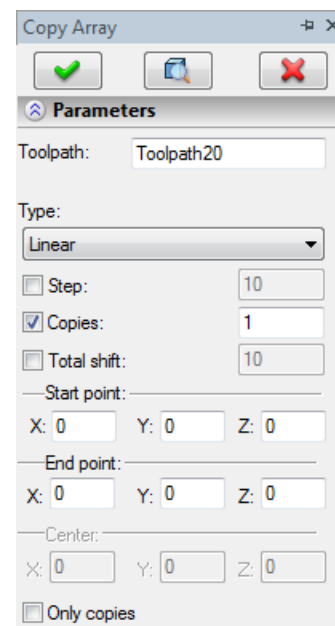
Total angle: 90

Step: 90

Center

X: 0 Y: 0

☐ End



Copy Array

Parameters

Toolpath: Toolpath20

Type: Linear

☐ Step: 10

☒ Copies: 1

☐ Total shift: 10

Start point:

X: 0 Y: 0 Z: 0


End point:

X: 0 Y: 0 Z: 0

Center:

X: 0 Y: 0 Z: 0

☐ Only copies

Center (the  icon in the automenu). It is specified in terms of coordinates or as a 3D node. The point of a center with respect to which the copying will be carried out (only for circular array).

Based on the coordinates of the start and end points of copying, the system calculates the vector (direction) of copying and the total shift (the length of the segment on which the copying will be carried out), and also the angle and the axis of copying (only for circular array).

For copying 2D and 3D toolpaths with the help of arrays, it is possible to select one or several toolpaths (using the keys <Shift> and <Ctrl>).

Creating the points of tool approach and retreat

To create additional segments at the start and end of the machining toolpath (for the purpose of bringing the tool to the machining area at fast feed rates or, vice versa, withdrawing the tool from the machining area), T-FLEX CAM has a provision for defining approach and retreat points. To use it, call the context menu of

the machining toolpath and in it select the “Approach/Moving Off” option. This action will bring up the nonmodal dialog window for defining the tool approach and retreat parameters.

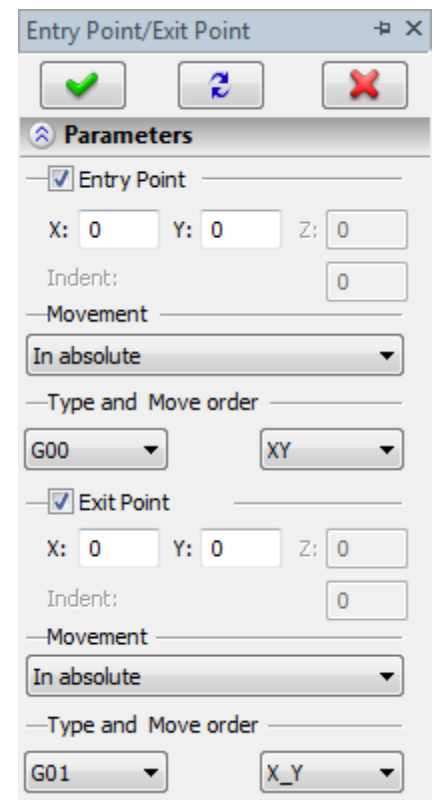
The start (approach) and return (retreat) points are defined explicitly by numerical values. Check “Start point” or “Return point” as appropriate, and then enter the values in the respective input box. The coordinates can be specified with respect to the global coordinate system of the drawing, by checking the “In absolute” option or in increments with respect to the calculated toolpath of the tool motion (here, the coordinates origin is considered at the start point of the toolpath in the case of the approach, and at the end point in the case of the retreat).

Point of approach/retreat. The flag of uploading the point of approach/retreat into the control sequence.

X,Y,Z. Specify the coordinates of the point of approach/retreat, if the **displacement** is specified in absolute coordinates. Specify the value of the increments relative to the calculated toolpath along the axes, if the **displacement** is specified in increments. In the last case, for the approach point the center of coordinate system is located at the first point of the toolpath, and for the retreat point – at the last point of the toolpath.

Offset. The length of the vector of the approach or retreat. The field is active, when the **displacement** is carried out along the tangent or along the normal.

Movements. Possible ways of calculation of the approach or retreat: in absolute coordinates, in increments, along the tangent, along the normal.



Type and move order. Here in the drop down lists the user can select the type of the G-command (G00 - rapid movement, G01 – movement on working feed rate) and the sequence of translations of the tool along the axes (simultaneous or separate - first along one axis, then along the other one).

Specify cut-in ramp motions

The "**Cut-in**" option from the context menu of the toolpath allows us to specify the way of cutting-in the tool for the toolpaths of milling.

The user can specify one of the four types of cutting-in: **cutting-in into the drilled hole, cutting-in along an arbitrary line, cutting-in along the first line of the toolpath, cutting-in along the spiral.**

When using the cutting-in into the drilled hole and along the arbitrary line, the base points of the cutting-in are specified in polar coordinates (R,A) with respect to the first point of the toolpath. In case of arbitrary tool's vector, for correct selection of the start point it is possible first to set A=0 and carry out the trial calculation by pressing the <Preview> button. By doing so, the line of cutting-in visible on the screen will coincide with the conditional X-axis with respect to which the angle A, which determines the direction of the line of cutting-in, must be specified. The initial point and the line of cutting-in always lie in the working plane (perpendicular to the tool's vector and passing through the initial point of the toolpath).

The final point of cutting-in will coincide with the first point of the machining toolpath on each layer being removed. The cutting-in is mainly used for rough and semi finishing machining.

Also, for passes that take into account limitations, the specification of **cutting-in by contour (traversal)** is available. This type of cutting-in will be applied when executing the transfer of the tool through the designated obstacle or upon transition from one layer of machining to another.

Parameters.

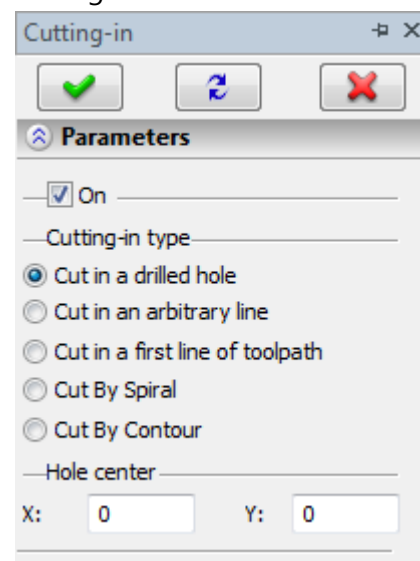
On. The flag of enabling/disabling the motions of cutting-in of the tool.

Cutting-in type. Enables the flag located opposite the desired type of cutting-in.

Hole center (cutting-in into the drilled hole). Specifies the length R of the segment of cutting-in and the angle of rotation A of the vector of cutting-in in the plane of machining.

Segment of cutting-in (cutting-in along the arbitrary line). Specifies two points in polar coordinates (R – shift of the point, A – the rotation of the vector of cutting-in constructed through the given point). Between the first and second points the passes are calculated with the magnitude of the deepening step, equal to the **depth of iteration**.

Depth of iteration (cutting-in along the arbitrary line and cutting-in along the first line of the toolpath) – here is specified the magnitude of the deepening step.



Step of spiral (cutting-in by spiral). The distance between the first and last points of one turn (coil) of the spiral is specified.

Cutting-in length/cutting-in segment (cutting-in along the arbitrary line and cutting-in along the first line of the toolpath). Specifies the ratio of the length of each cutting-in to the entire segment of cutting-in.

Segment length (cutting-in along the first line of the toolpath). Specifies the length of the segment of cutting-in.

Diameter of spiral (cutting-in by spiral). Specifies the magnitude of the diameter of the spiral's turns.

Spiral interpolation (cutting-in by spiral). This parameter forms the frames of the spiral interpolation when cutting-in along the spiral.

—Cutting-in section—

X1: X2:
 Y1: Y2:

—Cutting-in parameters—

Depth of every iteration:
 Cutting-in length/sect.:
☐ Cutting-in sect. length:

—Cutting-in parameters in one section—

Depth of every iteration:
 Cutting-in length/sect.:
 Cutting-in section length:

—Technological Parameters—

☐ Working feed rate:
☐ Spindle RPM:

☐ Spiral interpolation

For the toolpaths of 5D machining the specification of the movements of cutting-in is also available. This process is equivalent to setting up parameters of the toolpath of 5D machining in the dialog window of cutting-in. When the **Moving on** (Entry) or **Moving off** (Exit) options are enabled, the **length** parameter becomes available. If the user specifies the length and then finishes creation of the moving on/off, then the moving on/off will be built as a segment of the specified length along the tangent to the first/last point of the toolpath. This method of construction of the cutting-ins applies to all cases of creation of toolpaths of 5D machining by three paths. In case of construction of the motions of the cutting-in for the toolpaths of 5D machining geometrically described with the help of five paths, **Moving on/off vector** and **Tool's vector** should be used

Cutting-in

☐ ☐ ☐

Parameters

☒ Moving on: —

Length:

☐ Moving On vector —

VX: VY: VZ:

☒ Projection plane —

VX: VY: VZ:

☒ First face ☒ Second face

☐ Feed rate:

Length – the length of the segment of cutting-in.

Tool's vector – the vector along which the tool will be oriented in space when executing the cutting-in.

Moving on/off vector – direction by which the cutting-in will be carried out.

First face – the specified vector will be enabled for the first face.

Second face – the specified vector will be enabled for the second face.

Feed rate for moving on/off – the magnitude of the feed rate when executing the cutting-in.

Specifying the motions of moving on/off (leadin/leadout)

The “**Moving on/off**” option of the context menu of the toolpath invokes the window for specifying parameters of the cutting-in motion of the tool.

It is possible to specify the motion of cutting-in of the tool along the **tangent**, **normal** and **arc** of the circle. Depending on this choice, the parameters should define either the length of the segment (along the normal or tangent) that limits the motion of cutting-in or the radius of the circle whose arc will be taken as the basis of the motion of cutting-in and the angle that limits the arc of the circle of the given radius. Moreover, to specify the cutting-in along the arc of the circle, the user should also specify the direction of the cutting-in (clockwise or counterclockwise).

In all cases of construction of the motion of cutting-in, the first (for moving on) and the last (for moving off) points of the drawing will be taken as the origins.

Parameters.

Leadin (Leadout). This flag includes, into the control sequence, the motion of leadin of the tool on the contour being machined (leadout of the tool off the contour being machined).

X. The x-coordinate of the first point of the vector of leadin (leadout).

Y. The y-coordinate of the first point of the vector of leadin (leadout).

Length. The length of the tangent vector of leadout (leadin).

Radius. The radius of the circle for moving on (off) along the arc.

Angle. Central angle of the circle when performing leadin (leadout) along the arc.

Type of leadin (leadout). Selection of the type of leadin (leadout).


Clockwise. Direction of the motion along the arc for leadin (leadout) is clockwise.

Counterclockwise. Direction of the motion along the arc for leadin (leadout) is counterclockwise.







RELATIVE COORDINATE SYSTEM

RELATIVE COORDINATE SYSTEM FOR 2D, 2.5D AND 4D MACHINING

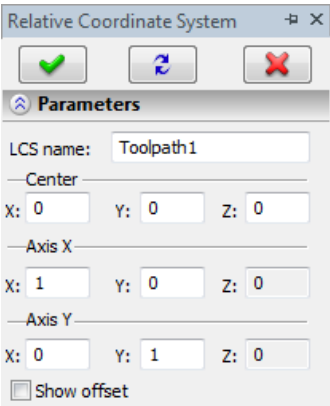
In the cases of flat types of machining, the user can use a relative coordinate system. This may be necessary quite often, as, for instance, in the cases of mismatches in design and technological data. To define a relative coordinate system, use the command:

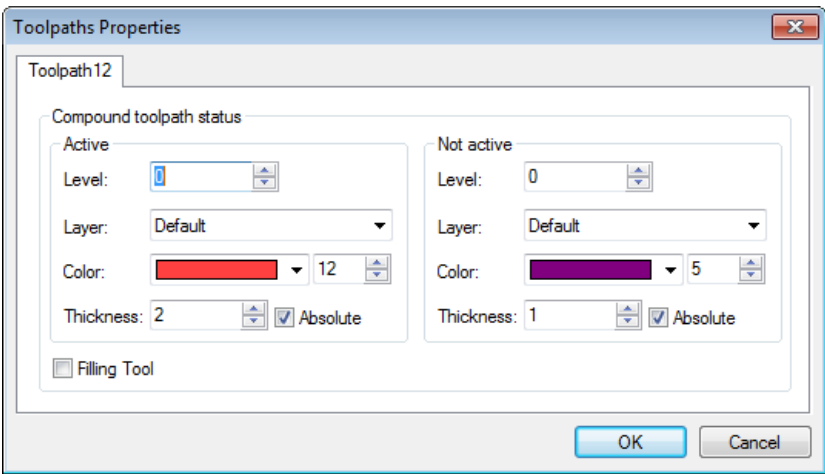
Icon	Ribbon
	CNC → Machining → Coordinate system
Keyboard	Textual Menu
	CNC> Relative coordinate system



Upon calling the command, the following options become available in the automenu:

	<End>	Finish editing
	<P>	Toolpath properties
	<O>	Coordinate system center
	<X>	Axis X of coordinate system
	<Y>	Axis Y of coordinate system
	<Esc>	Exit command

In the **Toolpath properties** window, the user can specify the properties for two groups of elements – active and nonactive toolpaths. Activity of toolpaths is described in more detail in the Machining manager section. It is possible to customize the thickness of displayed elements of the toolpath, their color, layer and level of location of the toolpath on the drawing.






The options with icons depicting coordinate axes ( and ) are for defining the center and the respective axes of the coordinate system. Usually, each of those elements is defined by selecting a node in the part drawing.

To define the relative coordinates system, specify just the center and one of the axes of the coordinate system being created.





To have the machining toolpath calculated according to the created relative coordinate system, move this coordinate system before the machining toolpath in the list of the toolpaths.

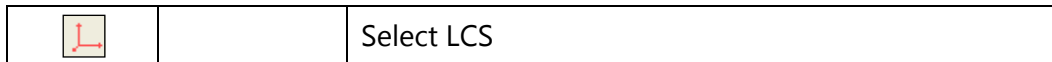
RELATIVE COORDINATE SYSTEM FOR 3D, 4D AND 5D MACHINING (3D VERSION)


In the cases of spatial machining types, the user can define a relative coordinate system. This may be necessary quite often, as, for instance, in the cases of disagreements between the design and the technological data. To define a relative coordinate system, use the command:

Icon	Ribbon
	CNC → Machinig → Relative coordinate system 3D
Keyboard	Textual Menu
	CNC > Relative coordinate system 3D

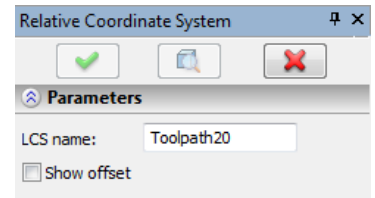
Upon calling the command, the following options become available in the automenu:

	<Y>	Finish input
	<P>	Set entity parameters
	<I>	Select Other Element
	<X>	Exit command



The parameters setup option  allows the user to define the parameters in the “Coordinate System Parameters” dialog window.


To define a relative coordinate system, select a local coordinate system previously created by a user within the T-FLEX CAD 3D system. The coordinates of the toolpaths which follow the relative coordinate system will be calculated with respect to that system. If the **“Show offset”** option is enabled, then the calculated toolpaths will be displayed with respect to the relative coordinate system. When the control sequence is being saved, the toolpath coordinates are written with respect to the relative coordinate system regardless of the state of the “Show offset” option.





NC PROGRAM GENERATION

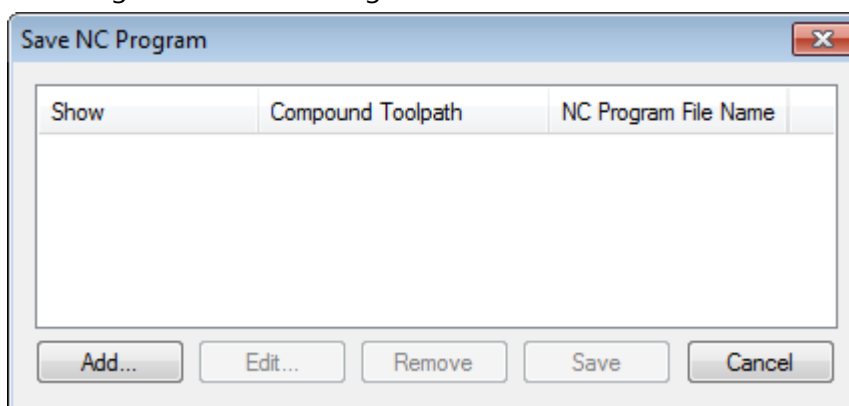
After creation of the machining toolpaths it is required to generate a file with a control sequence for the desired model of the machining station and CNC system. To do so, it is required to select the corresponding external or tabular postprocessor. If the processor is not selected, the control sequence will be created on the basis of the built-in basic postprocessor.

To create a control sequence, use the command:

Icon	Ribbon
	CNC → Output → NC program
Keyboard	Textual Menu
	CNC>Save NC program

Calling either of the commands brings up the "Save NC program" dialog box.

Double-click   inside the empty pane of the dialog or press the **[Add]** button. This will bring up the "Compound Toolpath Saving Parameters" dialog box.

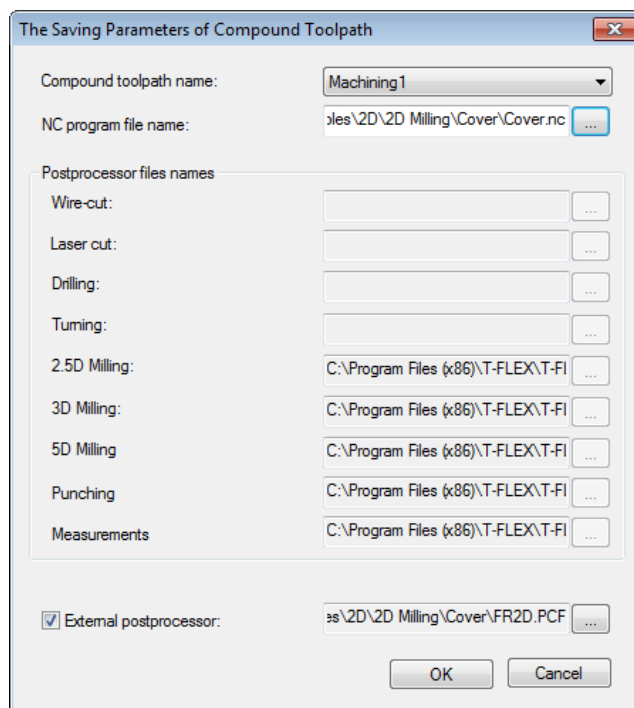



In this dialog box, specify one by one the names of the necessary postprocessors for the selected machining type, the name of the control sequence and the location where you want to save it.

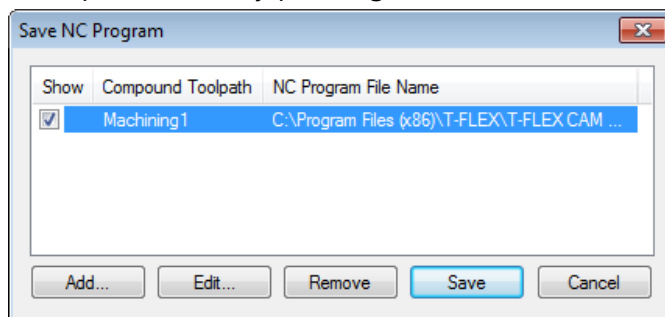
Note that the user can select either the postprocessors shipped with the system or the custom ones, built using the postprocessor generator. Control sequences for the same part and the same type of machining can be saved in different files with different postprocessors. This helps fully leverage the same equipment connected to different CNC centers.

Once the above steps are accomplished correctly, the dialog box will reappear on the screen with the specified entries.

Selecting the **"Show control sequence"** option makes the source text of the control sequence displayed in a separate window, right after the saving.



Note that any particular control sequence can be removed from the list. To do this, select it in the list by clicking  or by using the keys <↑> and <↓>, and then press the **[Remove]** button. You can also save all sequences shown in the list in separate files, by pressing the **[Save]** button.




MACHINING SIMULATOR

Once the user developed a machining toolpath and saved based on it control sequence for a CNC center, one can playback a quick machining simulation based on the saved control sequence. This is done by the built-in machining simulator.

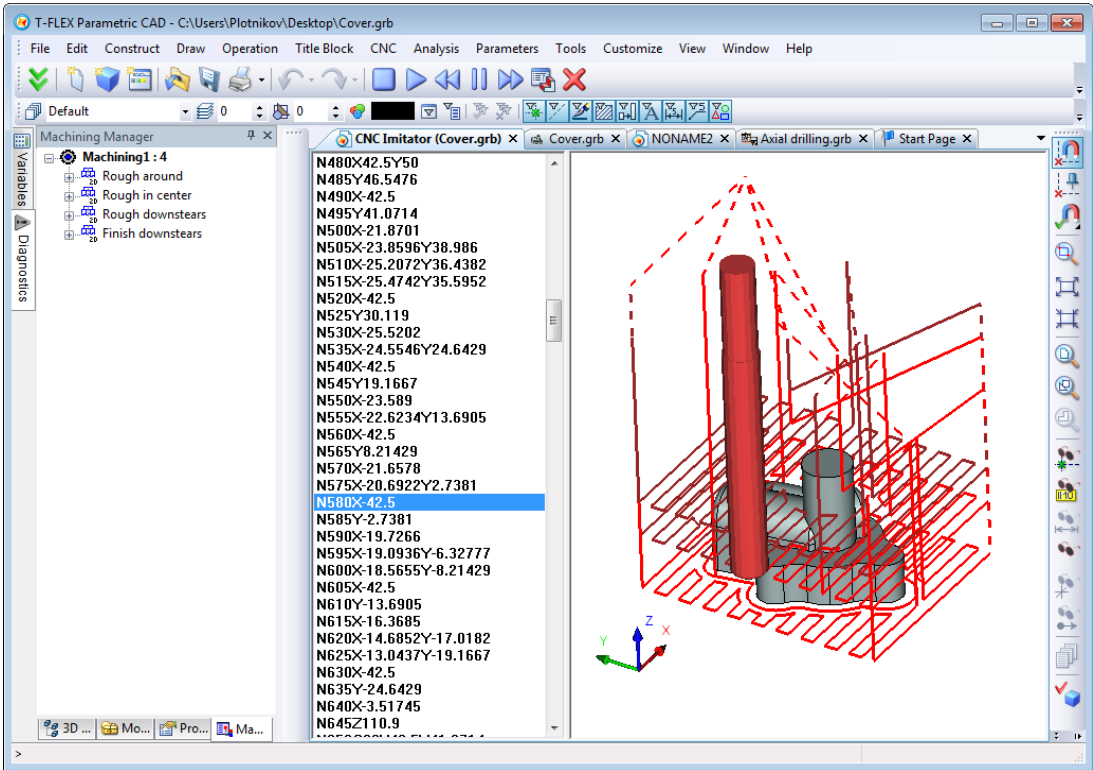
RUNNING THE MACHINING SIMULATOR

To run the machining simulator, use the command:

In the 2D version:

Icon	Ribbon
	CNC → Outline → Machining simulation
Keyboard	Textual Menu
	CNC>Machining simulation

After calling the command, the following appears in the 3D scene: the machined part, the machining toolpath and the tool used for machining. Also displayed is a pane in which the control sequence frames are displayed. The current frame processed by the simulator is marked with a blue cursor.



All machining simulation steps are done according to the control sequence. This means that first you need to create the model of the part within the system by drawing or importing supported formats. Next, using various commands in the “CNC” menu, generate a control sequence containing the toolpath.

You can control machining simulation playback using the “control deck”.



You can playback the whole machining process step by step, one motion frame at a time, even in the reverse order.

During the simulation, you can visually assess the toolpath. At the same time, the system conducts automatic verification of the machining toolpath validity.

The standard T-FLEX CAD 3D functions provide the means to control the view points, the light sources and the color scheme of the 3D scene.

If the user works with T-FLEX CAD 2D, then the machining simulation will open in a standalone window, and all manipulations with the view points and 3D scene layout preferences are done using the appropriate buttons on the toolbar and the context menu.

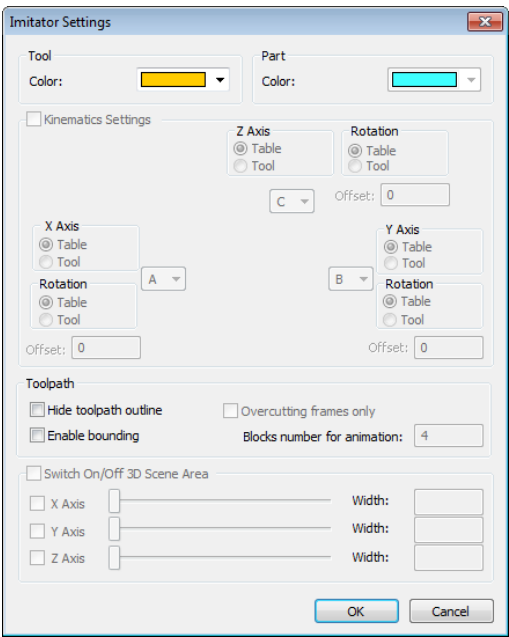
The description of toolbar button functions and context menu options used when running machining simulation in the 2D and 3D versions of the system is given below.

SETTING UP THE MACHINING SIMULATOR (2D VERSION)

To invoke the menu of the simulator’s settings, use the command:

Icon	Ribbon
	CNC → CNC Imitator → Imitator Settings
Keyboard	Textual Menu
	CNC > Imitator Settings

After selection of the command, the dialog window with machining simulator setting parameters will appear on the screen.



List of parameters:

Tool's color – color of the cutting part of the tool (selected from the drop down list of colors).

Part's color – color of the part being machined (selected from the drop down list of colors).

Font's color – the major color of the background (selected from the drop down list of colors).

Smooth transition to a color – additional color of the background. When this flag is enabled, the background will be drawn as a gradual transition from the basic color to the additional one.

Turn off contour of toolpath – when this option is enabled the toolpath of the tool's motion will not be displayed upon simulation.

Turn on cutoff – activate segment simulation of the machining along each of the axes.

Number of frames for animation – specify the number of frames for simulation taking into account the current one.


SETTING UP THE MACHINING SIMULATOR (3D VERSION)

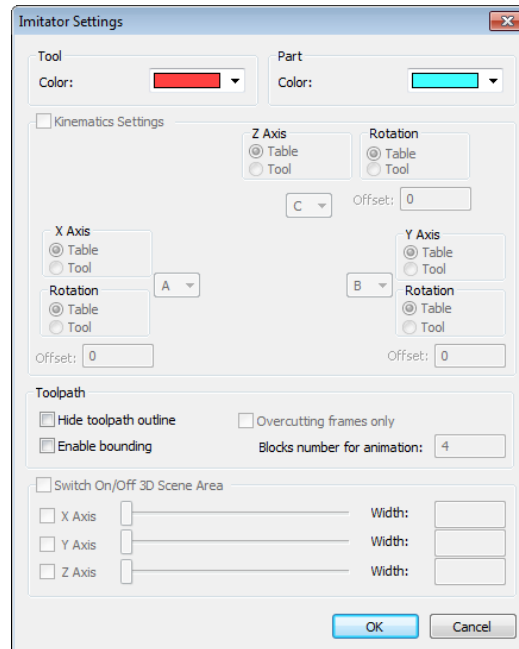
The general settings and control of the view points, of the sources of light and of the color range of the 3D scene is carried out with the help of standard functions of T-FLEX CAD 3D.

Special simulator settings

To access the simulator settings menu, use the command:

Icon	Ribbon
	CNC → CNC Imitator → Imitator Settings
Keyboard	Textual Menu
	CNC > Imitator Settings

When running a simulation of the machining of 2D toolpaths, pressing the  button will make the next dialog window appear:



List of parameters:

Tool's color – color of the cutting part of the tool (selected from the drop down list of colors).


Part's color – color of the part being machined (selected from the drop down list of colors).

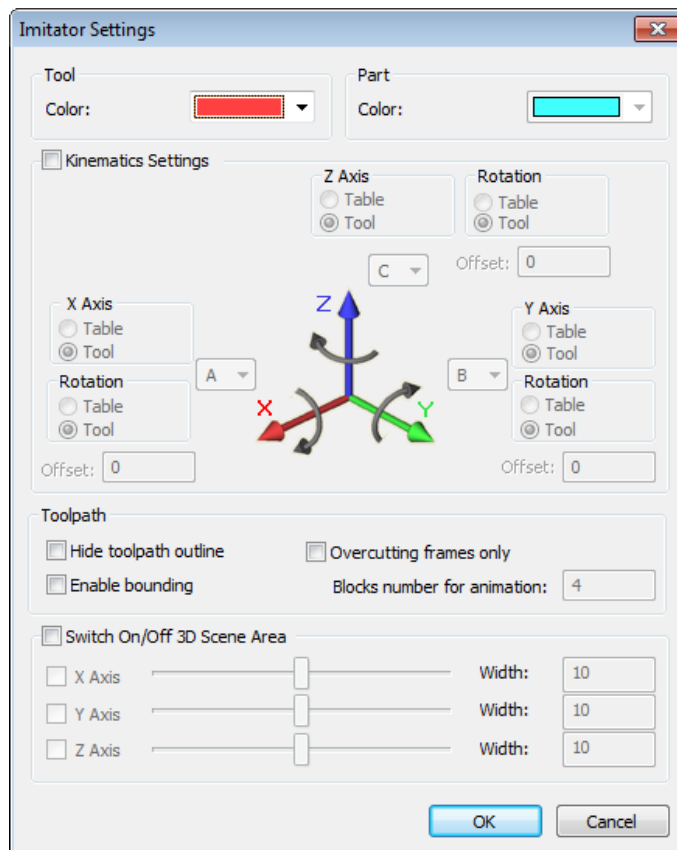
Turn off contour of toolpath – when this option is enabled the toolpath of the tool's motion will not be displayed upon simulation.

Turn on cutoff – activate segment simulation of the machining along each of the axes.

Number of frames for animation – specify the number of frames for simulation starting with the current one.

Furthermore, when the model of the part being machined is present in the 3D scene, it will automatically be uploaded into the 3D scene for simulation.

When running simulation of 3D toolpaths, pressing the  button will cause the dialog window to appear in which, in addition to the parameters described above, the following parameters of simulation will be available:



Only frames of overcuts – simulation of the tool's motion only in the segments of system-evaluated overcuts of the part produced by the tool.

Setting parameters for machining station kinematics:


Setting for translation/rotation of axis (table/tool) – with the help of this parameter the user defines how the translations (rotations) along (around) the selected axis will be carried out: by the table of the machining station or by the tool. For each axis, it is possible to specify the value of the offset of the tool. The user has an option to unfold any of the axes of the coordinate system by specifying, for example, the axis «-A» instead of the axis «A».

When the model of the part being machined is present in the 3D scene, it will still be displayed when running the machining simulation.

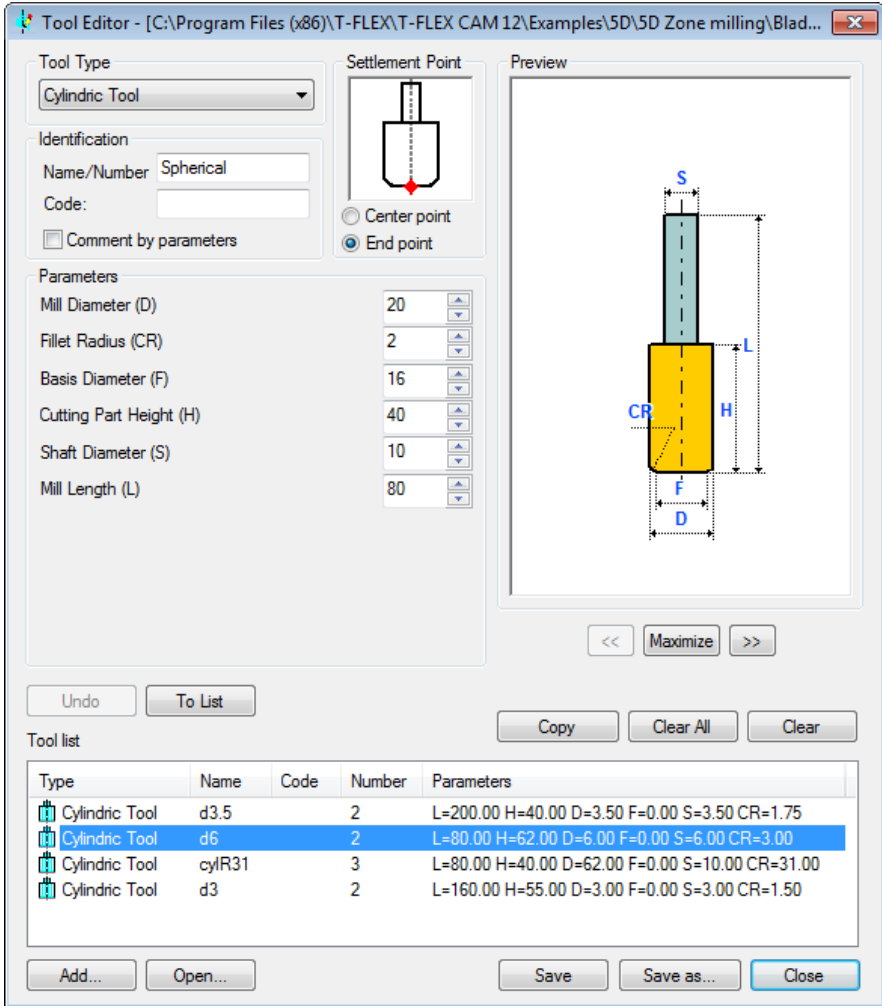
If 2D and 3D toolpaths are present in the same **Machining** (see the «Machining manager» section), they will also be displayed in the 3D version of the simulator.

TOOL EDITOR

The tool editor is provided for designing a cutting tool and creating a database for manufacturing units (for a particular workstation or a complete production line). The tool editor is invoked by the command:

Icon	Ribbon
	CNC → Tools → Tool Editor
Keyboard	Textual Menu
	CNC>Tool Editor

Upon calling the command, the tool editor window appears on the screen. The editor allows creating new tools and editing existing ones. Created tool databases are saved in the files with the extension "*.too".



The list of already existing tools with parameters is displayed at the bottom part of the window. When the editor window opens, the list is empty. A drop-down list of cutting tools available for use is located at the top part of the window.

To create a tool, the user shall select the desired type from the list. As a result, a sketch of the selected tool will be displayed with default parameters. The user remains in this mode throughout the process of designing a new or editing an existing tool. Simply change the parameters available in the window and graphically displayed on the tool sketch. The drawing will dynamically be recalculated according to the user's editions.

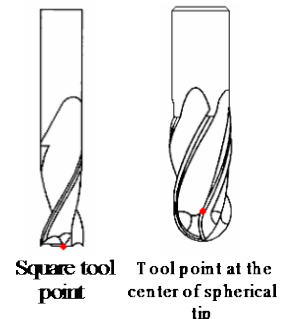
To save the changes done to the tool, the user shall click the **[Apply]** button. If the user is not satisfied with something in the new parameters, one can make a step back by clicking the **[Undo]** button.

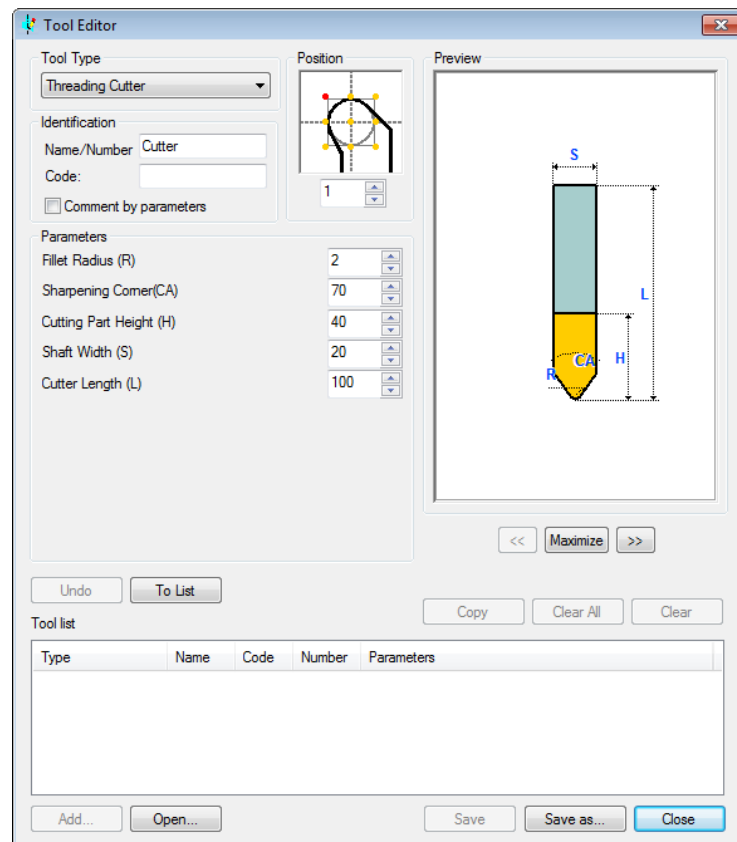
The **"Identification"** group of parameters allows us to modify the default tool assignments of the **"Name/Number"** of the tool settlement point in the tool cartridge of the machining station, **"Location"** and **"Code"**, initially assigned to the tool and saved in the list of tools in the tool editor and, later on, in a specific tool file.

The **"Position"** parameter value in the **"Identification"** group of parameters affects the tool display in "Machining simulator". If the "End point" position is set in the **"Position"** parameter, then the "Machining simulator" drives the tool through the frames of the control sequence with the end-face tool alignment. This type of tool

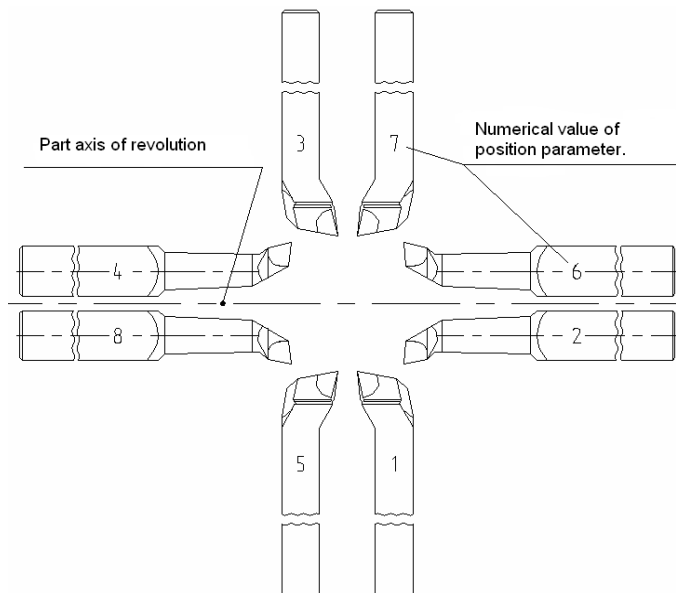
alignment is used for calculating toolpaths in 2D and 3D machining, for cylindrical, spherical, conic, mushroom and other mill types, and, in some cases, for calculating toolpaths in up to five-dimensional machining.

If the "Center point" position is set in the **"Position"** parameter, then the "Machining simulator" drives the tool through the frames of the control sequence with the center-of-sphere tool alignment. This type of tool alignment is used for calculating toolpaths for spherical (ball-nose) mills in 4D and 5D zone machining. With the given **"Position"** parameter value, the attachment point on the cylindrical tool will be shifted by the tool radius.






The “**Position**” parameter value also affects the cutter position in turning-type machining, when viewing the control sequence frames in “Machining simulator”. Eight positions are used for displaying the tool in “Machining simulator”.



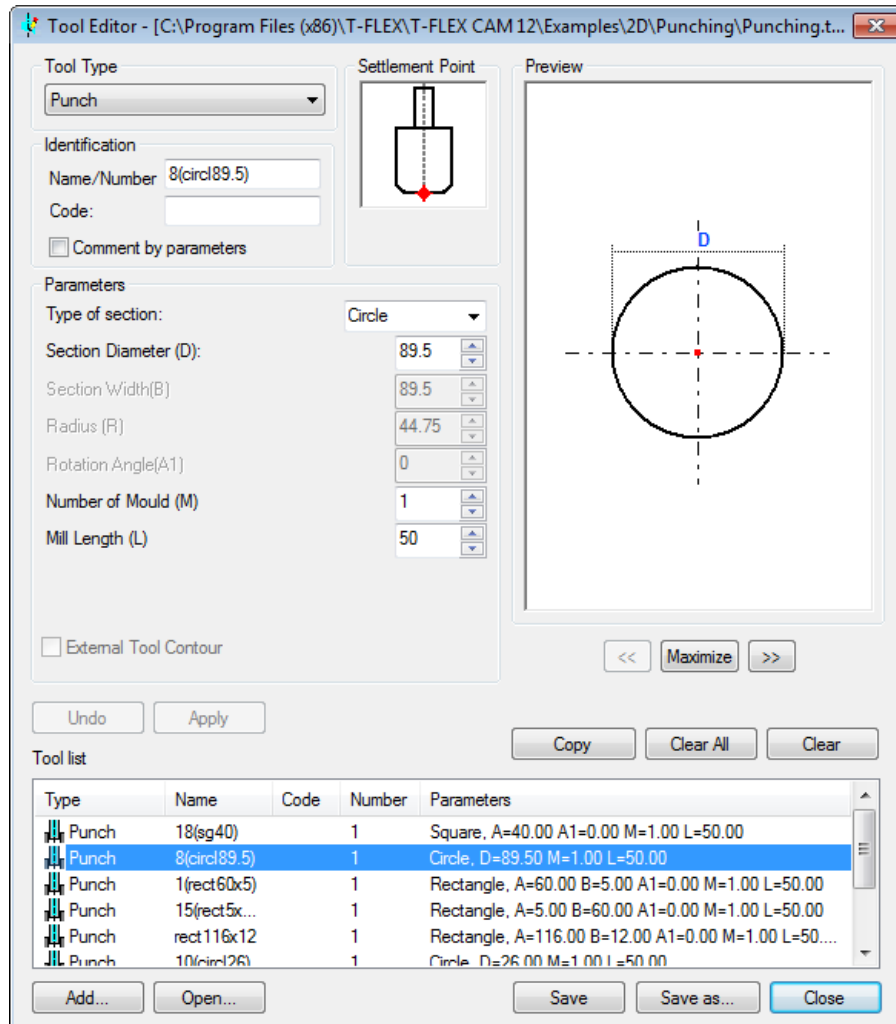
The “**Code**” field allows the user to enter additional information for the tool identification.

It is to be noted that the system automatically assigns the value to the “Position” parameter (by default, it is the «Cutting edge»), and therefore, attention should be paid to the proper selection of this parameter to avoid displaying of the toolpath incorrectly by the machining simulator.

For some types of tools (Punch), you can define arbitrary tool contours. To do this, you need to create a one-contour hatch in the T-FLEX CAD 2D window. Note that the tool attachment point will be represented by the (0, 0) point in the drawing. Next, select the hatch in the drawing and go into the «Tool Editor» . If the hatch was selected successfully, then the “External Contour” option will be provided in the upcoming Tool Editor window.




In this mode, the tool’s contour will be created following the selected hatch contour, and a number of parameters for the current-type tool will then become inaccessible for editing.

The tool created by an external contour is saved into the same tool file. The tool contour can be edited only by redefining the external contour again.




Just like other tools, this tool can be used when calculating a toolpath or when running a visualization in the T-FLEX CAM built-in simulator or in the external T-FLEX NC Tracer simulator.

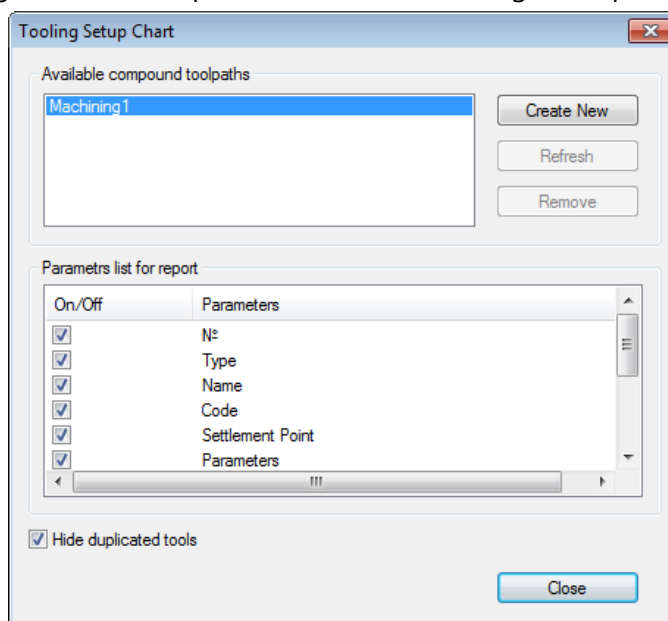
To complete the tool creation, click the **[To List]** button. The created tool will appear in the list of the tool editor.

To navigate in the list of tools, use  or the keyboard keys <↑> and <↓>. To edit a tool, select it in the list and double-click   or press [Enter]. This will bring back the window with the sketch of the selected tool. To delete a selected tool, use the **[Clear]** button. The **[Clear all]** button allows deleting all tools in the list.

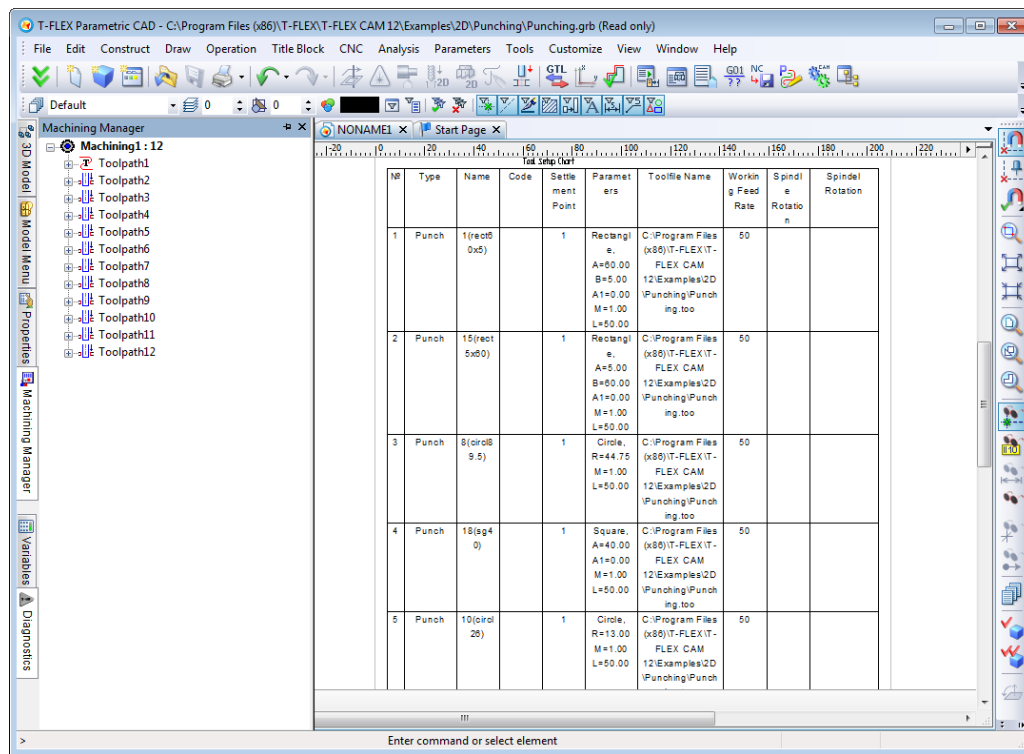
In addition to creating a new tool, the user can add to the current list an earlier designed tool, from another file, by using the **[Add...]** button. The created tools list can be saved to a file by the **[Save...]** button. If changes need to be done to an existing tool file, the file can be opened for editing using the **[Open...]** button.

TOOL SETUP CHART

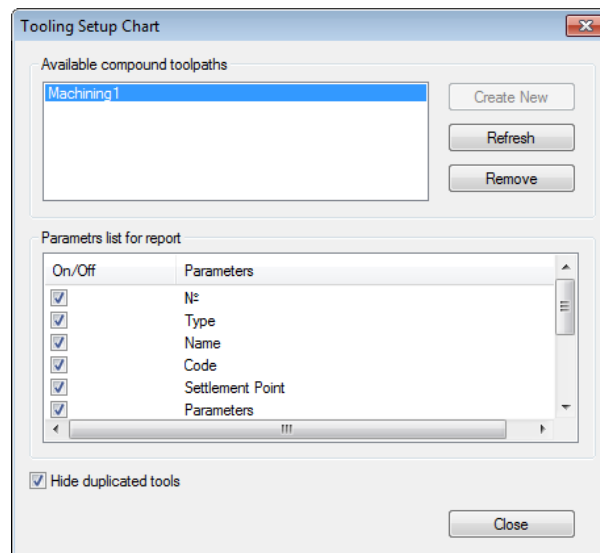
This option serves to create documentation on a tool. Upon clicking the  button of the base module, the system will create a new page in the current T-FLEX CAM document with a template of the tool chart, with all tools used for this machining entered in it. You can create a separate chart for each type of machining. To create a new chart, select a machining type from the list and click the **[Create New]** button. Clicking the button will bring up a window, in which the user will be offered to select the options of the technological chart template, and define the tool parameters to be entered in the template by enabling/disabling the flags in the list of parameters for formatting the report.



By default, the entire tool used for the given machining type will be added to the blank. To create a new chart, it is required to select a desired machining type from the list (if there are several of them) and press the **[Create New]** button. For each type of machining a separate chart can be created. Pressing the button will create in the T-FLEX CAD drawing window an additional page with the name of the machining that contains the blank of technological chart.



After that, the tool chart will be created in the document. After creating the chart, the user may proceed with creating toolpaths for this machining. To edit a chart, the user will have to call the option again, select the desired machining and click the **[Refresh]** button.



If you need to delete a tool chart for a certain machining type from the document, just call this option, select the machining type from the list and click the **[Remove]** button. A setup chart can also be deleted using the **[Delete Page]** command of T-FLEX CAD. Regardless of how the "Tool Setup Chart" option was used, to close the window, click the **[Close]** button.

POSTPROCESSORS

This chapter provides a brief description of commands used in various postprocessors for different types of machining. As was mentioned earlier in the manual, those are the commands that can be edited by the user.


POSTPROCESSOR EDITOR

When saving a control sequence, T-FLEX CAM system uses the modules of postprocessing (postprocessors). These modules allow the user to convert geometrical and technological information from the created by the user trajectory of motion of the tool to the set of G- and M-commands that form the control sequence for the CAM system of the machining station.

T-FLEX CAM allows us to use the table and external postprocessors when saving the control sequence.

T-FLEX CAM offers a default generic postprocessor for all types of machining. In addition, a number of specialized postprocessors are included in the installation for various types of machining. Those are specifically tuned up for supporting such CNC centers as "Power Automation", "FANUC", "Olivetti", etc. Moreover, the user can largely extend the provided set, using the postprocessor Editor.

The Editor is called by the command:

Icon	Ribbon
	CNC → Control → Postprocessor options
Keyboard	Textual Menu
	CNC>Postprocessor options

Upon calling the command, the dialog box "Postprocessor options" is displayed on the screen. This window provides the user with complete environment for editing existing postprocessors and creating custom ones.

A pane in the center of the Editor window displays the list of the current postprocessor commands. By moving through the list, the user can get explanations to commands in the provided pane, the full command name and its code in the active postprocessor. The code can be altered by typing, in the "New" field, the code used in the currently used CNC center.

In addition, the user can change postprocessors for all types of machining supported by T-FLEX CAM system. To do this, first change the machining type in the “**Machining type**” combobox, and then select the currently desired type in the list.

To save a newly developed postprocessor, press the **[Save as]** button, and then specify the name and location of the postprocessor file. We recommend saving postprocessors in the folder ...\\T-FLEX CAM\\ *Postprocessors*\\ *Postprocessors 2D* (*3D*), created at the time of the system installation.

The user can modify an existing postprocessor. To do this, first load the postprocessor file, using the **[Open]** button. If the file opens successfully, the box “**File**” will display the file name. After that, you can do desired editing of the loaded postprocessor, as described above. Once editing is done, just press the **[Save]** button. This will save all changes in the current postprocessor file.

Note that the “**Postprocessor comment**” field tracks the date and time of creation or last editing of the postprocessor.

Additional postprocessor parameters

By enabling the “**Show Extended Parameters**” option, the user can access the dialog serving to setup additional parameters of the postprocessor being created, which provides the greatest flexibility in setting up various postprocessor command combinations. If this option is not used, then any additional parameter values are ignored when saving a control sequence.

The groups of items in the dialog are subject to change depending on the current postprocessor command and the selected machining type. For example, in the case of the wire cutting command G00, the dialog will contain the following groups of commands: “**Coordinate System**”, “**Radius Correction**”, “**Additional Parameters**”, “**Output Format**” and “**Output String Format**”.

Postprocessor Options

Machining type: Wire-cut machining File:

Command remarks:
Travel to programmed point with given velocity (i.e. with rapid feed). Preprogrammed velocity of working feed is ignored, but not canceled. Travel by coordinate axes

Command code: Default: G00 New:

Positioning

Positioning	G00
Linear interpolation	G01
Circular clockwise interpolation	G02
Circular counter-clockwise interpolation	G03
Pause in milliseconds	G04

Additional parameters

Program header: Start number: 0

Program Numbering step: 5

Postprocessor comment
//Developed 10.8.2012 14:1

☐ Show Extended Parameters

☐ Add spaces to output strings in NC

Save as... Save Open Close

Coordinate system

☐ Absolute coordinate system in the first frame

☐ Local coordinate system in the first frame

☐ Working coordinate system in the first frame

Radius correction

☐ Turning on radius correction G41/G42 in the first frame

☐ Turning off radius correction G40 in the last frame

☐ Working plane system in the first frame

Additional parameters


Output format

#,###

Output string format

G00 XYZ

<< >>

By enabling the desired options, the user can have a set of additional parameters written into the row (string) of the control sequence along with the G-code command and coordinates. The obtained string is shown in the “**Output String Format**” field. To change the relative position of elements, use the [<<] and [>>] buttons. To do this, click  on the desired element in the string, and then use the buttons to specify in which direction the element shall be moved. For example, including the absolute coordinate system and the radius correction into the command G00 will produce the line G00 XYZ G90 G41. Moving the output coordinates “XYZ” to the end of the line with the [>>] button will result in G00 G90 G41 XYZ.

Coordinate system

☒ Absolute coordinate system in the first frame

☐ Local coordinate system in the first frame

☐ Working coordinate system in the first frame

Radius correction

☒ Turning on radius correction G41/G42 in the first frame

☐ Turning off radius correction G40 in the last frame

☐ Working plane system in the first frame

Additional parameters

Output format


#,###

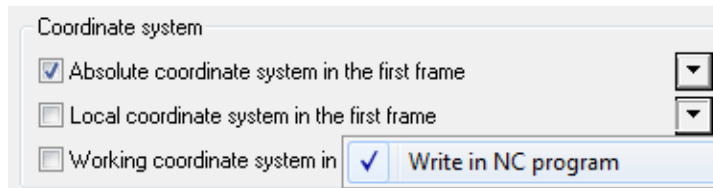
Output string format

G00 XYZ G90 G41

<< >>

For example, if a user-defined machining process contains the command G90 and the command for enabling the left radius correction G41, then those commands will be written to the control sequence together with the G00 command: N5 G00 G90 G41 X13.534 Y15.456.

Some elements in the dialog have extended parameters accessible by clicking the  button. For example, in the case of the absolute coordinate system, there is a **“Write in NC prog”** option. If this option is enabled, then the G90 command will not be written into the control sequence.



The **“Additional Parameters”** field allows you to enter the parameters for the current postprocessor command which are not available in the dialog's list of options. Once a parameter is entered in the **“Output String Format”** field, a new **“PAR”** item shows up.

The **“Output Format”** field displays the format of writing numerical values into the control sequence. This could be coordinate values or other parameters, depending on the current postprocessor command. Shown below is the table of correspondences between the masks and the numerical value output formats.

Mask	Numerical value output format	Keyboard
#	Whole part of a number	Any letter
#,###	Whole and Fraction parts of a number where each “#” character stands for a decimal digit in the output	“.” or “,” to indicate the decimal point
#(,)###	“(,)” means the fixed decimal point. Integers will be outputted in the format “5.”	“(” or “)” to set the fixed decimal point
+ #	“+” sign shall be used with positive numbers	“+” to set, “-” to unset
#*10	The numerical value is multiplied by the entered factor	“*” to set/unset the factor; use digits to specify the factor

Note that, when using a postprocessor saved with additional parameters, the generated control sequence is subject to the following conditions:

- If there is a relative coordinate system, then the coordinates are outputted in increments.
- Duplicate coordinates are not outputted.
- Frames with duplicate coordinates are not outputted.
- Names of G-commands and additional parameters are recorded only when they change.

When cooling is On and several toolpaths follow one after another with the same tool, then cooling will be turned off in the last toolpath.

Special Capabilities of Postprocessor Generator

The postprocessor generator supports macros for defining the format of the numerical output. The user is thus provided with full competence over shaping up the control sequence. The macros for internal representation provide the user with a new tool for customizing T-FLEX CAM system. Macros help define geometry, technology and the topology components of the control sequence, regardless of the part geometry and ways of its machining. The user is fully in control over defining the formatting rules, with the correction of the existing ones or introduction of new rules and laws for the control sequence description. The postprocessor generator is flexible in creating typical postprocessors, so that just a few lines of those need to be edited. Note also, that the user can create a library of postprocessors based on just one custom-built postprocessor. Control sequences can be created by modifying the format of numerical data. The internal numerical data of a control sequence is not limited to geometrical parameters (X, Y, Z, I, J, etc.). It also includes general machining parameters (S, F, T, etc.) and semantics (N, D, etc.).

Purpose and Use of Macros in Postprocessing

Description of the control sequence syntax is complicated by a large number of possible permutations. Adoption of unified rules for defining a postprocessor is challenging even for an experienced user. This takes the user's time and may finally force the user to call technical support (which usually means an extra charge). All of these factors delay a CAM system adoption to a new machining station. Use of macros helps resolve such problems. Not only the user has actual control over the postprocessing procedure, but one can also view each step of building a postprocessor, debug and edit it to perfection. Use of macros in tabulated settings is as simple as the following record: *X%+6:3.3i10*. Such macros describe the full range of numerical values.

Macros Structure and Syntax

<name>%<sign><number of digits before the decimal point>:<number of decimal digits>.<parameter calculation accuracy><parameter type><multiplier>

<name> is a string parameter preceding the numerical value of the calculated variable;

<sign> - sets the sign of the value;

<number of digits before the decimal point> - sets the required number of digits before the decimal point of the parameter being defined;

<number of decimal digits> - sets the required number of digits after the decimal point of the parameter being defined;

<parameter calculation precision> - is the precision of calculating the decimal fraction of the number;

<parameter type> - sets real or integer format of the value;

<multiplier> - the factor for multiplying the raw parameter when making its string representation.

Parameters Table

Parameters	Applicable for, depending on parameter type	Admissible values and brief description
Name	Unrestricted	Any character string that you wish to put before the numerical parameter
Sign	For f and i	"+" for defining the sign; use when you need to explicitly show the parameter sign
Number of digits before decimal point	For f and i	Any positive integer; use when you need to explicitly specify the number of digits in the whole part of the parameter, padded with zeros, if necessary
Number of decimal digits	For f	Any positive integer; use when you need to explicitly specify the number of decimal digits of the parameter, padded with zeros, if necessary
Parameter calculation precision	For f	A positive integer less than 15 (mathematical calculation precision); use when you need to round the fractional part of the parameter
Parameter type	For f and i	Use the character "f" or "i"; defines the parameter format type: integer – "i" or real – "f"
Multiplier	For f and i	An arbitrary integer; use when you need to scale the raw parameter before writing it out as a character string

Note that all above-listed parameters are optional when defining macros. You can use just the necessary parameters when defining macros, in any combination. When inserting macros in the tabulated settings of a postprocessor, take a notice of the default parameters listed below.

Default Macros Parameters

All macros entered in the tabulated settings of a postprocessor fit the following format: <parameter name>%.28f1, where <parameter name> is copied from the respective table entry. This formatting means that the input numerical parameter will be converted to a character string by the following rules:

- (i) the sign is written only for the values less than 0;
- (ii) only the whole part of the number is written, without padding with zeros;
- (iii) the decimal point and the fraction part are written only if they actually exist (based on the maximum precision);
- (iv) the output value is unchanged, as the multiplier is 1.

This format for numerical parameters in a control sequence is supported by all modern CNC centers. Here follows an example of a control sequence using this formatting:

N5T10

N400G00X10Y15.758

N405G01Z-26.1

N410G01X10.129Y18.02

Macros Examples and Their Impact on Control Sequence Appearance

Let's review the following sample task of describing a machining contour.

The tool sweeps the contour in the clockwise direction, beginning from the start point (-50, -40).

Example 1. The control sequence generated by default:

N20G00X-50Y-40

N25G01X-50Y60

N30G01X15Y60

N35G02X30Y45I15J45

N45G01X30Y-40

N50G01X-50Y-40

Example 2. The control sequence generated with the following macros settings:

- a) the "Line entry" – N%3i1;
- b) the "X-axis entry" – X%+4:2f10;
- c) the "Y-axis entry" – Y%+4:2f10;
- d) the "I-axis entry" – I%+4:2f10;
- b) the "J-axis entry" – J%+4:2f10, resulting in the following fragment:

N020G00X-500.00Y-400.00

N025G01X-500.00Y+600.00

N030G01X+150.00Y+600.00

N035G02X+300.00Y+450.00I+150.00J+450.00

N045G01X+300.00Y-400.00

N050G01X-500.00Y-400.00

Example 3. The control sequence generated with the following macros settings:

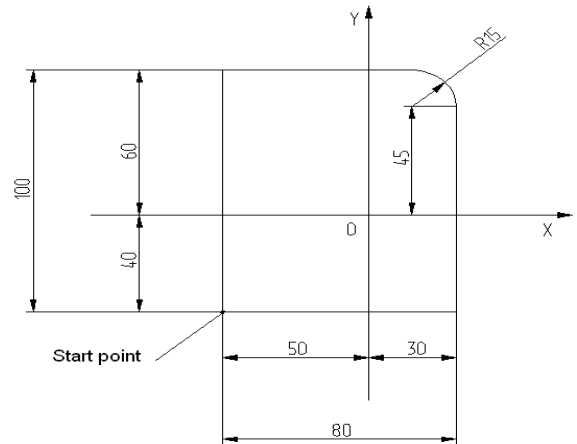
- a) the "Line entry" – N%3i1;
- b) the "X-axis entry" – X%+i100;
- c) the "Y-axis entry" – Y%+i100;
- d) the "I-axis entry" – I%+i100;
- e) the "J-axis entry" – J%+i100, resulting in the following fragment:

N020G00X-5000Y-4000

N025G01X-5000Y+6000

N030G01X+1500Y+6000

N035G02X+3000Y+4500I+1500J+4500



N045G01X+3000Y-4000

N050G01X-5000Y-4000

Predefined Postprocessors

Postprocessor for Wire Cutting

The specific wire-cutting commands are tuned by default to "DIPOL" CNC center, widely used with machining stations. Numerous special commands are also tuned for Russian CNC center brands "15 IPI-3-001" and "2M-43".

The default set of commands used in control sequences can be divided into preparatory commands and auxiliary commands.

Preparatory Commands

The names of preparatory commands begin with letter G, followed by two digits defining the command type. Additional commands could be instructions defining a particular mode of the control system. Here follows a detailed description of preparatory commands.

Rapid positioning command G00

This command positions the cutting wire at the point with specified coordinates at the rapid feed rate, without cutting. A specified maximum velocity is used for the rapid feed rate.

G00 command format:

Nn G00XxYy or Nn G00XxYyFf, where

- Nn – the number of the line in the sequence,
- X and Y are the coordinates of the target position,
- x and y are the respective distances to the target, in mm,
- F defines the feed rate,
- f – sets the numerical value of the feed velocity, in mm/minute.

Linear interpolation command G01

In the case of the linear interpolation, the motion occurs along a straight line, with constant velocity.

G01 command format:

Nn G01XxYy, where

- Nn – the number of the line in the sequence,
- X and Y are the coordinates of the target position,
- x and y are the respective distances to the target, in mm.

Circular interpolation commands G02, G03

The command G02 is used for programming motion along a circular arc, in clockwise direction from the start point. The command G03 defines motion in the counterclockwise direction.

G02 command format:

Nn G02XxYyliJj, where

Nn – the number of the line in the sequence,

X and Y are the coordinates of the target point on the arc,

I and J are the coordinates of the circle center, relative to the start point on the arc.

G02, G03 command parameters

Absolute coordinates	Relative coordinates
$X = x_1$	$X = x_1 - x_0$
$Y = y_1$	$Y = y_1 - y_0$
$I = x_c - x_0$	$I = x_c - x_0$
$J = y_c - y_0$	$J = y_c - y_0$

Pause command G04

The command G04 interrupts the motion with respect to the reference coordinates. It neither turns off the operating voltage nor winds the wire. The motion with respect to the reference axes will resume after the pausing specified by the parameter X.

G04 command format:

Nn G04Xx, where

Nn – the number of the line in the sequence,

X – pause duration, in milliseconds.

Parabolic (spline) interpolation command G06

The command G06 is used for programming motion along a second-degree parabolic arc, from the start point toward the target point. The intersection point of the endpoint tangency lines is entered as the second parameter of the G06 command.

G06 command format:

Nn G06XxYyliJj, where

Nn – the number of the line in the sequence,

X and Y are the coordinates of an intermediate point on the parabola,

I and J are the coordinates of the target point on the arc.

G06 command parameters

Absolute coordinates	Relative coordinates
$X = x_t$	$X = x_t - x_0$
$Y = y_t$	$Y = y_t - y_0$
$I = x_2$	$I = x_2 - x_0$
$J = y_2$	$J = y_2 - y_0$

Command for swapping axes X and Y, G08

This command swaps the X and Y coordinates in the control sequence. This makes a contour mirrored from the original one about the centerline between the X and Y axes.

G08 command format:

Nn G08, where Nn – the number of the line in the sequence.

The command G09 voids G08 command.

In the cases of generating complex shaping (the commands G23 and G51), the axes swapping command works on parameter axes U and V as well. Note that the use of the command G08 causes automatic replacement of G41 by G42 and, vice versa, G42 by G41, in order to maintain the side of contour traversal.

Command of coordinate axes rotation by an arbitrary angle, G14

The command for rotating coordinate axes by an arbitrary angle, G14, realizes rotation of the X-Y plane. In the cases of generating complex shaping, it also handles the U-V plane.

The rotation functionality makes a separate line in the control sequence.

G14 command format:

Nn G14QqliJ, where

Nn – the number of the line in the sequence,

I and J are the center of coordinates rotation.

Q – rotation angle.

I and J define the center of coordinates rotation, I in X direction, J – in Y. The rotation angle is identified by Q, with the precision 0.001°. The positive direction is counterclockwise rotation from the X-axis in the X-Y plane.

Scaling commands G20, G21, G22

The scaling commands make separate lines in the control sequence. Those simultaneously affect all coordinates.

G20 and G22 command format:

Nn G20Ss, Nn G22Ss, where

Nn – the number of the line in the sequence,

S – the scale factor written out as an integer, with the decimal point assumed to be between the last second and third positions.

The admissible range of the parameter S in the command G20 (scaling down) is from 1 to 99. This corresponds to the actual scale in the range from 0.01 to 0.99. The admissible range of the parameter S in the command G22 (scaling up) is from 1 to 9999. This corresponds to the actual scale in the range from 1.0 to 99.99. The scaling commands are voided by the command G21, simultaneously for all coordinates.

G23 and G24 command format

The command G23 enables CNC generation in the complex shaping mode. In this mode, the bottom (base) and the top (secondary) cutting contours are defined (programmed) separately. The number of base contour elements must be equal to the number of elements in the secondary contour. Use of two contours is required when the base and the secondary contours contain matches of different element types. This could be, for example, pairs of line-arc, or two opposite-directed arcs. This command is also used in the case when the dimensions of the top and bottom contours are known, while the tilt angle of the cutting wire is unknown in some positions.

The command G23 makes the upper guide of the wire cutter move in the U-V coordinate plane with respect to the part, while the bottom one in the X-Y plane. In this case, the specified in the control sequence cutting feed rate defines the motion in X-Y plane. The feed rate in U-V plane is calculated based on the ratio of matching element lengths in the X-Y and U-V planes.

G23 command format:

Nn G23, where Nn is the number of the line in the sequence.

The command G24 voids the command G23.

After the insertion point of G23 command in the control sequence, the further motion-defining lines must simultaneously turn on the motion commands in the bottom and the top planes.

The compensation (correction) commands for the radius of the wire cutter G41, G42, G40

G41, G42 and G40 command format:

Nn G41Dd – the left offset contour,

Nn G42Dd – the right offset contour,

Nn G40 – this command cancels offsetting, where

Nn – the number of the line in the sequence,

Dd – the number of the compensation, i.e. the number of the entry row in the table of compensations.

Angle cutting commands G50, G51

The command G51 makes the upper guide of the wire cutter (in the U-V coordinate plane) shift in the plane perpendicular to the toolpath. In this way, shift to the right corresponds to negative values, positive

to the left. The command G50 voids the command G51. The latter command restores the upper guide in the original position, making the cutting wire vertical.

G50 and G51 command format:

Nn G50, Nn G51Aa, where

Nn – the number of the line in the sequence,

A – the tilt angle parameter of the wire cutter,

a – sets the numerical value of tilt angle, in degrees.

Commands for setting up working coordinate systems, G54-G59

These commands modify the reference point (the coordinate system origin) of the current coordinates of the wire cutter.

G54 and G59 command format:

Nn G54, Nn G59, where Nn is the number of the line in the sequence.

A wire-cutting machine “zero” commonly refers to the origin of the machine coordinate system. The coordinate systems whose origins are shifted off the machine zero are called “working coordinate systems”.

Commands for enabling/disabling adaptation of cutting clearance, G67, G68

The commands are used in clean-up processes for controlling the adaptation of cutting clearance. The command G67 enables the mode, while the command G68 disables it.

G67 and G68 command format:

Nn G67, Nn G68, where Nn is the number of the line in the sequence.

Command for setting the target position in absolute coordinates, G90

This command sets position calculation with respect to the origin of the current coordinate system.

G90 command format:

Nn G90, where Nn is the number of the line in the sequence.

Command for setting the target position in increments, G91

This command sets position calculation from the current position by increments in the respective coordinates.

G91 command format:

Nn G91, where Nn is the number of the line in the sequence.

G92 command

The command sets up the new working coordinate system with the origin shifted off the absolute coordinate system.

G92 command format:

Nn G92, where Nn is the number of the line in the sequence.

Commands for mirroring about coordinate axes, G93, G94, G95 and G96

Some systems support mirroring in one or two coordinates simultaneously, about the specified axes. This affects primarily the moves defined by the commands G00, G01, G02, G03 and G06.

The command G93 mirrors the direction of a move along the X-axis.

The command G94 mirrors the direction of a move along the Y-axis.

The command G95 mirrors a bi-directional simultaneous move in X and Y.

The command G96 voids the commands G93, G94, G95.

The format of those commands does not include any parameters.

When mirroring one or more coordinates, keep in mind the following rules:

- the signs of all mirrored coordinates are flipped;
- when mirroring just one of the two axes in the plane of circular interpolation, the traversal direction is flipped. For example, mirroring only X in the X-Y plane would alter G02 to G03, G03 to G02;
- when mirroring two axes in the plane of circular interpolation, the arc traversal direction stays unchanged (G02 stays G02, G03 stays G03);
- when mirroring just one of the two axes in the plane of the offset, the arc traversal direction (left or right) is flipped (G41 switches to G42, G42 - to G41);
- when mirroring two axes in the plane of the offset, the arc traversal direction (left or right) stays unchanged (G41 and G42 stay unchanged);

Auxiliary Commands

The names of auxiliary commands begin with letter M, followed by two digits defining the command type. Auxiliary commands control the modes of CNC center operation. Here follows a detailed description of auxiliary commands.

Programmed stop command M00

The command M00 can be inserted anywhere within the line in which the program stop is intended, or in a separate line following right after. The command M00 stops motion in both coordinates.

M00 command format:

Nn M00, where Nn is the number of the line in the sequence.

Stop with confirmation command M01

The command M01 must be placed within the line where the stop with confirmation is intended. Before executing the line, the system will display a query for the operator to confirm stopping of the machine. Upon the confirmation, the system processes all commands in the line, and then stops the motion.

M01 command format:

Nn M01, where Nn is the number of the line in the sequence.

End of program command M02

The line with the command M02 is the last command in a control sequence. It completes the control sequence of a given technological process. The command M02 can be placed in a separate line or inserted at the end of the last line in the control sequence. Any command that follows the command M02 is ignored by the CNC IMT208SOFT system and other systems, and won't be processed.

M02 command format:

Nn M02, where Nn is the number of the line in the sequence.

Operating voltage generator On command M10

M10 command format:

Nn M10Ss, where

Nn – the number of the line in the sequence,

Ss – the command parameter defining the number of the entry row in the table of generator states.

Operating voltage generator Off command M11

This command voids the command M10 (turns Off the voltage generator).

M11 command format:

Nn M11, where Nn is the number of the line in the sequence.

End of subroutine command M17

This command completes the current subroutine and returns control to the line of the main control sequence that follows the subroutine call line.

M17 command format:

Nn M17, where Nn is the number of the line in the sequence.

End of cycle command M20

This command defines the end of a cycle in a process control sequence.

M20 command format:

Nn M20, where Nn is the number of the line in the sequence.

End of control sequence file command M30

This command marks the end of the control sequence file and is used in combination with the command M02.

M30 command format:

Nn M30, where Nn is the number of the line in the sequence.

Feed rate scaling commands M36, M37

The command M37 reduces the feed rate defined by the parameter F, by 100 times. The command M36 voids the command M37.

M36 and M37 command format:

Nn M36, Nn M37, where Nn is the number of the line in the sequence.

Dielectric system control commands, M48, M49

M48 command format:

Nn M48Ss, where

Nn – the number of the line in the sequence,

Ss – the command parameters defining the dielectric system state.

The S parameter values and their respective states of operation with filling the reservoir are as follows:

- 0 – switches off the high- and low-pressure flushing valves;
- 1 – switches on the high-pressure flushing valve;
- 2 – switches on the low-pressure flushing valve;
- 3 – switches on both flushing valves;

The s parameter values and their respective states of operation without filling the reservoir are as follows:

- 4 – switches off the high- and low-pressure flushing valves;
- 5 – switches on the high-pressure flushing valve;
- 6 – switches on the low-pressure flushing valve;
- 7 – switches on both flushing valves.

Executing the command M48 with the S parameter values from 0 to 7 is accompanied in some systems by automatic turning on of the circulation pump and the flushing pump, if those were not already turned on. When operating with reservoir filling, turning on the pumps also turns on the reservoir filling valve.

Additional values of the parameter S:

- 8 – switches on all valves and pumps;
- 9 – switches off all valves and pumps, except for the circulation pump.

Wire winding On/Off commands M50, M51

The command M51 voids the command M50. Wire winding stops, and wire tension is switched from operational to idle (shade), that is maintained while CNC is off.

M50 and M51 command format:

Nn M50Ss, Nn M51, where

Nn – the number of the line in the sequence,

Ss – the number of the entry row in the table of winding states.

The entries in the table of winding states specify velocity in meters/minute, tension - in Newtons (N).

Command for canceling cutting clearance adaptation, M54

The command M54 voids the command G67, canceling cutting clearance adaptation mode.

M54 command format:

Nn M54, where Nn is the number of the line in the sequence.

Acceleration/Deceleration ban commands M55, M56

The command M55 sets a ban on accelerations/decelerations along X, Y and U, V axes.

The command M56 voids the command M55, removing the ban on accelerations/decelerations.

M55 and M56 command format:

Nn M55, Nn M56, where Nn is the number of the line in the sequence.

Postprocessor for Laser Cutting

The default set of commands used in control sequences can be divided into the preparatory commands and auxiliary commands.

Preparatory Commands

The names of preparatory commands begin with letter G, followed by two digits defining the command type. Additional commands could be instructions defining a particular mode of the control system. Here follows a detailed description of preparatory commands. (Note that the command format has been described in the "Postprocessor for wire cutting " section).

Rapid positioning command G00

This command moves the tool to the programmed point with a specified velocity (such as the maximum feed rate). The preprogrammed velocity of the working feed is ignored, but not canceled. The components of the move by the coordinate axes could be uncoordinated. Example: *N10 G01 X10 Y25*.

Linear interpolation command G01

This command moves the tool along a straight line at a specified rate (in an orthogonal coordinate system). In this case, a constant ratio must be maintained between the velocity components by the coordinate axes. The ratio should be equal to that of the tool move components by the coordinate axes. Example: *N10 G01 X10 Y25*.

Circular interpolation commands G02, G03

The commands define a circular interpolation. The tool motion direction can be clockwise when watched from the positive normal side of the machined surface, or counterclockwise. Motion is done with a programmed feed rate. Example: *N10 G02 X10 Y25 I5 J2* or *N10 G03 X10 Y25 I5 J2*.

Pause command G04

Pauses machining for the duration specified in the line of the control sequence, in milliseconds. Example: *N20 G04 X1000*.

Command for swapping axes X and Y, G08

Swaps X and Y-axis. The command G09 voids G08 command.

Command of coordinate axes rotation by an arbitrary angle, G14

Rotates coordinate axes by an arbitrary angle. The command G15 voids the command G14.

Scaling commands G20, G21, G22

The scaling commands make separate lines in the control sequence. Those simultaneously affect all coordinates. The admissible range of the parameter S in the command G20 (scaling down) is from 1 to 99. This corresponds to actual scaling in the range from 0.01 to 0.99. The admissible range of the parameter S in the command G22 (scaling up) is from 1 to 9999. This corresponds to actual scaling in the range from 1.0 to 99.99. The command G21 voids scaling commands, simultaneously for all coordinates.

The compensation (correction) commands for the tool radius G41, G42, G40

The command G41 sets the left correction for the tool radius (example: *N100 G41 D4*). The command G42 sets the right correction for the tool radius (example: *N100 G42 D4*). The command G40 cancels left and right correction for the tool radius (example: *N100 G40*).

The commands for setting up working coordinate systems, G54-G59

These commands modify the reference point (the coordinate system origin) of the current coordinates of the tool. The commands set working coordinate systems enumerated from 0 to 5, respectively.

Command for setting the target position in absolute coordinates, G90

This command sets position calculation with respect to the origin of the current coordinate system.

Command for setting the target position in increments, G91

This command sets position calculation from the current position by increments in the respective coordinates.

G92 command

The command sets up the new working coordinate system with the origin shifted off the absolute coordinate system.

The commands for mirroring about coordinate axes, G93, G94, G95 and G96

Some systems support mirroring in one or two coordinates simultaneously, about the specified axes. This affects primarily the moves defined by the commands G00, G01, G02, G03 and G06.

The command G93 mirrors the direction of a move along the X-axis.

The command G94 mirrors the direction of a move along the Y-axis.

The command G95 mirrors a bi-directional simultaneous move in X and Y.

The command G96 voids the commands G93, G94, G95.

The format of these commands does not include any parameters.

When mirroring one or more coordinates, keep in mind the following rules:

- The signs of all mirrored coordinates are flipped;
- When mirroring just one of the two axes in the plane of circular interpolation, the traversal direction is flipped. For example, mirroring only X in the X-Y plane would alter G02 to G03, G03 to G02;
- When mirroring two axes in the plane of circular interpolation, the arc traversal direction stays unchanged (G02 stays G02, G03 stays G03);
- When mirroring just one of the two axes in the plane of the offset, the arc traversal direction (left or right) is flipped (G41 switches to G42, G42 - to G41);
- When mirroring two axes in the plane of the offset, the arc traversal direction (left or right) stays unchanged (G41 and G42 stay intact);

Auxiliary Commands

The names of auxiliary commands begin with letter M, followed by two digits defining the command type. Auxiliary commands control the modes of CNC center operation. Here follows a detailed description of auxiliary commands. (Note that the command format has been described in the "Postprocessor for wire cutting " section).

The program stop command M00

The command M00 can be inserted anywhere within the line in which the program stop is intended, or in a separate line following right after. The command M00 stops motion by both coordinates.

Stop with confirmation command M01

The command M01 must be placed within the line where the stop with confirmation is intended. Before executing the line, the system will display a query for the operator to confirm stopping of the machine. Upon the confirmation, the system processes all commands in the line, and then stops the motion.

Program stop cancellation command M02

The line with the command M02 is the last command in a control sequence. It completes the control sequence of a given technological process. The command M02 can be placed in a separate line or inserted at the end of the last line in the control sequence. Any command that follows the command M02 is ignored by the CNC system and won't be processed.

Operating voltage generator command M10

Sets the number of the generator state. The command turns the generator on in the specified mode. Example: *N100 M10 S2*.

Operating voltage generator Off command M11

This command cancels the command M10 (turns Off the voltage generator).

End of control sequence file command M30

This command marks the end of the control sequence file and is used in combination with the command M02.

Feed rate scaling commands M36, M37

The command M37 reduces the feed rate defined by the parameter F, by 100 times. The command M36 voids the command M37.

Acceleration/Deceleration ban commands M55, M56

The command M55 sets a ban on accelerations/decelerations along X, Y and U, V axes.

The command M56 voids the command M55, removing the ban on accelerations/decelerations.

Postprocessor for Turning

The default set of commands used in control sequences can be divided into the preparatory commands and auxiliary commands.

Preparatory Commands

The names of preparatory commands begin with letter G, followed by two digits defining the command type. Additional commands could be instructions defining a particular mode of the control system. Here follows a detailed description of preparatory commands. (Note that the command format has been described in the "Postprocessor for wire cutting " section).

Rapid positioning command G00

This command moves the tool to the programmed position with a specified velocity (such as the maximum feed rate). The preprogrammed velocity of the working feed is ignored, but not canceled. Travel by coordinate axes could be uncoordinated, for example: *N10 G01 X10 Y25*.

Linear interpolation command G01

This command moves the tool along a straight line at a specified rate (in an orthogonal coordinate system). In this case, a constant ratio must be maintained between the velocity components by the coordinate axes. The ratio should be equal to that of the tool move components by the coordinate axes. Example: *N10 G01 X10 Y25*.

Circular interpolation commands G02, G03

The commands define a circular interpolation. The tool motion direction can be clockwise when viewed from the positive normal side of the machined surface, or counterclockwise. Motion is done with a programmed feed rate. Example: *N10 G02 X10 Y25 I5 J2* or *N10 G03 X10 Y25 I5 J2*.

Pause command G04

Pauses machining for the duration specified in the line of the control sequence, in milliseconds. Example: *N20 G04 X1000*.

Command for setting the target position in absolute coordinates, G90

This command sets position calculation with respect to the origin of the current coordinate system.

Command for setting the target position in increments, G91

This command sets position calculation from the current position by increments in the respective coordinates.

Auxiliary Commands

The names of auxiliary commands begin with letter M, followed by two digits defining the command type. Auxiliary commands control the modes of CNC center operation. Here follows a detailed description of auxiliary commands. Note that the command format has been described in the "Postprocessor for wire cutting " section.

The program stop command M00

The command M00 can be inserted anywhere within the line in which the program stop is intended, or in a separate line following right after. The command M00 stops motion by both coordinates.

Stop with confirmation command M01

The command M01 must be placed within the line where the stop with confirmation is intended. Before executing the line, the system will display a query for the operator to confirm stopping of the machine. Upon the confirmation, the system processes all commands in the line, and then stops the motion.

Program stop cancellation command M02

The line with the command M02 is the last command in a control sequence. It completes the control sequence of a given technological process. The command M02 can be placed in a separate line or inserted at the end of the last line in the control sequence. Any command that follows the command M02 is ignored by the CNC system and won't be processed.

Commands for turning on spindle rotation with specified RPM, M03, M04

The commands turn on spindle rotation at a specified RPM, clockwise and counterclockwise, respectively.

Cooler On command M07

The command turns on the cooling pump. Example: *N100 M07 S2*, where S is a cooling system parameter.

The command M08 voids the command M07, turning off the cooling pump.

Spindle Off command M11

Turns off spindle rotation.

End of control sequence file command M30

This command marks the end of the control sequence file and is used in combination with the command M02.

Acceleration/Deceleration ban commands M55, M56

The command M55 sets a ban on accelerations/decelerations along X, Y and U, V axes.

The command M56 voids the command M55, removing the ban on accelerations/decelerations.

Postprocessor for Drilling

The default set of commands used in control sequences can be divided into the preparatory commands and auxiliary commands.

Preparatory Commands

The names of preparatory commands begin with letter G, followed by two digits defining the command type. Additional commands could be instructions defining a particular mode of the control system. Here follows a detailed description of preparatory commands. Note that the command format has been described in the "Postprocessor for wire cutting " section.

Rapid positioning command G00

This command moves the tool to the programmed position with a specified velocity (such as the maximum feed rate). The preprogrammed velocity of the working feed is ignored, but not canceled. Travel by coordinate axes could be uncoordinated, for example: *N10 G01 X10 Y25*.

Linear interpolation command G01

This command moves the tool along a straight line at a specified rate (in an orthogonal coordinate system). In this case, a constant ratio must be maintained between the velocity components by the coordinate axes. The ratio should be equal to that of the tool move components by the coordinate axes. Example: *N10 G01 X10 Y25*.

Circular interpolation commands G02, G03

The commands define a circular interpolation. The tool motion direction can be clockwise when viewed from the positive normal side of the machined surface, or counterclockwise. Motion is done with a programmed feed rate. Example: *N10 G02 X10 Y25 I5 J2* or *N10 G03 X10 Y25 I5 J2*.

The pause command G04

Pauses machining for the duration specified in the line of the control sequence, in milliseconds. Example: *N20 G04 X1000*.

Skip line prefixed by </> command G11, G12

The command G11 instructs skipping lines prefixed with </>, when executing the control sequence. The command G12 voids the command G11.

Command for setting the target position in absolute coordinates, G90

This command sets position calculation with respect to the origin of the current coordinate system.

Command for setting the target position in relative coordinates, G91

This command sets position calculation from the current position by increments in the respective coordinates.

Auxiliary Commands

The names of auxiliary commands begin with letter M, followed by two digits defining the command type. Auxiliary commands control the modes of CNC center operation. Here follows a detailed description of auxiliary commands. (Note that the command format has been described in the "Postprocessor for wire cutting " section).

The program stop command M00

The command M00 can be inserted anywhere within the frame in which the program stop is intended, or in a separate frame following right after. The command M00 stops motion in both coordinates.

Stop with confirmation command M01

The command M01 must be placed within the line where the stop with confirmation is intended. Before executing the line, the system will display a query for the operator to confirm stopping of the machine. Upon the confirmation, the system processes all commands in the line, and then stops the motion.

Program stop cancellation command M02

The line with the command M02 is the last command in a control sequence. It completes the control sequence of a given technological process. The command M02 can be placed in a separate line or inserted at the end of the last line in the control sequence. Any command that follows the command M02 is ignored by the CNC system and won't be processed.

Commands for turning on spindle rotation with specified RPM, M03, M04

The commands turn on spindle rotation at a specified RPM, clockwise and counterclockwise, respectively.

Cooler On command M07

The command turns on the cooling pump. Example: *N100 M07 S2*, where S is a cooling system parameter.

The command M08 voids the command M07, turning off the cooling pump.

Spindle Off command M11

This command turns off spindle rotation.

End of control sequence file command M30

This command marks the end of the control sequence file and is used in combination with the command M02.

Postprocessor for 2-2.5D Milling

The default set of commands used in control sequences can be divided into the preparatory commands and auxiliary commands.

Preparatory Commands

The names of preparatory commands begin with letter G, followed by two digits defining the command type. Additional commands could be instructions defining a particular mode of the control system. Here follows a detailed description of preparatory commands. (Note that the format of some commands has been described in the "Postprocessor for wire cutting " section).

Rapid positioning command G00

This command moves the tool to the programmed position with a specified velocity (such as the maximum feed rate). The preprogrammed velocity of the working feed is ignored, but not canceled. Travel by coordinate axes could be uncoordinated, for example: *N10 G01 X10 Y25*.

Linear interpolation command G01

This command moves the tool along a straight line at a specified rate (in an orthogonal coordinate system). In this case, a constant ratio must be maintained between the velocity components by the coordinate axes. The ratio should be equal to that of the tool move components by the coordinate axes. Example: *N10 G01 X10 Y25*.

Circular interpolation commands G02, G03

The commands define a circular interpolation. The tool motion direction can be clockwise when viewed from the positive normal side of the machined surface, or counterclockwise. Motion is done with a programmed feed rate. Example: *N10 G02 X10 Y25 I5 J2* or *N10 G03 X10 Y25 I5 J2*.

The pause command G04

Pauses machining for the duration specified in the line of the control sequence, in milliseconds. Example: *N20 G04 X1000*.

Spline type definition command G05

This command allows the user to specify the type of spline to be used in machining.

The following spline types are supported: Natural spline; spline with start tangency condition; spline with end tangency condition; spline with start and end tangency conditions.

Spline interpolation command G06

The command turns on the mode of using spline interpolation in machining. Example: *N10 G06 X... Y...*

Tangency interpolation command G07

The command turns on the mode of using tangency interpolation in machining. This means, transition from a point specified in the control sequence to the next point will be done along a circular arc. Example: *N10 G07 X... Y...*

Circular interpolation with specified radius commands, G12 and G13 (clockwise and counterclockwise, respectively)

The command turns on the mode of circular interpolation by an arc of specified radius, clockwise or counterclockwise. Example: *N10 G13 X... Y... K...*

Commands for defining next point in absolute or relative polar coordinates, G14 and G15

These commands prescribe the use of absolute or relative polar coordinates.

Polar coordinate system center definition command G16

Defines the center of a polar coordinate system. Example: *10 G16 X... Y...*, where X is the angle, and Y is the radius.

Working plane selection commands G17, G18, G19

These commands allow selecting the desired working plane among the X-Y, Z-X and Y-Z planes.

Free working plane definition command G20

This command defines a working plane through the specified axes. Example: *N10 G20 I4 J3*, where *I4* is the number of the principal axis, *J3* – the number of the secondary axis.

Working space bounds commands G24, G25

The commands define the lower and the upper bounds of the working space, respectively.

Working space bounds On/Off commands G26, G27

The commands enable/disable the bounds on the working space, respectively.

Command of mirroring about lines G38

The command sets mirroring about vertical and horizontal lines. The lines are defined by the parameter X – for horizontal lines, Y – for vertical lines.

The command G39 voids the command G38, turning off the mirroring.

Left/right correction commands G41, G42

The commands set the toolpath compensation for the tool radius on the left- and right-hand side, respectively. Example: *N10 G41 D3*, where *D3* is the correction number.

The command G40 voids the commands G41 and G42, turning off the toolpath correction.

“Part rotation” commands G51, G52, G22

These commands are used in the case when the user needs to run an existing control sequence again, but with a different part orientation. The part can be rotated about the center of the working plane by the angle specified in the described commands. The angle can be in radians or in degrees. Example: *N10 G52 R...*, where *R* is the rotation angle in radians.

The commands for setting up working coordinate systems, G54-G59

Those commands modify the reference point (the coordinate system origin) of the current coordinates of the tool. These commands can be used, for example, for machining parts in pallets.

G54 and G59 command format:

Nn G54, Nn G59, where Nn is the number of the line in the sequence.

A machine “zero” commonly refers to the origin of the machine coordinate system. The coordinate systems whose origins are shifted off the machine zero are called “working coordinate systems”.

The command G53 voids commands G54 – G59, canceling modifications of the reference point (the coordinate system origin) of the current coordinates of the tool.

Spindle RPM and feed rate control command, G63

This command allows the user to modify the programmed spindle RPM in the range from 63% to 120%, reducing or increasing the spindle RPM. Additionally, this command allows the user to scale already programmed feed rate, that is, increase or decrease it, in the range from 1% to 120%.

The command G66 cancels this mode, restoring the initially programmed spindle RPM and feed rate.

Programming units commands (inches or metric), G71 and G70

By default, metric system is used across machining stations. The user, however, can use inches at one’s preference, with the help of the command G70. To switch back to the metric system, use the command G71.

Filleting error handling command, G73

Use of this command prevents filleting error.

The command G72 voids the command G73.

Machining “zero” setting command G74

This command reassigns the position of the machining “zero”. Example: *N10 G74 X... Y... ...* .

Angular acceleration and machining precision command G86

Angular acceleration affects the precision of machining a contour. The command G86 serves for programming both parameters. The value of angular acceleration is set by the parameter E and can take arbitrary values. There are, however, several tabulated values:

- 1 – doubles the maximum acceleration

- 0.5 – sets maximum acceleration

- 0.25 – halves maximum acceleration

- 0.05 – quarters maximum acceleration

The parameter K sets the precision of contour machining. Its value can be set arbitrarily. Example: *N10 G86 E0.5 K0.05*.

Command for setting the target position in absolute coordinates, G90

This command sets position calculation with respect to the origin of the current coordinate system.

G90 command format:

Nn G90, where Nn is the number of the line in the sequence.

Command for setting the target position in relative coordinates, G91

This command sets position calculation from the current position by increments in the respective coordinates.

G91 command format:

Nn G91, where Nn is the number of the line in the sequence.

G92 command

The command sets up the new working coordinate system with the origin shifted off the absolute coordinate system.

G92 command format:

Nn G92, where Nn is the number of the line in the sequence.

Auxiliary Commands

The names of auxiliary commands begin with letter M, followed by two digits defining the command type. Auxiliary commands control the modes of CNC center operation. Here follows a detailed description of auxiliary commands. (Note that the format of some commands has been described in the "Postprocessor for wire cutting " section).

The program stop command M00

The command M00 can be inserted anywhere within the frame in which the program stop is intended, or in a separate frame following right after. The command M00 stops motion in both coordinates.

Stop with confirmation command M01

The command M01 must be placed within the line where the stop with confirmation is intended. Before executing the line, the system will display a query for the operator to confirm stopping of the machine. Upon the confirmation, the system processes all commands in the line, and then stops the motion.

Program stop cancellation command M02

The line with the command M02 is the last command in a control sequence. It completes the control sequence of a given technological process. The command M02 can be placed in a separate line or inserted at the end of the last line in the control sequence. Any command that follows the command M02 is ignored by the CNC system and won't be processed.

Commands for turning on spindle rotation with specified RPM, M03, M04

The commands turn on spindle rotation at a specified RPM, clockwise and counterclockwise, respectively.

Cooler On command M07

The command turns the cooling pump on in the specified mode. Example: *N100 M07 S2*, where *S* is a cooling system parameter.

The command *M08* voids the command *M07*, turning off the cooling pump.

Spindle Off command M11

This command turns off spindle rotation.

End of control sequence file command M30

This command marks the end of the control sequence file and is used in combination with the command *M02*.

Postprocessor for 3D Milling

The default set of commands used in control sequences can be divided into the preparatory commands and auxiliary commands.

Preparatory Commands

The names of preparatory commands begin with letter *G*, followed by two digits defining the command type. Additional commands could be instructions defining a particular mode of the control system. Here follows a detailed description of preparatory commands. (Note that the format of some commands has been described in the "Postprocessor for wire cutting " section).

Rapid positioning command G00

This command moves the tool to the programmed position with a specified velocity (such as the maximum feed rate). The preprogrammed velocity of the working feed is ignored, but not canceled. Travel by coordinate axes could be uncoordinated, for example: *N10 G01 X10 Y25*.

Linear interpolation command G01

This command moves the tool along a straight line at a specified rate (in an orthogonal coordinate system). In this case, a constant ratio must be maintained between the velocity components by the coordinate axes. The ratio should be equal to that of the tool move components by the coordinate axes. Example: *N10 G01 X10 Y25*.

The pause command G04

Pauses machining for the duration specified in the line of the control sequence, in milliseconds. Example: *N20 G04 X1000*.

Left/right correction commands G41, G42

The commands set the toolpath compensation for the tool radius on the left- and right-hand side, respectively. Example: *N10 G41 D3*, where *D3* is the correction number.

The command *G40* voids the commands *G41* and *G42*, turning off the toolpath correction.

Spindle RPM and feed rate control command, G63

This command allows the user to modify the programmed spindle RPM in the range from 63% to 120%, reducing or increasing the spindle RPM. Additionally, this command allows the user to scale already programmed feed rate, that is, increase or decrease it, in the range from 1% to 120%.

The command G66 cancels this mode, restoring the initially programmed spindle RPM and feed rate.

Programming units commands (inches or metric), G70 and G71

By default, metric system is used across machining stations. The user, however, can use inches at one's preference, with the help of the command G70. To switch back to the metric system, use the command G71.

Command for setting the target position in absolute coordinates, G90

This command sets position calculation with respect to the origin of the current coordinate system.

G90 command format:

Nn G90, where Nn is the number of the line in the sequence.

Command for setting the target position in relative coordinates, G91

This command sets position calculation from the current position by increments in the respective coordinates.

G91 command format:

Nn G91, where Nn is the number of the line in the sequence.

Auxiliary Commands

The names of auxiliary commands begin with letter M, followed by two digits defining the command type. Auxiliary commands control the modes of CNC center operation. Here follows a detailed description of auxiliary commands. Note that the format of some commands has been described in the "Postprocessor for wire cutting " section.

The program stop command M00

The command M00 can be inserted anywhere within the frame in which the program stop is intended, or in a separate frame following right after. The command M00 stops motion in both coordinates.

Stop with confirmation command M01

The command M01 must be placed within the line where the stop with confirmation is intended. Before executing the line, the system will display a query for the operator to confirm stopping of the machine. Upon the confirmation, the system processes all commands in the line, and then stops the motion.

Program stop cancellation command M02

The line with the command M02 is the last command in a control sequence. It completes the control sequence of a given technological process. The command M02 can be placed in a separate line or inserted at the end of the last line in the control sequence. Any command that follows the command M02 is ignored by the CNC system and won't be processed.

Commands for turning on spindle rotation with specified RPM, M03, M04

The commands turn on spindle rotation at a specified RPM, clockwise and counterclockwise, respectively.

Cooler On command M07

The command turns the cooling pump on in the specified mode. Example: *N100 M07 S2*, where *S* is a cooling system parameter.

The command M08 voids the command M07, turning off the cooling pump.

Spindle Off command M11

This command turns off spindle rotation.

End of control sequence file command M30

This command marks the end of the control sequence file and is used in combination with the command M02.

Postprocessor for 5D Milling

The default set of commands used in control sequences can be divided into the preparatory commands and auxiliary commands.

Preparatory Commands

The names of preparatory commands begin with letter G, followed by two digits defining the command type. Additional commands could be instructions defining a particular mode of the control system. Here follows a detailed description of preparatory commands. Note that the format of some commands has been described in the "Postprocessor for wire cutting " section.

Rapid positioning command G00

This command moves the tool to the programmed position with a specified velocity (such as the maximum feed rate). The preprogrammed velocity of the working feed is ignored, but not canceled. Travel by coordinate axes could be uncoordinated, for example: *N10 G00 X10 Y25*.

Linear interpolation command G01

This command moves the tool along a straight line at a specified rate (in an orthogonal coordinate system). In this case, a constant ratio must be maintained between the velocity components by the coordinate axes. The ratio should be equal to that of the tool move components by the coordinate axes. Example: *N10 G01 X10 Y25*.

The pause command G04

Pauses machining for the duration specified in the line of the control sequence, in milliseconds. Example: *N20 G04 X1000*.

Left/right correction commands G41, G42

The commands set the toolpath compensation for the tool radius on the left- and right-hand side, respectively. Example: *N10 G41 D3*, where *D3* is the correction number.

The command G40 voids the commands G41 and G42, turning off the toolpath correction.

Spindle RPM and feed rate control command, G63

This command allows the user to modify the programmed spindle RPM in the range from 63% to 120%, reducing or increasing the spindle RPM. Additionally, this command allows the user to scale already programmed feed rate, that is, increase or decrease it, in the range from 1% to 120%.

The command G66 cancels this mode, restoring the initially programmed spindle RPM and feed rate.

Programming units commands (inches or metric), G71 and G70

By default, metric system is used across machining stations. The user, however, can use inches at one's preference, with the help of the command G70. To switch back to the metric system, use the command G71.

5-axis transformation command, G81

This command sets the machining station to operate in the coordinate system of the machined part. It also engages the 5-axis control transformation.

The command G80 voids the command G81, turning off the 5-axis transformation.

Command for setting the target position in absolute coordinates, G90

This command sets position calculation with respect to the origin of the current coordinate system.

G90 command format:

Nn G90, where Nn is the number of the line in the sequence.

Command for setting the target position in relative coordinates, G91

This command sets position calculation from the current position by increments in the respective coordinates.

G91 command format:

Nn G91, where Nn is the number of the line in the sequence.

Auxiliary Commands

The names of auxiliary commands begin with letter M, followed by two digits defining the command type. Auxiliary commands control the modes of CNC center operation. Here follows a detailed description of auxiliary commands. (Note that the format of some commands has been described in the "Postprocessor for wire cutting " section).

The program stop command M00

The command M00 can be inserted anywhere within the frame in which the program stop is intended, or in a separate frame following right after. The command M00 stops motion in both coordinates.

Stop with confirmation command M01

The command M01 must be placed within the line where the stop with confirmation is intended. Before executing the line, the system will display a query for the operator to confirm stopping of the machine. Upon the confirmation, the system processes all commands in the line, and then stops the motion.

Program stop cancellation command M02

The line with the command M02 is the last command in a control sequence. It completes the control sequence of a given technological process. The command M02 can be placed in a separate line or inserted at the end of the last line in the control sequence. Any command that follows the command M02 is ignored by the CNC system and won't be processed.

Commands for turning on spindle rotation with specified RPM, M03, M04

The commands turn on spindle rotation at a specified RPM, clockwise and counterclockwise, respectively.

Cooler On command M07

The command turns the cooling pump on in the specified mode. Example: *N100 M07 S2*, where *S* is a cooling system parameter.

The command M08 voids the command M07, turning off the cooling pump.

Spindle Off command M11

This command turns off spindle rotation.

End of control sequence file command M30

This command marks the end of the control sequence file and is used in combination with the command M02.

EXTERNAL POSTPROCESSORS

Creating an External Postprocessor in Microsoft Visual C++ 6.0

Postprocessing stages

1. **Reading data** from file *defttemp.cld* is carried out in the static library *CLDLoader.lib*.

This stage does not require modifications.

2. **Data transformation.**

Presence of this stage is determined by the necessity to transform coordinates, for example, in case of five-coordinate toolpath. The function *GTransform()* is added to the main class of the postprocessor.

3. **Formatting rows of the output file** and writing them to the output text file.

Modification of functions *Get_()* in the main class of the postprocessor.

Project description

External postprocessor – MFC dialog based application (Dialog based Application) with the statically linked library. It performs reading of the input file (*.cld), coordinate transformation and saving the text file of the control sequence.

Structure of catalogs

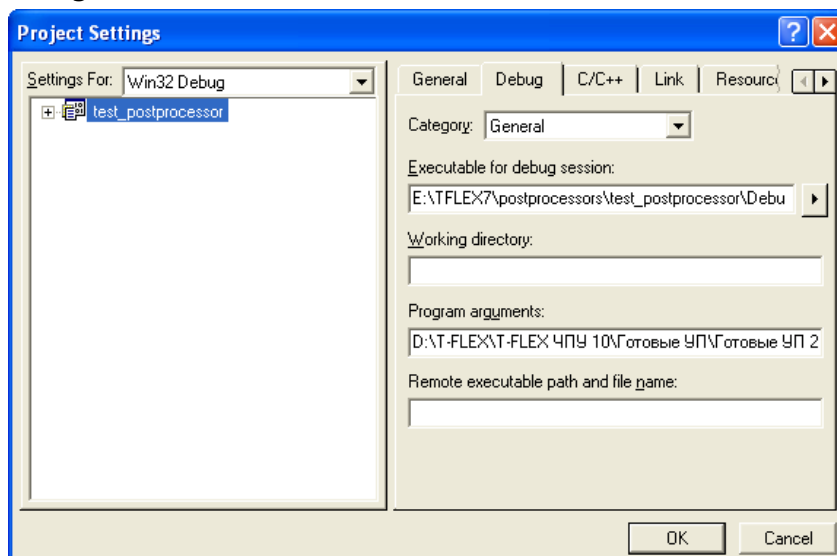
At the initial stage there are the following catalogs:

- \readCLD – project of reading CLD-file that contains the initial files and compiled static library \Release\cldloader.lib.
- \NewPost\ - project of the postprocessor being developed.

Setting a project

Select properties of the created project.

1. Project Settings/Debug

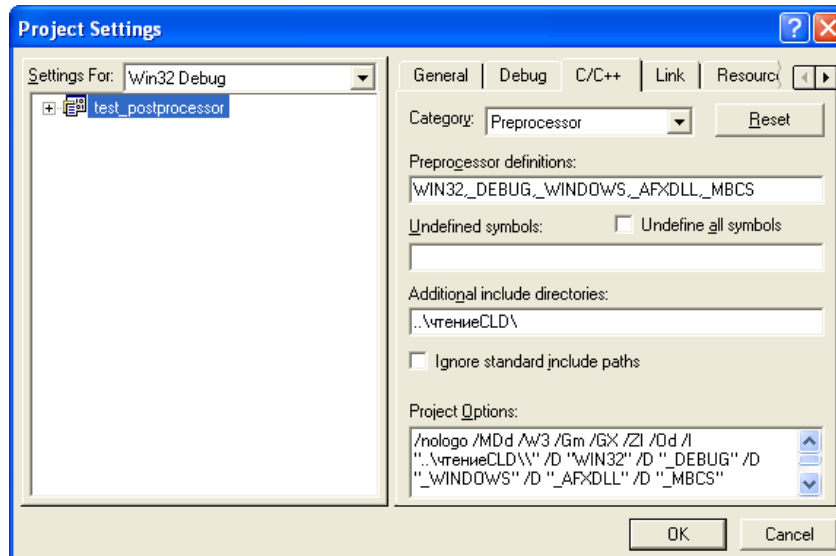


We select **General** and in **Program Arguments** we write D:\T-FLEX\T-FLEX CAM 10\Ready CS\Ready CS 2D\deftemp.cld_D:\Temp \test.nc

D:\T-FLEX\T-FLEX CAM 10\Ready CS\Ready CS 2D\deftemp.cld – path of the file deftemp.cld.

D:\Temp \test.nc – path of the saved file with a control sequence.

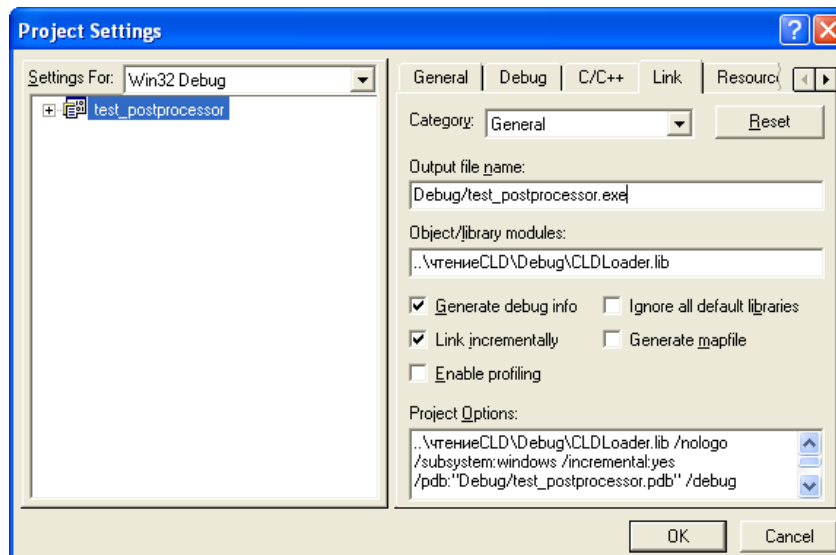
2. Project Settings/C/C++



Select option **Preprocessor** and write in **Include additional directories** the following data:

- ..\readCLD\

3. Project Settings/Link



Select option **General** and write in **Object/library modules** the following data:

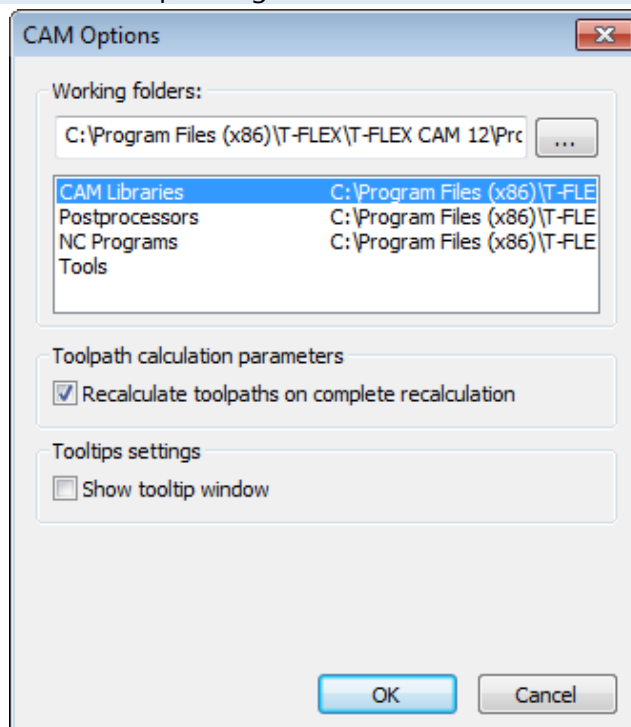
- ..\readCLD\Debug\CLDLoader.lib (for Debug)

Project "ReadCLD"

Project "ReadCLD" is required for assembling the project of the postprocessor. It contains program classes whose functions perform reading of technological and geometric information about the

toolpaths calculated in T-FLEX CAM system. This information is contained in the binary file **defttemp.cld**. This file is created when saving the control sequence and external postprocessor in T-FLEX CAM system and it is stored on the disk in the work directory specified in settings of CAM-system in the "G-programs" row.

T-FLEX CAM system saves the coordinates of points and vector with respect to the global system of coordinates in CLData-file. However, the control sequence must contain the commands for control of translations for the corresponding coordinates.



Main program classes

It is possible to select the classes of toolpaths and the class of frames of the toolpaths from the program classes of the project.

1. **Classes of toolpaths** are inherited from the base class **Sm2DTraject**. They contain technological parameters (feed rate, allowance, revolutions, etc).

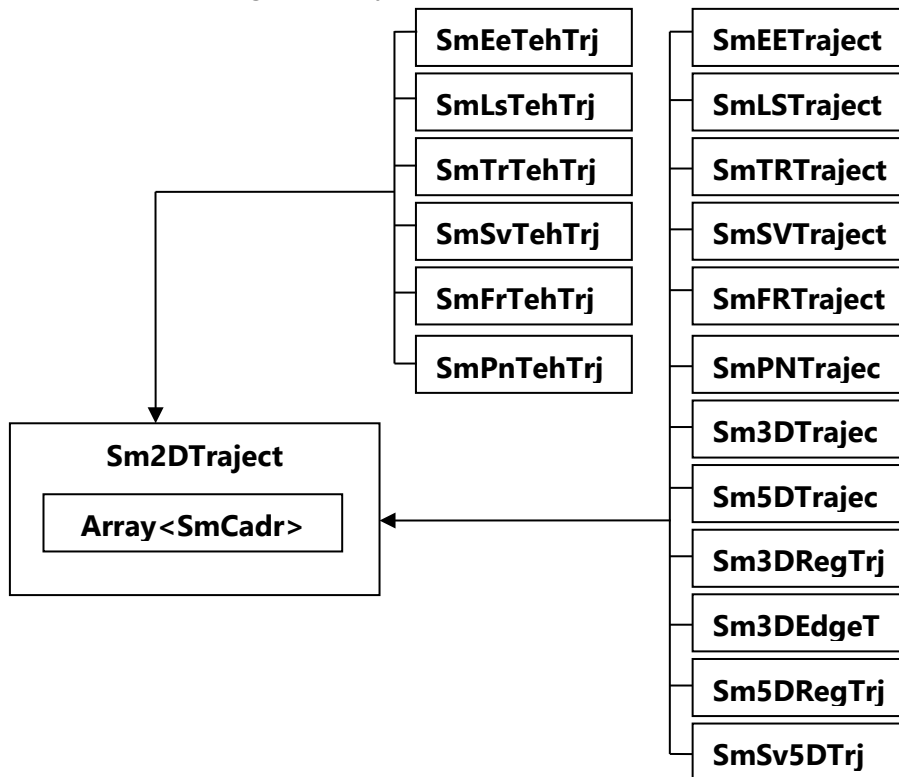
There are classes of the following toolpaths:

- SmEeTehTrj** - technological wire-cutting toolpath
- SmLsTehTrj** - technological laser-cutting toolpath
- SmTrTehTrj** – technological turning toolpath
- SmSvTehTrj** - technological drilling toolpath
- SmFrTehTrj** - technological milling toolpath
- SmPnTehTrj** - technological punching toolpath

SmEETraject – wire-cutting toolpath
SmLSTraject – laser-cutting toolpath
SmTRTraject – turning toolpath
SmSVTraject - drilling toolpath
SmFRTraject - milling toolpath
SmPNTraject - punching toolpath
Sm3DTraject - 3D milling toolpath
Sm5DTraject - 5D milling toolpath
Sm3DRegTrj - 3D zone milling toolpath
Sm3DEdgeTrj - 3D toolpath of milling the edges
Sm5DRegTrj - 5D zone milling toolpath
SmSv5DTrj - 5D drilling toolpath

The base class **Sm2DTraject** includes an array of calculated frames of the toolpath the sequence of which forms the basis for generation of the output text of the control sequence.

2. **Class of frames of toolpaths SmCadr.** This class contains the type and number of the frame, geometric information (coordinates of points of translation) and various parameters that are initialized depending on the type of the frame.



On the basis of unique identifiers described in files SmCadr.h and SmPostProcessor.h, in the project of postprocessor the analysis of types of the toolpaths (FR_TRAJECT, SV_TRAJECT etc.) and frames of toolpaths (FG00_TYPE, SG00_Fanuc_TYPE etc.) occurs. From the classes of toolpaths it is possible to get the required technological parameters, and the sequence of frames allows us to create functions-handlers for each frame with subsequent generation of rows of the control sequence.

Main functions

Class **SmPostProcessor** contains declarations of several main functions whose definitions are used in the project of postprocessor.

SaveGCode – the main function of postprocessor in which the successive machining of frames of the toolpaths and calling of the functions-handlers of the class of postprocessor occurs. Definition of this function has to be present in any project of the postprocessor.

SaveEndGCode – function that is invoked before the end of work of the postprocessor, after the function SaveGCode has been executed. Usually, with the help of this function, the commands of termination of the program and end of the file are uploaded to the control sequence.

Assembling a project

As a result of the assembly of the project "ReadCLD", the file of the static library CLDLoader.lib is created. This library is used when assembling the project of the postprocessor.

Wizard for creation of postprocessor project (Custom Postprocessor Wizard)

Installing a wizard (version for Visual Studio 6.0)

To install a wizard it is required in the Microsoft Visual Studio 6.0 programming environment to open the file of the project named Custom Postprocessor.dsp and make the assembly with the configuration "Release".

Installing a wizard (version for Visual Studio 2005)

To install a wizard, it is required in the Microsoft Visual Studio 2005 programming environment to open the file of the project named Custom Postprocessor.sln and make the assembly with the configuration "Release".

In the file *Custom Postprocessor 2005.vsz* in the row Param = "ABSOLUTE_PATH = it is necessary to specify the path to the catalog that contains the project of the postprocessor creation wizard:

```
Param="WIZARD_NAME = Custom Postprocessor 2005"  
Param="ABSOLUTE_PATH = E:\TFLEX7\Постпроцессоры\БАЗОВЫЙ 2005\Custom Postprocessor 2005"  
Param="FALLBACK_LCID = 1033"
```

Next, it is required to copy the files

Custom Postprocessor 2005.ico

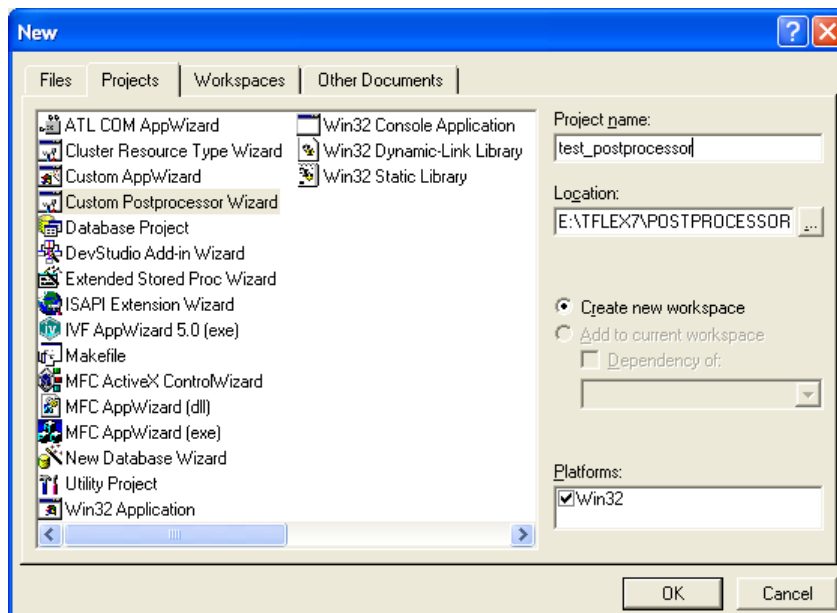
Custom Postprocessor 2005.vmdir

Custom Postprocessor 2005.vsz

to the catalog «disk»:\Microsoft Visual Studio 8\VC\vcprojects\

Creation of postprocessor project with the help of wizard (version for Visual Studio 6.0)

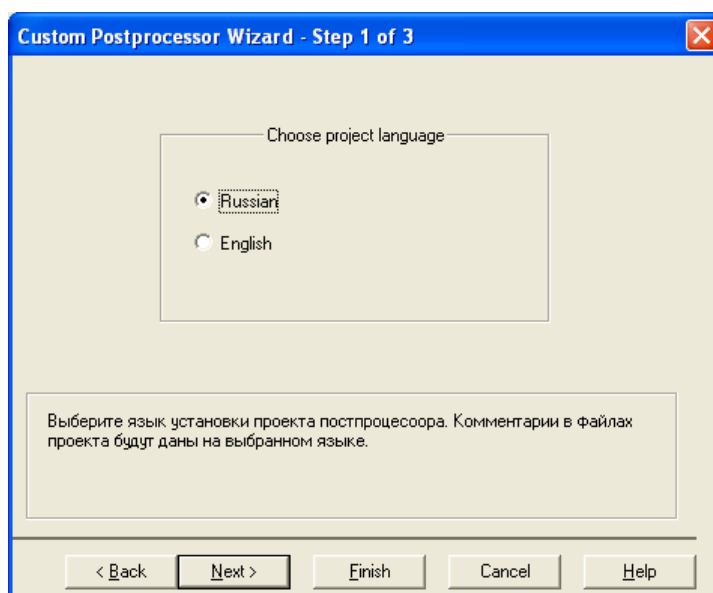
Creation of a new postprocessor project is carried out via the options of the menu “File/New” of the Visual Studio environment.



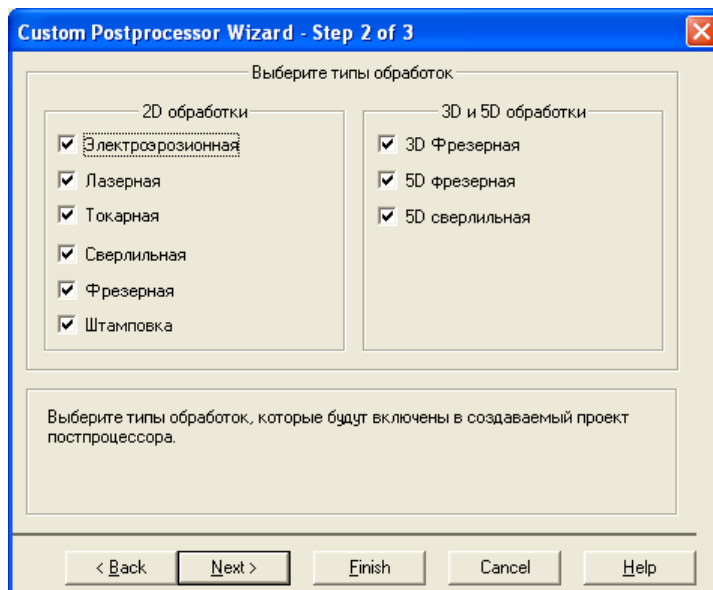
In case of successful assembly of the project of postprocessor creation wizard, a new wizard – “**Custom Postprocessor Wizard**” will appear in the list. In the “**Project name**” field, the user specifies the name of the postprocessor project being created, and in the “**Location**” field – the path to the catalog that contains the project.

After pressing the [OK] button, the postprocessor creation wizard will be launched. Creation of the postprocessor project consists of three stages. Transition to the next stage is carried out with the help of the [Next >] button, to the previous step – the [< Back] button, to the beginning of installation – the [Finish] button.

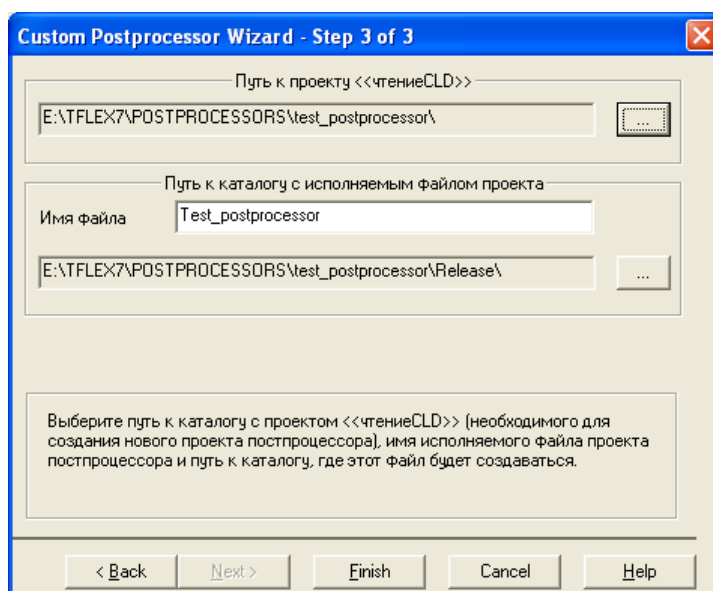
At the first stage we select the language for the installation wizard. The comments in the files of the postprocessor project will be written in this language.




After finishing selection of the language, the user has to select the **[Next >]** button.

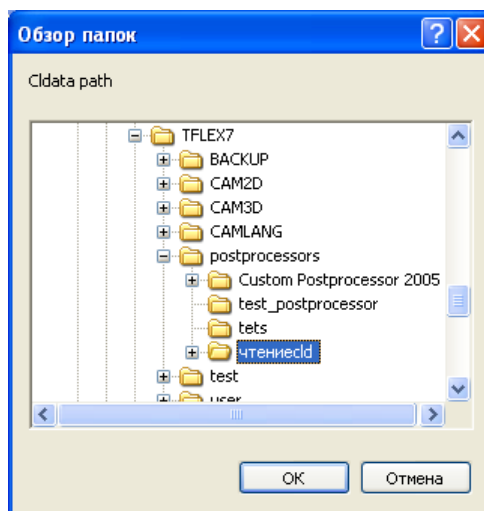


At the second stage, the user selects the types of machining, which he needs in the postprocessor project. Here it is important to note that if upon further work with the postprocessor project the user needs the types of machining not selected during this stage, he will have to add them to the project manually. After finishing selection of the types of machining, press the **[Next >]** button.



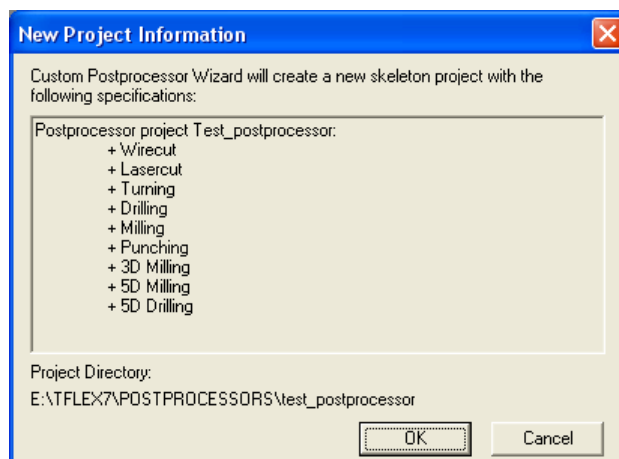
On the third stage, setting of parameters of the postprocessor project is carried out.

The important step is the selection of the catalog with the project "readCLD" that is made by pressing the  button.



The next step is the specification of the name of the executable file of the postprocessor project and the catalog in which this file will be created. The executable file is the result of the assembly of the postprocessor project.

Selection of the catalog is carried out by pressing the  button opposite the corresponding field. To complete settings for the project, press the **[Finish]** button.

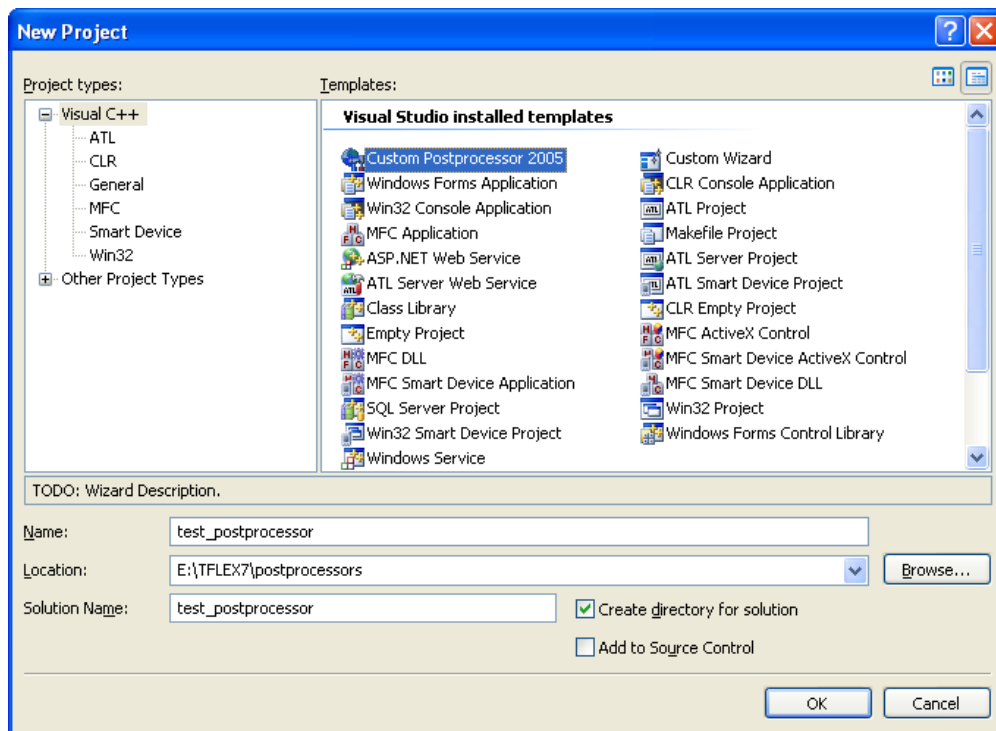


Before starting to install the files of the project, the information about settings performed by the user will be shown to him. Pressing the [OK] button will start the installation, pressing the [Cancel] button will return the user to the previous stage.

Creation of postprocessor project with the help of a wizard (version for Visual Studio 2005)

Creation of a new postprocessor project is carried out via the options of the menu "File/New/Project..." of the Visual Studio environment:

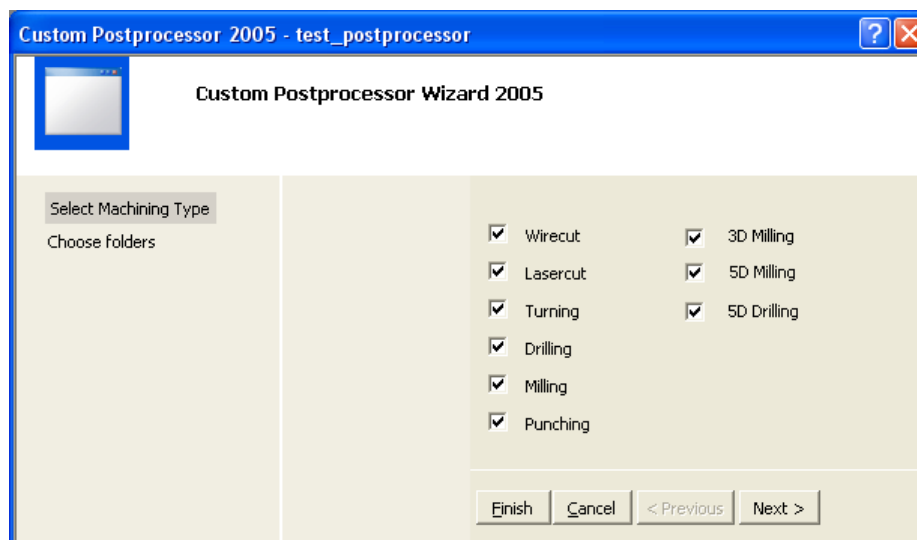




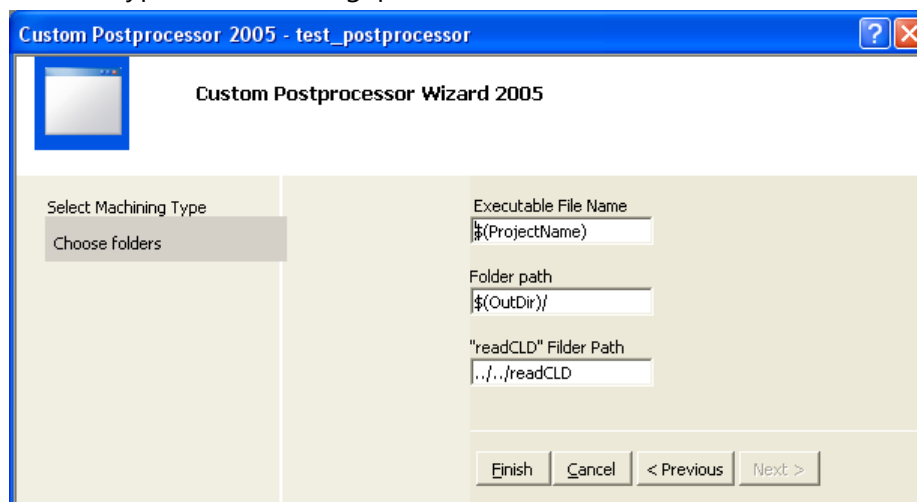
In the dialog window that appears in the tree-like list on the left (Project types) it is required to select “Visual C++”, and in the right list (Templates) in case of successful assembly of the project of the postprocessor creation wizard, a new wizard will appear – “**Custom Postprocessor 2005**”. In the “**Name**” field specify the name of the postprocessor project being created, and in the “**Location**” field – path to the catalog that contains it.

After pressing the [OK] button, the postprocessor creation wizard will be launched.

Creation of the postprocessor project consists of two stages. Transition to the next stage is carried out with the help of the [Next >] button, to the previous step – the [< Back] button, to the beginning of installation – the [Finish] button.



On the first stage, the user selects the types of machining, which he needs in the postprocessor project. Here it is important to note that if, in further work with the postprocessor project, the user needs the types of machining not selected during this stage, he will have to add them to the project manually. After finishing selection of the types of machining, press the **[Next >]** button.



At the second stage, setting of parameters of the postprocessor project is carried out. In the **“Executable File Name”** field, the name of the executable file of the postprocessor is specified. The executable file is a result of the assembly of the postprocessor project. The macro `$(ProjectName)` selected here by default corresponds to the name of the postprocessor project that is specified before the start of the work of the wizard.

In the **“Folder path”** field specify the path to the catalog where the file of the postprocessor is located. By default the macro `$(OutDir)/` corresponds to the full path to the catalog where the postprocessor project is located.

The important step is the correct specification of the path to the catalog with the **«ReadCLD»** project.

To complete settings and begin installation of the project, press the **[Finish]** button.

Structure of postprocessor project created with the help of a wizard

On the basis of the settings selected by the user, the postprocessor creation wizard will create a new project in the Visual Studio programming environment. The project will consist of the classes of the following types:

1. **Required classes.** Classes that are always created regardless of the user's settings of the wizard. The main classes that provide exchange of the data and processing of commands of the postprocessor are **SmBase_Post** and **Data**.

SmBase_Post – the main class of the postprocessor project. The file `Postprocessor.cpp` contains definitions of the basic functions of the class `SmPostprocessor`: **SaveGCode** and **SaveEndGCode**. The function **SaveGCode** contains callings of functions-handlers for all types of frames existing in T-FLEX CAM system. Function **SaveEndGCode** is invoked at the end of work of the postprocessor and uploads commands of ending the file of the control sequence, for example, the command M30.

Data – class for exchange of data between the machining classes that are selected by the user. It contains the current values of the coordinates, of various technological parameters, the current frame number and other items.

2. **Machining classes.** Classes of types of machining selected by the user during the stages of project creation with help of a wizard. They contain the variables required for work and functions-handlers for processing the frames of the corresponding type.

SmEe_Post (files `ee.cpp` and `ee.h`) – class of wire-cutting.

SmLas_Post (files `las.cpp` and `las.h`) – class of laser cutting.

SmTok_Post (files `tok.cpp` and `tok.h`) – class of machining by turning

SmSv_Post (files `sv.cpp` and `sv.h`) – class of machining by drilling

SmFr_Post (files `fr.cpp` and `fr.h`) – class of machining by milling

SmPn_Post (files `pn.cpp` and `pn.h`) – class of machining by punching

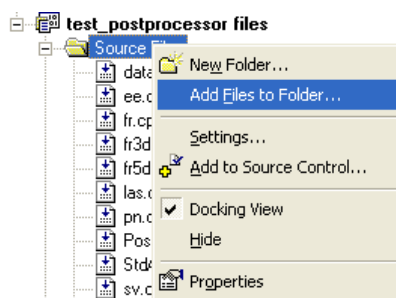
SmFr3D_Post (files `fr3d.cpp` and `fr3d.h`) – class of 3-coordinate machining by milling

SmFr5D_Post (files `fr5d.cpp` and `fr5d.h`) – class of 5-coordinate machining by milling

SmSv5D_Post (files `sv5d.cpp` and `fr3d.h`) – class of 5-coordinate machining by drilling

If the user wants to add to the postprocessor the machining types that were not included by him during the stage of setting a wizard, he has to perform the following steps.

- Copy from the catalog **Template**, which is located inside the catalog containing the project of a postprocessor creation wizard («disk»:/«path to a wizard»/Template), the files of the required machining type to the catalog with the created postprocessor project.
- Add these files to the project.



Editing postprocessor project created with the help of a wizard

Using the class of the wire-cutting named “SmEe_Post” as an example, let us consider the capabilities of quick editing of the postprocessor project that are provided by the developers. The class consists of the files ee.h (declaration of variables and functions of the class) and ee.cpp (definition of variables and functions of the class).

1. **Macros.** At the beginning of the file ee.cpp we can see the macros that begin with command #define.

```
#include "stdafx.h"
#include "ee.h"
#include "math_defines.h"

#ifdef _DEBUG
#define new DEBUG_NEW
#undef THIS_FILE
static char THIS_FILE[] = __FILE__;
#endif

#define G02G03_R      0 //1 - вычислять радиус
#define G02G03_IJ_ABS 0 //1 - вычислять I и J в абсолютных коорг
#define CORR_R        1 //1 - выгружать корректор на радиус
#define G90_STD        1 //1 - выгружать G90 кадр (стандартный ка
// G90 выгр. только при смене системы
#define HP_EN          1 //1 - выгружать высоту детали
#define M02_EN         1 //1 - выгружать конец программы
#define M10_EN         0 //1 - выгружать включение/выключение ген
#define M30_EN         1 //1 - выгружать конец файла
#define M48_EN         1 //1 - выгружать включение/выключение gиз
#define M50_EN         1 //1 - выгружать включение/выключение пер
```

By changing the numerical value of the macro from 0 to 1 and from 1 to 0, it is possible to quickly change the format of recording of several parameters into the control sequence. For example, if in the macro G02G03_R zero is replaced with 1,

```
#define G02G03_R      1 //1 - вычислять радиус
```

then in the frames of circular interpolation G02 and G03, the radius will be uploaded instead of coordinates I, J, i.e.,

```
N40 X149.91
N45 G02 X263.83 R56.96
N50 G1 X293.95
```

If in the comments to the macro it is indicated that 1 – signifies uploading a certain parameter, then selection of zero value will allow us to remove this parameter from the control sequence.

2. **Functions of formatting numerical values.**

These functions are used for formatted uploading of the numerical values into the control sequence.

GetX_String – formatted output of the numerical value of the X-coordinate

GetY_String – formatted output of the numerical value of the Y-coordinate

GetI_String – formatted output of the numerical value of the I-coordinate

GetJ_String – formatted output of the numerical value of the J-coordinate

GetString – formatted output of the numerical value.

For example, if it is required to change, in the X-coordinate, the number of significant digits after the comma from three (default value) to five, then the user has to change the number 3 to 5 in the row `str_tmp.Format("%.5f", fabs(dbl_x))` inside the function `GetX_String`:

```
////////////////////////////////////  
// получить строку с числом X  
// --  
CString SmEe_Post::GetX_String(double dbl_x) {  
    CString str_out = "";  
    CString str_tmp;  
    str_tmp.Format("%.5f", fabs(dbl_x));  
    CString str_1, str_2, str_ss;
```

3. Functions-handlers.

For all G-commands, M-commands, tool's frames and other types of frames the corresponding functions-handlers are defined. For example,

```
// получить коррекцию на радиус G41/G42  
BOOL GetG41G42(BOOL b_write_N, CString &str_out, Data &data);
```

– function-handler of the command of correction (compensation) on the radius of the wire electrode.

Here `b_write_N` – parameter that determines whether this command will be uploaded to the control sequence in the form of a separate string (TRUE), or it will be added to already existing string (FALSE). This parameter is introduced for cases when it is required to upload G or M commands into the structure of other commands.

Let us consider a case when the left correction on the radius G41 must be uploaded inside the command of linear interpolation G01.

- a) At the first step it is required to find handlers for frames of the wire-cut machining (wire-cutting) in the file `Postprocessor.cpp`. They begin with the service macro `#ifdef EE`.

```

//кадры
for(i = 0; i < trj[i_curt]->trjlen; i++) {
    data.i_N_cur = i;
    switch(trj[i_curt]->trj[i].type) {
        //электроэрозия -----
        #ifdef EE
            //G-команды
            case G00_TYPE: // позиционирование
                if(ee_post.GetG00(trj[i_curt]->trj[i].P[0],
                                TRUE, str_out, data) == TI
                    outf->WriteString(str_out);
                break;
            case G01_TYPE: // Линейная интерполяция
                if(ee_post.GetG01(trj[i_curt]->trj[i].P[0],
                                trj[i_curt]->trj[i].P[2],
                                TRUE, str_out, data) == TI
                    outf->WriteString(str_out);
                break;
            case G01G02_TYPE: // линейная и круговая
                if(ee_post.GetG01G02(trj[i_curt]->trj[i].P[0],
                                    trj[i_curt]->trj[i].P[2],
                                    trj[i_curt]->trj[i].P[4],
                                    TRUE, str_out, data) == TI
                    outf->WriteString(str_out);

```

Next let us find the handler for the frame of the left correction on the radius G41_TYPE

```

case G41_TYPE: // Включение левой коррекции на радиус
    data.i_G41 = 0;
    data.i_N_cor = (int)trj[i_curt]->trj[i].P[0];
    if(ee_post.GetG41G42(TRUE, str_out, data) == TRUE)
        outf->WriteString(str_out);
    break;

```

In this handler it is required to comment the call to the function GetG41G42 since it will be invoked from the handler of the command of linear interpolation G01.

```

case G41_TYPE: // Включение левой коррекции на радиус
    data.i_G41 = 0;
    data.i_N_cor = (int)trj[i_curt]->trj[i].P[0];
    //if(ee_post.GetG41G42(TRUE, str_out, data) == TRUE)
    // outf->WriteString(str_out);
    break;

```

- b) The next step is enabling the editing of the function-handler GetG01 of the command of linear interpolation inside the file ee.cpp

```

//////////
// получить G01 кадр (линейная интерполяция)
//---
BOOL SmEe_Post::GetG01(double dbl_x, double dbl_y,
                      double dbl_u, double dbl_v,
                      BOOL b_write_N, CString &str_out, Data &data)

```

To this function it is required to add the call to the function-handler of the commands of correction on radius GetG41G42 with parameter b_write_N = FALSE so that the correction is uploaded to the control sequence in the same row as the linear interpolation.

```

if(b_write_N == TRUE){ // новая строка
    str_out = "";
    str_out += data.GetN() + data.str_sep;
}

if(data.i_GT != 1)
    str_out += str_G1 + data.str_sep;

    GetG41G42(FALSE, str_out, data);

if(fabs(dbl_x - data.dbl_X) > MAX_GDOP){ //X

```

Therefore, correction on radius will be uploaded to the control sequence in the same row with linear interpolation before writing the coordinates of translation.


```

test
N5 T1
N10 HP10
N15 G90
N20 M48 S1
N25 M50 S1
N30 F50
N35 G01 G41 D1 X166.94 Y236.06
N40 X98.14 Y134.55
N45 X289.62
N50 X356.49 Y236.06
N55 X166.94
N60 G40
N65 M51
N70 M49
N75 M30

```

In a similar way to the example considered, the user can upload various combinations of G and M commands in the same row.

4. Notation for parameters of machining.

The postprocessor project is created in such a way that the user can quickly change notations of G-commands, M-commands, axes and other variables of the class. To do so, it is sufficient to replace one value with another in the function Init.

For example, by default, the command of linear interpolation is denoted by "G01":

```

////////////////////////////////////
// уникализация
//--
void SmEe_Post::Init(void)
{
    str_G0      = "G00";    // обозначение перемещения
    str_G1      = "G01";    // обозначение линейной интерполяции

```

If the user replaces the expression "G01" with, for example, "G1", the row with linear interpolation in the control sequence will appear as:

```

                                N35 G1 X166.94 Y236.06
////////////////////////////////////
// уникализация
//--
void SmEe_Post::Init(void)
{
    str_G0      = "G00";    // обозначение перемещения
    str_G1      = "G1";     // обозначение линейной интерполяции

```

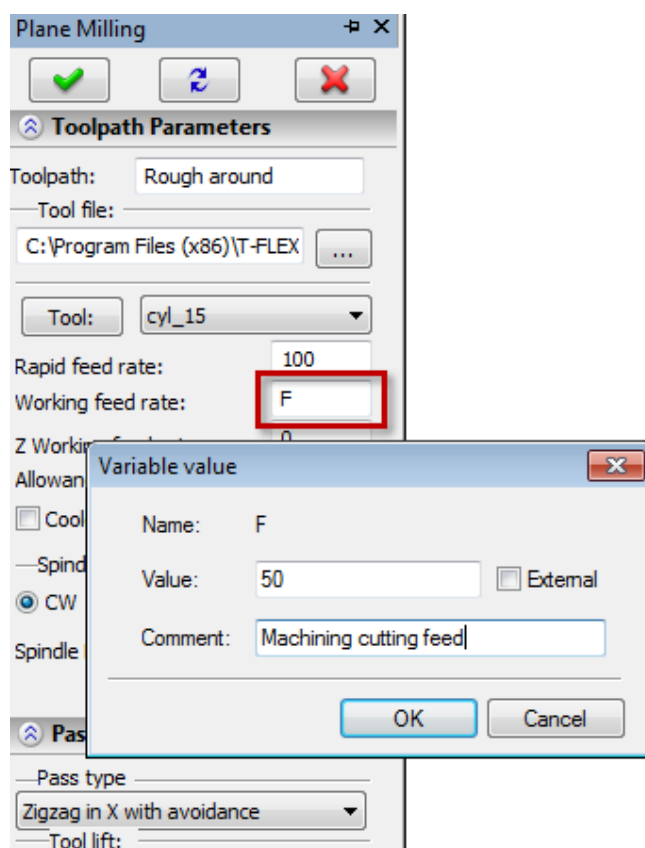
Debugging postprocessor project

In order to have the capability of debugging, it is required to perform setting of the postprocessor project, which is described in the section "Setting a project. Project Settings/C/C++".

ADDITIONAL SYSTEM FUNCTIONS

T-FLEX CAM VARIABLES

Just like in T-FLEX CAD, T-FLEX CAM lets you define real variables instead of specific values of geometrical and technological parameters. Each variable has a unique name and value, which is calculated according to a mathematical expression. In addition, a variable has a comment, in which you can specify the meaning of this variable (feed rate, spindle RPM, height of the machining plane, allowance, etc.). The variable type determines what values this variable can assume. The values of a real variable are numbers.



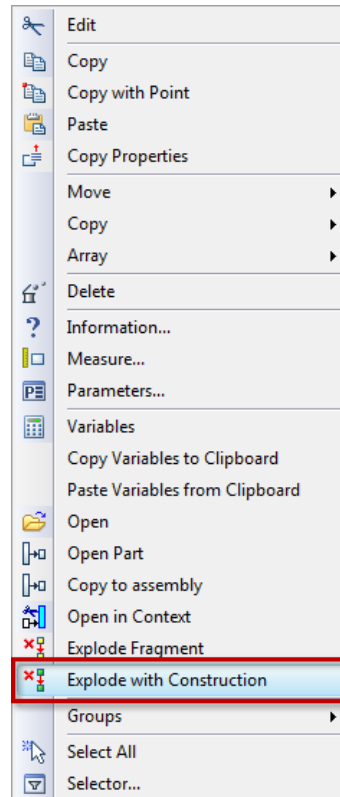
An expression is a mathematical formula containing standard algebraical operations, logical operations, conditional operations, calls to mathematical functions and functions of T-FLEX CAD. Expression evaluation results in a value corresponding to the variable type. An expression may contain operands (members) - real constants and variables, and operations (a set of actions applied to those operands). Expressions may also include functions. Function names are case-insensitive, so you can use either upper- or the lowercase letters in the function names. You can create T-FLEX CAM variables either in the dialog windows at the time of creating a machining path, or using the variables editor.

For details on working with variables, refer to the "Parameterization" section of the T-FLEX CAD Help.

T-FLEX CAM FRAGMENTS (*.GRB FILE)


When adding a T-FLEX CAD file to a document (the **Draw-Fragment** command), a fragment is inserted into the document, which contains geometrical elements and calculated toolpaths. All added elements are displayed in the drawing as one group without editing provisions.

To add the inserted toolpaths into the document's main list of toolpaths (Machining Manager), you need to select the context menu "**Explode with Construction**".

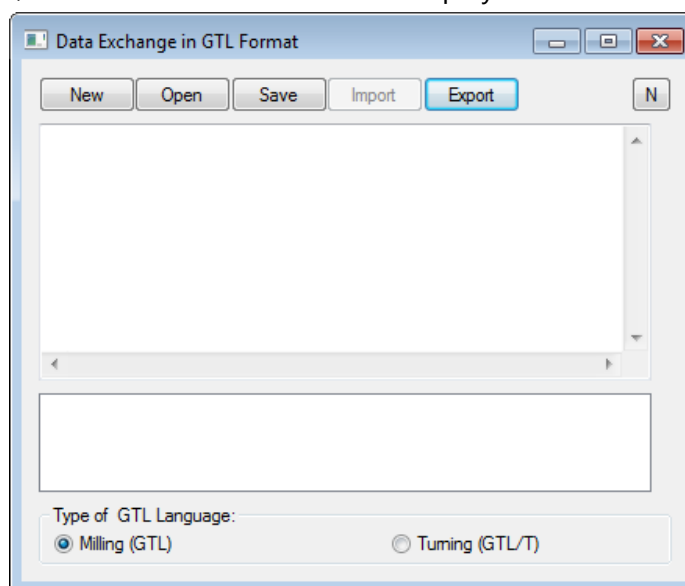


DATA EXCHANGE IN THE GTL FORMAT

A CNC 'subsystem' called GTL was at a certain time supplied together with its CNC centers by Italian manufacturer OLIVETTI; the 'subsystem' appeared as an early implementation of what is now known as CAM systems. Numerous shops widely used that system and over the time built up extensive libraries of their dedicated NC sequences. Nowadays, this system is aged and is virtually out of use. Yet the shops that have been using it do face a need to machine some parts of an older design. T-FLEX CAM provides the "**Data Exchange in the GTL Format**" option to import control sequences from the GTL format. This option is accessed by the respective icon on the toolbar or via a textual menu item:

Icon	Ribbon
	CNC → Control → GTL Data Exchange
Keyboard	Textual Menu
< >	CNC>GTL Subsystem

After calling the command, the GTL converter window is displayed.



In this dialog, it is possible to create a new NC in the GTL format by clicking the **New** button and entering the source numerical sequence. To save the created program, click the **Save** button and specify the name and the path to the file. To import an existing NC in the GTL format into T-FLEX system, click the **Open** button and specify the file in the GTL format. Once the file is loaded, click the **Import** button, and the system will convert the NC into a file with the extension ".grb" (the format of T-FLEX CAD/CAM files), and will create the rendering and the toolpath based on the parameters contained in the control sequence. By clicking the button marked with the letter "N", you access the control sequence renumbering option.

In case the reverse operation is required (converting a T-FLEX CAM-generated NC into the GTL format), load the control sequence and then click the **Export** button. In either case, you should click the **Save** button after completing the conversion.

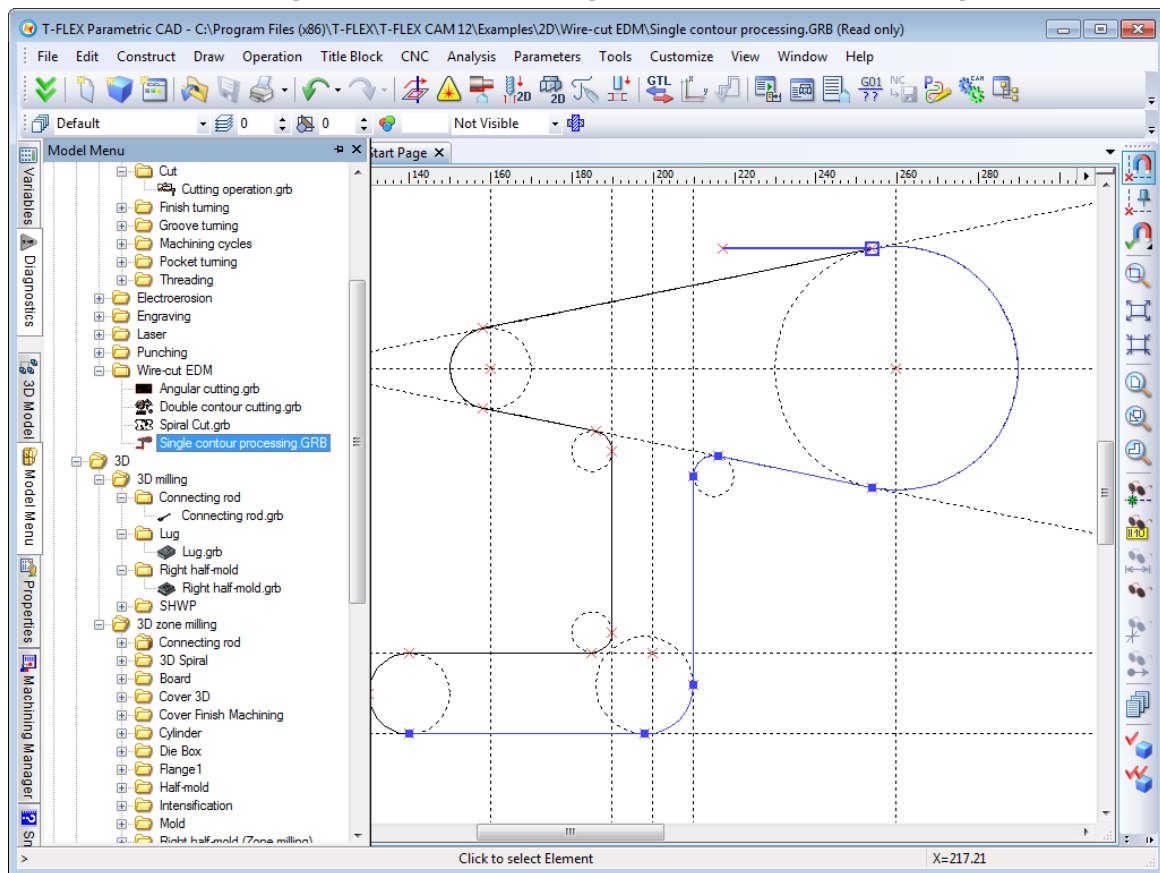
EXAMPLES OF USE





WIRE CUTTING AND LASER CUTTING

Note that these types of machining are reviewed together since they share common machining subtypes as:


- Single-contour cutting;
- Angular cutting;
- Double-contour cutting;
- Spiral material removal.

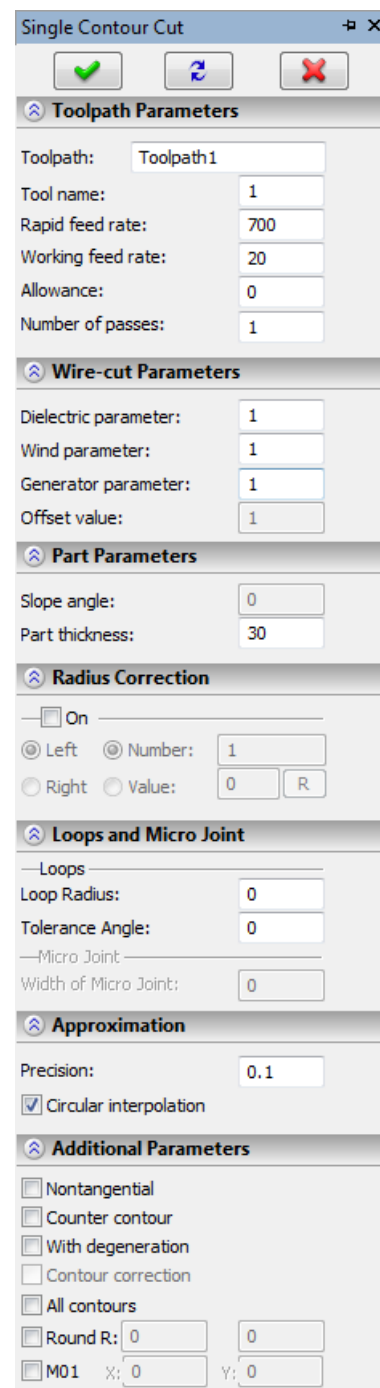
Let's review an instance of single-contour wire cutting of the part shown on the figure.



After constructing a hatch by a specified contour, proceed to the menu of the wire cutting module commands by clicking the  found on the main toolbar in the «CNC 2D» mode, or by selecting the textual menu "CNC>2D, 2.5D and 4D Machining>Wire-cut". Next, to create a single-contour wire cutting toolpath, click the  button on the toolbar. The wire cutting module provides for a capability to define part geometry either by using a hatch or by using the "Path" element, therefore, the respective buttons will be available on the toolbar. In this case, click the  button (hatch). After that the system will automatically transition to the mode of waiting for contour (hatch) selection for machining the part, with the toolpath parameters dialog window being displayed simultaneously. It is possible to select the hatch first, and then specify the toolpath parameters, while it is also possible to first define the toolpath parameters, and then select the machining contour (hatch) by clicking  inside the hatch contour.

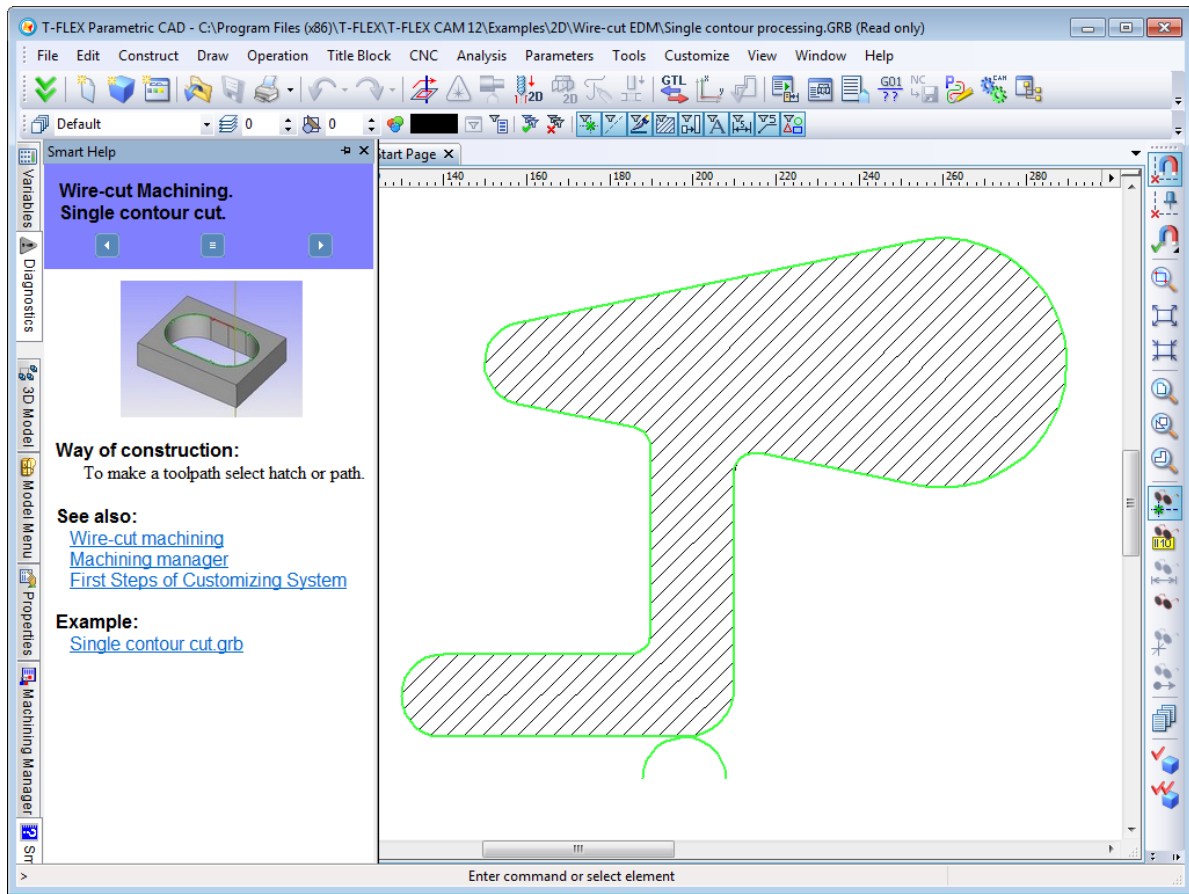
The machining toolpath parameters are described in the first part of the manual.

Once all necessary changes are made to the machining parameters, you just need to click the  button to have the system calculate the toolpath. The toolpath will appear in the drawing as a red line.




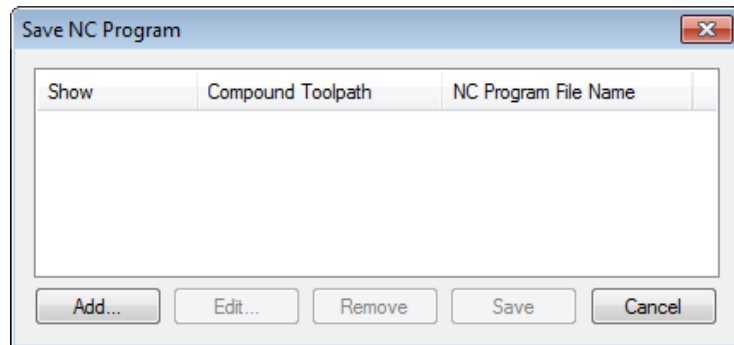
The dialog window titled "Single Contour Cut" contains several sections for configuring wire-cut parameters:


- Toolpath Parameters:**
 - Toolpath: Toolpath1
 - Tool name: 1
 - Rapid feed rate: 700
 - Working feed rate: 20
 - Allowance: 0
 - Number of passes: 1
- Wire-cut Parameters:**
 - Dielectric parameter: 1
 - Wind parameter: 1
 - Generator parameter: 1
 - Offset value: 1
- Part Parameters:**
 - Slope angle: 0
 - Part thickness: 30
- Radius Correction:**
 - ☐ On
 - ☒ Left ☐ Number: 1
 - ☐ Right ☐ Value: 0
- Loops and Micro Joint:**
 - Loops—
 - Loop Radius: 0
 - Tolerance Angle: 0
 - Micro Joint—
 - Width of Micro Joint: 0
- Approximation:**
 - Precision: 0.1
 - ☒ Circular interpolation
- Additional Parameters:**
 - ☐ Nontangential
 - ☐ Counter contour
 - ☐ With degeneration
 - ☐ Contour correction
 - ☐ All contours
 - ☐ Round R: 0 0
 - ☐ M01 X: 0 Y: 0




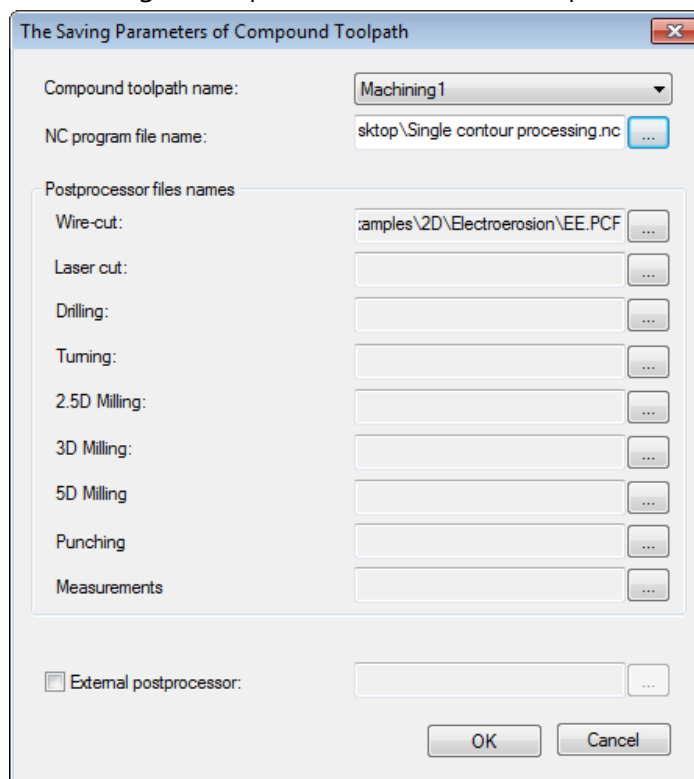
Once the toolpath is calculated, the user can save the respective control sequence. A new postprocessor can be created with the help of the "Postprocessor Editor", if it was not yet available in the postprocessors library shipped with the system.



Once the desired postprocessor is selected, the user can run the control sequence generation (saving). To do this, select the textual menu command "CNC>Save NC Program" or click the  button on the "Base Module" toolbar.

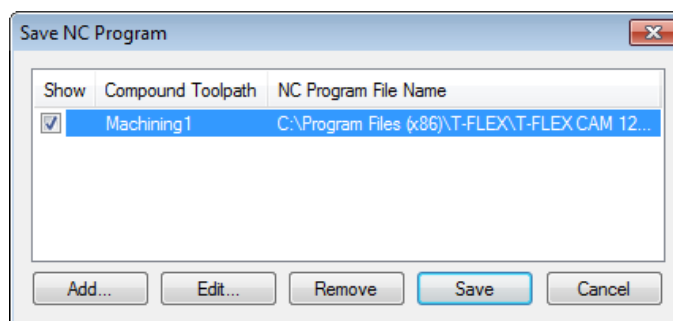



Double-click  inside the empty pane of the dialog. This will bring up the "Compound Toolpath Saving Parameters" dialog box. In this dialog box, specify one by one the names of the necessary postprocessors


for the selected machining type, the name of the control sequence and the location where you want to save it. Whenever you need to change a file path, use the  button provided next to relevant boxes.

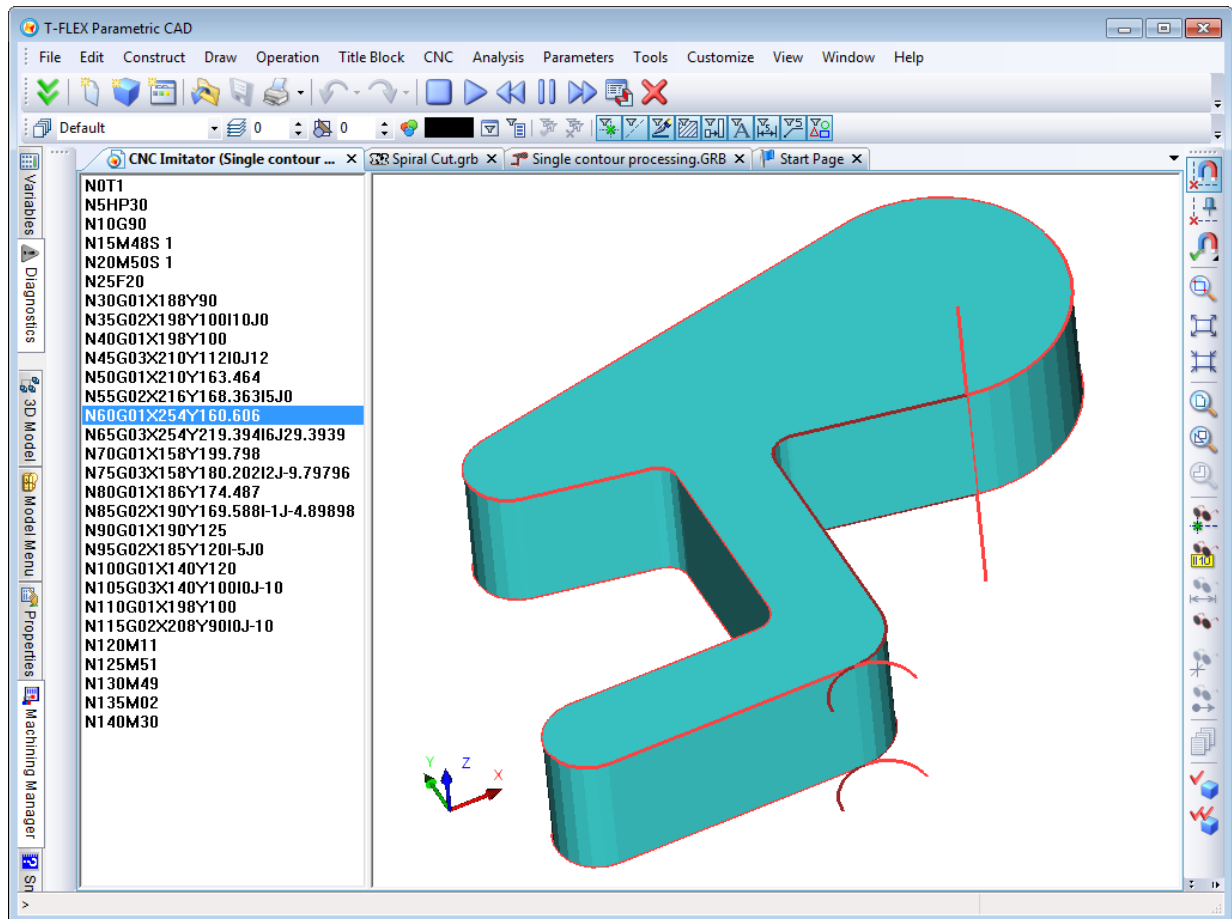


Once done with creating the paths, the user shall click the  button in order to save the control sequence saving parameters. Clicking this button brings the user to the previous dialog window, where a line is displayed with a brief account of main control sequence parameters. To save a control sequence, including the specified postprocessor, the user shall click the  button. We recommend enabling also the "Show" option. In this case, the saved control sequence will automatically open in the "Notepad" MS Windows editor.

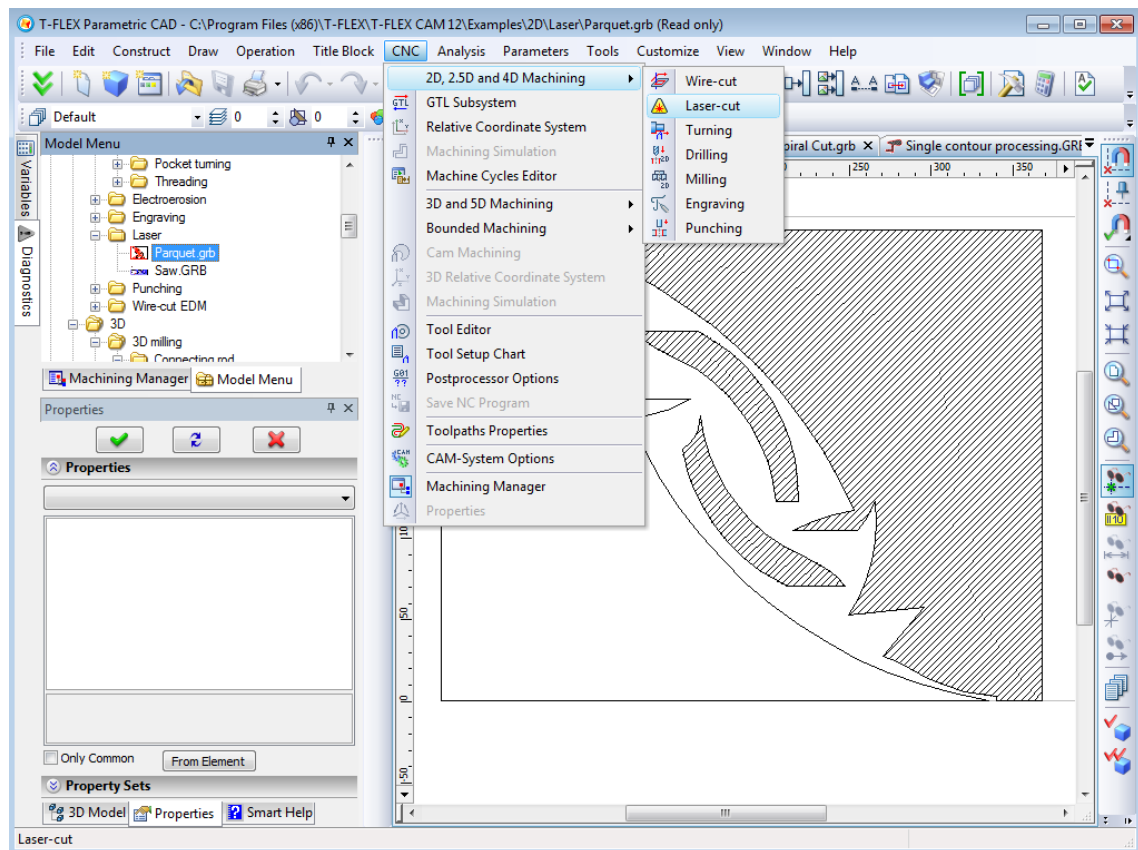






After performing all the steps described above, the user will be able to watch the playback of the generated control sequence by calling **"CNC>Machining simulation"** or by clicking the  button on



the main toolbar. For launching the machining simulator the user needs to press the  button on the main toolbar in the «CNC Simulation» mode, which appears automatically.




Let's review the example of angular cutting on a more complex part. To demonstrate similarity of the two types of machining, we will review the angular cutting toolpath calculation and the control sequence generation on the example of laser cutting. The difference from wire cutting is that the controlled tool in laser cutting is the laser beam. There are also a number of specific parameters that need to be defined before calculating the part machining toolpath. A detailed list of such parameters was presented in the first part of the manual.

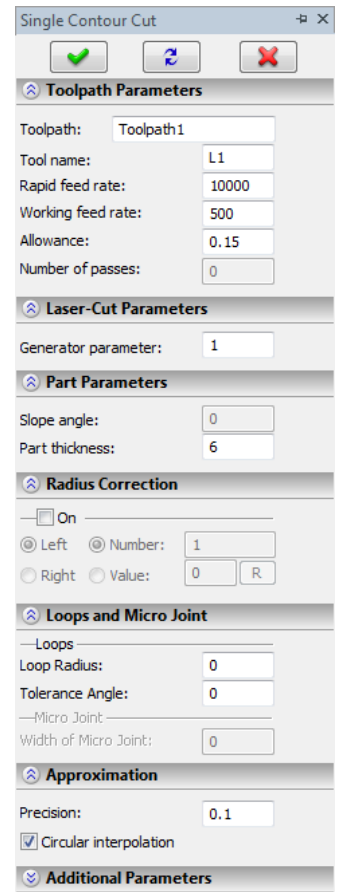


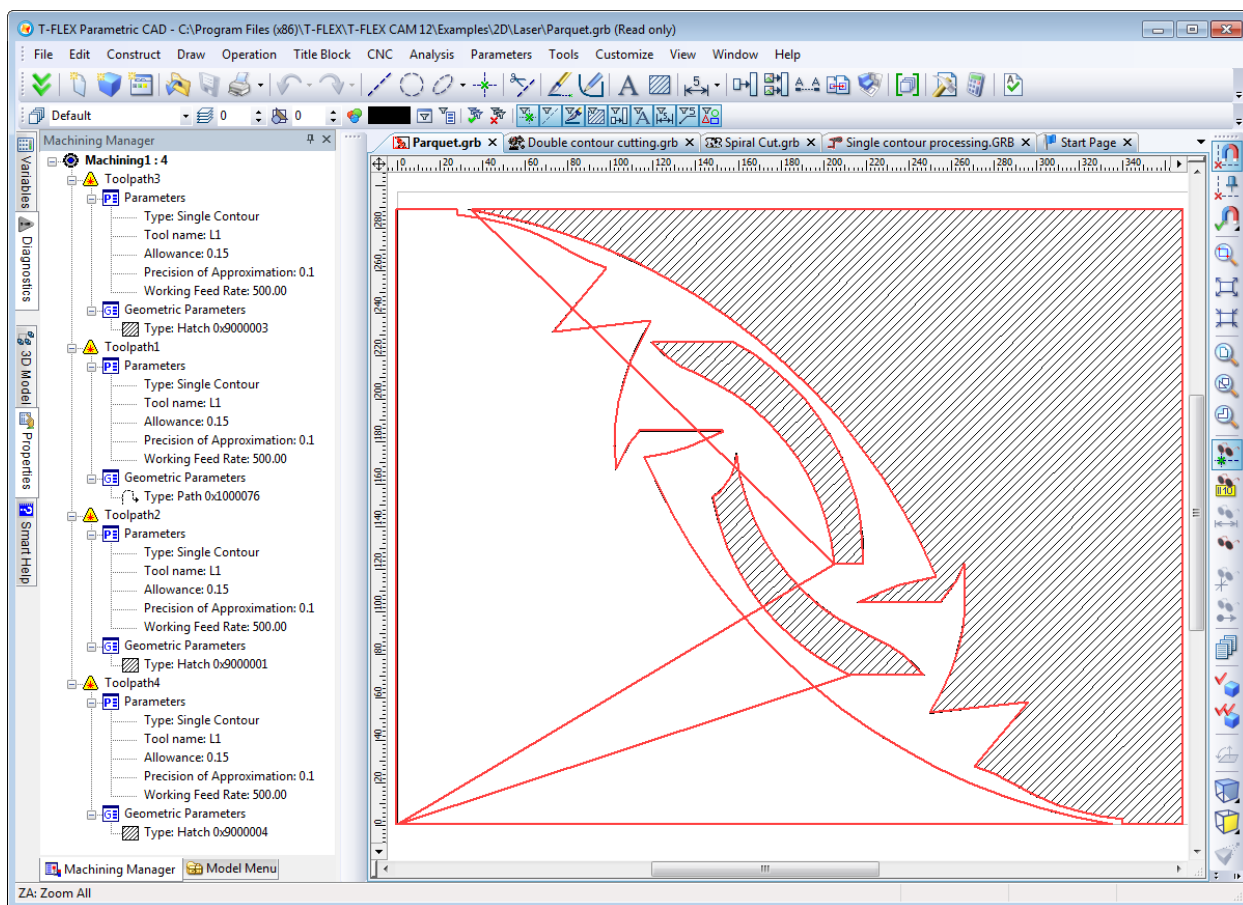
After constructing hatches and paths by a specified contour, proceed to the menu of the laser cutting module commands by clicking the  button on the toolbar or by selecting the textual menu **"CNC>2D, 2.5D and 4D Machining>Laser-cut"**. Next, to create an angular laser-cutting toolpath, click the  button on the tool bar. The laser cutting module provides for a capability to define part geometry either by using a hatch or by using the "Path" element, therefore, all the respective buttons will be available on the toolbar. In this case, we will use both the hatches and the path, since the parquet contour involves several contours to be machined. In the case of creating hatch-based toolpaths, click the  button. After that the system will automatically transition to the mode of waiting for contour (hatch) selection for machining the part, with the toolpath parameters dialog window being displayed simultaneously. It is possible to select the hatch first, and then specify the toolpath parameters, while it is also possible to first define the toolpath parameters, and then select the machining contour (hatch) by clicking  inside the hatch contour. You shall create a new toolpath for each of the contours by repeating the actions described above. Several separate selected contours cannot be machined using a single created toolpath.

In case of creating a path-based toolpath, click the  button. After that the system will automatically transition to the mode of waiting for contour (path) selection for machining the part, with the toolpath parameters dialog window being displayed simultaneously. To select a path, the user needs to click  on any of the path's segments.


After selecting the machining contour, the user defines all the rest of the necessary machining parameters in the coming up dialog box, as was described in the first part of this manual.

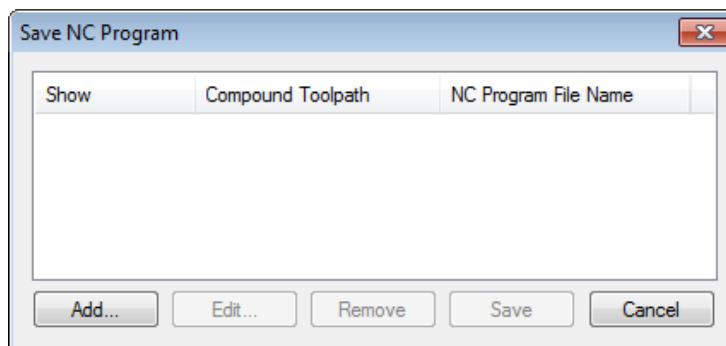
Once all necessary changes are done to the machining parameters, the user just needs to click the  button to have the system calculate the toolpath. The toolpath will appear on the drawing as a colored line.







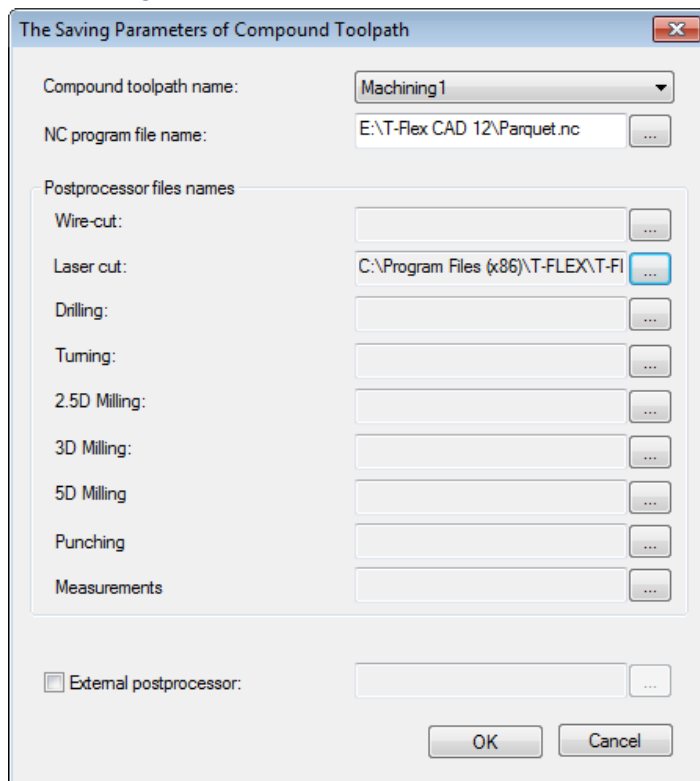
Once the toolpath is calculated, the user can save the respective control sequence. A new postprocessor can be created with the help of the "Postprocessor Editor", if it was not yet available in the postprocessors library shipped with the system.



Once the desired postprocessor is selected, the user can run the control sequence generation (saving). You can use the **"CNC>Save NC Program"** command for this purpose, or, alternatively, click the  button on the "CAM" bar.

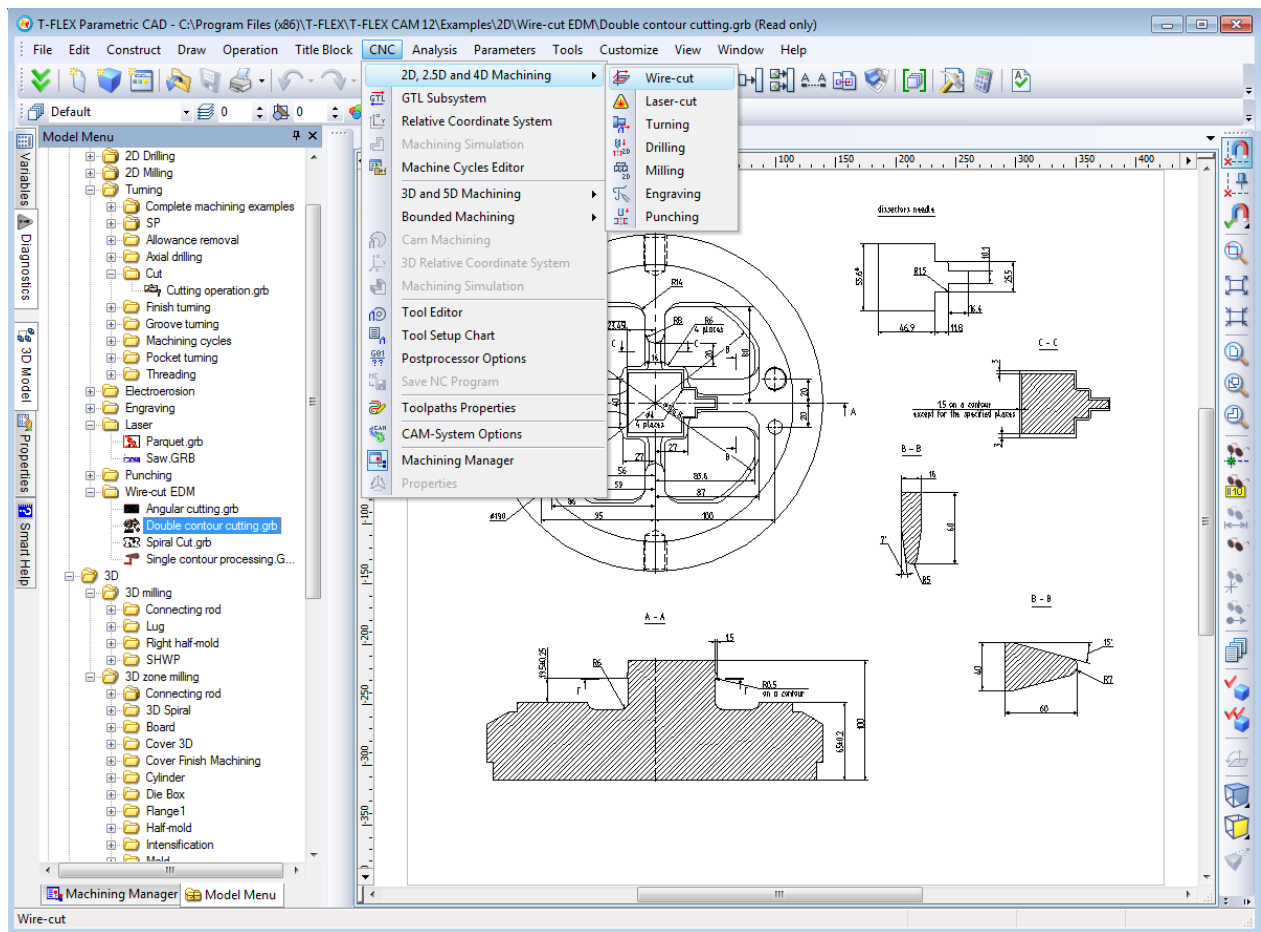







Double-click  inside the empty pane of the dialog. This will bring up the "Compound Toolpath Saving Parameters" dialog box. In this dialog box, specify one by one the names of the necessary postprocessors

for the selected machining type, the name of the control sequence and the location where you want to save it. Whenever you need to change a file path, use the  button provided next to relevant boxes.




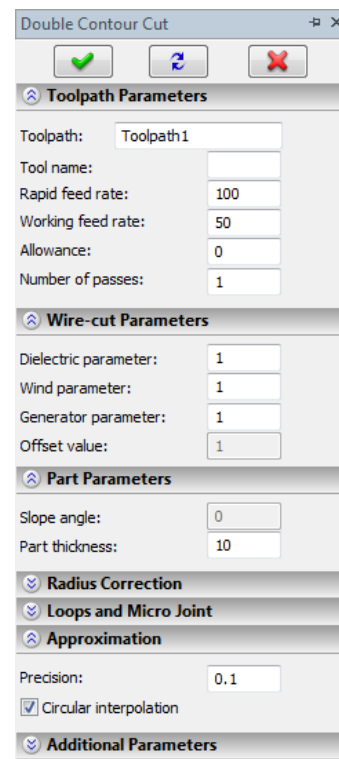
After performing all the steps described above, the user can view the playback of the generated control sequence by using the **"CNC>Machining simulation"** command or by clicking the  button on the main toolbar. For launching the machining simulator the user should press the  button on the main toolbar in the «CNC Simulation» mode, which appears automatically.



After constructing the two required hatches based on the top and the bottom contours of the punch being cut out, proceed to the menu of the wire cutting module commands by clicking the  on the main toolbar in the «CNC 2D» mode, or by selecting the textual menu "CNC>2D, 2.5D and 4D Machining>Wire-cut". Next, to create a double-contour wire cutting toolpath, click the button  on the tool bar. The wire cutting module provides for a capability to define part geometry either by using a hatch or by using the "Path" element, therefore the respective buttons will be available on the toolbar. In this case, click the  button (hatch). After that the system will automatically transition to the mode of waiting for the first contour (hatch) selection for machining the part, with the toolpath parameters dialog window being displayed simultaneously. It is possible to select the hatches first, and then specify the toolpath parameters, while it is also possible to first define the toolpath parameters, and then select the machining contour (hatches) by clicking  once and inside each hatch contour again. As soon as the user selects the first part contour, the system automatically goes into the second hatch selection mode (the  button becomes pushed automatically).

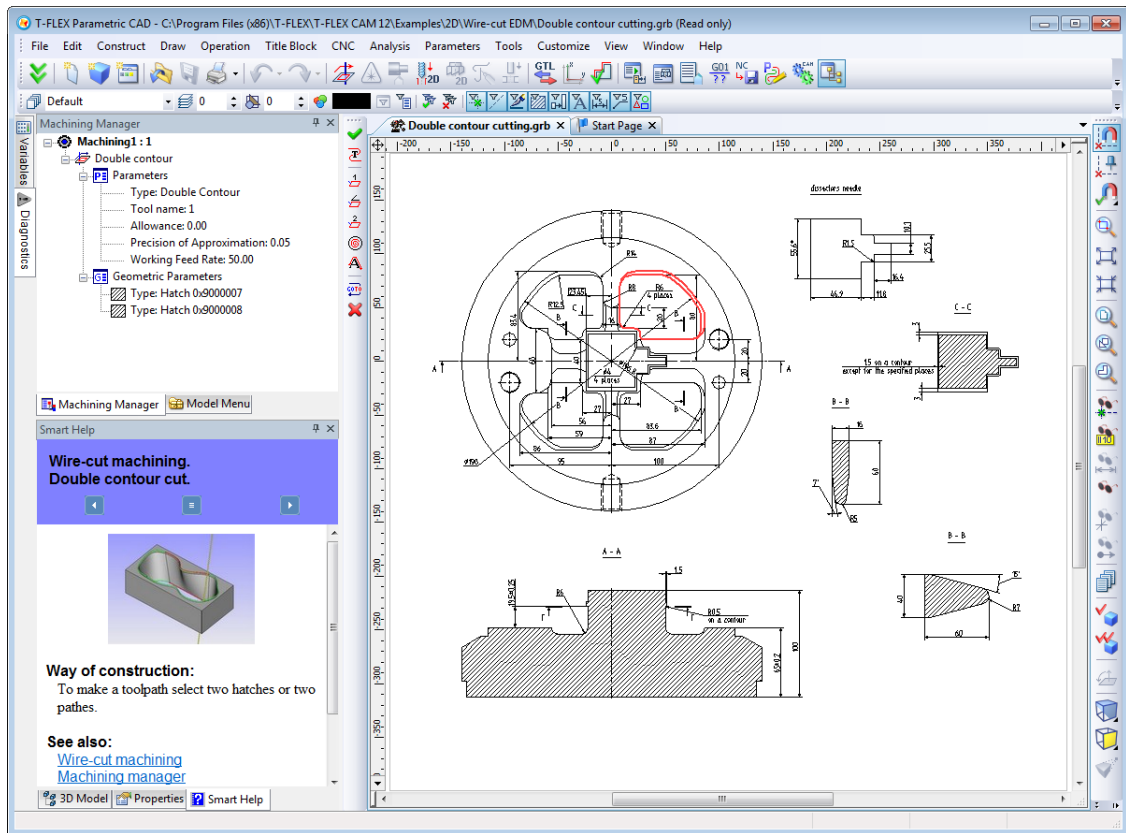
The machining toolpath parameters are described in the first part of the manual.

Once all necessary changes are made to the machining parameters, you just need to click the  button to have the system calculate the toolpath. The toolpath will appear in the drawing as a red line.




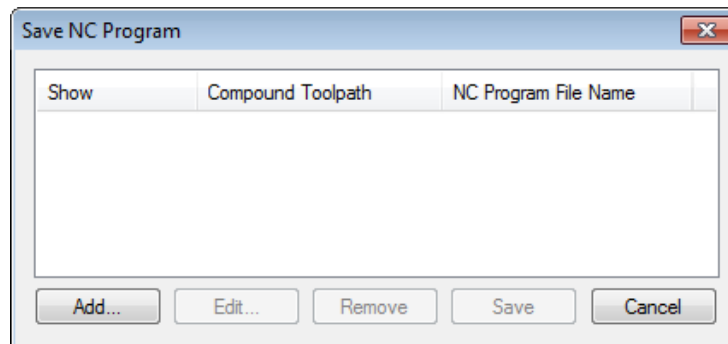
The screenshot shows the 'Double Contour Cut' dialog box with the following sections and parameters:


- Toolpath Parameters**
 - Toolpath: Toolpath1
 - Tool name:
 - Rapid feed rate: 100
 - Working feed rate: 50
 - Allowance: 0
 - Number of passes: 1
- Wire-cut Parameters**
 - Dielectric parameter: 1
 - Wind parameter: 1
 - Generator parameter: 1
 - Offset value: 1
- Part Parameters**
 - Slope angle: 0
 - Part thickness: 10
- Radius Correction**
- Loops and Micro Joint**
- Approximation**
 - Precision: 0.1
 - ☒ Circular interpolation
- Additional Parameters**




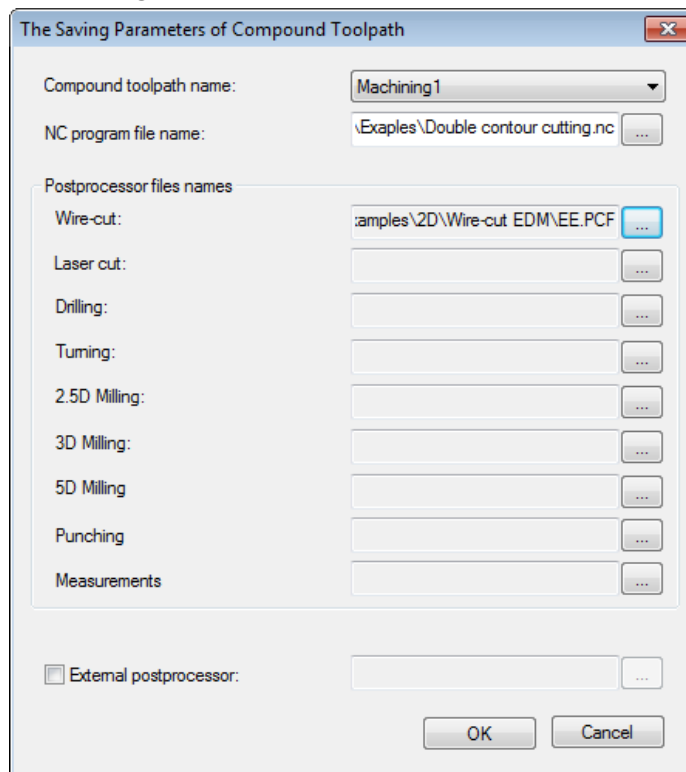
Once the toolpath is calculated, the user can save the respective control sequence. A new postprocessor can be created with the help of the "Postprocessor Editor", if it was not yet available in the postprocessors library shipped with the system.



Once the desired postprocessor is selected, the user can run the control sequence generation (saving). The **"CNC>Save NC Program"** command is used for this purpose, or, alternatively, click the  button on the "CNC" bar.

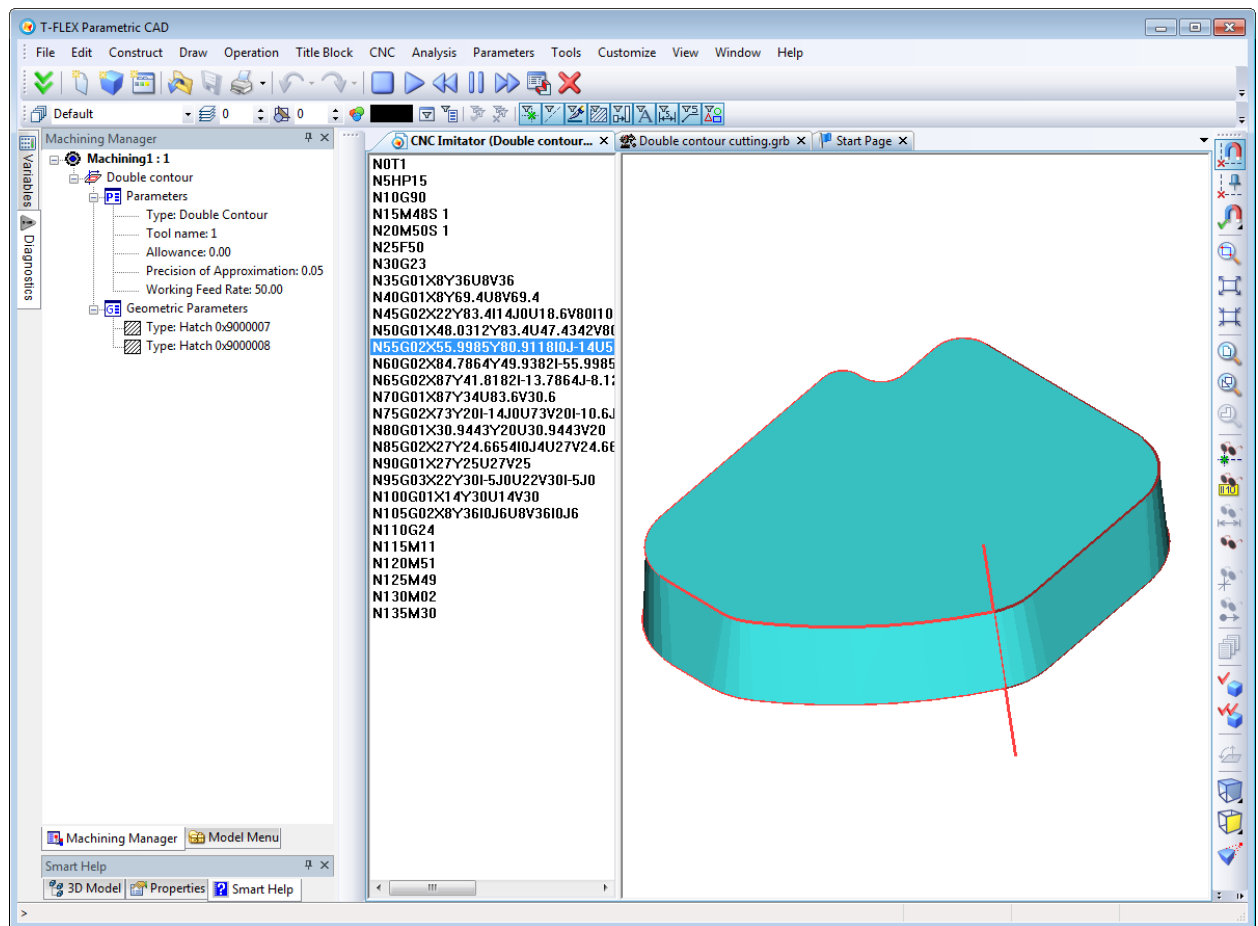



Double-click  inside the empty pane of the dialog. This will bring up the "Compound Toolpath Saving Parameters" dialog box. In this dialog box, specify one by one the names of the necessary postprocessors

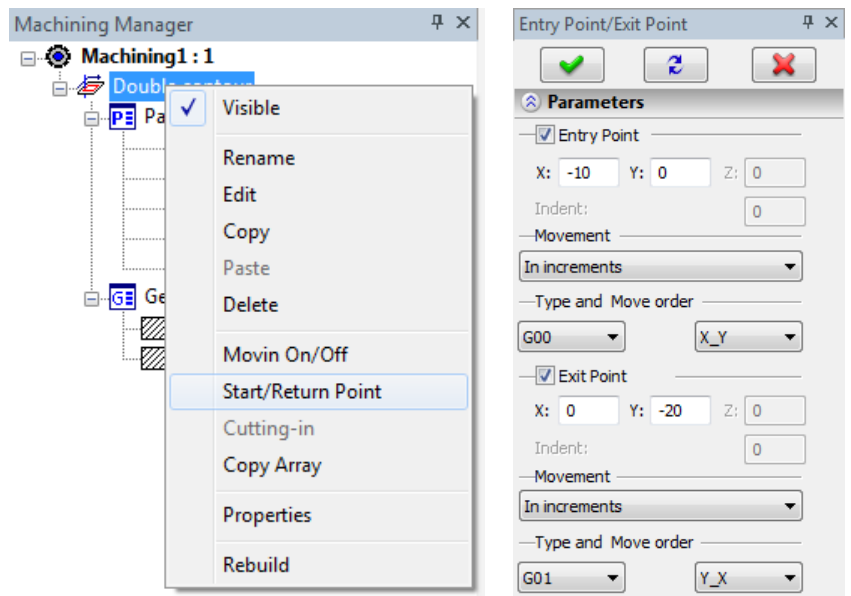
for the selected machining type, the name of the control sequence and the location where you want to save it. Whenever you need to change a file path, use the  button provided next to relevant boxes.



After performing all the steps described above, the user can view the playback of the generated control sequence by using the “**CNC>Machining simulation**” command or by clicking the  button on the main toolbar. For launching machining simulator the user needs to press the  button on the main toolbar in the «CNC Simulation» mode.

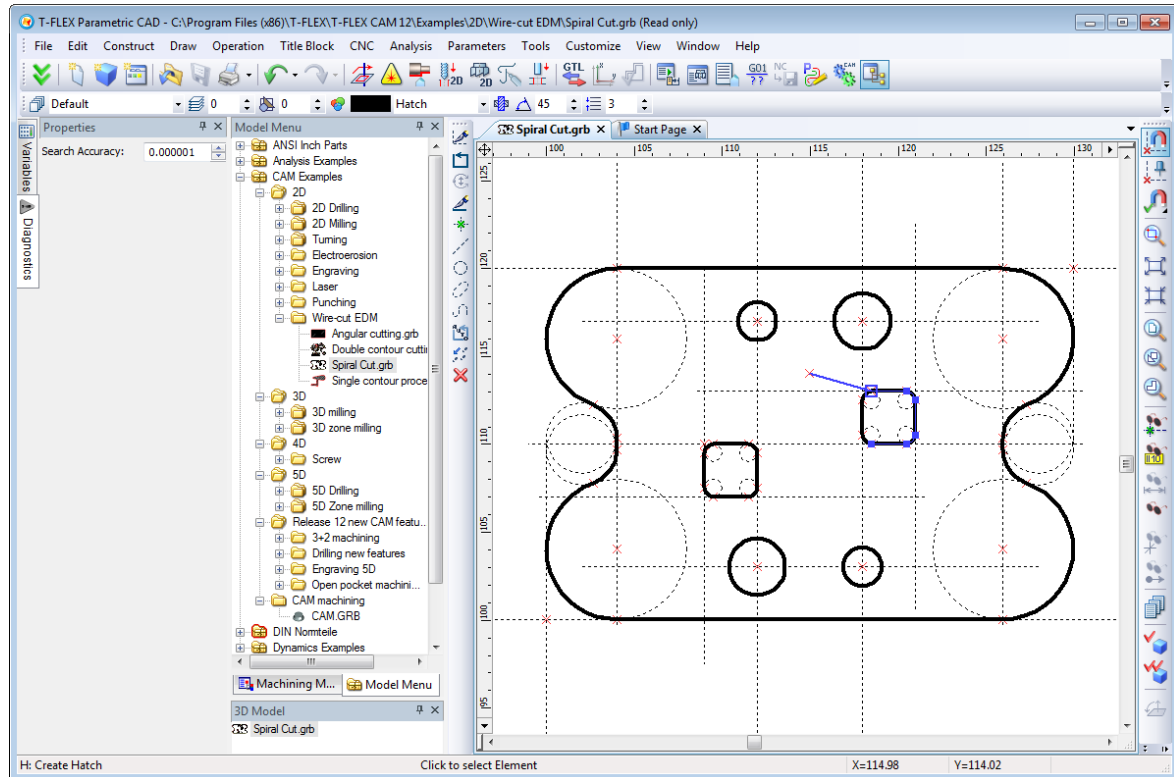






The user can specify the tool approach and retreat points within the “Machining Manager” . The particular numerical values of the approach and retreat points are entered in the provided dialog box.




We will now review the final example provided in the current section of the second part of the manual.

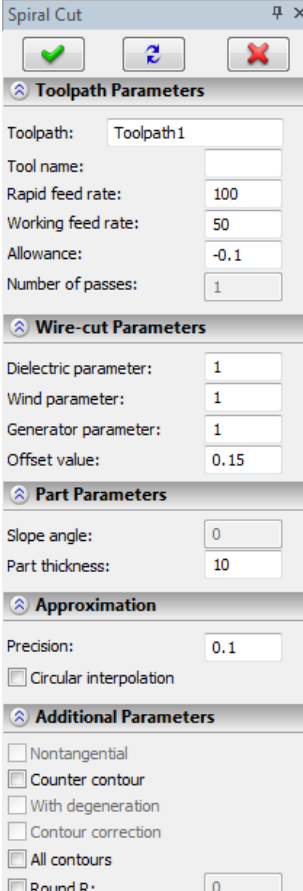
This is spiral material removal by wire cutting. Let us select another part. Note that the present machining type is used rather seldom in production.



After constructing a hatch by the contour of the hole containing the material to be removed, proceed to the menu of the wire cutting module commands by clicking the  on the main toolbar in the «CNC 2D» mode or by selecting the textual menu "CNC>2D, 2.5D and 4D Machining>Wire-cut". Next, to create a spiral material removal toolpath, click the  button on the tool bar. In this case, the wire cutting module provides for geometry to be defined by a hatch only, therefore the  button will automatically be pushed. After that, the system will automatically transition to the mode of waiting for contour (hatch) selection for machining the part, with the toolpath parameters dialog window being displayed simultaneously. It is possible to select the hatch first, and then specify the toolpath parameters, while it is also possible to first define the toolpath parameters, and then select the machining contour (hatch) by clicking  inside the hatch contour.

The machining toolpath parameters are described in the first part of the manual.

Once all necessary changes are made to the machining parameters, you just need to click the  button to have the system calculate the toolpath. The toolpath will appear in the drawing as a red line.



Spiral Cut

✓ ↺ ✕

Toolpath Parameters

Toolpath: Toolpath1

Tool name:

Rapid feed rate:

Working feed rate:

Allowance:

Number of passes:

Wire-cut Parameters

Dielectric parameter:

Wind parameter:

Generator parameter:

Offset value:

Part Parameters

Slope angle:

Part thickness:

Approximation

Precision:

☐ Circular interpolation

Additional Parameters

☐ Nontangential

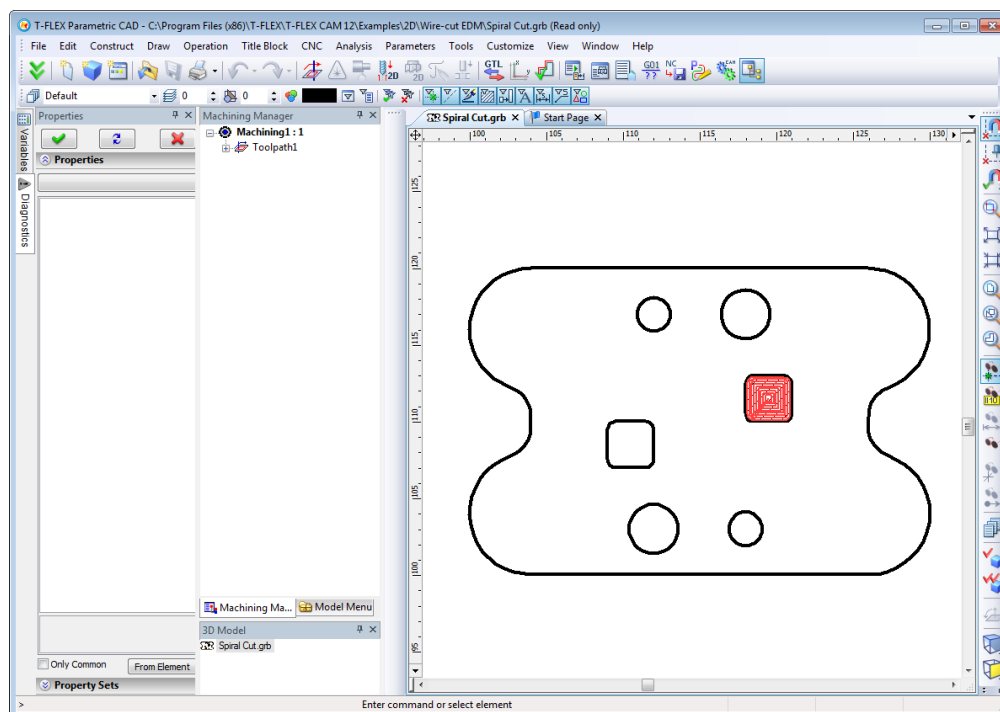
☐ Counter contour

☐ With degeneration


☐ Contour correction

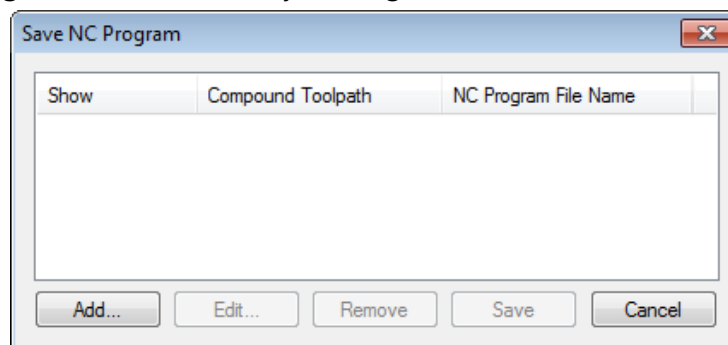
☒ All contours



☐ Round R:

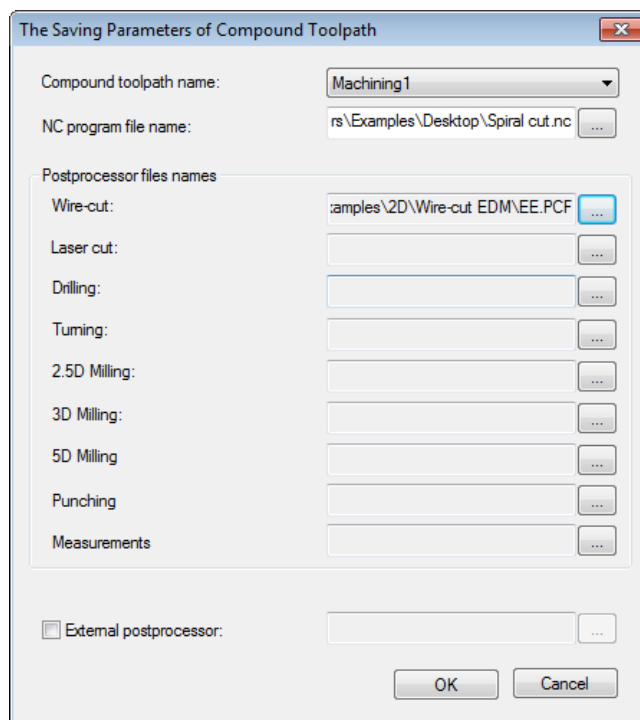




Once the toolpath is calculated, the user can save the respective control sequence. A new postprocessor can be created with the help of the "Postprocessor Editor", if it was not yet available in the postprocessors library shipped with the system.

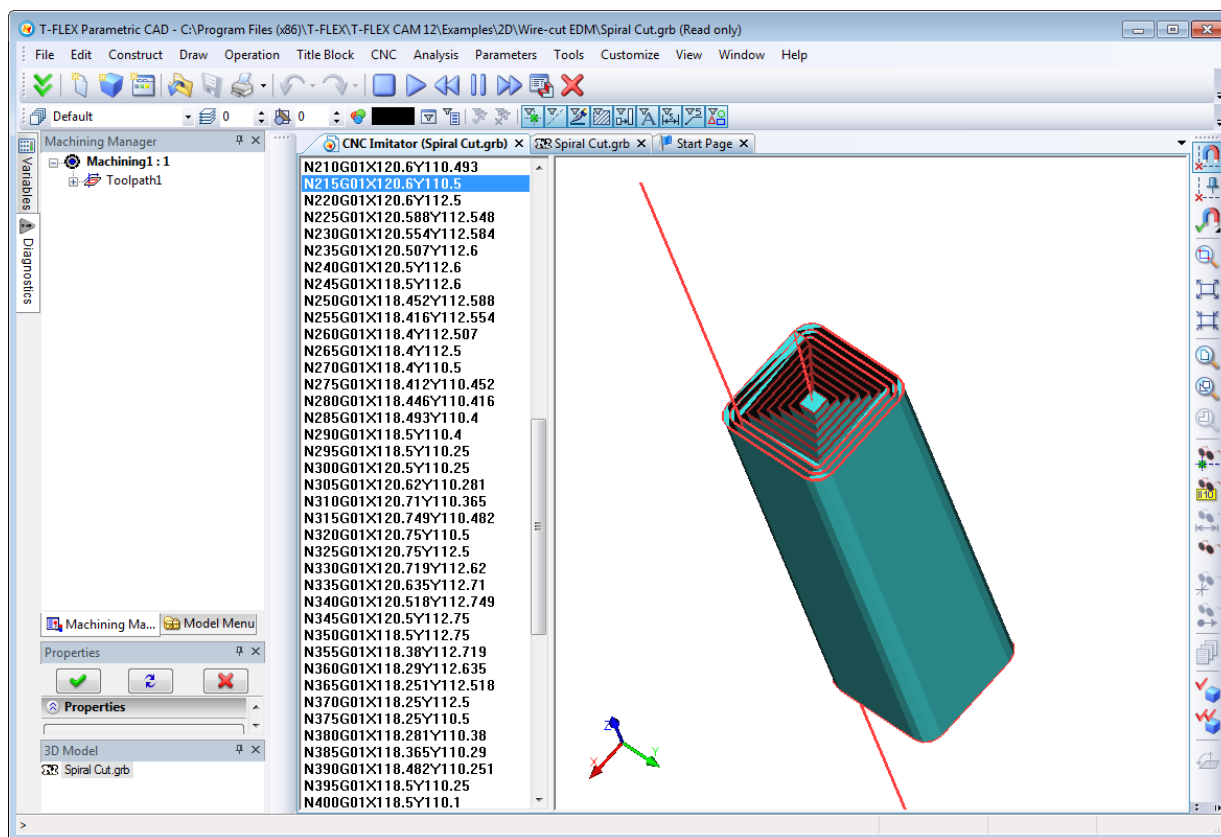
Once the desired postprocessor is selected, the user can run the control sequence generation (saving) by the "CNC>Save NC program" command or by clicking the  button on the toolbar.



Double-click  inside the empty pane of the coming up "Save NC program" dialog. This will bring up the "Compound Toolpath Saving Parameters" dialog box. In this dialog box, specify one by one the names of the necessary postprocessors for the selected machining type, the name of the control sequence and the location where you want to save it. Whenever you need to change a file path, use the  button provided next to relevant boxes.

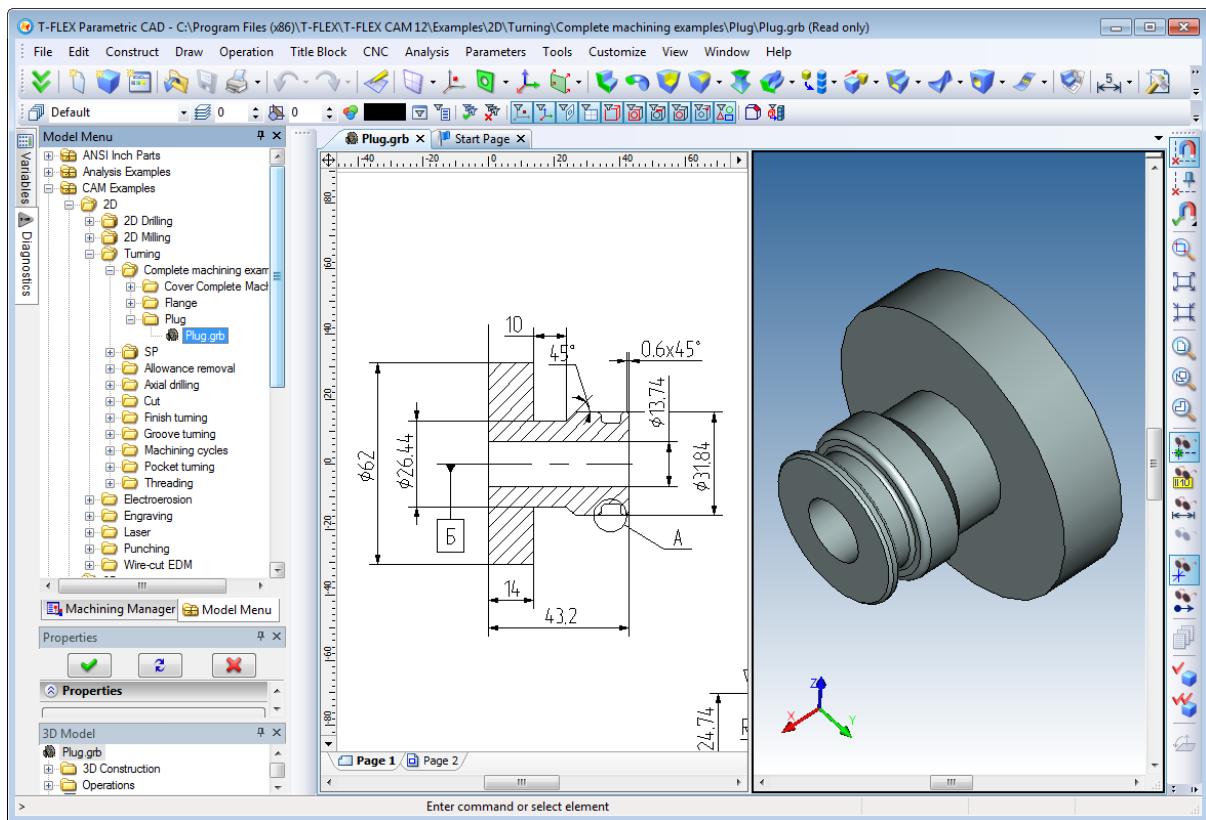


In addition, after performing all the steps described above, the user can view the playback of the generated control sequence by using the “**CNC>Machining simulation**” command or by clicking the  button on the main toolbar. For launching machining simulator the user should press the  button on the main toolbar in the «CNC Simulation» mode. Similar to the machining examples described above, the simulator will only display the region of the part defined by a closed path or a hatch.






TURNING

Let's review a turning example on a sample bushing (shown on the figure below). We will calculate the toolpaths for allowance removal, finishing along the contour, and grooving. After that, we will generate the control sequence for the part shown on the figure.








To begin with, construct several paths in the drawing. To perform machining by turning, we shall construct the path that is the part contour as intended to be obtained as a result of the machining. After that, we shall specify the contour of the bar from which this part will be cut. The workpiece contour shall start at the first node of the contour path (as described above) and end at its last node. Grooves will be cut using the specialized operations from the "Groove Turning" group. Their use does not require construction of any additional elements. At this point, we can regard the geometrical, or preliminary, step of toolpath creation as completed. Let's proceed to creating the actual toolpaths and then to creating the control sequence.

For allowance removal, in the turning machining functionality select the  group, and then the «Allowance removal by passing cutter» option . Upon selecting this machining mode, subsequently select two paths. First, select the part contour, which is the target of machining, and then specify the original workpiece contour subject to allowance removal. Note that, unlike the case of the contour turning machining, the system will automatically prompt for the second machining path. At the same time, a window will open with the toolpath parameters, which the user can define before selecting, while selecting or after selecting the paths.

Note that the user can preset the default machining parameters in a special dialog box that appears upon calling the  option. In this dialog box, the user can define such parameters as the file with tool data, the tool name and several other parameters.

After selecting the machining contour, the user defines all the rest of the necessary machining parameters in the coming up dialog box, as described in the first part of this manual.

Allowance Removal (Pass cutter)  

Toolpath Parameters

Toolpath: Toolpath2

Tool file: C:\Program Files (x86)\T-FLEX ...

Tool: prohod

Rapid feed rate: 100

Working feed rate on X dir.: 30

Working feed rate on Z dir.: 30

Coefficient for X/Z: 1

Allowance: 0.5

☐ Cooler ON

Spindle rotation direction:
☒ CW ☐ CCW

Spindle RPM: 200

Pass Parameters

Undertravel value: 2

Overtravel value: 1

Overtravel angle: 270

☐ Through pass 0

☒ Ignore pockets

Offset parameters:
☐ Auto ☒ Value: 0.01

Turning Direction

☐ In X direction ☐ To the right
☒ In Z direction ☐ Downwards

Stock Removal Parameters

Stock Type:
☐ On passes: 4
☒ By value: 1.5

Semi-Finished Pass



Radius Correction




☐ On
☒ Left ☐ Number: 0
☐ Right ☐ Value: 0 R

Approximation

Precision: 0.01

☐ Circular interpolation

Finish Turning  

Toolpath Parameters

Toolpath: Toolpath3

Tool file: C:\Program Files (x86)\T-FLEX ...

Tool: prohod

Rapid feed rate: 100

Working feed rate on X dir.: 30

Working feed rate on Z dir.: 30

Coefficient for X/Z: 1

Allowance: 0

☐ Cooler ON

Spindle rotation direction:
☒ CW ☐ CCW

Spindle RPM: 200

Counter Parameters

☐ Elementwise

Radius Correction

☐ On
☒ Left ☐ Number: 0
☐ Right ☐ Value: 0 R

Approximation


Precision: 0.1

☒ Circular interpolation

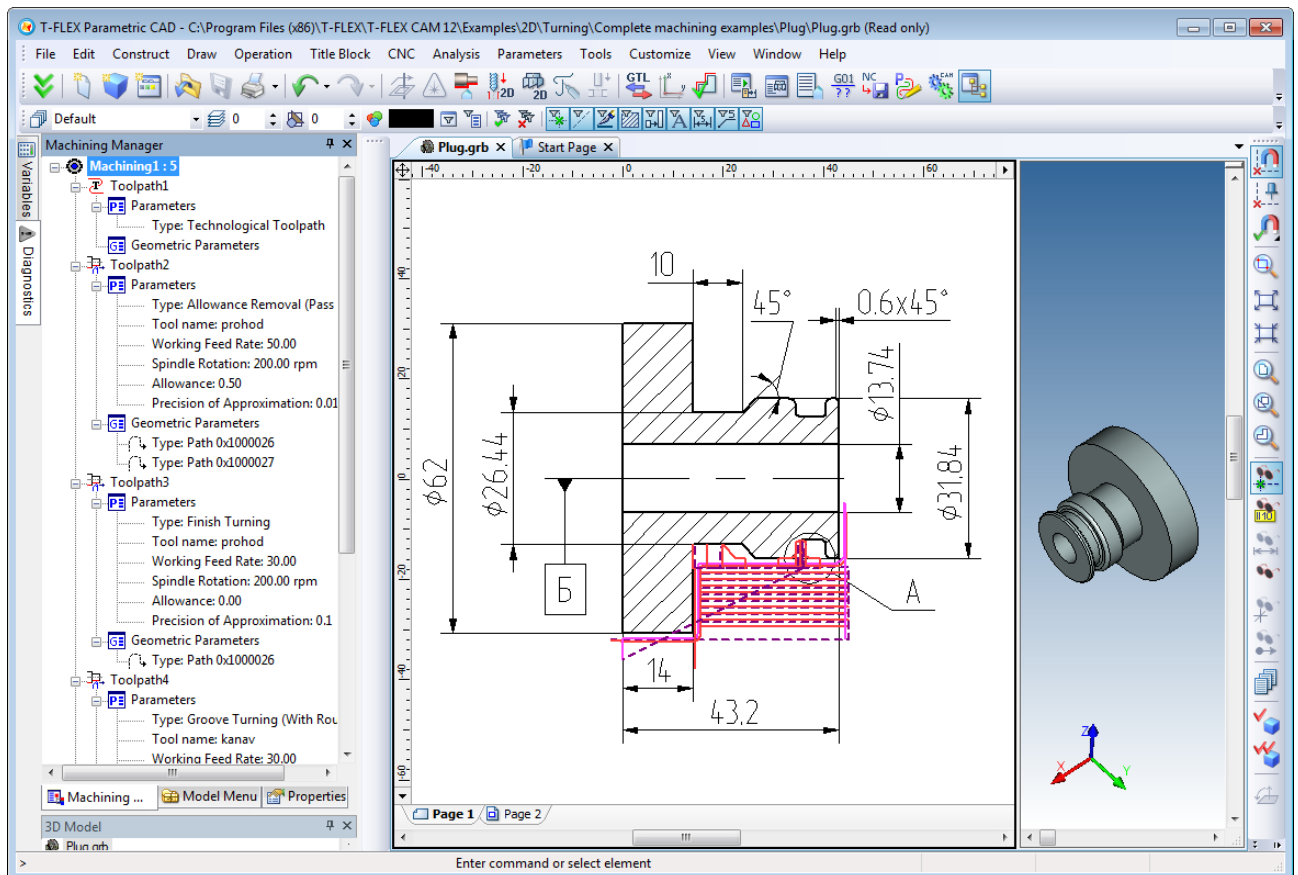
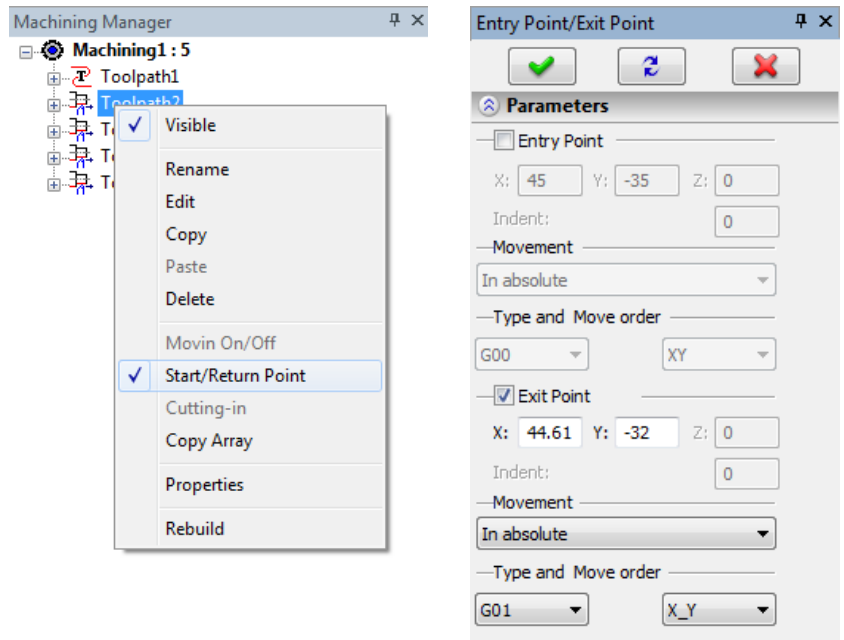
Additional Parameters


☒ Nontangential
☒ Counter contour
☐ With degeneration
☐ Contour correction

☐ Round R: 0

Once the machining toolpath has been created, the user can specify the approach and retreat points in the appropriate dialog box. To call this dialog, click the  icon on the main toolbar in the «CNC 2D» mode or call the context menu of the given toolpath in the Machining Manager window, and in it select the [Start/Return Point] option.



As a result of all the above steps performed in the described order, the user will get the calculated toolpaths displayed on the screen as colored lines.



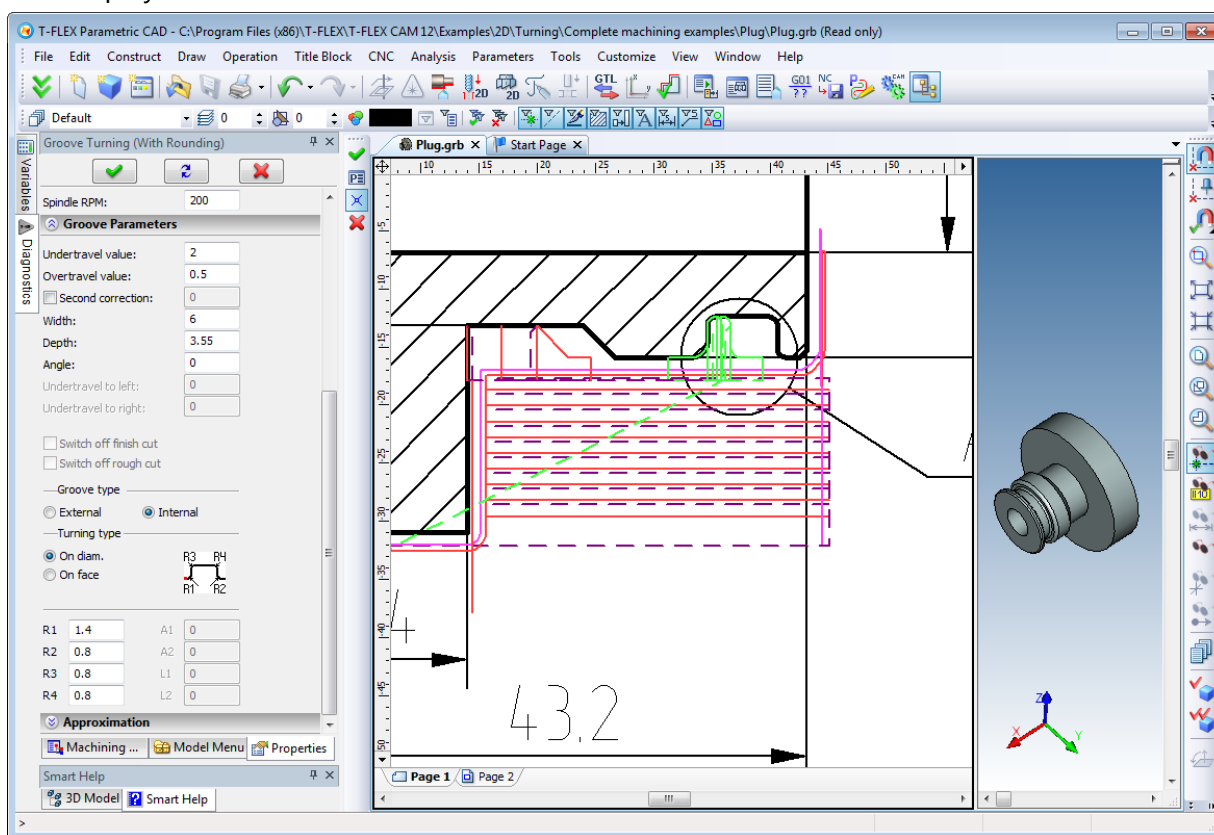
For finishing, in the «Turning» menu select the «Finish Turning of contour» option . Upon selecting this machining mode, specify the path that defines the final part contour. To do this, one can use the same path of the part contour as was used when creating the allowance removal toolpath. After selecting the machining contour, the user defines all the rest of the necessary machining parameters in the coming up dialog box, as described in the first part of this manual.



As a result of all the above steps performed in the described order, the user will get the calculated toolpaths displayed on the screen as colored lines.

Once the machining toolpath has been modeled, the user specifies the coordinates of the approach and retreat points in a special dialog box.

For grooving with rounds, select the «Finish turning of groove» group  and then operation – «Finish turning with rounding» . Upon selecting this machining mode, select a node. The requirements for selecting a node are described in the first part of the manual. After selecting the node, the user defines all the rest of the necessary machining parameters in the coming up dialog box, as described in the first part of this manual.

As a result of all the above steps performed in the described order, the user will get the calculated toolpaths displayed on the screen as colored lines.



As in the previous operation, for finish turning of groove with facets select the «Finish turning of groove» group  and then select operation – «Finish turning with facets» . Upon selecting the machining mode, select a node. The node selection procedure is described in the first part of the manual. After selecting the node, the user defines all the rest of the necessary machining parameters in the upcoming dialog window, as described in the first part of this manual.

As a result of all of the above steps performed in the described order, the user will get the calculated toolpaths displayed in the screen.

Groove Turning (With Rounding) [Maximize] [Close]

☒ Cooler ON

— Spindle rotation direction —

☒ CW ☐ CCW

Spindle RPM: 200

Groove Parameters

Undertavel value: 2

Overtravel value: 0.5

☐ Second correction: 0

Width: 6

Depth: 3.55

Angle: 0

Undertavel to left: 0

Undertavel to right: 0


☐ Switch off finish cut

☐ Switch off rough cut

— Groove type —

☐ External ☒ Internal

— Turning type —

☒ On diam. 

☐ On face

R1	1.4	A1	0
R2	0.8	A2	0
R3	0.8	L1	0
R4	0.8	L2	0

Approximation

Precision: 0.1

☒ Circular interpolation

Groove Turning (With Rounding) [Maximize] [Close]

☒ Cooler ON

— Spindle rotation direction —

☒ CW ☐ CCW

Spindle RPM: 200

Toolpath Parameters

Toolpath: Toolpath4

— Tool file: —

C:\Program Files (x86)\T-FLEX ...

Tool: kanav

Rapid feed rate: 100

Working feed rate on X dir.: 30

Working feed rate on Z dir.: 30

Coefficient for X/Z: 1

Allowance: 0

☐ Cooler ON

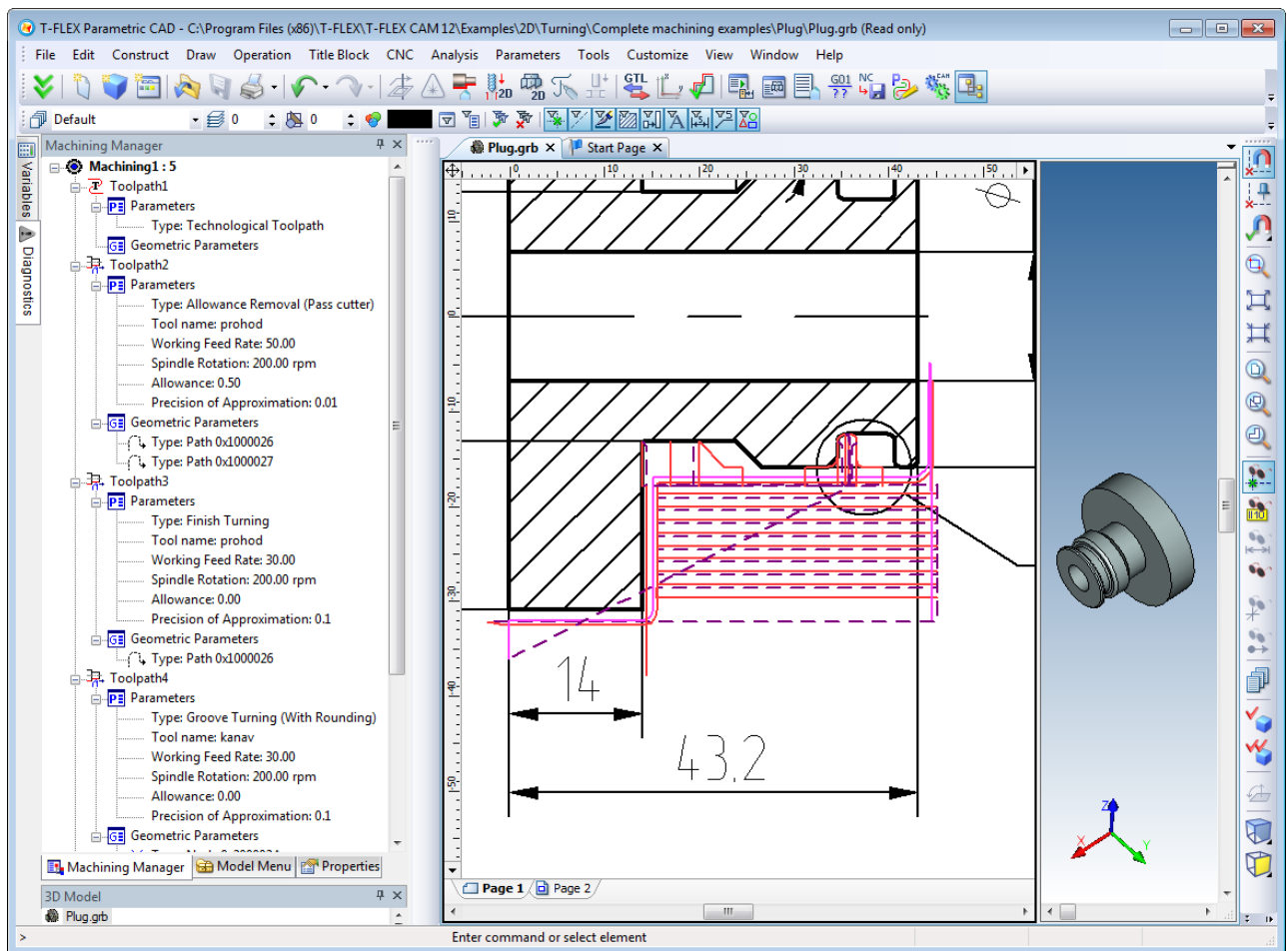
— Spindle rotation direction —

☒ CW ☐ CCW

Spindle RPM: 200

Groove Parameters



Approximation

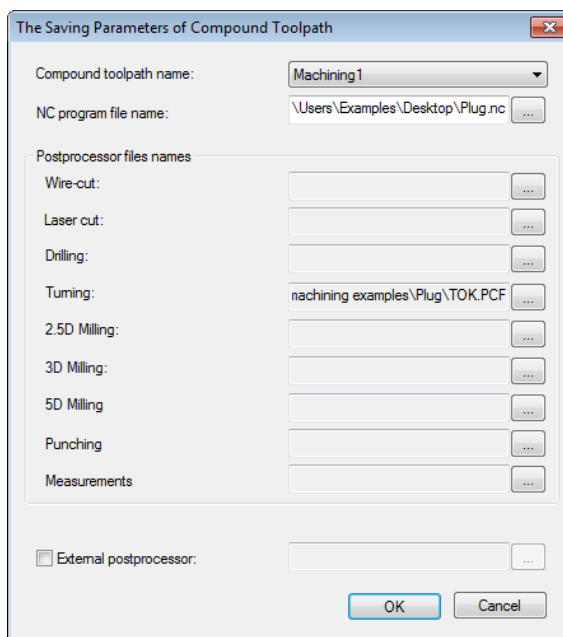


Once the compound toolpath is calculated, the user can save the respective control sequence. For this purpose, a new postprocessor shall be created within "Postprocessor Editor", if necessary. In this case, additional control sequence options for turning-type machining are available, which you can use when creating the postprocessor.

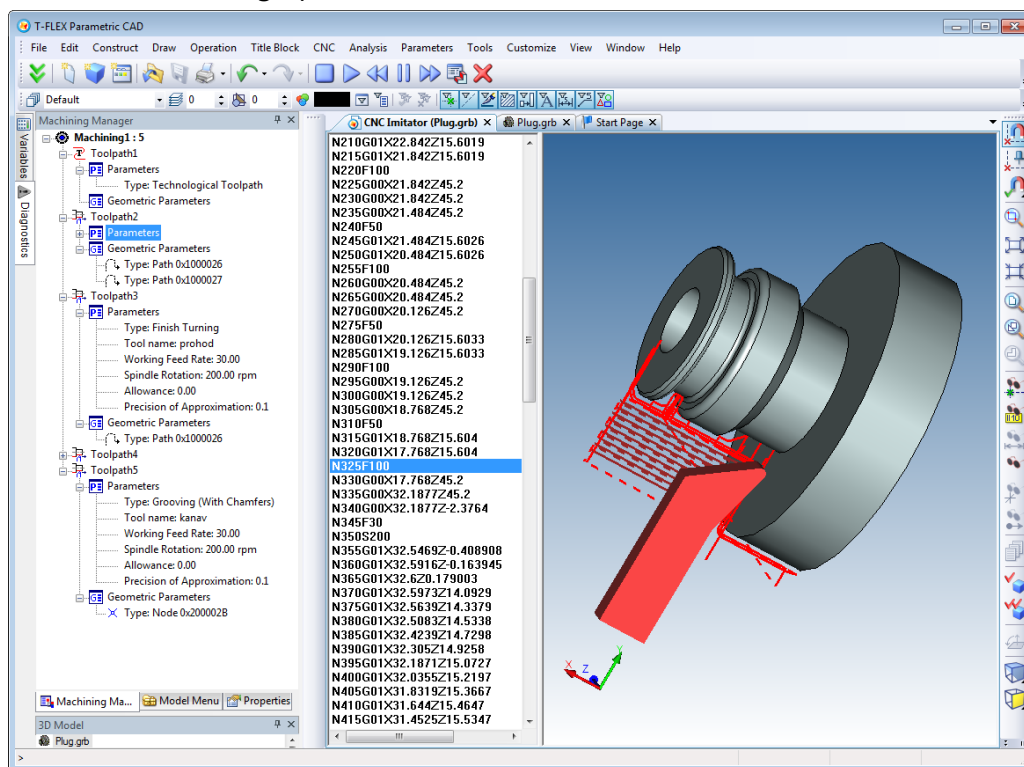
Otherwise, the user shall select a postprocessor from the library, which will be used to save the control sequence.

Once the desired postprocessor is selected, the user can run the control sequence generation (saving) by calling the **"CNC>Save NC program"** command.


Double-click   inside the empty pane of the dialog coming up on the screen. This will bring up the "Compound Toolpath Saving Parameters" dialog box. In this dialog box, specify one by one the names of the necessary postprocessors for the selected machining type, the name of the control sequence and the location where you want to save it.

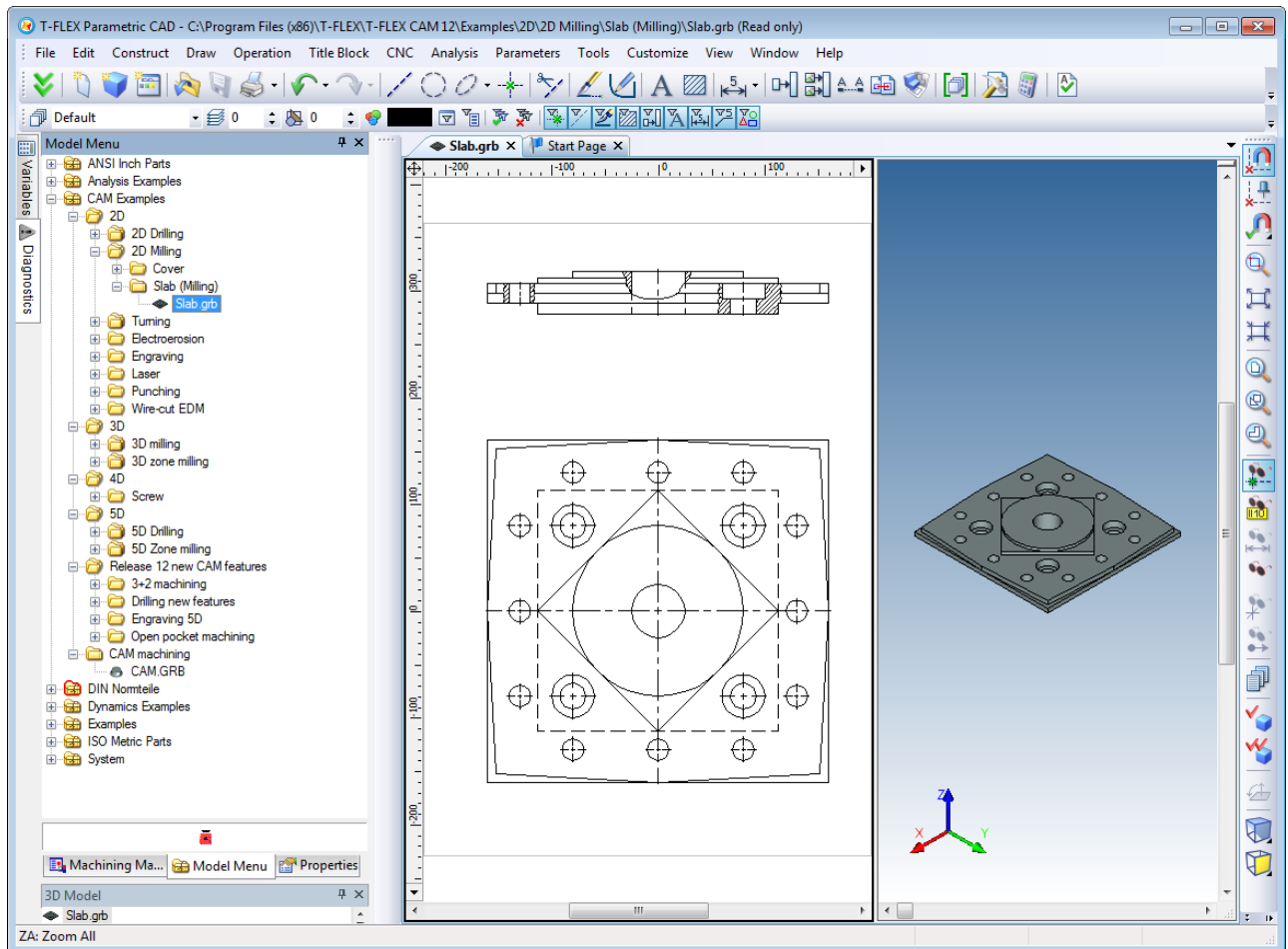



After performing all the steps described above, the user can view the playback of the generated control sequence, by calling the **"CNC>Machining simulation"** command. The user-created control sequence will be played back in the coming up window.

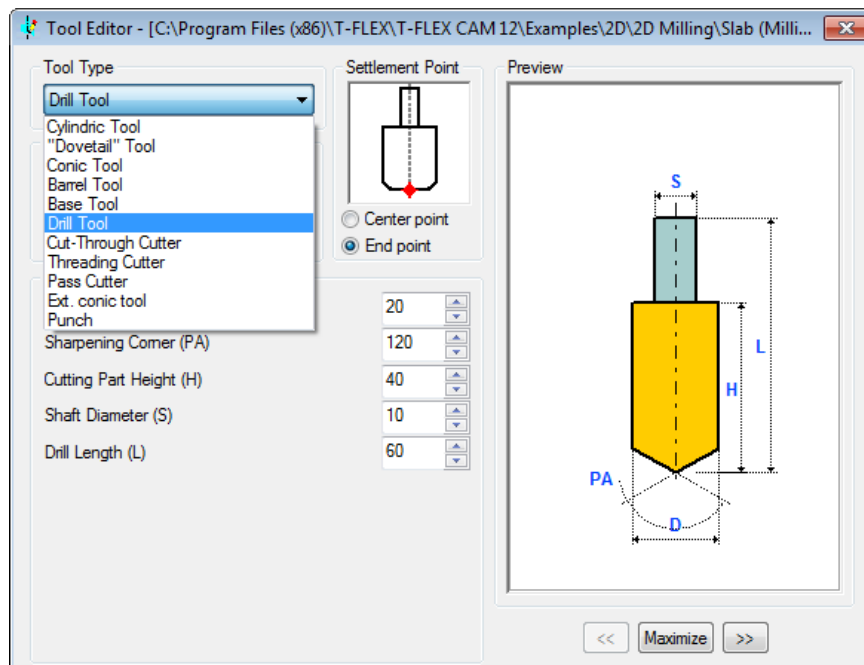


DRILLING AND 2.5D MILLING

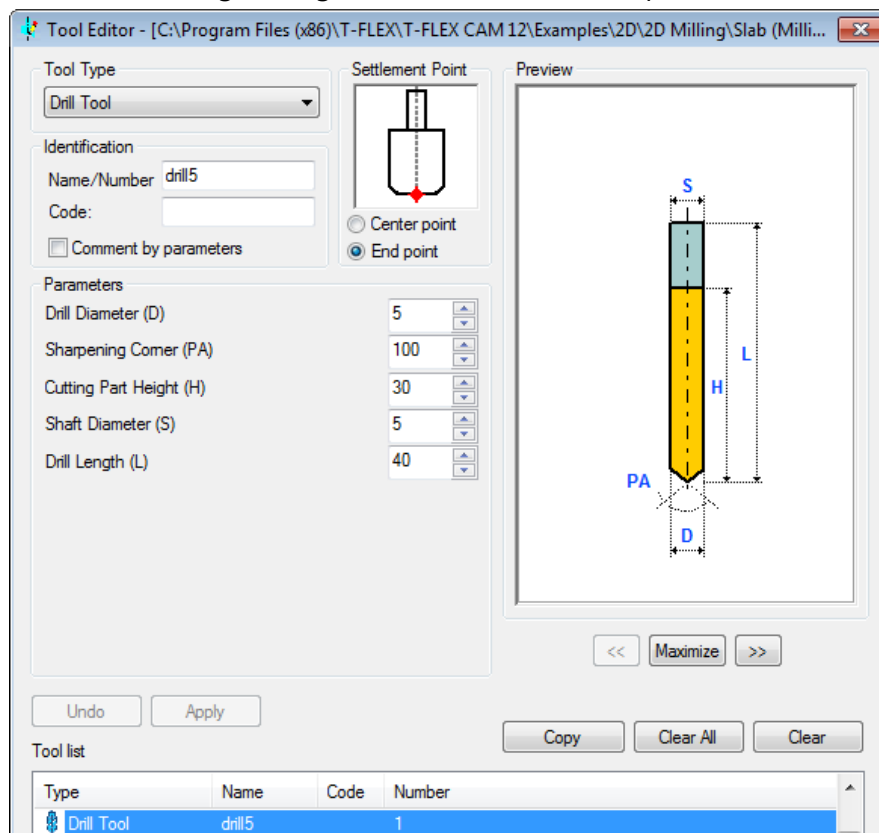
To calculate drilling toolpath, let us choose the part of the plate with several holes present in it. One would need to define the toolpaths of the drill bit motion when machining the chosen part. In other words, the user defines the order of drilling the holes using paths. Next, we shall design the cutting tool. Then, we proceed with creating the toolpaths, simulate them and save the control sequence. To create toolpaths for drilling, we use the  command or an appropriate machining cycle command.





Let us proceed with creating the cutting tool. It is better if we first drill a hole of diameter 50 by the bit of the half the diameter, and then we enlarge it by reaming to the desired size. Let us create a tool file based on these requirements. To do this, go into the tool editor by clicking the  button on the main toolbar. The editor window will appear. To create one bit, select the "Bit" option from the drop-down list in the tool editor dialog window.






To begin, let us create the centering drilling bit with the diameter equal to 5.




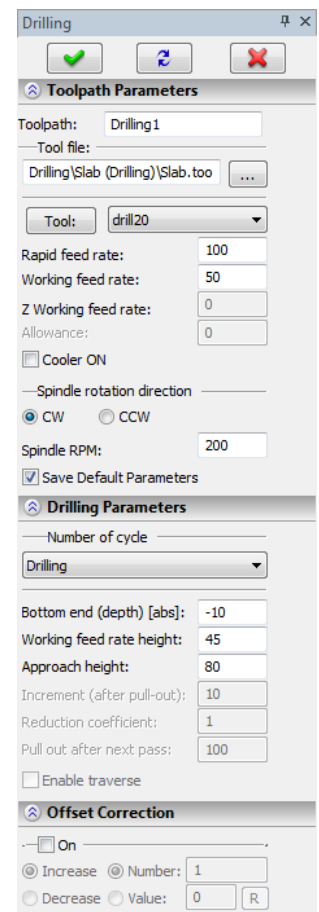
Avoid using "space" characters in tool names, because this may complicate reading the sequence by the CNC center and T-FLEX NC Tracer Machining Simulator. Let us change the default parameter and then click the  button to complete this tool creation. The created tool will be added to the tool list. The list shows its name in the set of parameters. By performing similar actions, we will create the rest of the tools required for this machining. When creating the next tool, keep in mind that the system will be defining the calculated point in the "Calculated point" field by the end point or by the center of sphere in the same way as for the tool created before this one. Please, do not ignore this issue. To complete the tool file creation, click the  button and specify the name of the file to save.

In the next step, we will construct the paths required for completing all drilling on this part. Four holes with diameters equal to 40 and 24 may rely on the same path, just like drilling and boring of the central hole may use the same path. This system feature helps avoid cluttering of the drawing and "mixing" paths when making a selection. Let's construct the drilling paths taking into the account specifics of their creation. The path for centering must pass through the centers of all holes, since all of them are centered using the same-diameter bit. The rest of the paths are also constructed along the centers of the holes that will be drilled by the bit of the same diameter.




When creating paths, keep in mind that the first segment of the path (between the first and the second nodes) is handled as an additional transition, and drilling is not performed in the first node.

Upon finishing constructing paths, proceed to creating the toolpaths properties. Let's invoke the drilling menu by clicking the  button on the main toolbar. In the menu that appears, select the standard drilling operation  and specify the path for centering holes. Upon selecting the path, a window appears with parameters of the standard drilling operation. We will use "Drilling" for centering. Specify certain machining parameters. In the "Tool file" field, click the  button and select the path to the tool file created earlier in the editor.

After that, the file will be loaded, and the names of the tools from the file suitable for this type of machining will appear in the "Tool name" list. You shall select a tool from the list that will be used in the given machining (in our case, that is sverlo5). Upon defining all parameters, complete the centering toolpath creation by clicking the  button. Graphic rendering of the machining toolpath will be displayed on the screen, while the toolpath itself will automatically be added to the tree of toolpaths in the Machining Manager. The drilling menu will transition to starting mode, awaiting for a new operation definition to start.



Drilling

Toolpath Parameters

Toolpath: Drilling1

Tool file: Drilling\Slab (Drilling)\Slab.tool

Tool: drill20

Rapid feed rate: 100

Working feed rate: 50

Z Working feed rate: 0

Allowance: 0

☐ Cooler ON

Spindle rotation direction: ☒ CW ☐ CCW

Spindle RPM: 200

☒ Save Default Parameters

Drilling Parameters

Number of cycle: 1

Drilling

Bottom end (depth) [abs]: -10

Working feed rate height: 45

Approach height: 80

Increment (after pull-out): 10

Reduction coefficient: 1

Pull out after next pass: 100

☐ Enable traverse

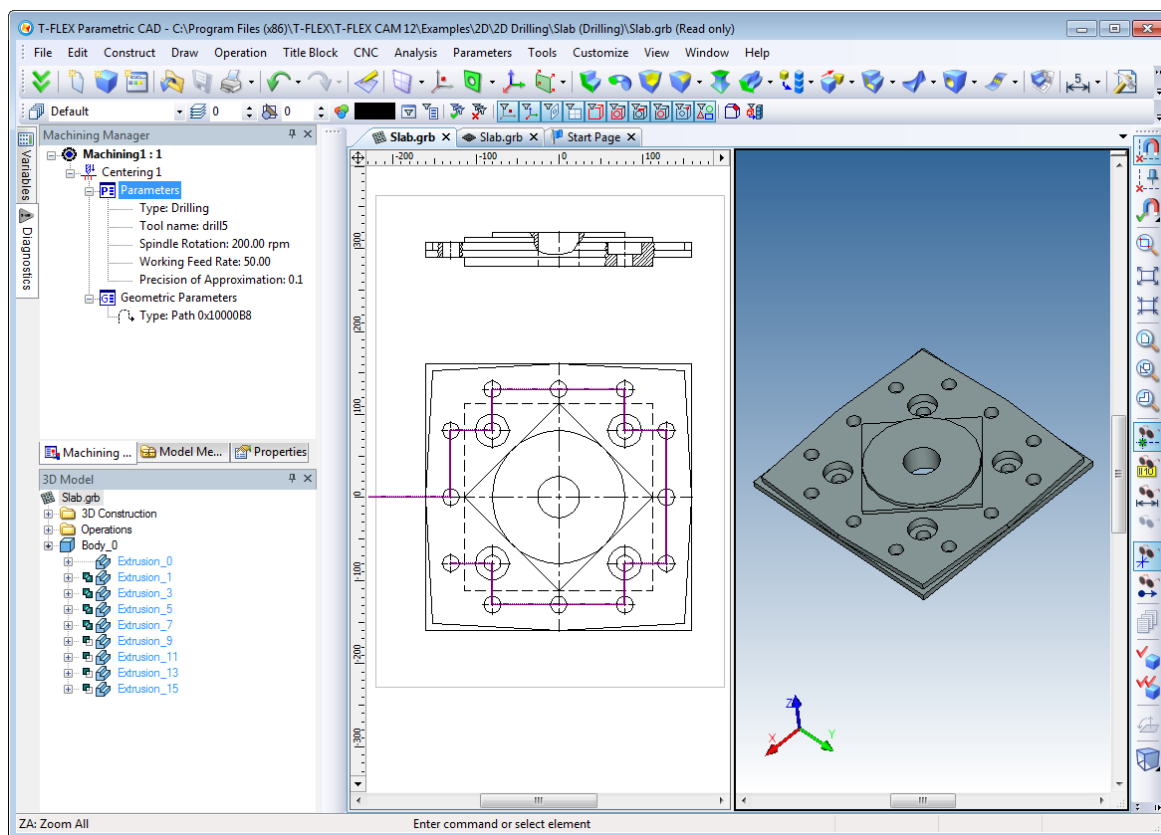
Offset Correction

☒ On


☒ Increase ☐ Decrease

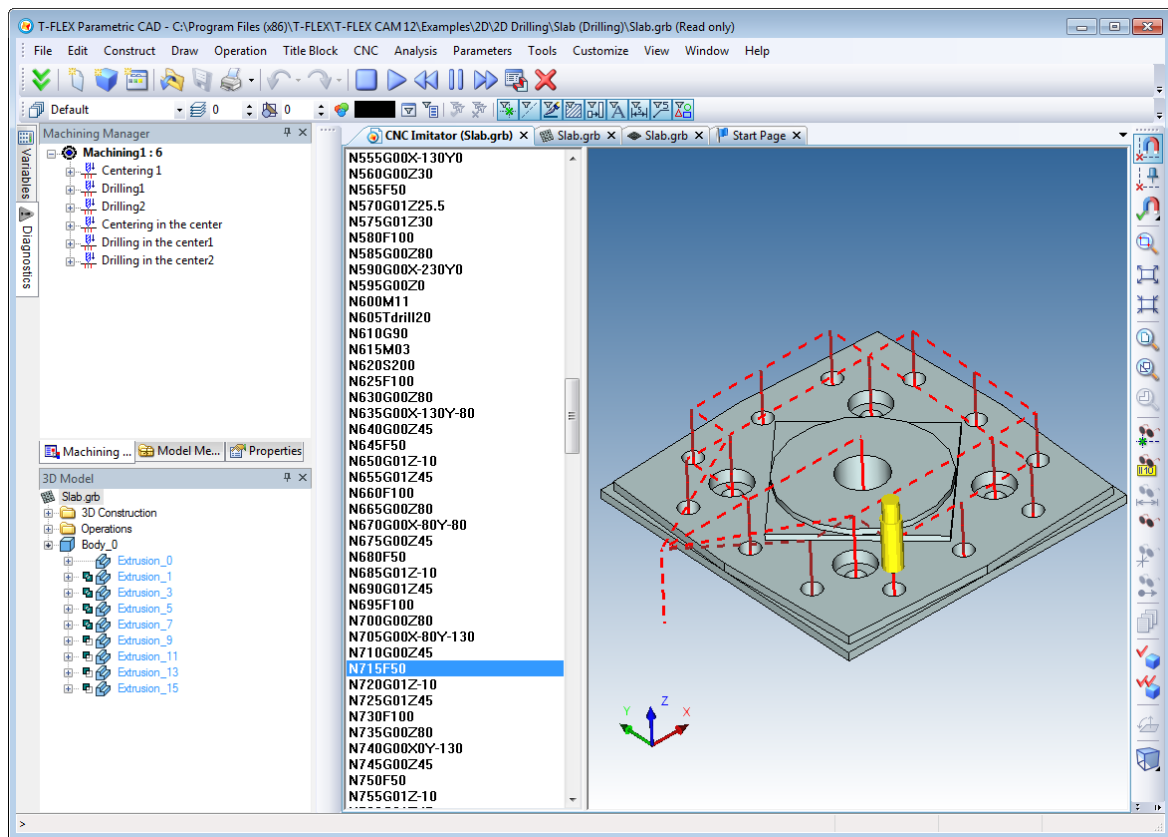
Number: 1

Value: 0




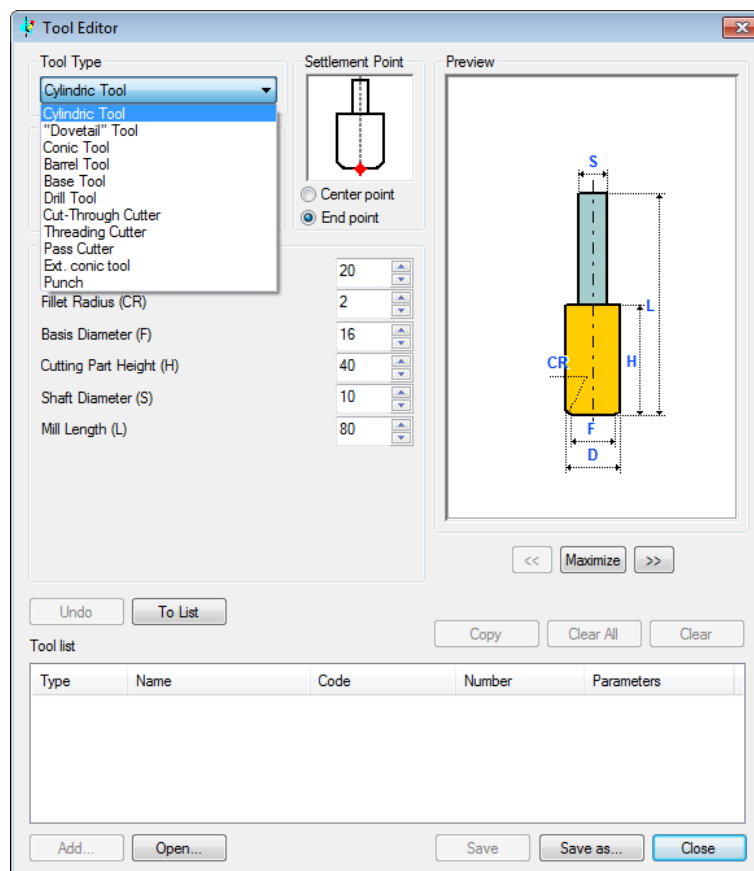
Create the rest of toolpaths similarly. Since it is never required to drill deeper than 35 mm while machining this part (meaning the drilling depth increments from the plane of the tool entering material), the use of “Deep drilling” option is not justified. When machining toolpaths creation is finished, the results of the efforts can be viewed with the help of the built-in machining simulator without material removal. As long as the constructed 3D model of the part is present within the file, it will automatically be loaded into the simulator's 3D scene.

To save the control sequence for this machining, go to the control sequence creation menu  and complete the steps similar to those described above, in the laser cutting example.



For an acquaintance with 2.5D milling, let us continue with machining toolpath creation for the “Slab” example.

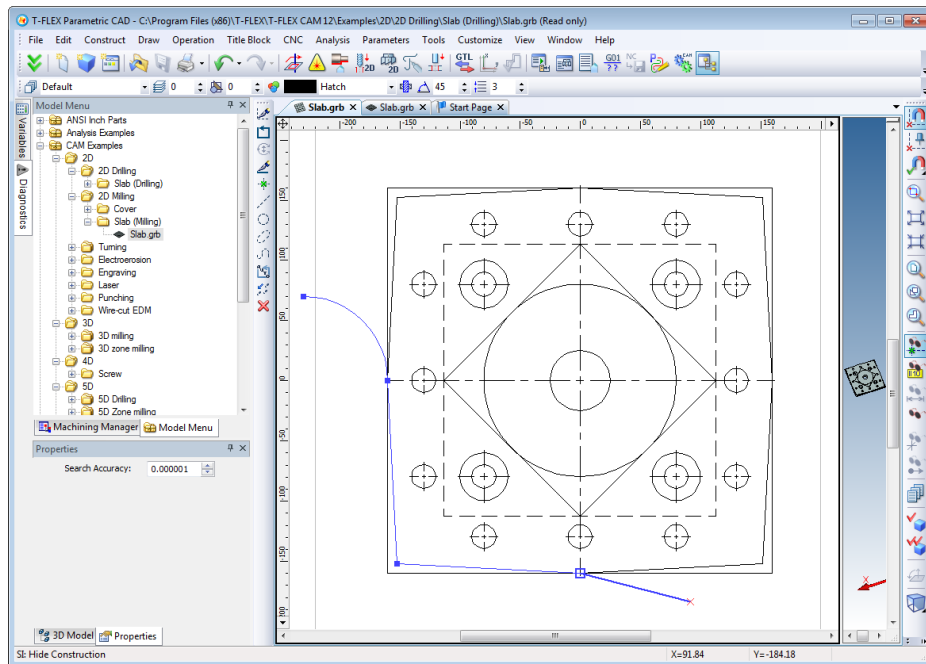
As appears from the drawing and the auxiliary 3D model (whose creation was possible only upon the availability of T-FLEX CAD 3D), the given part requires a common contour in place, all “flat spots” created (by plane milling), the circle and square contours milled, together with common chamfers. Let’s start creating the tool. Let’s determine the required tools based on the plan of machining we created above. To mill the planes, we would need a mill with diameter 40 and a cylindrical mill with diameter 30, for machining the contours of the square and the circle. Let’s use the tool editor . In the editor window, click the button and select a previously created file for the drilling operation. Upon loading the specified file, select the “Cylindrical Mill” tool from the list. In the menu of tool creation that appears, set the parameters of the mill with diameter 40.



This is similar to creating a drilling tool. Then, we will similarly create a mill with diameter equal to 60. The created tools will automatically be added to the loaded file (and, respectively, to the list of tools contained in the file). To save the file with the changes introduced in it, click the [Save...](#) button.

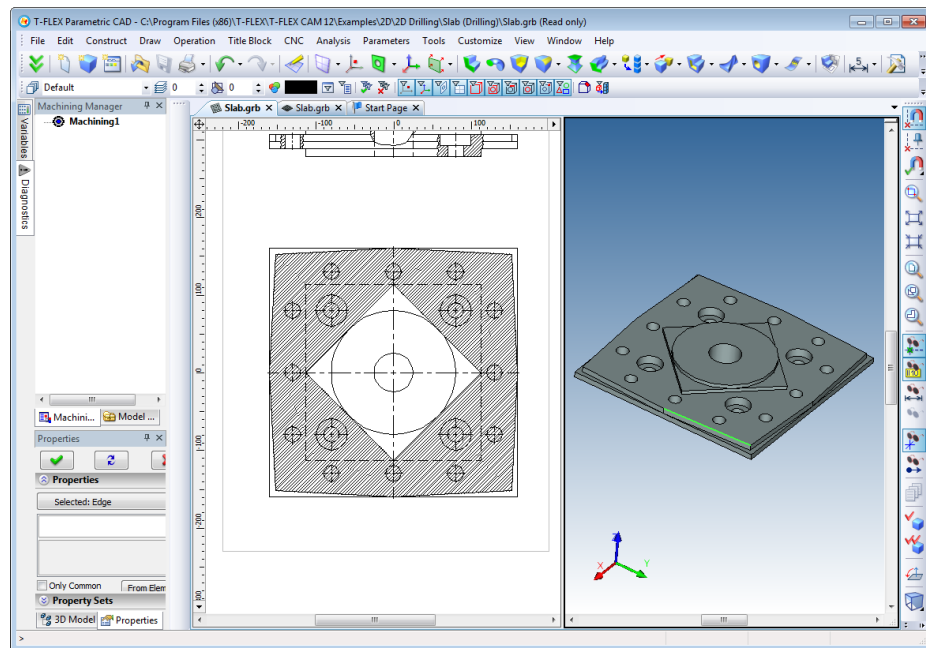
Upon saving the tool file, let's proceed with constructing geometrical elements required for building the toolpath. To create the contour of the larger square, let's make a pass along its contour with the mill number 40. We will define the contour by a path. In addition, to avoid extra steps when defining the mill approach and retreat, let's use the circle created as part of the drawing. This circle is tangent to the contour of the square. Let's construct the path starting with the 90 degree arc from the tangency point, and ending at the point which is the vertex of the 90 degree arc, after traversing the entire square contour.

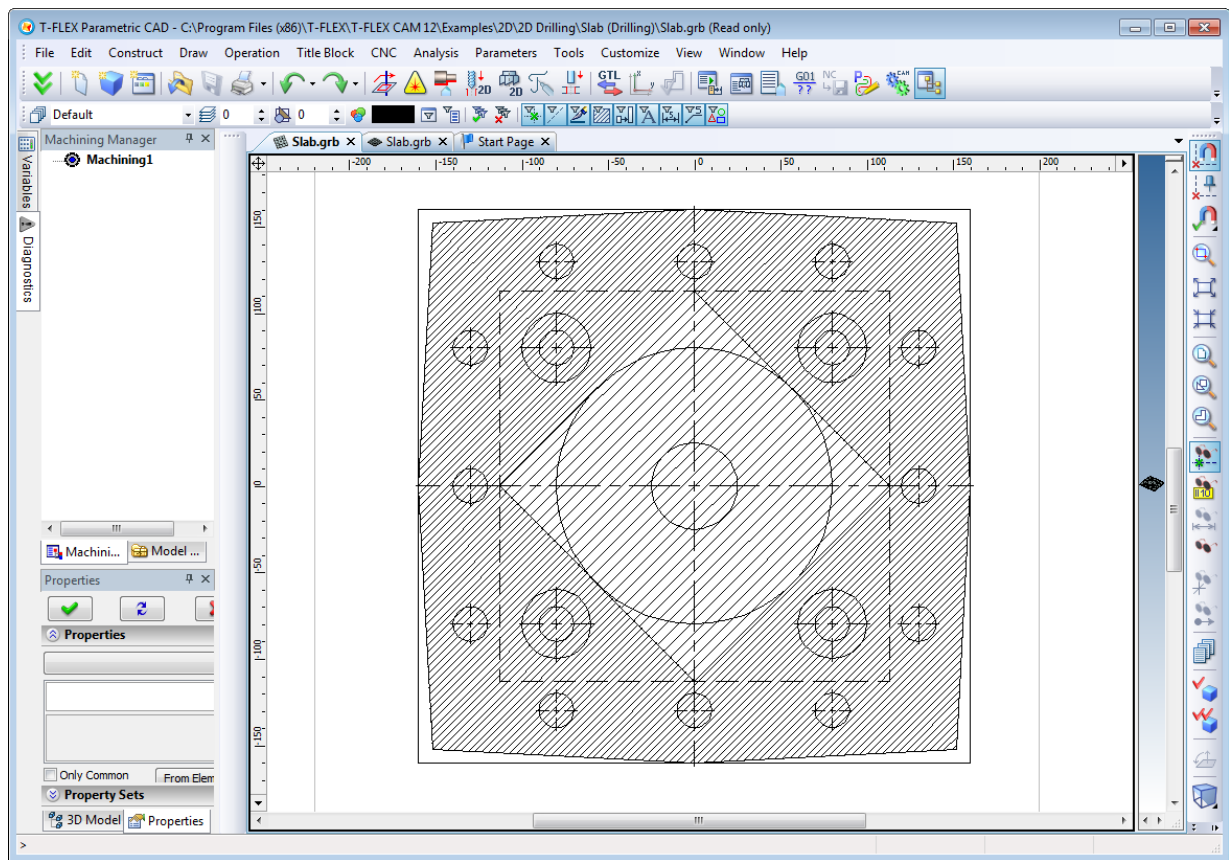
Thus, we have specified geometrically the approach and retreat along a circle, by using construction elements, i.e., without using the "Approach/Retreat" option of the Machining Manager.



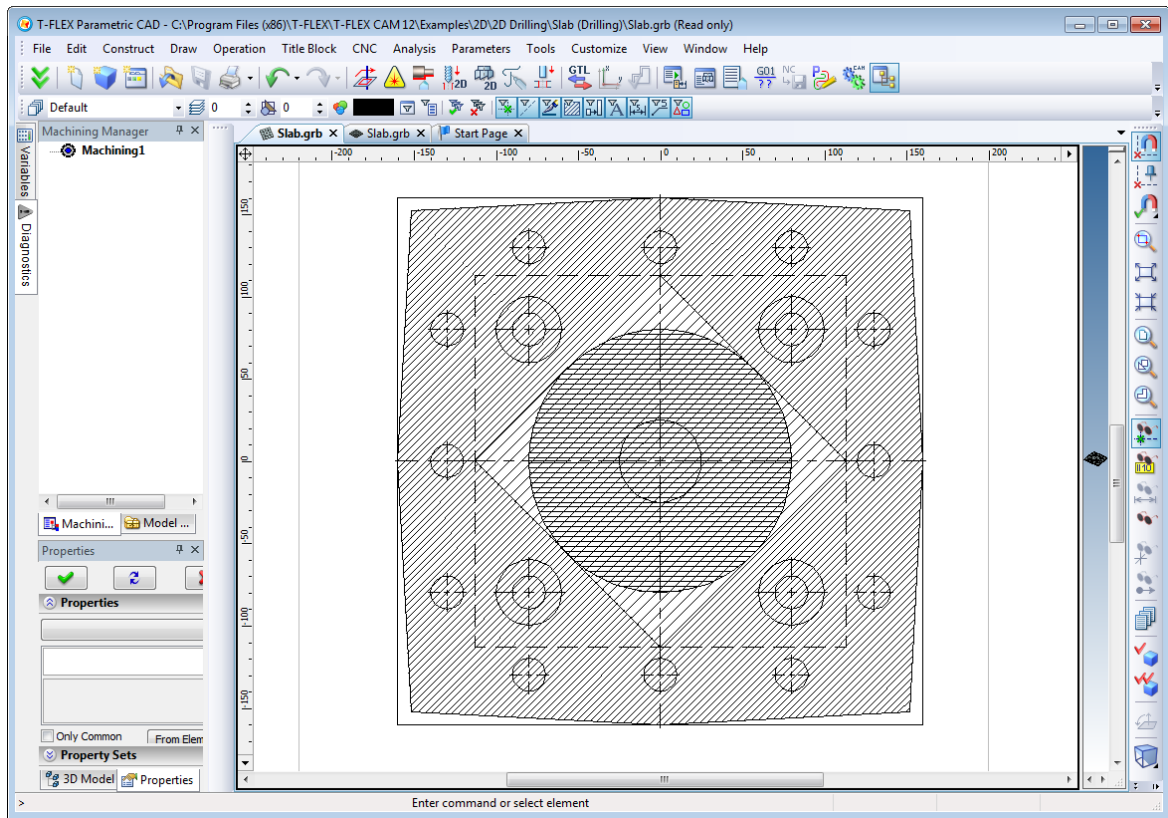
Next, you would need to define the contour with chamfers, and then mill the plane, avoiding the inner square. To simultaneously fulfill both tasks and avoid drawing's cluttering, we will use a hatch for creating toolpaths whose external boundary will be the square with clipped corners, and the internal boundary – the smaller square. Next, just let the system create the toolpaths.






For machining the smaller square and the circle contour, let's create hatches based on them.






You should not machine the flat area of the smaller square, since when milling the contour of the circle by the bit with the diameter 30, it will get to the shape by itself. Let us create the contours. First for the square, and then for the circle.








At this point, the process of creating the required construction can be considered finished. Let's proceed from the preparation to the creation of toolpaths. Let's invoke the milling menu by clicking the  button on the main toolbar in the «CNC 2D» mode. To begin with, let's create contour milling. Select the contour milling operation by clicking the  button in the milling menu. The system will prompt for selecting the element that user would need to use when creating the toolpath: a path  or a hatch . By clicking the  button, we will select the path constructed along the contour of the larger square.

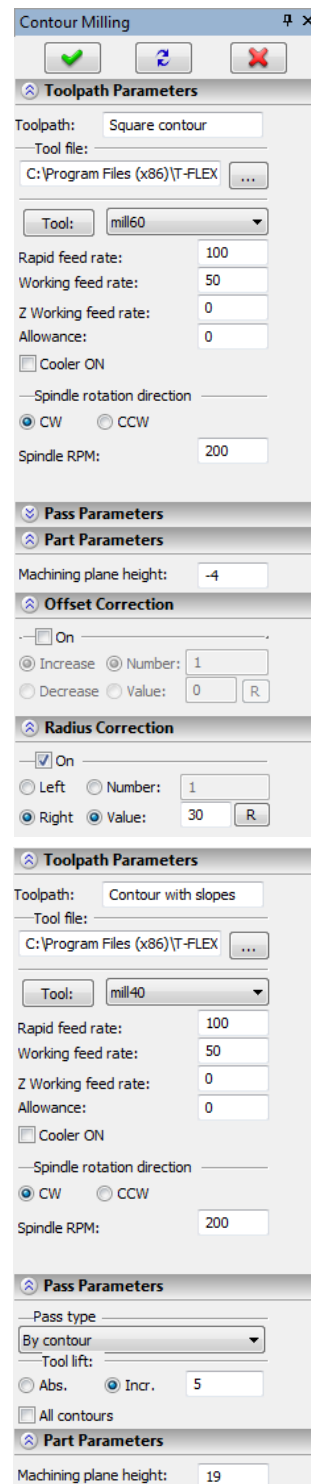
Upon selecting the path, the milling parameters window will appear. First of all, select the path to the tool file created in advance. After that, select in the list the tool from file, which is required for this type of machining. You can specify the pass type in the identically-named entry.



It is very important to specify the height of the machining plane. In our example, the part is based not on the table itself, but, rather, on a special plate whose top is accepted as the machining station zero. But if you set the height equal to zero, then the tool will pass this height with its outermost cutting point. Since the tool has a rounding radius, some material may be left out after cutting. Accordingly, the tool is required to pass below. This is exactly the reason why the negative height of the machining plane is chosen. To insure correct toolpath calculation accounting for the tool geometry, set the correction for radius. Set the radius correction according to the tool radius value.

Let's define various machining parameters: fast feedrate, working feedrate, the spindle rotation direction, etc. To complete toolpath creation, click the  button. Graphic rendering of the machining toolpath will be displayed on the screen, while the toolpath itself will automatically be added to Available toolpaths. The contour milling menu will return to its original state, waiting for a path or hatch selection for machining the contour.

Next, create milling of the square with chamfers. To do so, let's click the  button and select an already created hatch . In the window that appears subsequently with the contour machining parameters, specify parameters similar to those described above. Please give a special regard to the height of the machining plane. As appears from the drawing, chamfers are positioned at a nonzero level. It is important to avoid accidental enabling of the "All contours" option. In this case, the contour of the smaller square, which is the inner hatch boundary, will also be used in the machining.

After defining all required parameters, complete the machining toolpath creation by clicking the  button. Graphic rendering of the machining toolpath will be displayed on the screen, while the toolpath itself will automatically be added to the tree of toolpaths in the Machining Manager. The contour milling menu will return to its original state, waiting for the machining type to be defined. To exit the milling menu, click the  button or  over the drawing area.

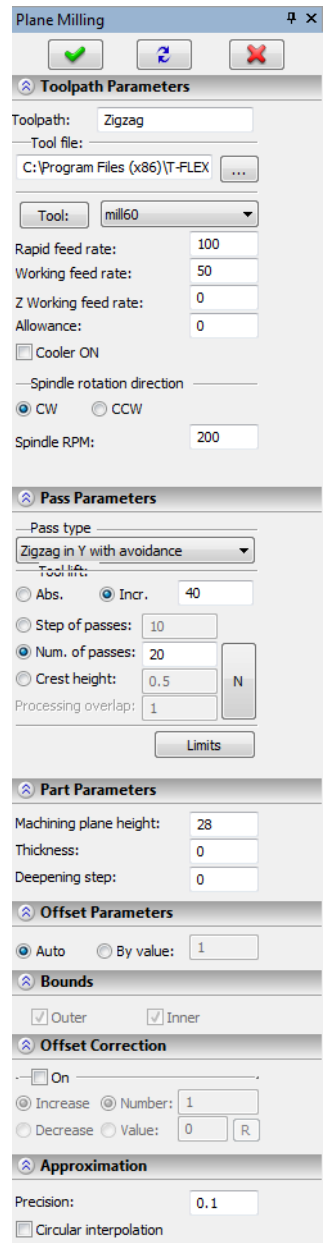
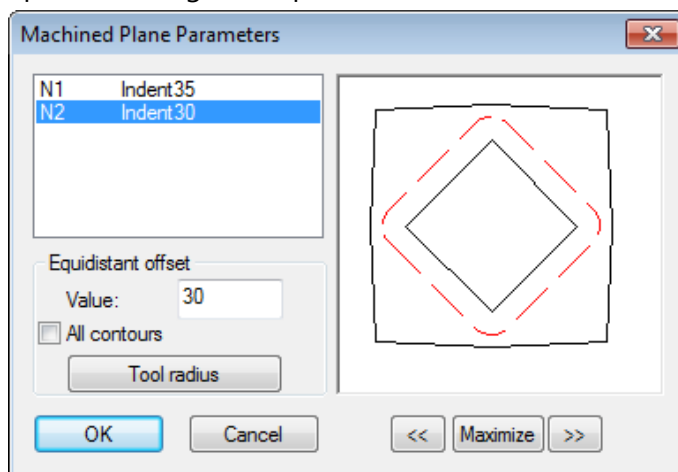



The next step in constructing a compound toolpath is creating the plane milling toolpath. To do this, let's invoke the menu of creating the plane milling operation with the help of the  button. In the operation menu that appears, the "Select hatch" command  is available. Click the button and select the same hatch as used in the previous machining. A window will appear with toolpath creation parameters. Upon selecting the tool, proceed with defining the pass type.

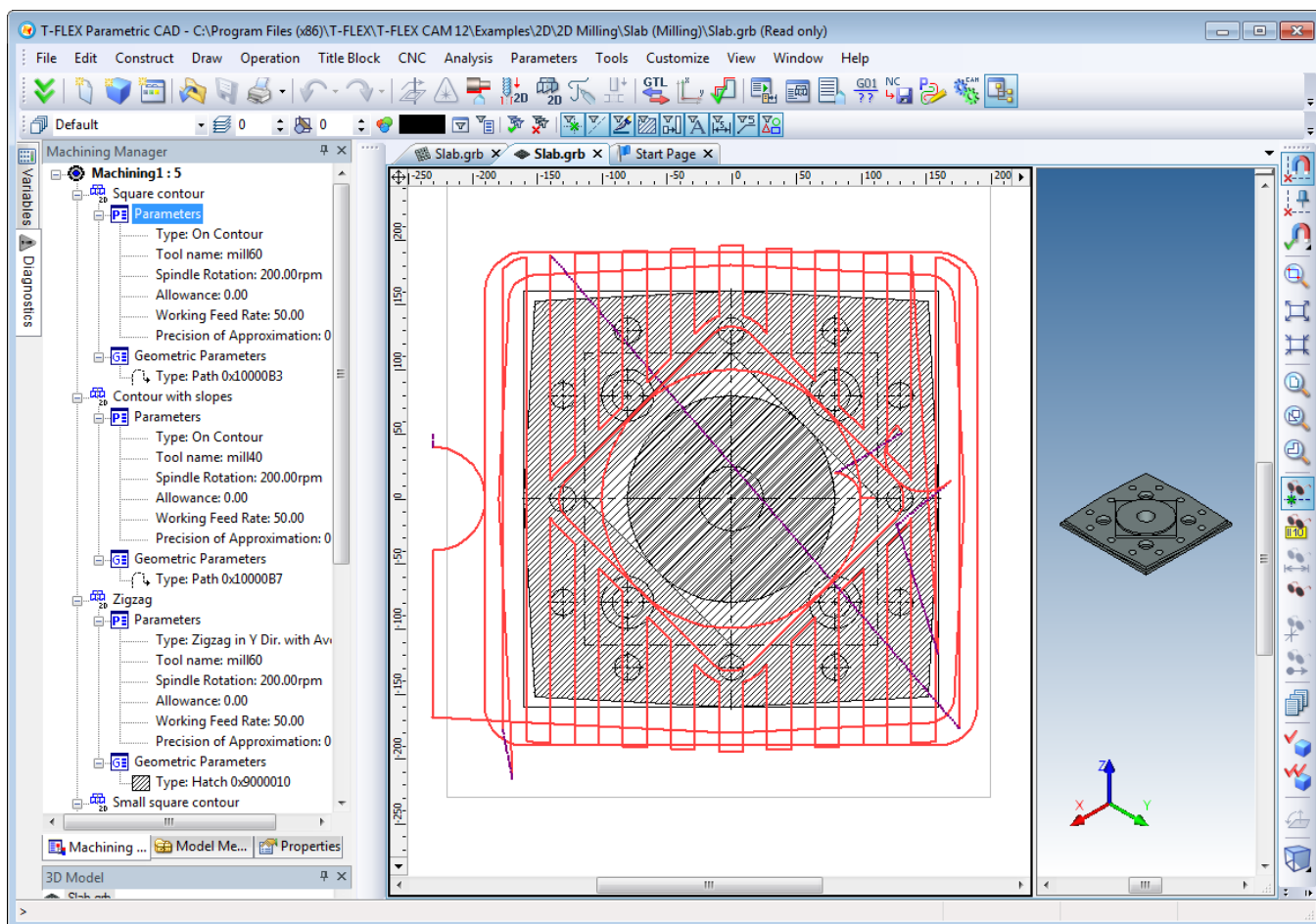
To avoid vertical tool plunging in the metal during transitions, you should set any type of passing with avoidance. In this case, the system will calculate the correct toolpath of the tool motion. Set the machining height according to the drawing. Select the method of calculating the toolpath by passes. For this purpose, let's specify the number of passes in the "Number of passes" field, and then click the [N] button. The system will automatically calculate the rest of the required parameters.

On the "Offset parameters" tab leave the selector in the "Auto" mode.


Additionally, on the "Pass parameters" tab specify limitations in the Limits dialog window. Let's call it with the help of the [Limits] button. Here, you should set the offset parameters, navigating through the system identified limitations. Set the offset by the tool radius for both bounds. Click the [OK] button to complete defining offset parameters.



Specify the appropriate spindle RPM and the values of the working and fast feed. Start feeding the cooling agent by checking the "Cooler On" option. To complete machining toolpath creation, click the  button. Graphic rendering of the created machining toolpaths will be displayed on the screen.

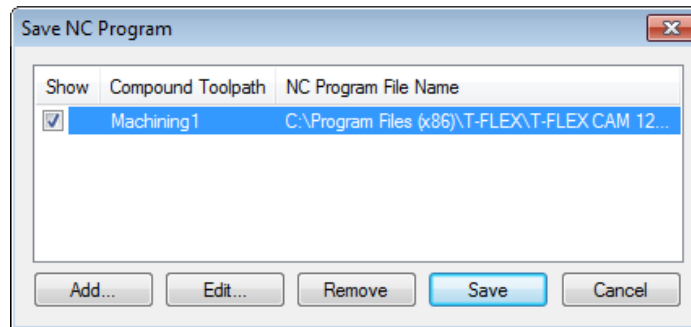


By completing similar steps, let us create toolpaths for machining square and round contours. Thus, we obtain a compound toolpath for milling the plate. But this file also contains drilling toolpaths that were created earlier. Accordingly, these toolpaths will appear before the milling toolpaths in the toolpaths tree as well. That, in turn, means that the drilling will be done prior to milling in the control sequence, which will be created on the way. But this position is not correct. There is an easy way out of this situation. You need to highlight the milling toolpaths in the list of toolpaths and move them one by one with the mouse to the desired location in the tree. The compound toolpath resulting from these actions is fully correct.


To save the control sequence for machining this part, let's call the window of parameters for saving the control sequence with the  button. In the window that appears, save the machining as: "Machining 1". Suppose, that the given part is machined on different machining stations (drilling and milling ones). Then, we would need two control sequences. To create those without cluttering the drawing with toolpaths and without extra efforts, you should use the built-in project management system. We will call the context menu in the Toolpath Manager window and pick the "New machining" option. After that, another machining with no toolpaths, will be added to the tree. For convenience, let's call it "Drilling".

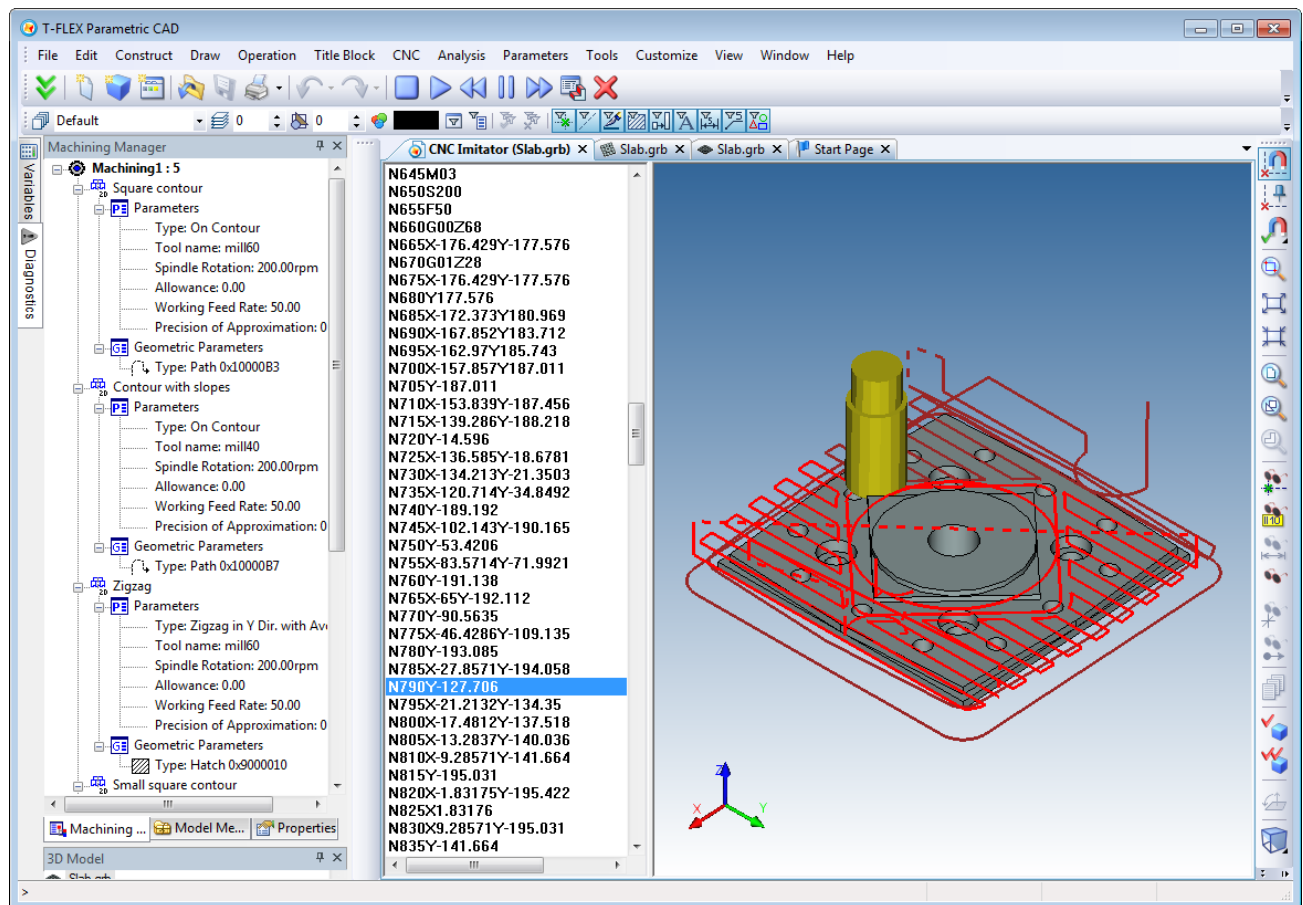
Now, let's copy drilling toolpaths from the former machining into the new one, by subsequently calling the context menu for each toolpath and using with the "Copy" and "Paste" commands.

Now, let's add one more machining. For convenience, let's call it "Milling". In this way, it will be easier to handle. Similar to the drilling toolpaths, we will copy milling toolpaths into the "Milling" machining. Thus, we have created three different compound toolpaths for machining this part: a combined machining (combined execution of drilling and milling), milling (only milling is executed), and drilling (only drilling is executed).



By saving these machining types with different postprocessors, we obtain control sequences for machining a part at a machining center, and two control sequences for machining at a milling and a drilling stations separately. To save the control sequence, enter the menu of creation of control sequences and, by clicking the **[Add]** button, select from list the desired toolpath and define the saving options. Thus, the T-FLEX CNC system supports development of part-machining toolpaths for machining stations of various types and sharing toolpaths across various machining types. The project management system is an excellent solution for most tasks that the user can face when working with machining stations at an enterprise.

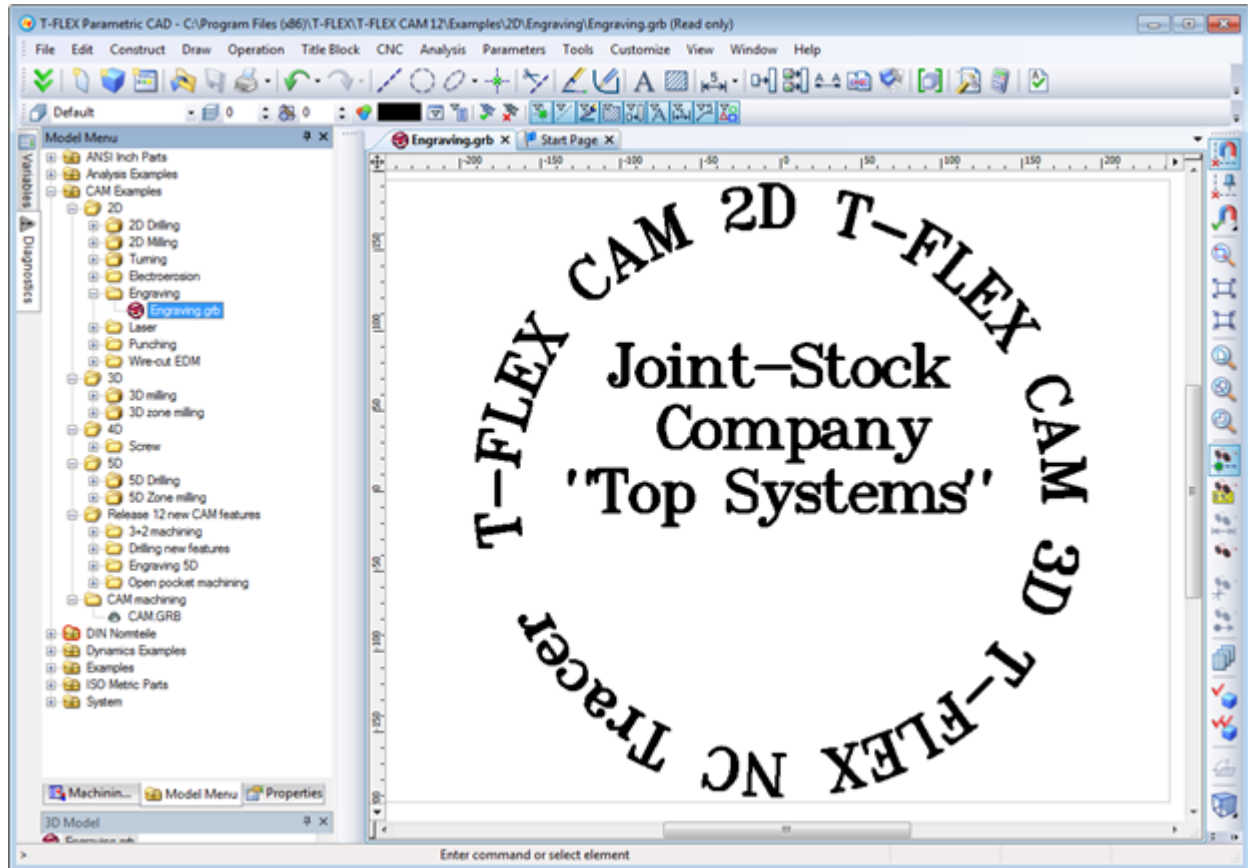
Let's use the automatic tool setup chart functionality. To do this, click the  button on the "Base Module" toolbar, and in the upcoming dialog window click the **[Create New]** button. Then close the dialog window. An additional page will appear in the document with the filled-in tool setup chart form which can be printed out.





Similar to previous machining types, the result of these actions can be viewed with the help of the built-in simulator, without material removal.


ENGRAVING

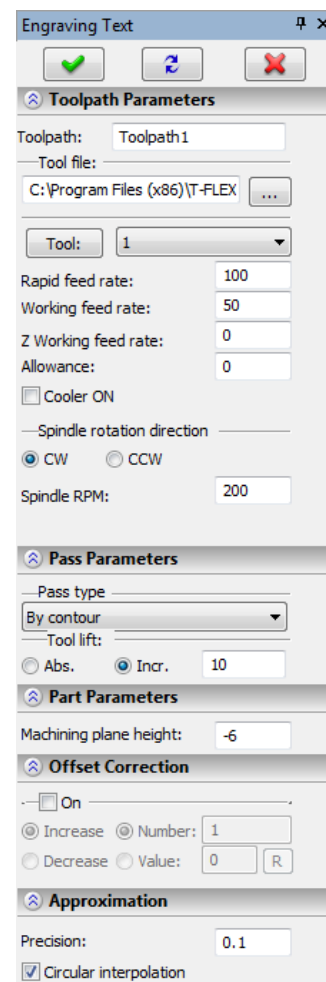
Let us calculate the engraving toolpath for a sample logo engraving.



Similar to the previous machining examples, before starting, the user can define the default machining parameters by clicking the  icon in the automenu. One may want to set up the file with the tool data and so on. The user inputs the values of the preset parameters in the appropriate input boxes of the dialog that appears on the screen.

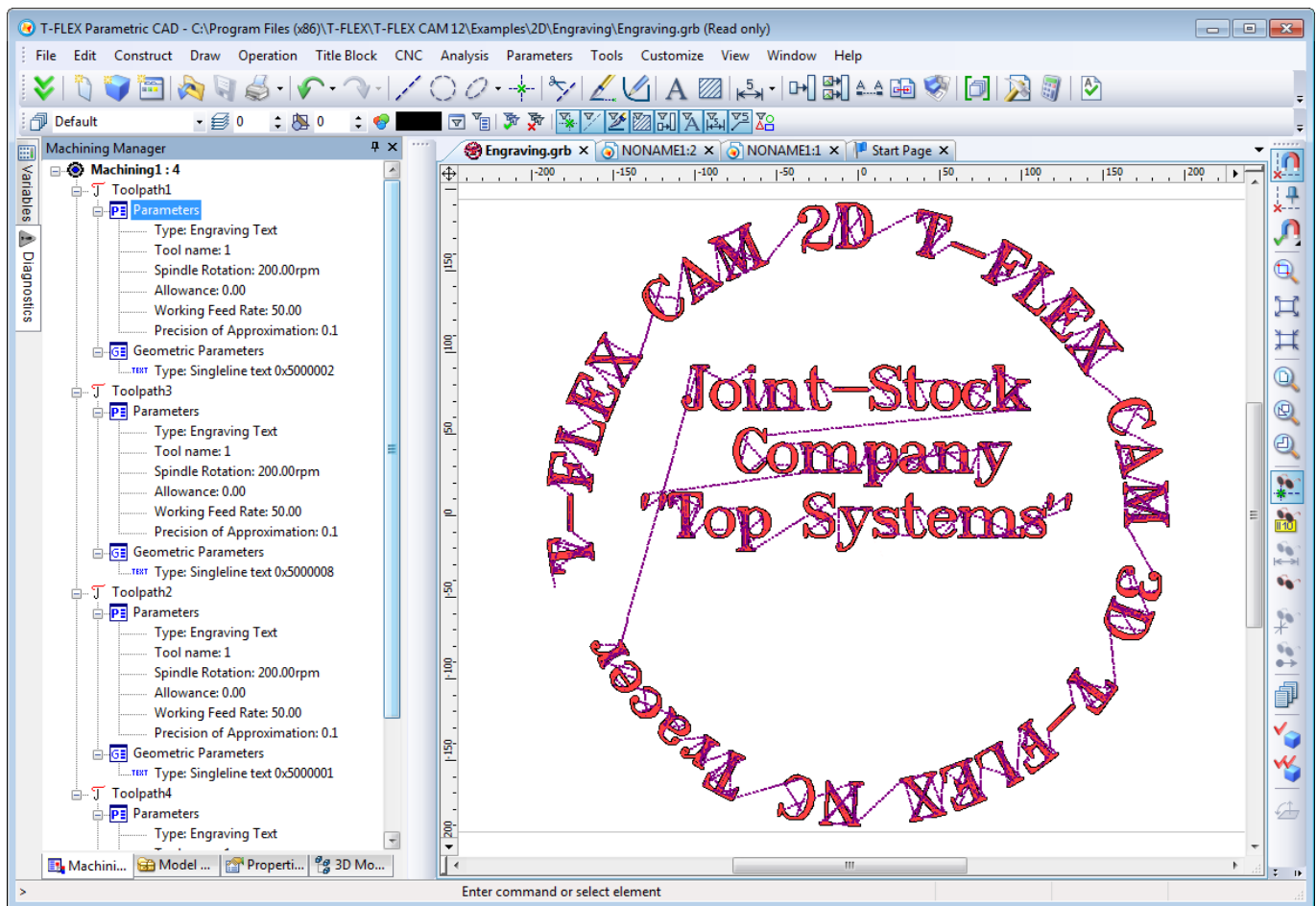
Next, select the text to be machined  in the part drawing, and then define the rest of machining parameters in the coming up dialog box. The parameters were described in detail in the first part of this manual.


Once all necessary changes are done to the machining parameters, the user just needs to click the  button to have the system calculate the toolpath. The toolpath will appear on the drawing as a colored line.





The image shows a software dialog box titled "Engraving Text". At the top, there are three buttons: a green checkmark, a circular arrow, and a red X. Below these are expandable sections: "Toolpath Parameters", "Pass Parameters", "Part Parameters", "Offset Correction", and "Approximation".

- Toolpath Parameters:** Includes fields for "Toolpath:" (Toolpath1), "Tool file:" (C:\Program Files (x86)\T-FLEX), "Tool:" (1), "Rapid feed rate:" (100), "Working feed rate:" (50), "Z Working feed rate:" (0), "Allowance:" (0), a "Cooler ON" checkbox, "Spindle rotation direction" (radio buttons for CW and CCW, with CW selected), and "Spindle RPM:" (200).
- Pass Parameters:** Includes "Pass type" (By contour), "Tool lift:" (radio buttons for Abs. and Incr., with Incr. selected and a value of 10), and "Machining plane height:" (-6).
- Part Parameters:** Includes "Offset Correction" (checkbox for On, with radio buttons for Increase and Decrease, and a "Number:" field set to 1, and a "Value:" field set to 0 with an "R" button).
- Approximation:** Includes "Precision:" (0.1) and a checked "Circular interpolation" checkbox.

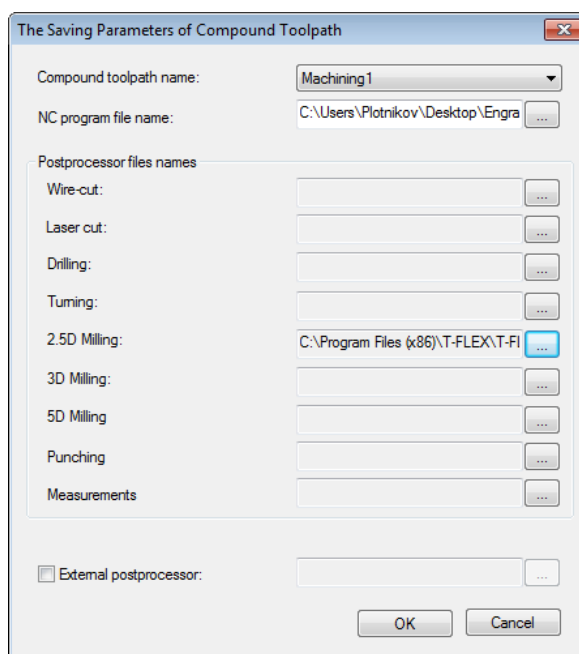
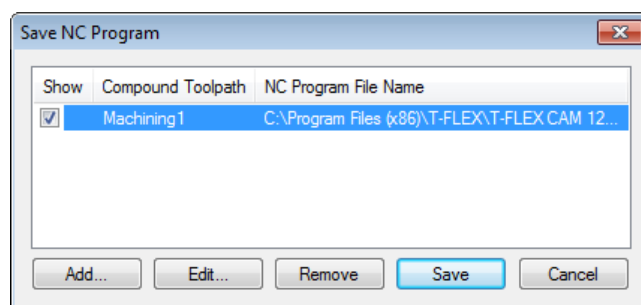


After the system creates the machining toolpath, the user can define the approach and retreat motions in a special window that appears when called from the context menu of the toolpath in the Machining Manager (the  button).

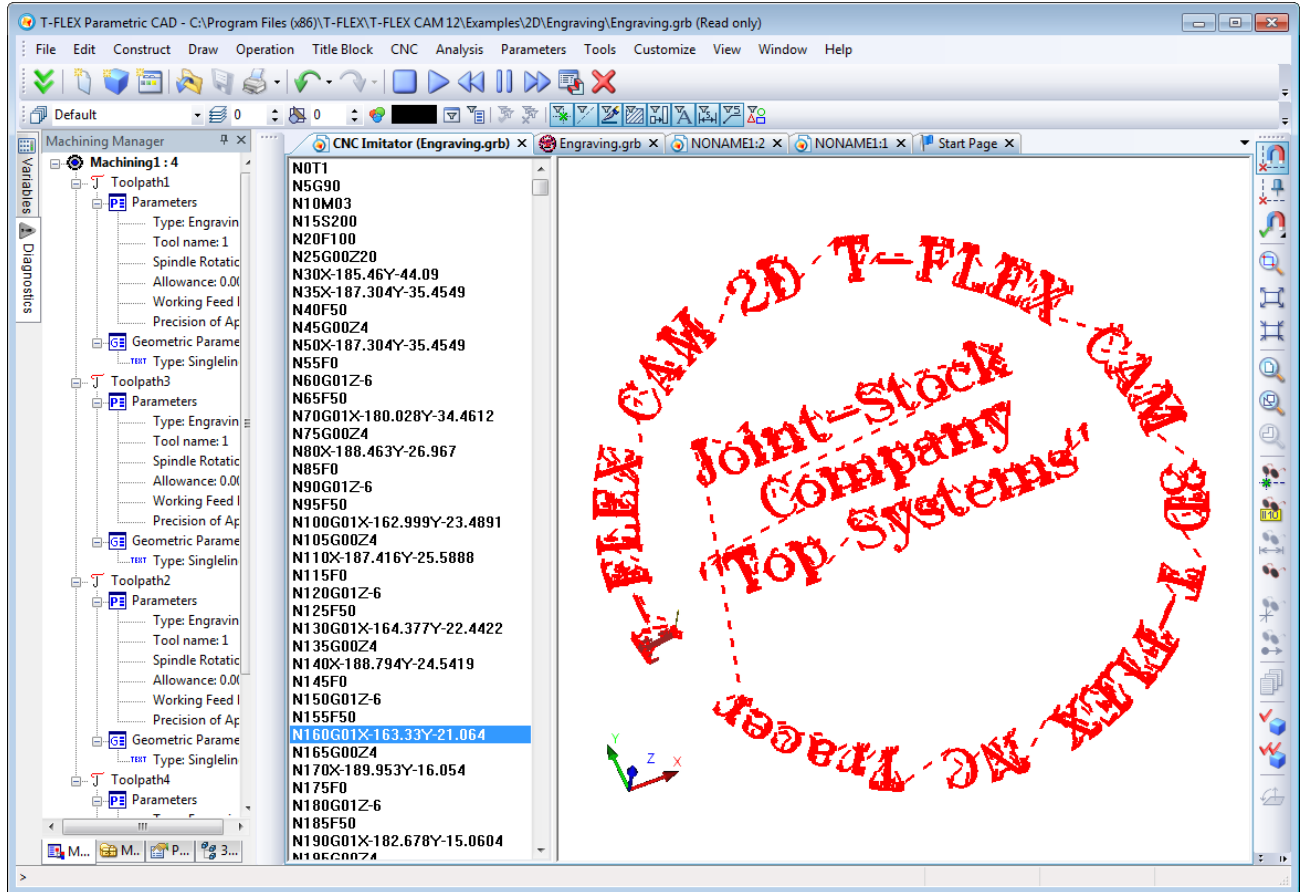
Once the toolpath is calculated, the user can save the respective control sequence. To do so, a new postprocessor is created in advance with the "Postprocessor options" command, if necessary to the user. Otherwise, the user can select a predefined postprocessor from the library supplied with the system.

Once the desired postprocessor is selected, the user can run the control sequence generation (saving) by using the **"CNC>Save NC program"** command. Double-click   inside the window that comes up on the screen. The "Compound Toolpath Saving Parameters" dialog box will appear.



Next, in this dialog box, specify one by one the names of the necessary postprocessors for the selected machining type, the name of the control sequence and the location where you want to save it.



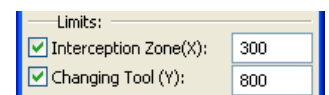
After performing all steps described above, the user can view the playback of the generated control sequence in the “CNC>Machining simulation” command.




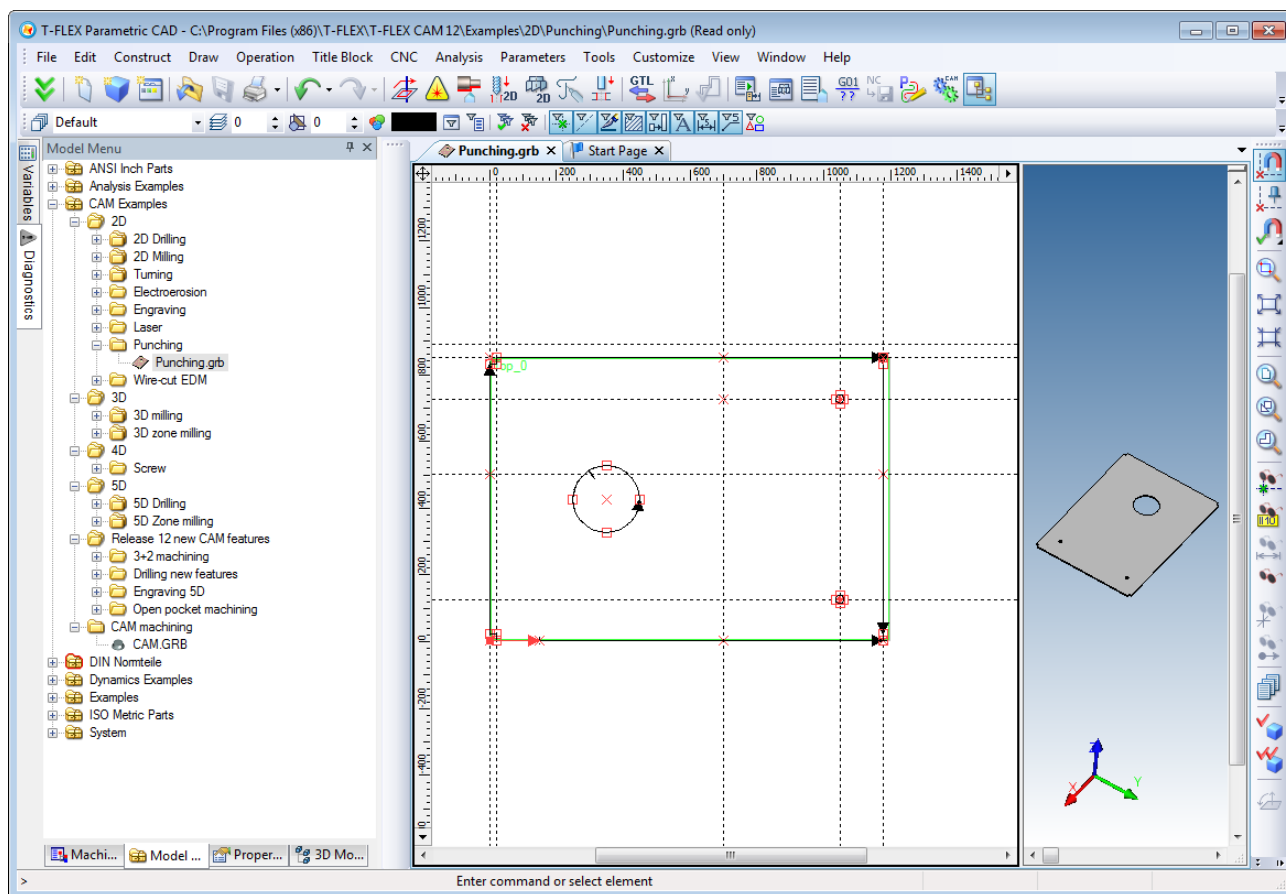
PUNCHING


Let's create a set of punching toolpaths for creating the part shown on the figure below. The dimensions of the part being machined may exceed the sheet machining zone in the case of using a single tool without interception, therefore, the sheet interception boundary and the tool change boundary shall be specified. T-FLEX CAM solves this by means of a technological toolpath. To create such a toolpath, go into the punch process menu by clicking the  button on the tool bar. In the upcoming commands menu click the  button. Clicking this button will bring up a dialog window of the technological toolpath parameters.

Define the sheet interception and tool change parameters in the appropriate sections of the dialog window.

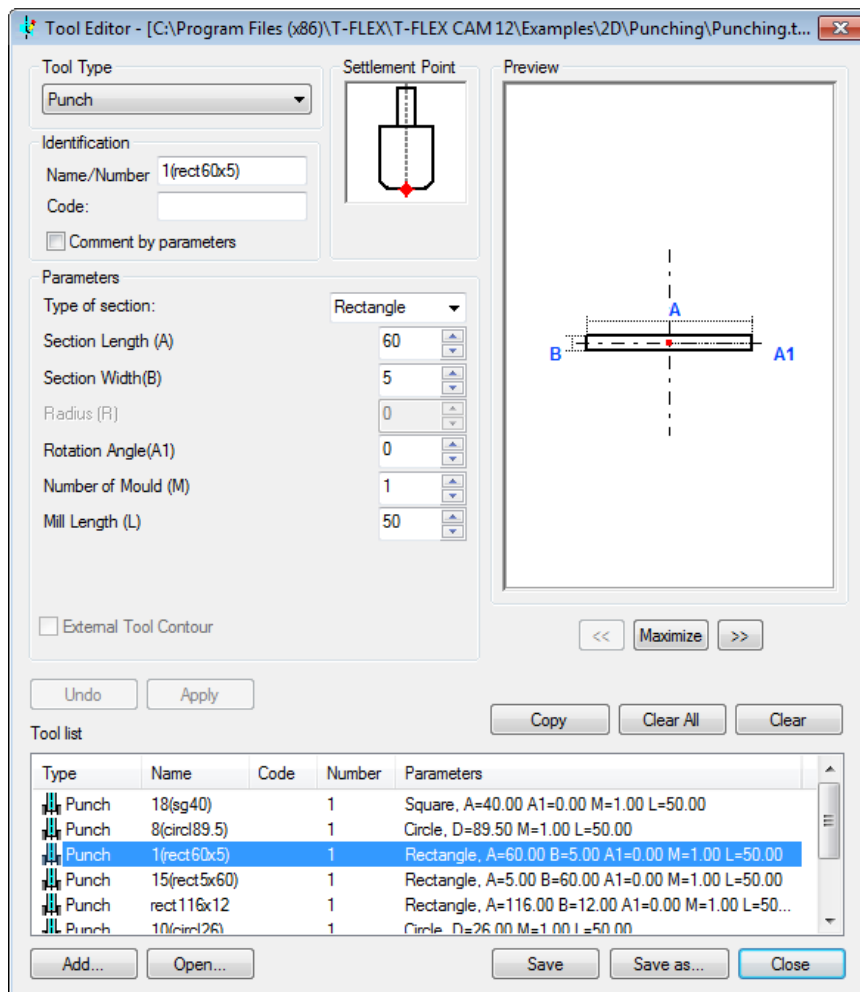


After that, click the  button and finish the technological toolpath creation. The subsequent toolpaths will require additional constructions in the drawing.



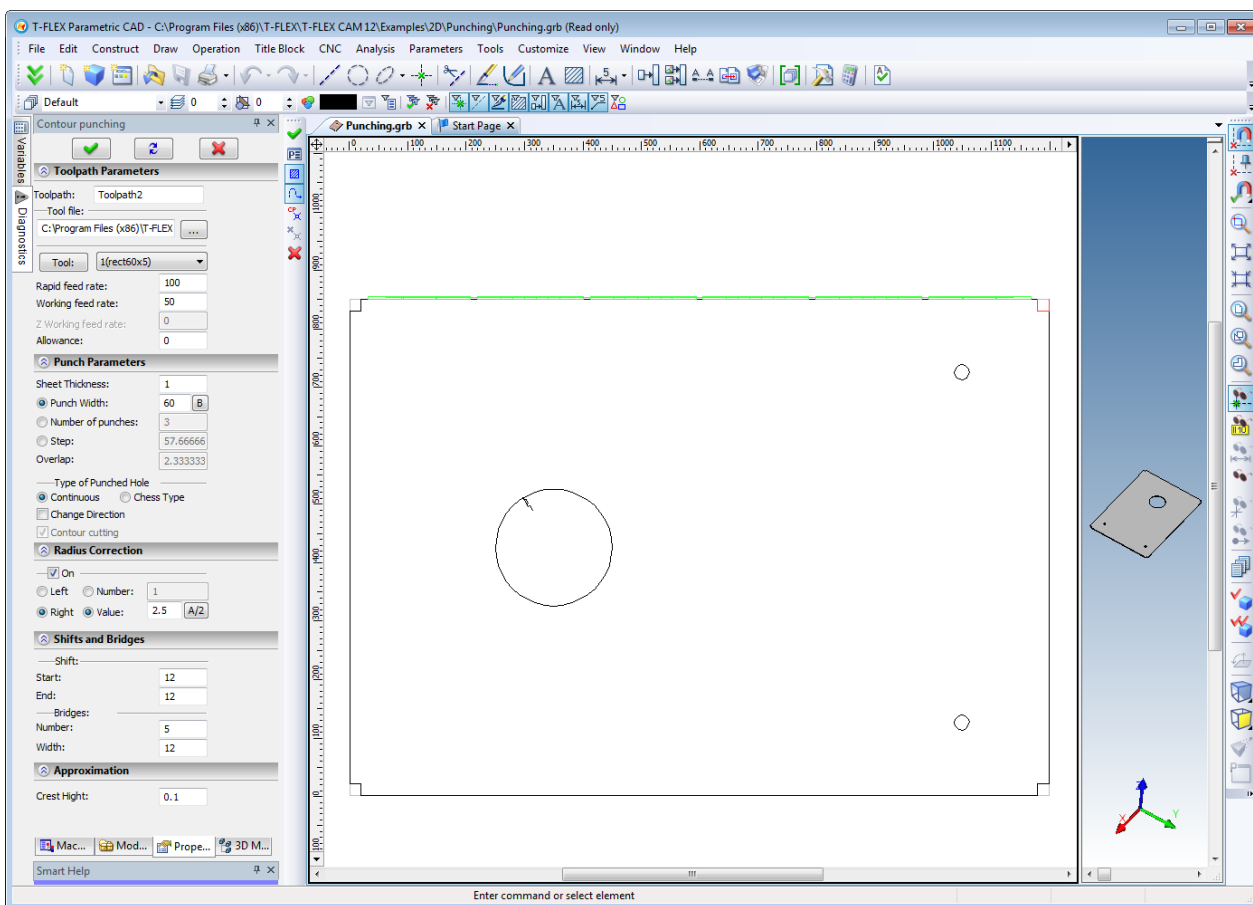
Paths shall be created following the part contour. The paths may consist of a single segment. Next, a hatch shall be created based on the contour of the larger hole in the sheet. When creating a toolpath for machining small holes resulting from a single tool punch, one can use the nodes at the circle centers which are created automatically as part of the circles' creation. If no such nodes exist, then those shall be created at the centers of the smaller circles. After performing all of the above constructions, one can proceed with the tool creation procedure. To do this, open the tool editor window by clicking the  button on the "CNC" tool bar.


Upon clicking this button, a dialog window of the tool editor appears on the screen, where you shall select the punch from the list of tools, specify the desired punch geometry to be used for shaping the part contour, modify the relevant parameters and add the punch to the list of tools. Apply the similar procedure to the tool used for punching holes. After that, save the entire created tool to a file. The procedure of working with the tool editor is described in detail in the first part of this manual.



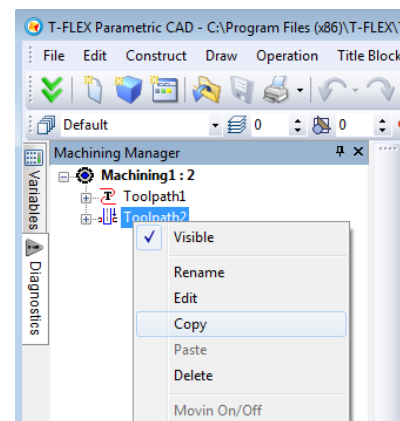
Let's proceed with defining the machining technological parameters and create the toolpaths.



To do this, go into the punching menu by clicking the button on the toolbar. In the upcoming commands menu click the button. After clicking this button, the dialog window of punch parameters appears on the screen, and, at the same time, the system prompts the user to select a construction element to define the toolpath geometry. To create a toolpath based on a path, click the button and select the path constructed based on the part contour. After this selection, specify the toolpath technological parameters. In the present case, as the part is being formed from a sheet, select the continuous-type punching. Other toolpath parameters are described in detail in the first part of this manual and are defined based on the part and the punch dimensions. Once done with defining parameters, click the button and complete the punching toolpath creation.






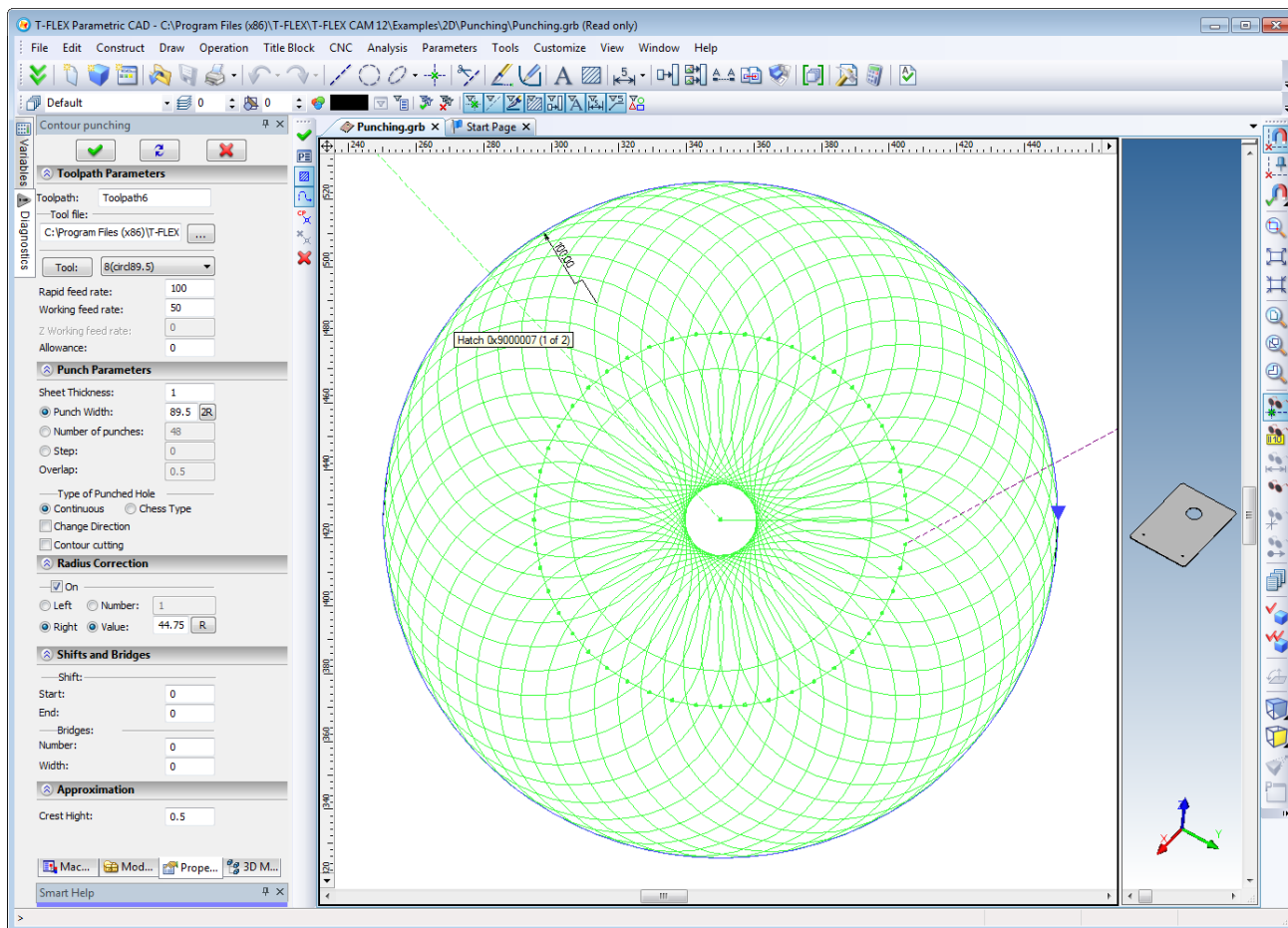
You do not have to define again the technological parameters in order to create subsequent toolpaths for contour punching. The optimal method of their creation is copying the just created toolpath. To do this, select the latter in the Machining Manager's tree of toolpaths and right-click . The toolpath's context menu will appear, with the available **"Copy"** command. Select this command from the menu, and the toolpath will be copied to the Clipboard. Next, similarly access the context menu of the machining to which the toolpath belongs.






The **"Paste"** command will be available in the context menu of the machining: select this command. Repeat the similar actions twice again (since four toolpaths are required to cut the four sides of the part).



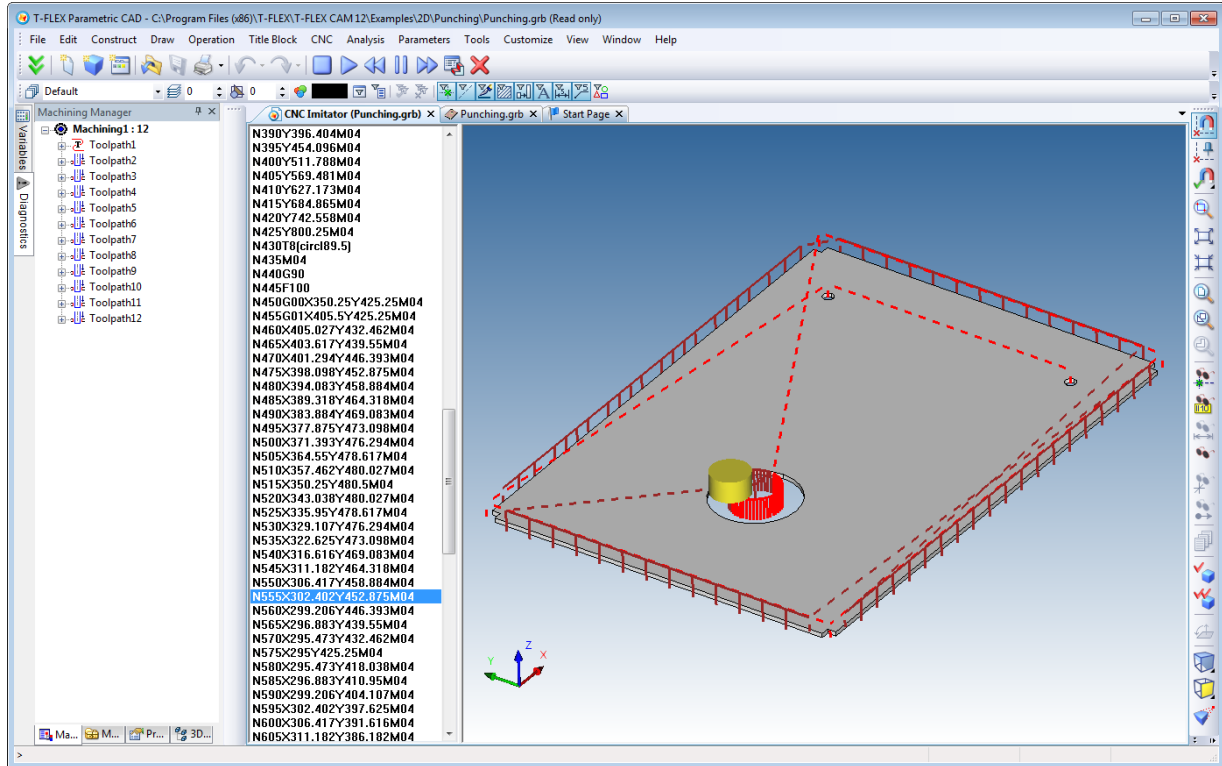
Next, open the first of the copied toolpaths and, by clicking the  button, specify the second path out of the four, constructed along the contour. After that, click the  button and complete the punching toolpath creation. Thus, we have obtained the second toolpath for the continued punching of the part contour. We will then proceed in the same manner with the rest of the copied toolpaths and with the paths constructed based on the contour. As a result, we obtain the entire punched contour of the part.

To punch the larger hole, we will use a round-profile punch created by using the tool editor. To do this, go into the punching menu by clicking the  button on the toolbar when it is in the "CNC 2D" mode. In the upcoming commands menu click the  button. After clicking this button, the dialog window of punching parameters appears on the screen, and, at the same time, the system prompts the user to select a construction element to define the toolpath geometry. In the case of a hole (when the contour is described by a hatch), click the  button and select the hatch which defines the hole's contour. In the parameters, use the continuous punching type, set the correction for tool dimensions and the punch dimensions.



After that, click the  button and complete the punching toolpath creation. This completes the main step of creation of the toolpaths forming the part's contour. To create the toolpaths for punching smaller holes with a single tool stroke, go into the punching menu () and select the GOTO operation by clicking the  button. The system will automatically transition to the mode of selecting a node for a single-stroke punch . You shall select the node constructed at the center of a smaller hole. After that, click the  button and complete the punching toolpath creation. To construct the punching toolpath

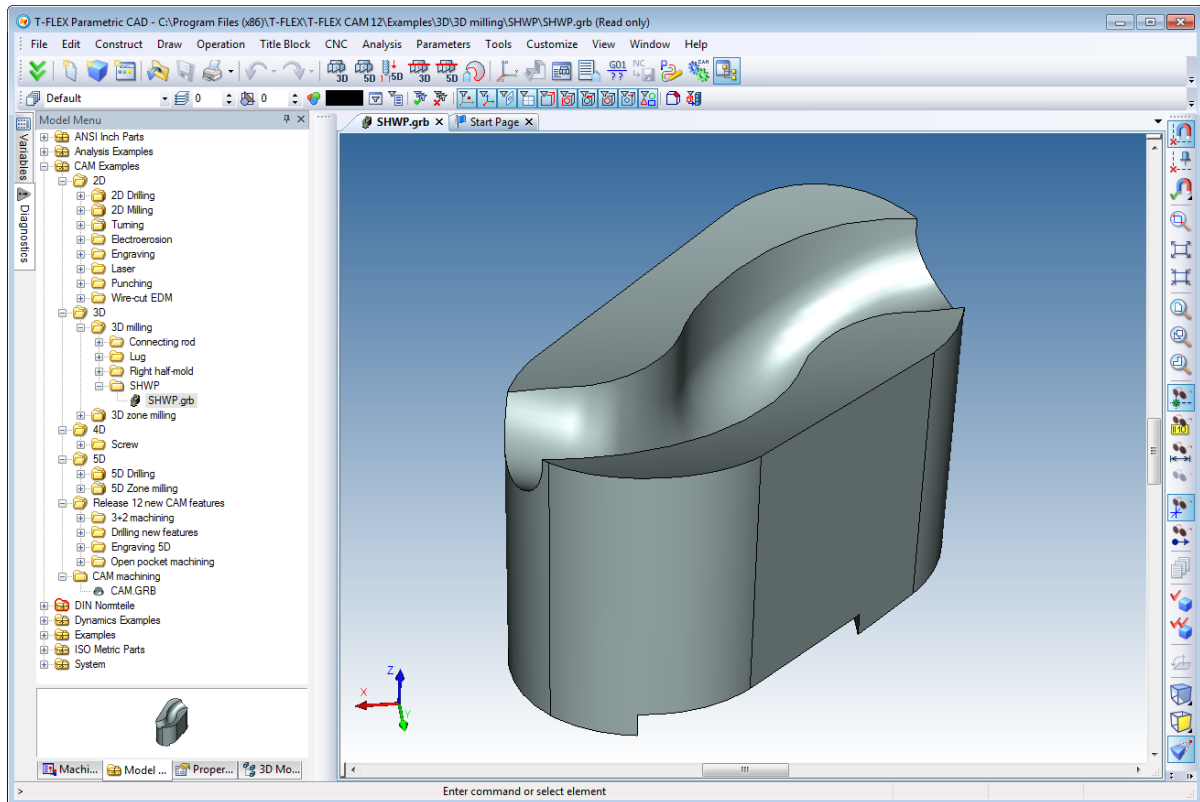
for the second hole, again use the functionality of toolpath copying and geometrical element replacement, as described above.



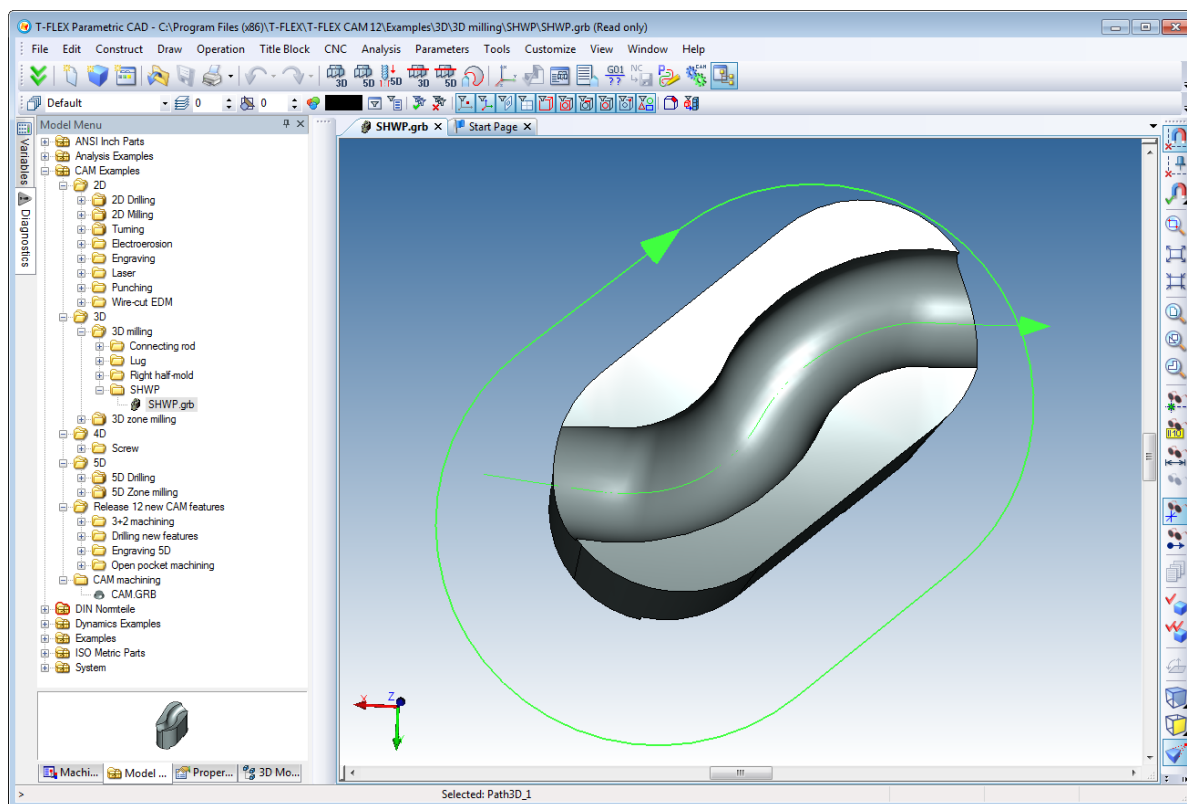
You can view the results of the performed steps using the built-in machining simulator. The control sequence can be saved using the appropriate command on the toolbar (see above for the detailed description of this machining).



3D MILLING

As an example of 3D milling machining, let's consider the SHWP part of a machining station. The 3D model of the part was built based on the 2D drawing. Normally, a table of reference points is supplied with such part drawings. Let's select from the table the points located at the "bottom" of the groove. Depending on the way of creating the model, 3D nodes at those points can be obtained as early as their construction stage. Otherwise, the first step in 3D milling toolpath creation will be constructing 3D nodes at the points mentioned above.





In the next step, we will construct a 3D path based on already created 3D nodes. The path must contain each of the nodes, and those are positioned exactly in the same order on the surface of the groove from its start to the end. Also, you can create a toolpath for forming the part's overall shape. To do this, you would need to create the projection of the part on the plane beneath it. Then, construct an offset to the common contour at a distance from it equal to the tool's radius. Next, build the 2D path as an offset, and the 3D path based on it. By using this path, we will actually get a 2D contour milling toolpath, that could also be achieved within the scope of the 2D module. Therefore, you should not regard this approach of using 3D milling capabilities as the only one and necessary.

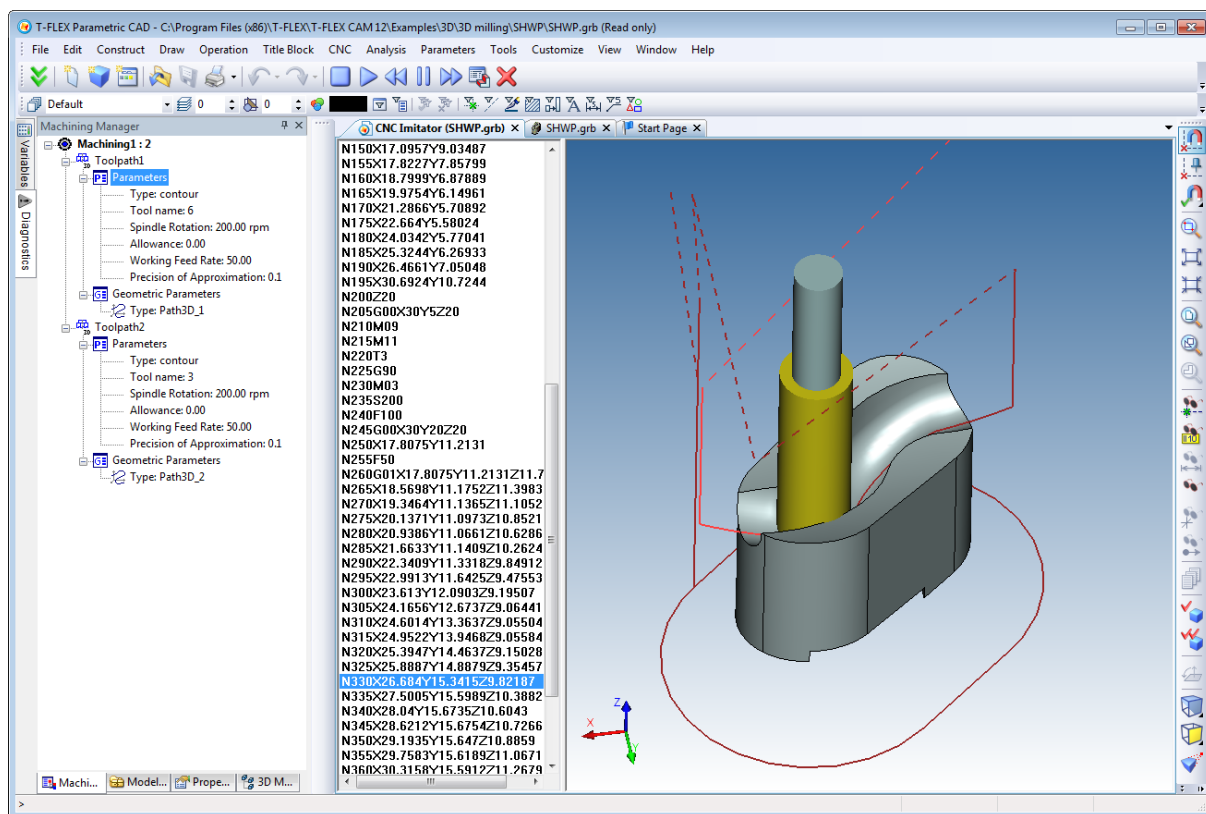



Next, let's call the 3D milling menu by clicking the  button, and in it select the "3D Contour Milling" operation . After that, a dialog window will appear with 3D contour milling parameters, and the system will also expect the user to select a 3D path for machining. Select the 3D path constructed along the bottom of the groove. Specify the feed and other machining parameters.

Another important step is to specify the path to the tool file (the method of creation of mills with the help of the tool editor is described above, in the 2D milling example) and, accordingly, the tool name from this file.

To complete the machining toolpath creation, click the  button. Graphic rendering of the machining toolpath will be displayed. Additionally, the toolpath will be added to the Machining Manager tree, whose window can be accessed by clicking the .

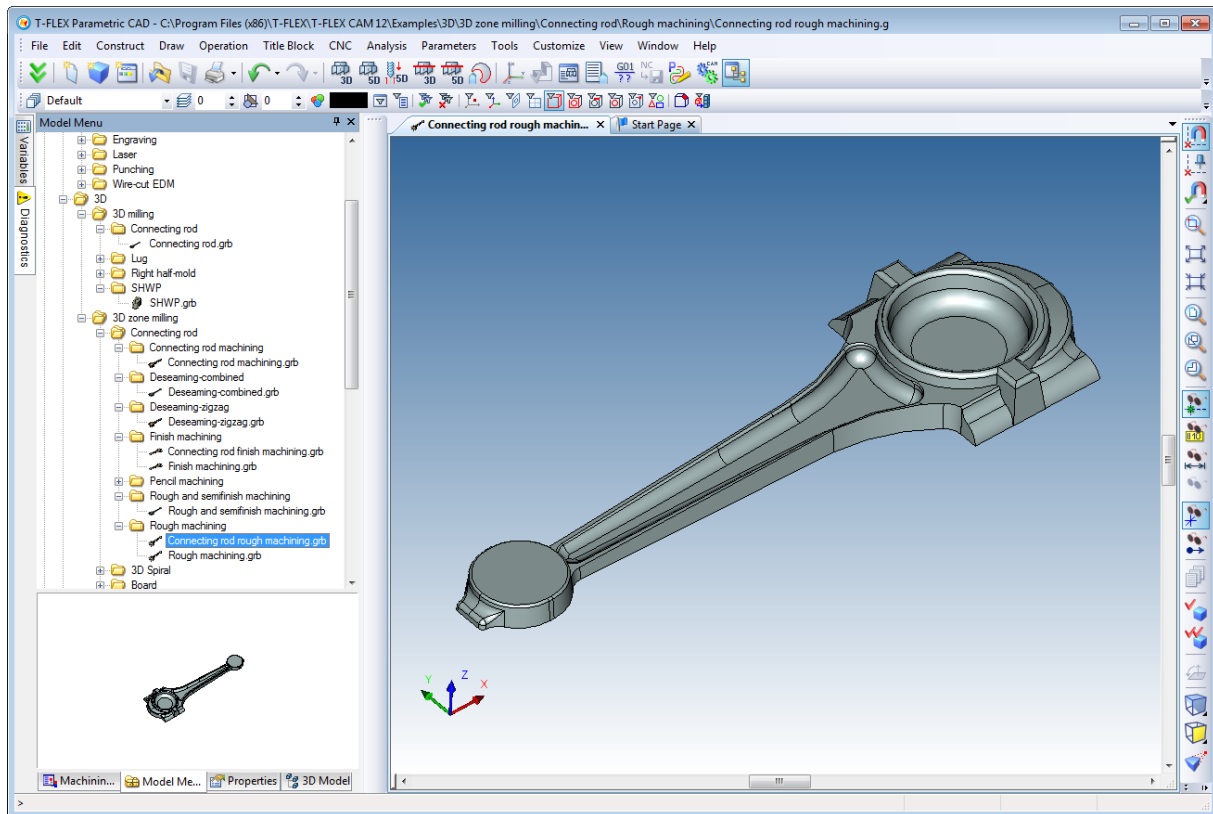
The following auxiliary tool motions can be defined by calling the context menu of some of the toolpaths in the Machining Manager window: approaches and retreats, as well as combinations thereof for specifying desired toolpath of tool transitions between operations.



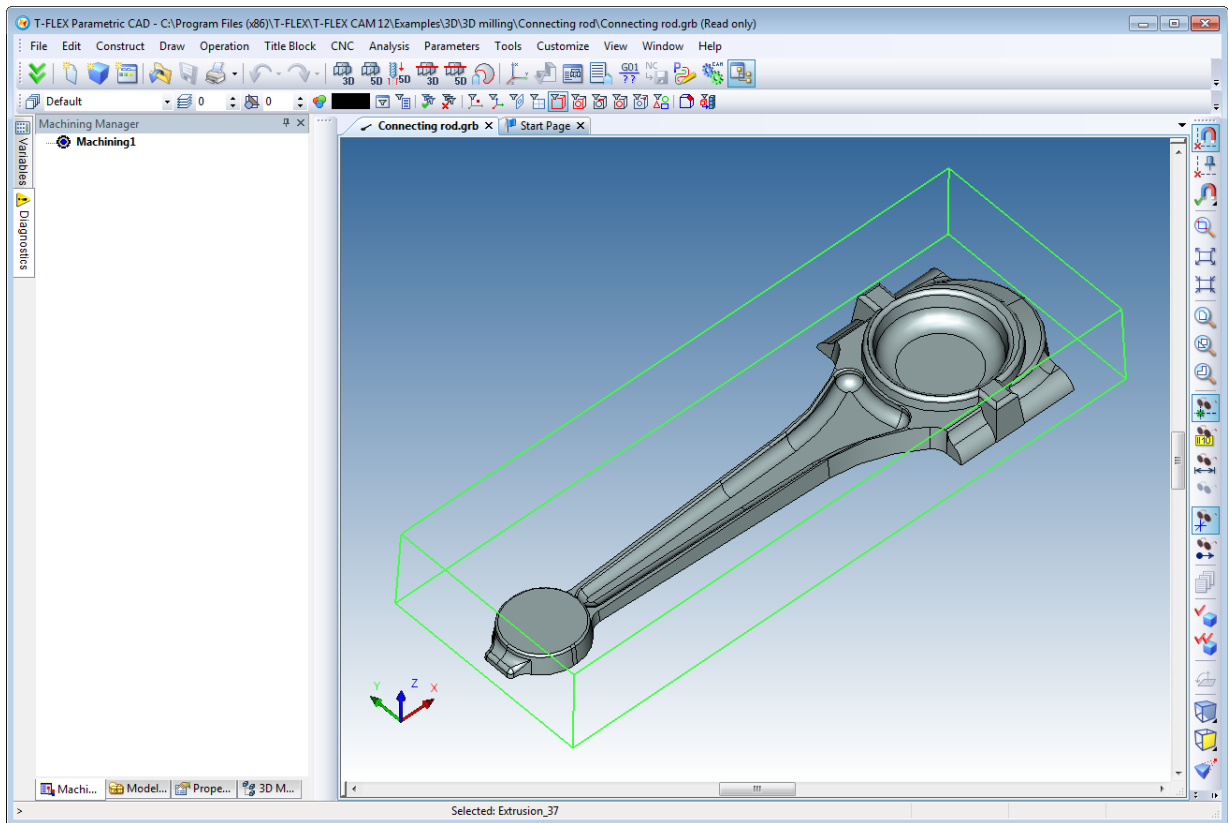
The created machining toolpaths can be viewed with the help of the built-in simulator , without stock removal.





BOUNDED MACHINING. 3D MILLING

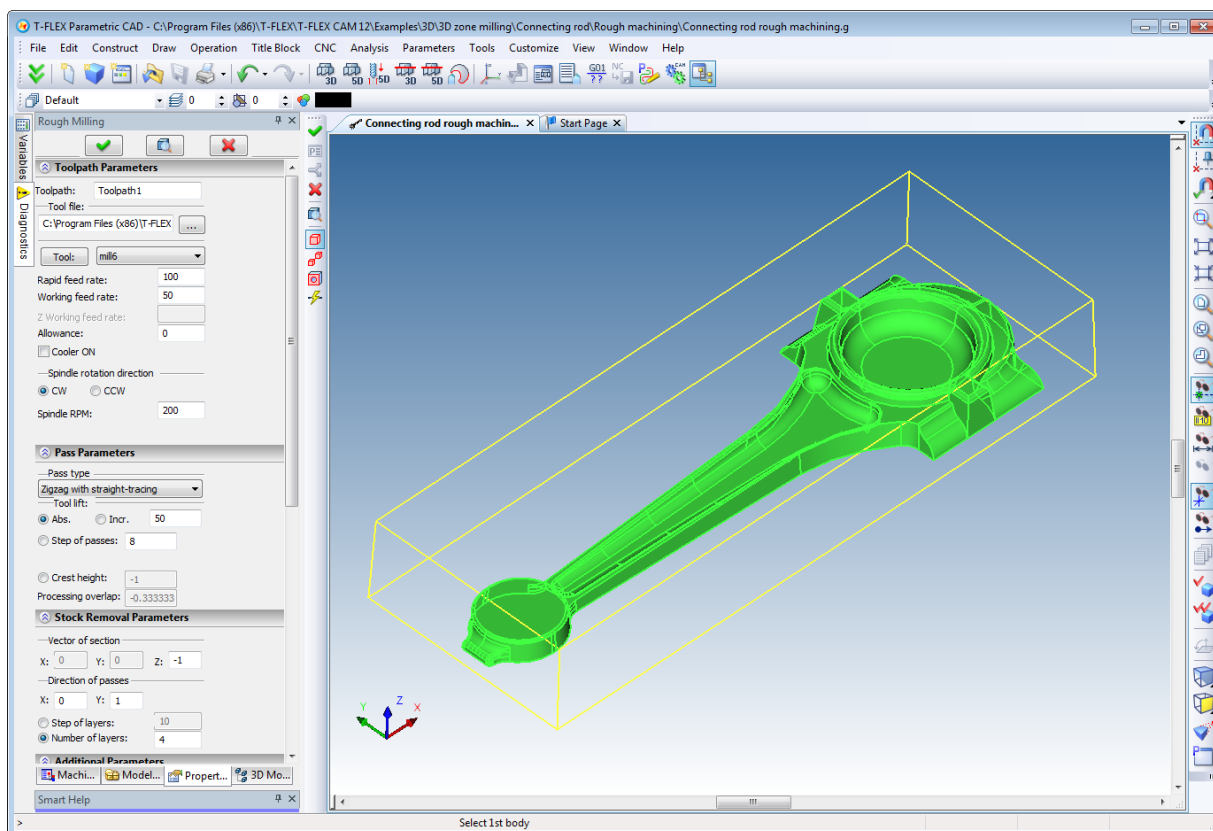
A “Rod” part, or, rather, its finishing is presented as an example. To define a bounded milling process, call the “CNC>Bounded machining>3D milling” command.





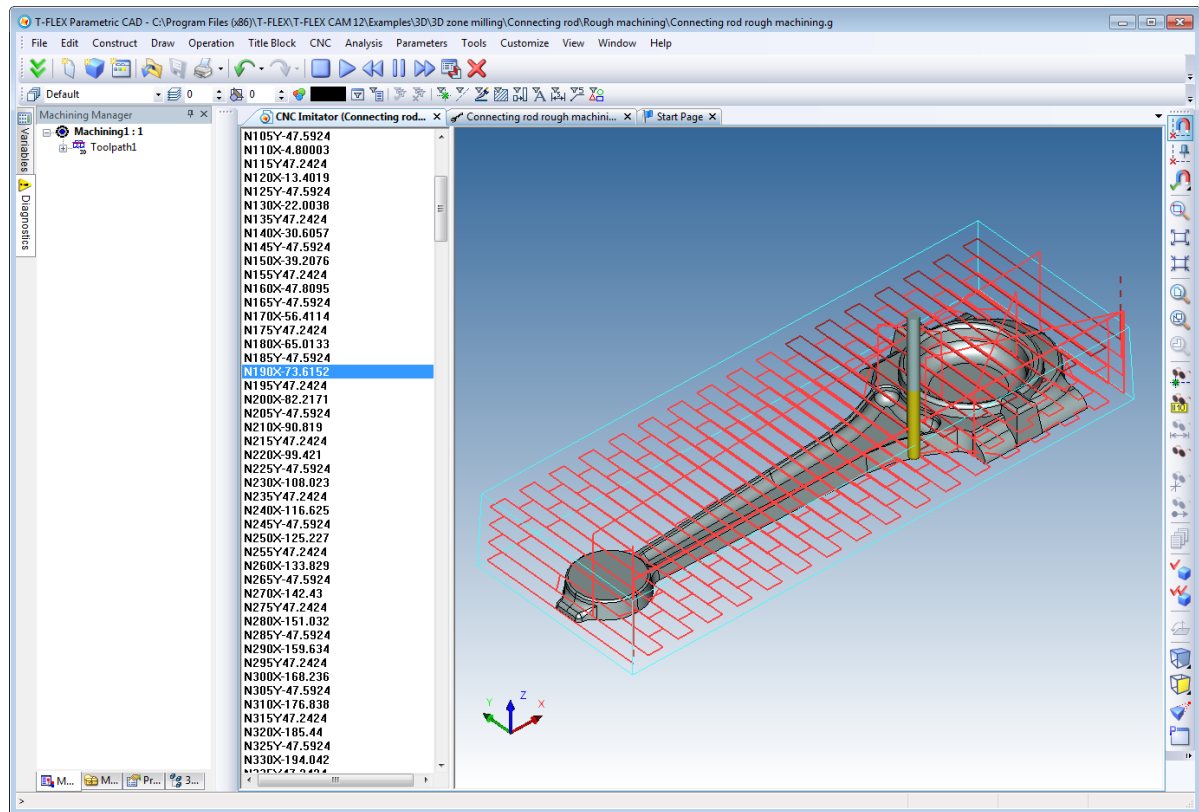
In this case, before editing machining parameters, you would need to do certain additional constructions. First, the user needs to construct the boundary of the area to be machined in the finishing mode, as shown on the figure. The zone is the second 3D body shown in the wireframe mode (which is not required), which provides the upper bound of the space around the rod and the four side bounds (the bottom face lies at the same level as the base of the part). Therefore, the rod is “fixed” within the space comprised by the box.



To create a finishing toolpath, select the option of the 3D zone milling . Upon clicking the button, a dialog window appears with 3D zone milling parameters. Additionally, as the mouse pointer is moved over the 3D scene, the system automatically activates body selection for the operation. Select one after another: first, the body to be machined, and then the body that serves as the bound of the machining zone. It is up to the user what to do first: select bodies or define machining parameters (the order does not matter). However, with the bodies selected, it becomes possible to use the option of previewing the resulting toolpath . Upon clicking the  button, the parameters window becomes locked, and the machining toolpath appears on the screen, created using the parameters entered by the user prior to calling the command. To exit the preview mode, click the  button again. The meaning and the purpose of each parameter are described in detail in the first part of the present manual. Therefore, we will leave their installation process out of the scope.



Upon defining all necessary machining parameters, just click the  button. The system will then calculate the final machining toolpath. The calculated toolpath is added to the Machining Manager's toolpaths tree, and its graphic image will appear on the screen. The tool motion along the created toolpath can be viewed in the built-in machining simulator, without stock removal. For this, click the  button.



Once the toolpaths are calculated, the user can save the resulting control sequence. For this purpose, a new postprocessor could be created in the "Postprocessor options" command, if necessary to the user. Additionally, the user can select a ready-to-use postprocessor from the library shipped with the system. The postprocessor is required for generating control sequences with a correct syntax.

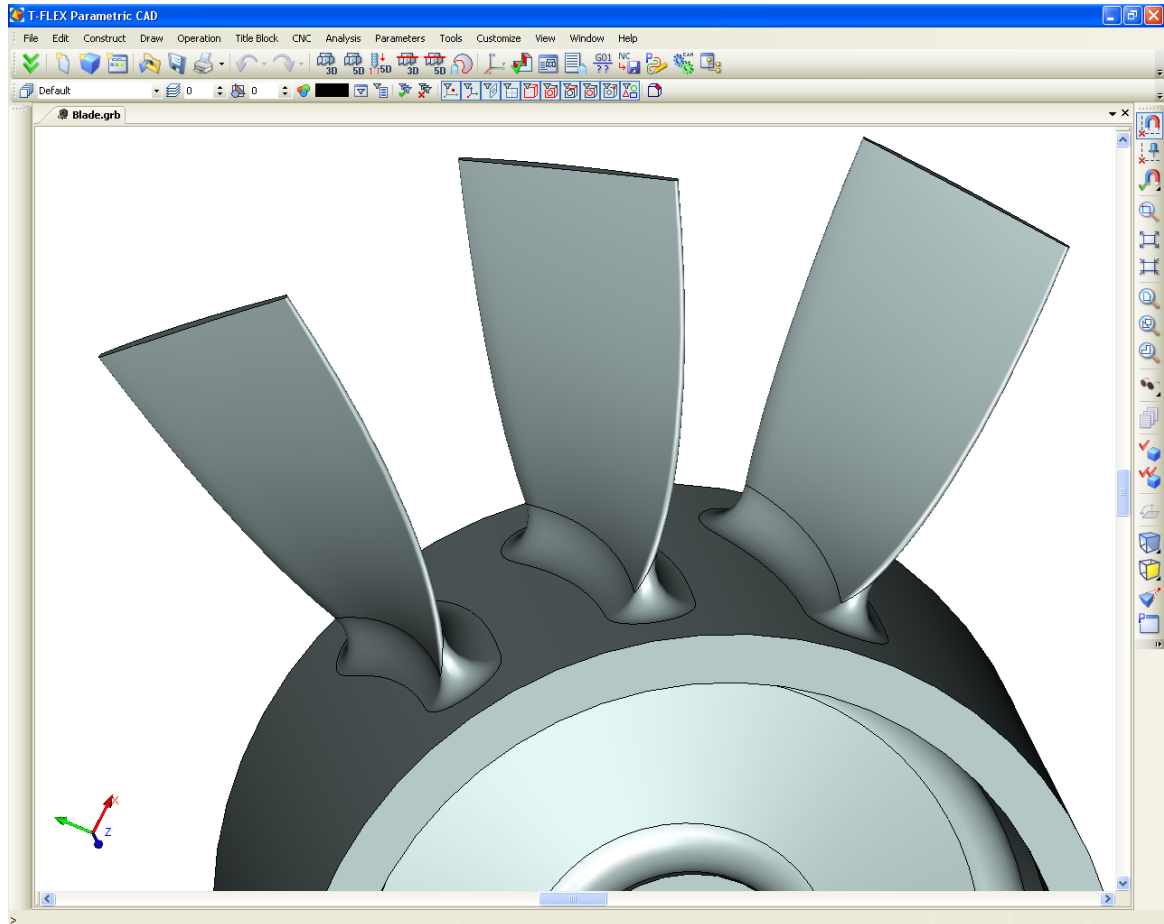
Once the desired postprocessor is selected, the user can run the control sequence generation (saving) by using the **"CNC>Save NC program"** command.

Double-click inside the window that comes up on the screen. The "Compound Toolpath Saving Parameters" dialog box will be displayed. In this dialog box, specify one by one the names of the necessary postprocessors for the selected machining type, the name of the control sequence and the location where you want to save it (see examples above).

After all steps are completed, the user can view the resulting control sequence working, with the help of the T-FLEX NC Tracer machining simulator developed by Top Systems Ltd., with stock removal. The files related to this example are provided.

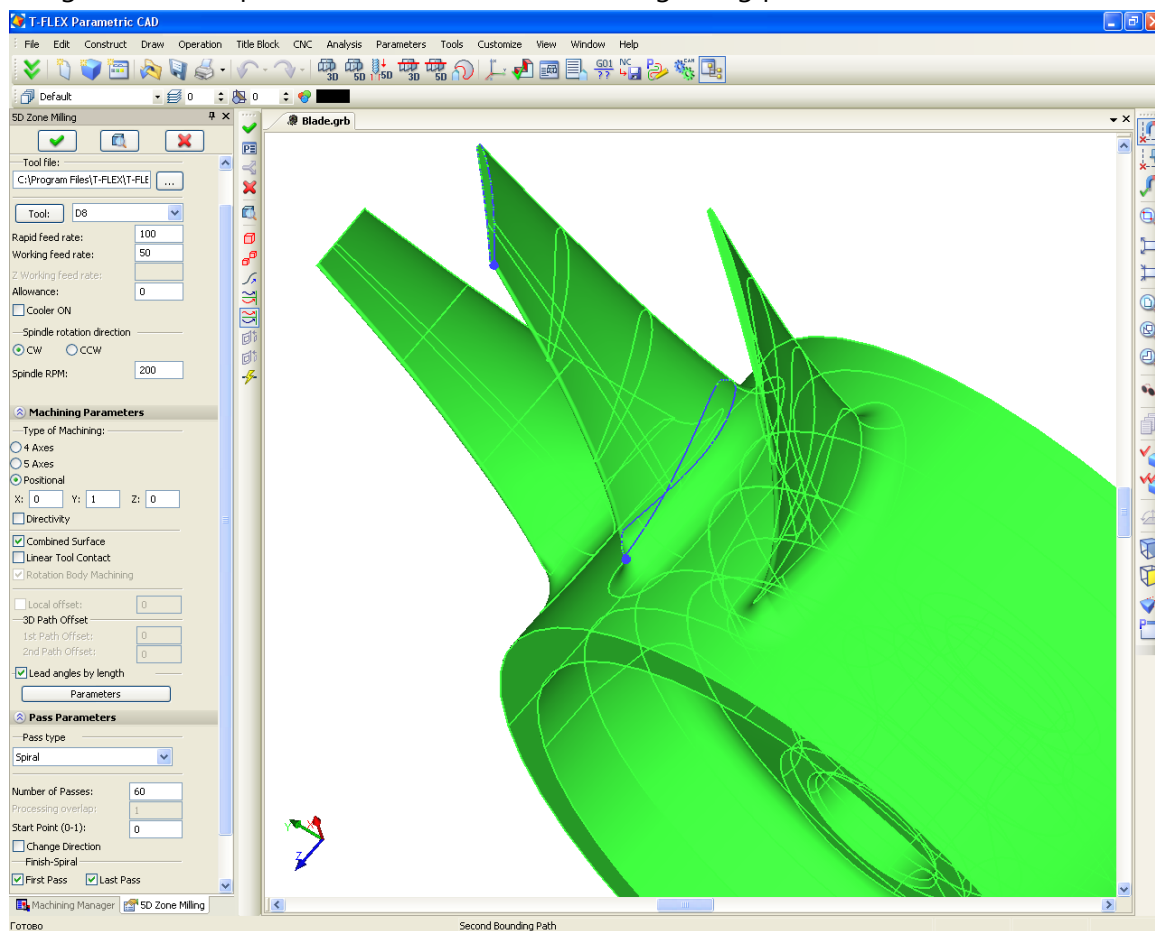
BOUNDED MACHINING. 5D-MILLING



Let's consider a "turbine rotor" part example. This kind of parts possesses spatially complex surfaces, which makes a particular concern regarding an ability to machine them. A rotor with blades of double curvature was selected as an example. Due to severe conditions the parts of this kind are subjected to, ~~these~~ often required, rehabilitation machining, realized by five-axis bounded machining. To define the bounded milling of the given sample, call the **"CNC>Bounded machining>5D - milling"** command.



When creating a finishing toolpath, you can use the "three paths" approach for this part, or, alternatively, use the system functionality of creating 5-axis milling toolpaths and create the toolpath by two paths. Since all rotor blades are identical, you need to create the toolpath for just one of them, and then, when machining the part, simply turn the rotor by the specified angle. When constructing a toolpath by three paths, the first two of them serve as bounds, while the third one – as the guide. Accordingly, the surface of the blade needs to be bounded using two paths; a third path shall be used for orienting the position of the tool in the space (limit the range of its motion and, as a result, mitigate possible collision of the tool's peripheral regions with other surfaces). All three paths must be co-directional, otherwise the system will not be able to construct the toolpath. The direction of the two passes is derived from the direction of the paths. Just like in the machining examples described above, the order of creating paths is

not significant. What matters is solely their selection order. Since we are to deal with the case of utilizing 5-axis milling, we will use parameters instead of the third, guiding path.



To calculate the toolpath of blade finishing, select 5D zone milling  from the 5D zone milling menu and set up all the required parameters in the toolpath parameters dialog window. The upper path is selected first, the lower one - second. When using the "three paths" approach, the third path shall be created above the first two and selected last. It limits the tool motion area and the tilt angle. Once the paths are selected, call the window with the angle parameters of the advancing angles. To do this, enable the "Lead angles by length" flag. After that, click the **[Parameters]** button. After defining the advancing angle parameters in this window and setting up other machining parameters in the main window, complete the toolpath creation by clicking the  button.

Lead Angles Modification

Length (%)	Angle Along	Angle Across	Allowance	Feed Rate	Slope Angle
0	35	90	0	0	0
0.35	35	90	0	0	0
0.45	35	130	0	0	0
0.6	85	90	0	0	0
0.88	85	90	0	0	0
0.92	65	60	0	0	0
0.95	35	90	0	0	0

0 35 90 0 0 0

☒ Smoothness of transition ☒ Smoothness of allowance

Parametrization Type: On number of points and search for breaking on OZ, explicit angle demar

Remove Calculate OK Cancel

Once the toolpaths are calculated, the user can save the resulting control sequence. For this purpose, a new postprocessor could be created in the "Postprocessor options" command, if necessary to the user. Additionally, the user can select a ready-to-use postprocessor from the library shipped with the system. The postprocessor is required for generating control sequences with a correct syntax.

Postprocessor Options

Machining type: 5D milling File:

Command remarks: Straight line travel with given velocity (in orthogonal coordinate system). This is the case of control that gives constant ratio between the velocities components.

Command code: Default: New :
 Linear interpolation G01

Positioning G00
 Linear interpolation G01
 Pause in milliseconds G04
 Cancels left and right radius correction. G40
 Left radius correction. G41

Additional parameters

Program header: Start number: 0
 Program Numbering step: 5

Postprocessor comment

//Developed 22.4.2009 17:9

☒ Show Extended Parameters
☐ Add spaces to output strings in NC

Save as... Save Open Close

Coordinate system

☒ Absolute coordinate system in the first frame
☐ Local coordinate system in the first frame

Cooling

☒ Turn on cooling in the first G01/G02/G03 frame
☐ Turn off cooling in the last G01/G02/G03 frame

☒ Working feed rate F in the first frame



Additional parameters

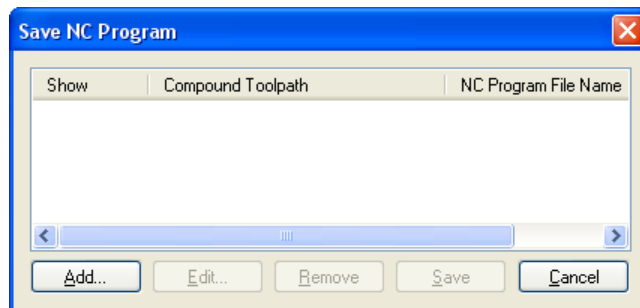
Output format

#,###

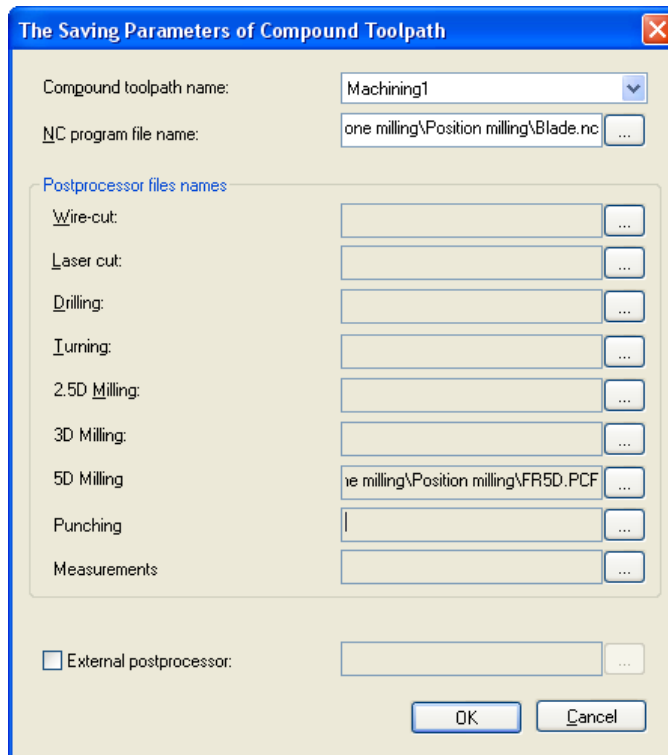
Output string format

G01 XYZ G90 M08 F << >>

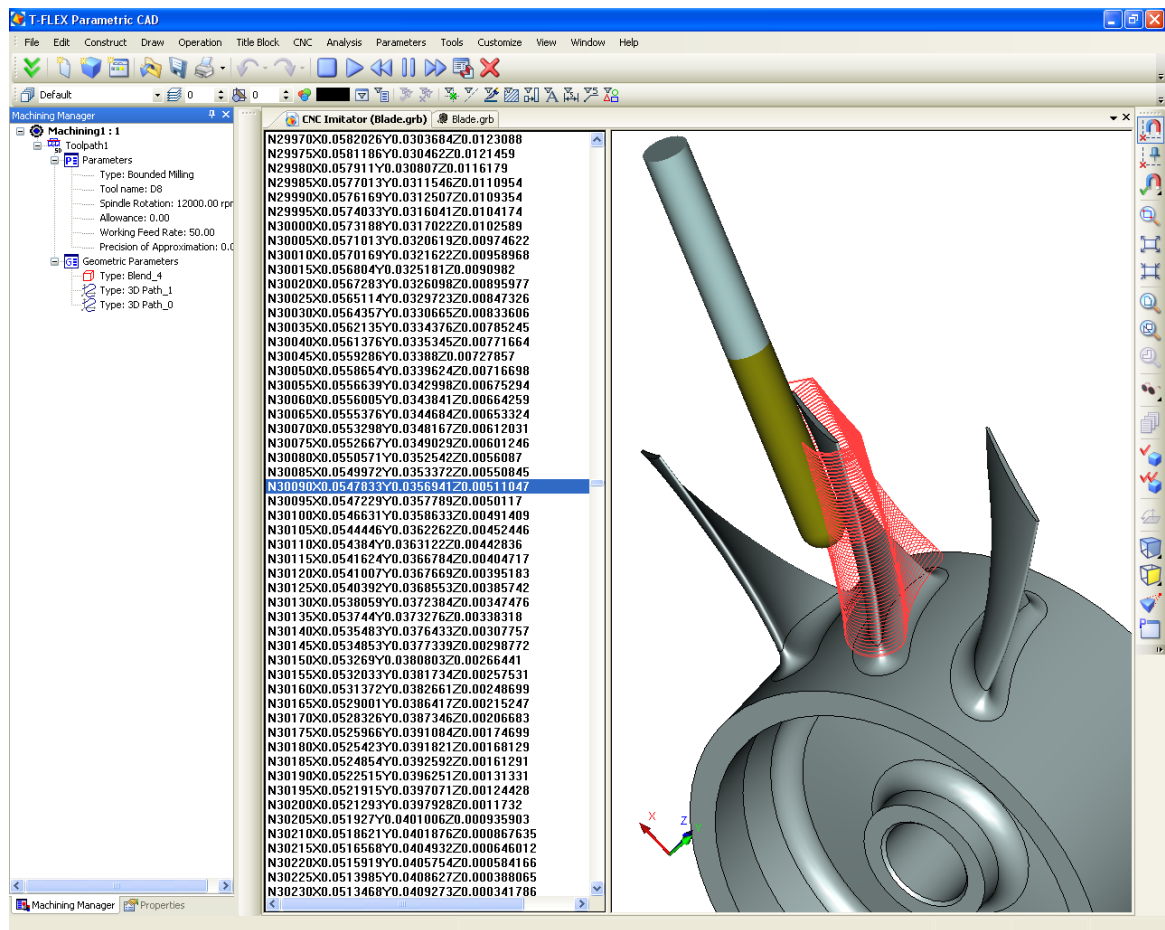
Once the desired postprocessor is selected, the user can run the control sequence generation (saving) by using the “CNC>Save NC program” command. Double-click   inside the window that comes up on the screen.



The “Compound Toolpath Saving Parameters” dialog box will be displayed. Next, in this dialog box, specify one by one the names of the necessary postprocessors for the selected machining type, the name of the control sequence and the location where you want to save it.



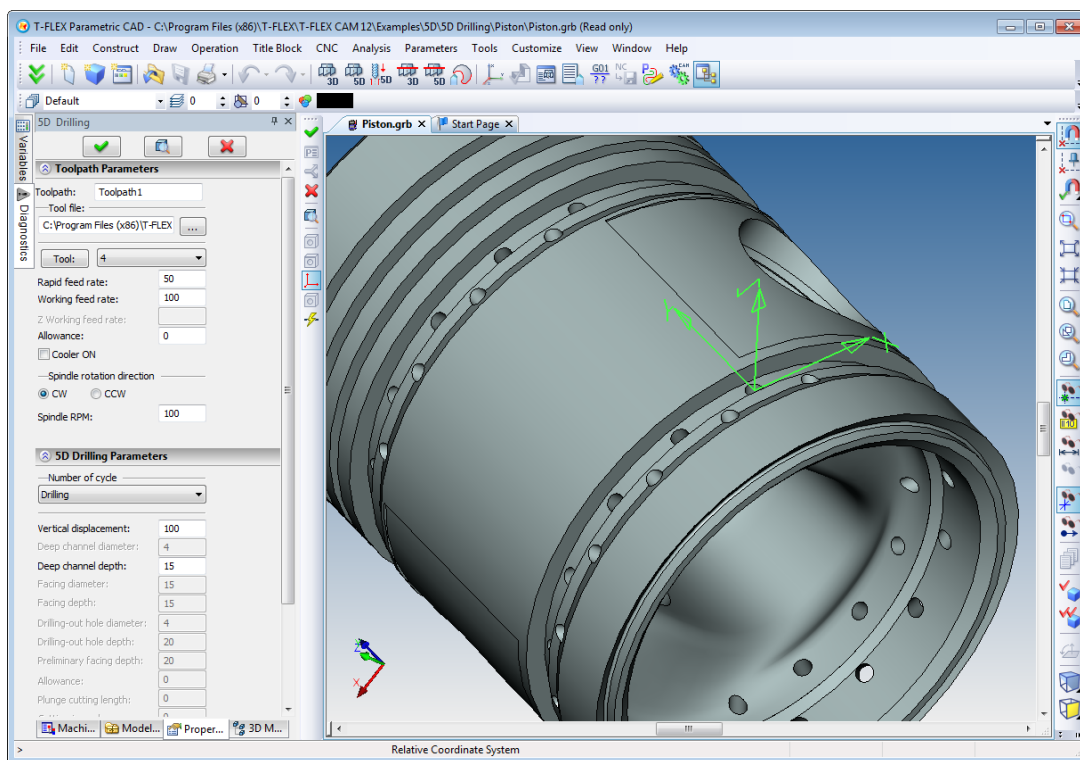
After performing all steps described above, the user can view the playback of the generated control sequence, by using the “CNC>Machining simulation” command. The user-generated control sequence will be played back in the upcoming window (working with the machining simulator and its setup process are described in detail in the respective sections of this manual and T-FLEX CAM system help).





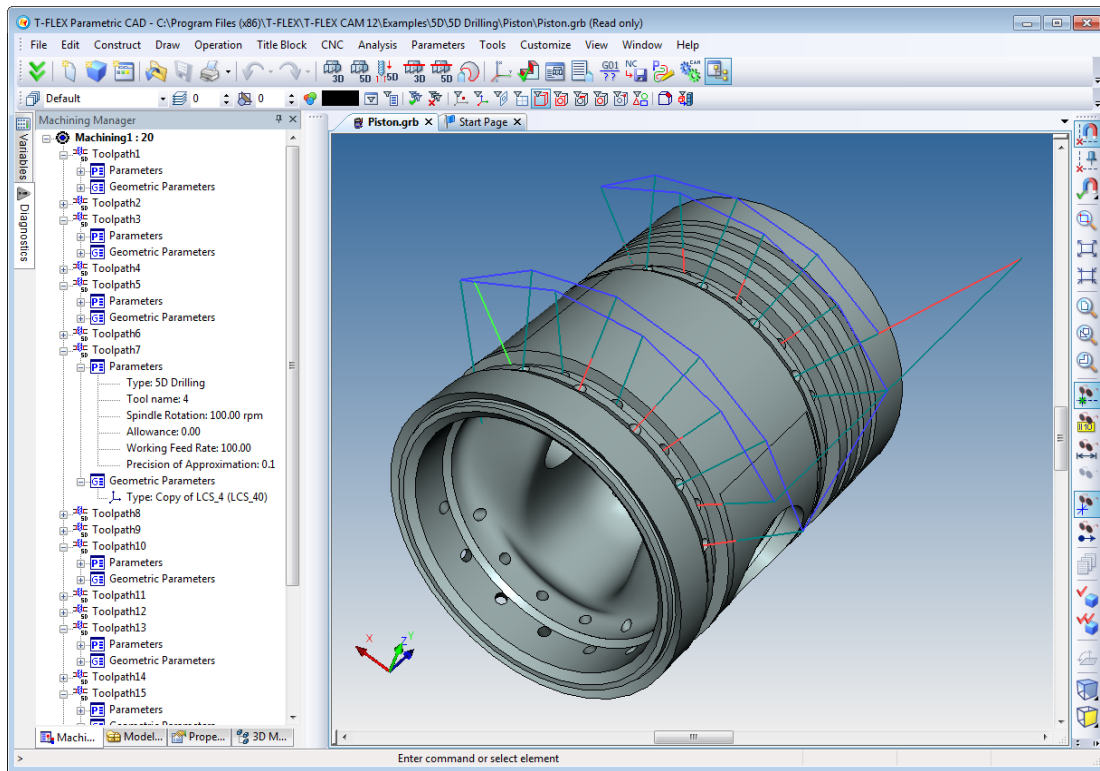
POSITION DRILLING

An internal combustion engine piston with oblique passages arranged along the circumferential is used as an example.



Before defining the machining toolpath parameters, you would need to do certain additional constructions. First, the user needs to construct a coordinate system at the reference point of the hole, using the T-FLEX CAD command "**Construct>LCS**". The reference point must be positioned at the "entrance" to the hole, while the Z- axis of the LCS must be directed towards the "exit" from the hole (towards the side opposite to the direction of the tool's entry ramp).



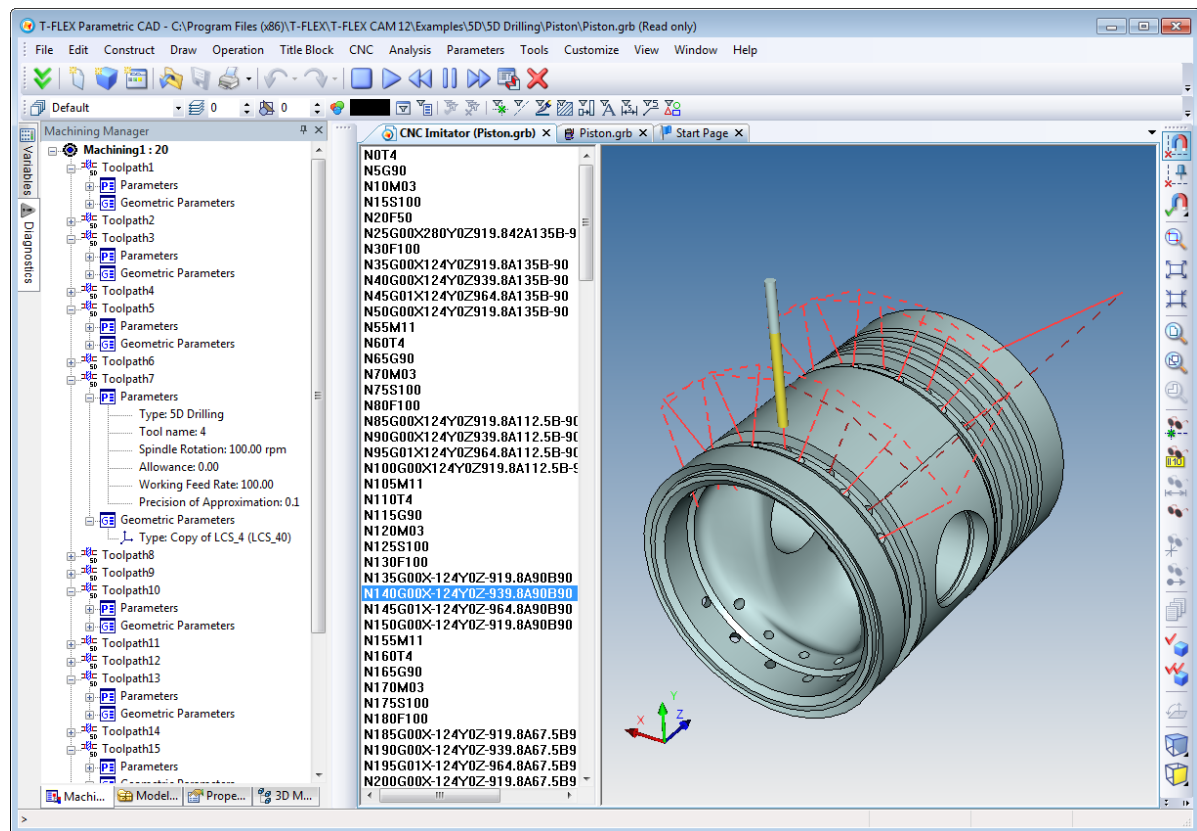
To develop a five-coordinate drilling process, call the textual menu command **"CNC>3D and 5D machining>Drilling 5D"** or use the  button on the toolbar. Upon clicking the button, a dialog window appears with 3D zone milling parameters. Additionally, as the mouse pointer is moved to the 3D scene, the system automatically activates LCS selection. 5D drilling parameters and their function were described in detail in the first part of this manual. Therefore, we will not concentrate on details of their definition. In addition to defining parameters, the user shall select an existing LCS. You can complete the toolpath creation by clicking the  button. The calculated toolpath will appear on the screen, and, at the same time, will automatically be added to the Machining Manager's toolpaths tree.



Once the toolpaths are calculated, the user can save the resulting control sequence. For this purpose, a new postprocessor could be created in the "Postprocessor options" command, if necessary to the user. Additionally, the user can select a ready-to-use postprocessor from the library shipped with the system. The postprocessor is required for generating control sequences with a correct syntax.

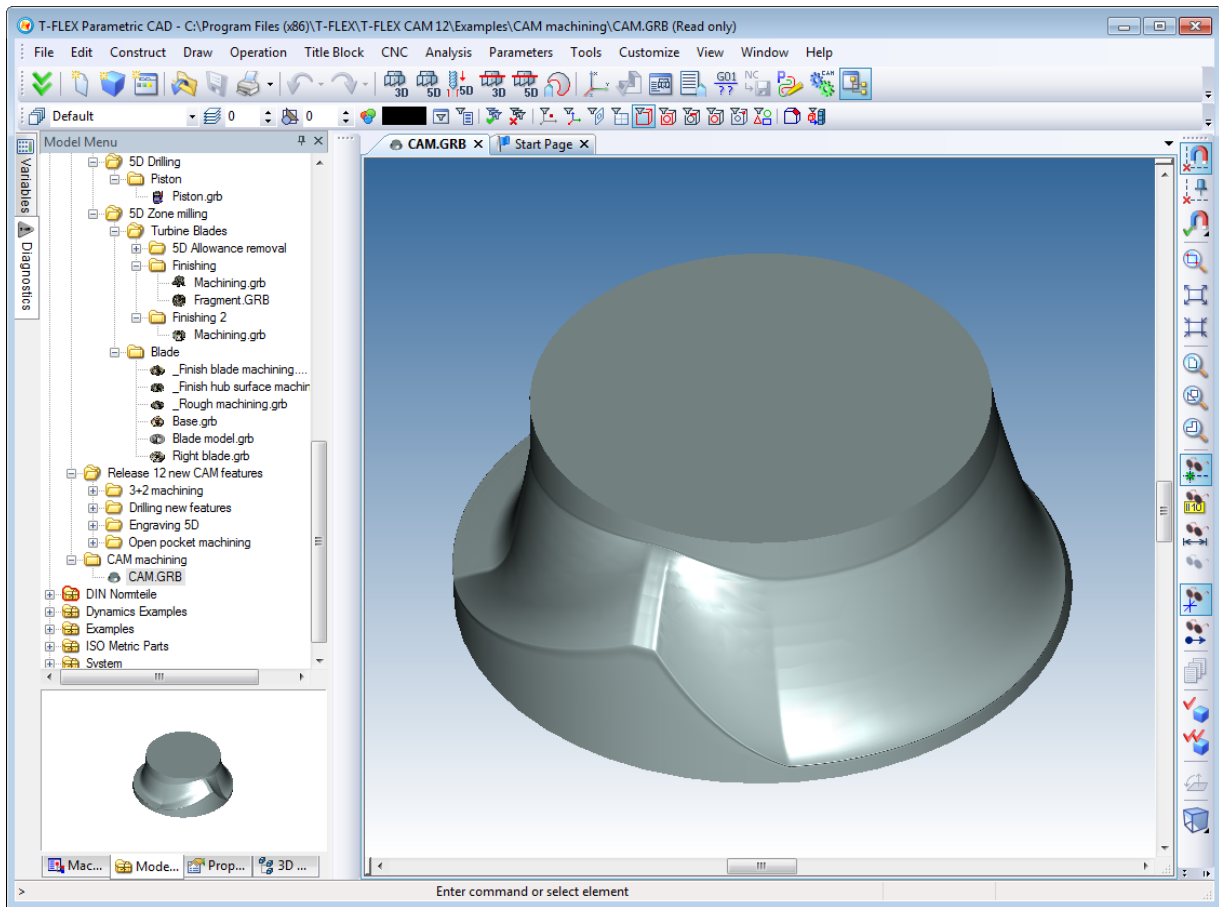
Once the desired postprocessor is selected, the user can run the control sequence generation (saving) by using the **"CNC>Save NC program"** command. Double-click   inside the window that comes up on the screen. The "Compound Toolpath Saving Parameters" dialog box will be displayed. Next, in this dialog box, specify one by one the names of the necessary postprocessors for the selected machining type, the name of the control sequence and the location where you want to save it (see the example above).



After performing all steps described above, the user can view the playback of the generated control sequence, by calling the **"CNC>Machining simulation"** command. The user-generated control sequence will be played back in the upcoming window (working with the machining simulator and its setup process are described in detail in the respective sections of this manual and T-FLEX CAM system help).




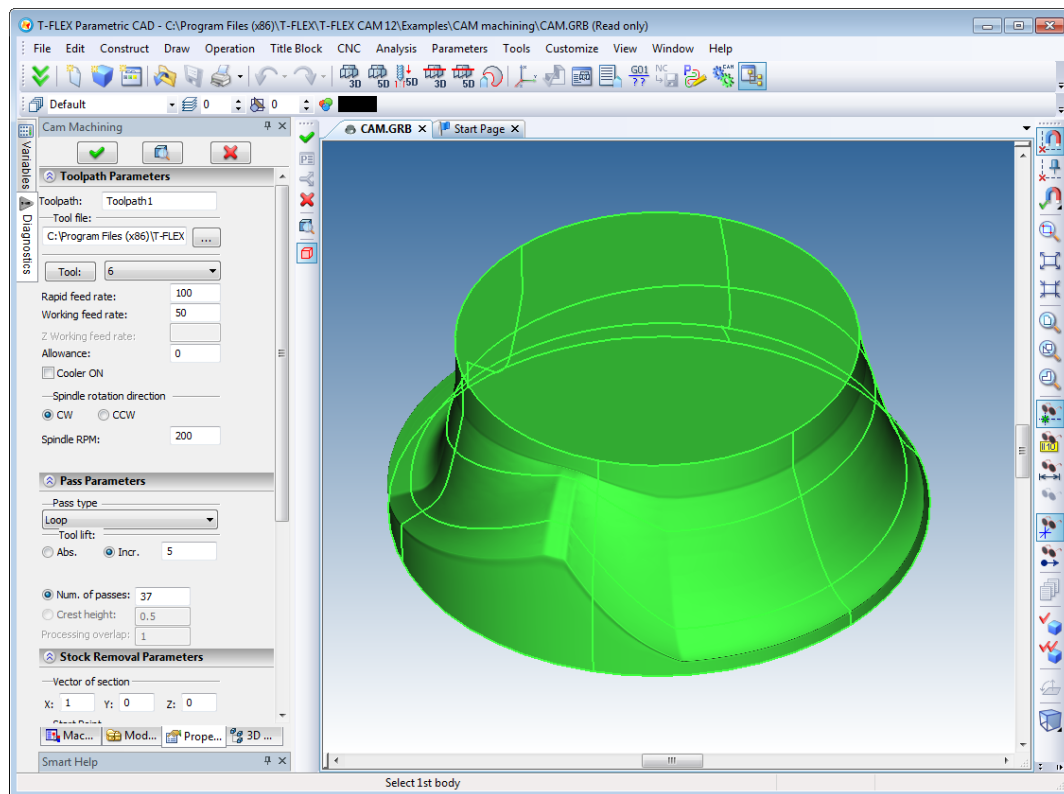
CAM MILLING (4D MACHINING)

A cam is used as an example. To define 4D milling machining of the given sample, call the “CNC>Cam milling” command.



Additionally, the user can select the Cam Milling option . In the menu that appears, select the Cam machining command . This command will bring up the dialog window of cam machining parameters, and will also activate selection of a body for machining. Note that the model of the cam being machined must form a united surface, not divided into operations. Note also that the imported profile of the cam should always be augmented to become a body of revolution.

After selecting the body and defining parameters, you can click the  button, and thus complete the toolpath creation. The created toolpath will be added to the Machining Manager's toolpaths tree, and its graphic image will appear on the screen.



Once the toolpath is calculated, the user can save the respective control sequence. To do this, first create a new postprocessor or select an existing one in the “Postprocessor options” command, which will define the control sequence.

Postprocessor Options

Machining type: 5D milling File:

Command remarks:
Straight line travel with given velocity (in orthogonal coordinate system). This is the case of control that gives constant ratio between the velocities components.

Command code: Default: New :
Linear interpolation G01

Positioning	G00
Linear interpolation	G01
Pause in milliseconds	G04
Cancels left and right radius correction.	G40
Left radius correction.	G41

Additional parameters

Program header: Start number: 0

Program Numbering step: 5

Postprocessor comment
//Developed17.8.2012 12:5

☒ Show Extended Parameters

☐ Add spaces to output strings in NC

Save as... Save Open Close

Coordinate system
☐ Absolute coordinate system in the first frame
☐ Local coordinate system in the first frame

Cooling
☐ Turn on cooling in the first G01/G02/G03 frame
☐ Turn off cooling in the last G01/G02/G03 frame

☐ Working feed rate F in the first frame

Additional parameters

Output format
#,###

Output string format
G01 XYZ << >>

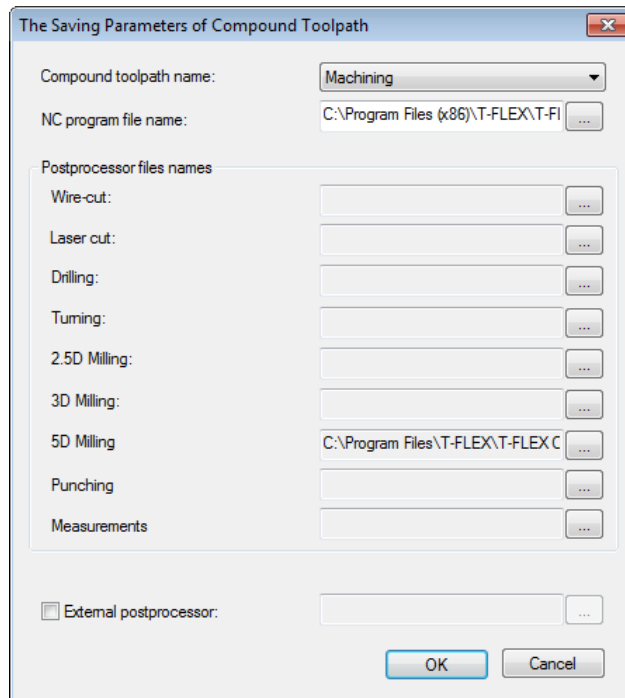
Once the desired postprocessor is selected, the user can run the control sequence generation (saving) by using the **"CNC>Save NC program"** command. Double-click inside the window that comes up on the screen. The **"Compound Toolpath Saving Parameters"** dialog box will be displayed.

Save NC Program

Show	Compound Toolpath	NC Program File Name

Add... Edit... Remove Save Cancel

In this dialog box, specify one by one the names of the necessary postprocessors for the selected machining type, the name of the control sequence and the location where you want to save it.



The dialog box is titled "The Saving Parameters of Compound Toolpath". It contains the following fields and controls:

- Compound toolpath name:** A dropdown menu with "Machining" selected.
- NC program file name:** A text field containing "C:\Program Files (x86)\T-FLEX\T-FI" and a browse button (...).
- Postprocessor files names:** A list of machining types, each with a text field and a browse button (...):
 - Wire-cut:
 - Laser cut:
 - Drilling:
 - Turning:
 - 2.5D Milling:
 - 3D Milling:
 - 5D Milling: C:\Program Files\T-FLEX\T-FLEX C
 - Punching:
 - Measurements:
- External postprocessor:** A checkbox (unchecked) and a text field with a browse button (...).
- Buttons:** "OK" and "Cancel" at the bottom right.

After performing all steps described above, the user can view the playback of the generated control sequence, by calling the **"CNC>Machining simulation"** command. The user-generated control sequence will be played back in the coming up window.

